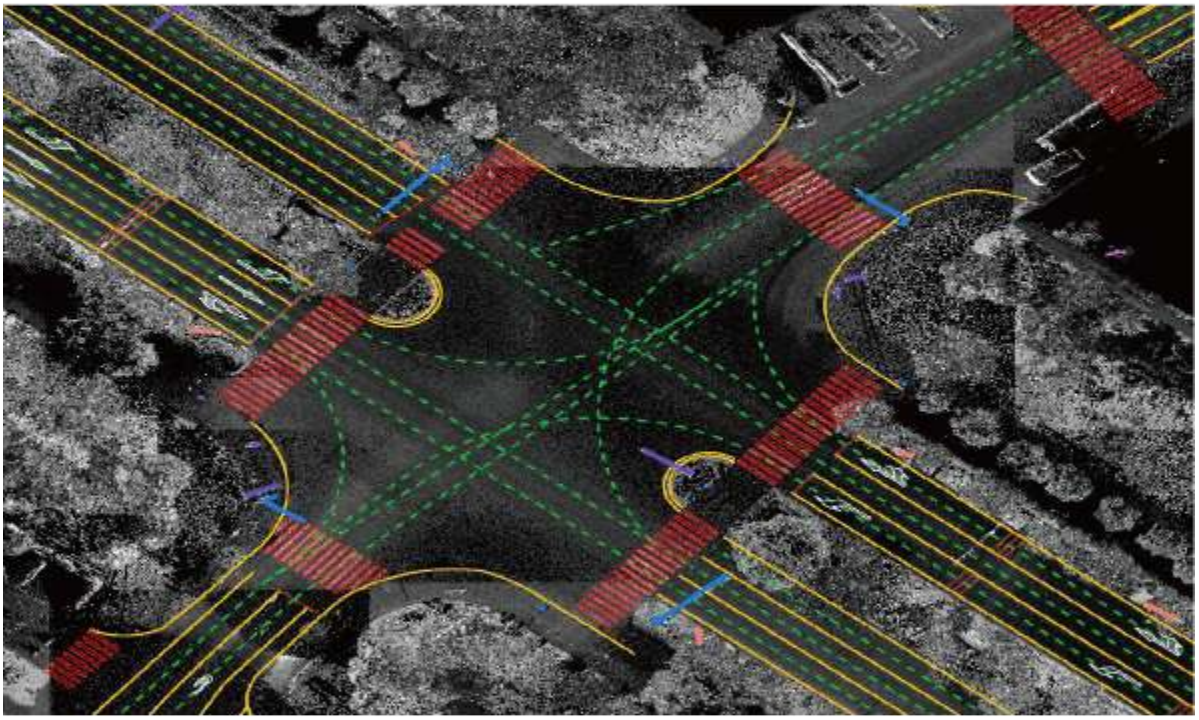
GreenValley International

LiDAR360 MLS User Guide

Terrestrial Point Cloud Feature Extraction and Analysis Software



http://www.greenvalleyintl.com

Table of Contents

Copyright

Beijing Greenvalley International Co., Ltd.

LiDAR360MLS Terrestrial Point Cloud Feature Extraction and Analysis Software

User Guide

Imprint and Version

Document Version 2.0

Published by:

Beijing GreenValley International, Ltd.

Website: https://greenvalleyintl.com

Dear Users,

Thank you for using LiDAR360MLS Terrestrial Point Cloud Feature Extraction and Analysis Software. It is a pleasure to provide you with high-precision map making and editing servicess. GreenValley International constantly strives to improve its products. We therefore appreciate all comments and suggestions for improvements concerning our software, training, and documentation. Feel free to contact us via info@greenvalleyintl.com. Thank you.

Legal Notes

Date printed: 22th October 2022

Introduction

LiDAR360 MLS Terrestrial Point Cloud Feature Extraction and Analysis Software developed by Beijing Greenvalley International Co., Ltd.

The main functions of the software include：

- File
- Tools
- Classification
- Preprocessing
- CutBlock
- MapElement
- Facility
- VectorEditor
- FacadeSurvey
- RoadAnalysis
- Attributes

Commonly used tools include：

- Profile
- Panorama
- Image
- ViewingTools
- ColorTools

Software Installation and Activation

Operation Environment

We recommend the use of a high-performance workstation, the configuration requirements are as follows：

- **RAM**：16GB and above。
- **CPU**：Intel® Core ™ i5 / i7 is recommended; eight-core and sixteen-thread processors with single-core processing performance of 4GHz and above.
- **Hard disk**：SSD is recommended with a transmission speed of 100M/s or above.
- **Display adapter**： NVIDIA graphics card is recommended, video memory no less than 8GB.
- **Operating system**：Microsoft Windows 7 (64-bit), Microsoft Windows 8 (64-bit), Microsoft Windows 10 (64-bit) or Windows Server 2012 and above.

> Note: For Windows 8 and Windows 10 system, if the software is installed in the system disk, you need to run it in administrator mode. Note: Please use the high-performance graphics mode to run the software, the operation steps are as follows High-Performance Graphics Mode.

Installation

1.Launch LiDAR360 MLS installation wizard.

2.Click next step when the installation dialog appears.

3.Click "I agree" to proceed if you agree with the license aggreement.

4.Choose installation path (or accept default path), and click "install".

5.Click "Finish".

License

LiDAR360 MLS accepts two kinds of licenses: Hard lock and soft lock. Hard lock provides USB drive while soft lock provides license key. As for hard lock, users should keep it safe and are not allowed to performe some operations including format/delete/copy.

1）Hard lock license

Currently, LiDAR360 MLS can detect hard lock license at real time. Users need to plug in the USB drive into the port.

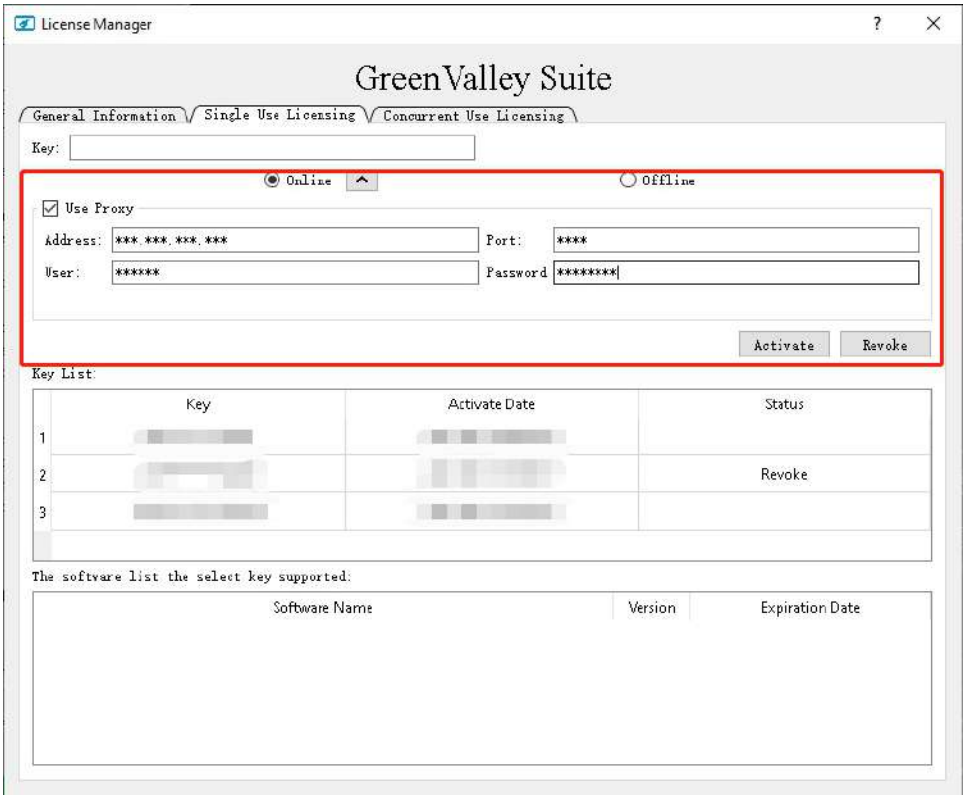
2）License key

The authorization code is generated based on the activation information provided by the LiDAR360 MLS user. After purchasing the authorization code, follow the steps below to activate LiDAR360 MLS.

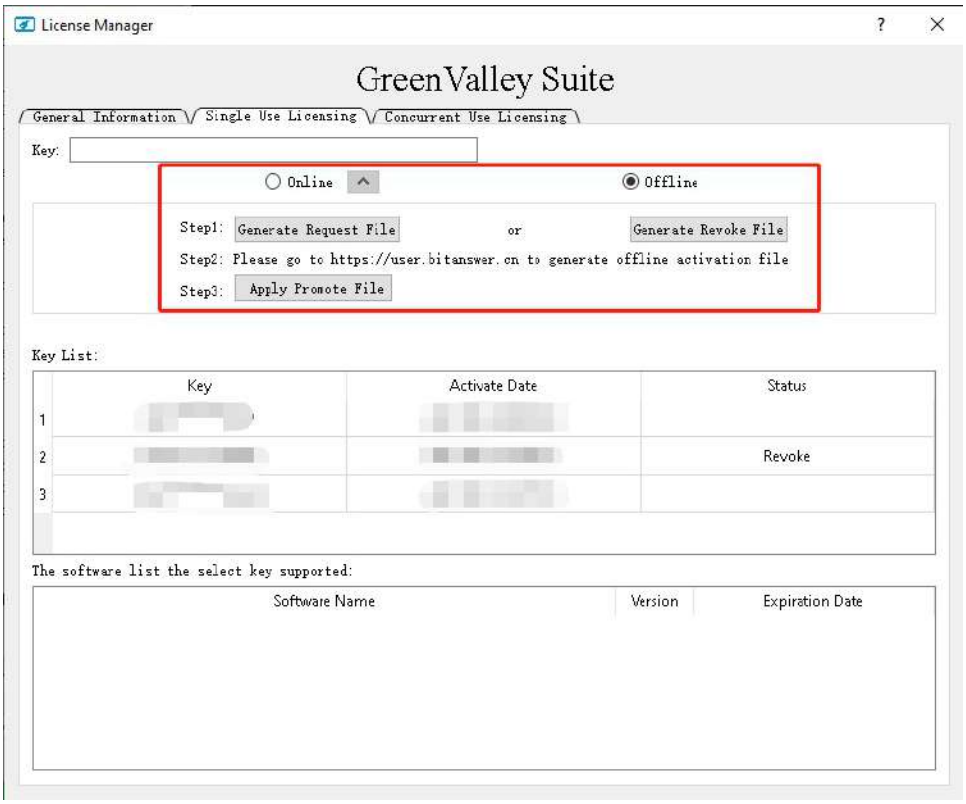
1. Run LiDAR360 MLS, the license management dialog box will appear.
2. Fill in the name and company name, select the module that needs to be activated, and then click "Copy".
3. Paste the copied information into the body of the email and send it to info@lidar360.com.
4. The software authorization mode is divided into stand-alone authorization and group authorization. After receiving the authorization code, it can be activated and moved out online or offline.

- Single Use License
  - Activation/Update

    Online Activate/Update: When having networking, enter the key under Single Use License tab, select "Online", and click "Activate", Or select the authorization code in the Key list, right-click, and select "Activate Key" from the pop-up menu. Users can check valid date in basic information module. If you need to use proxity, click  , and set address, port, user name and password of the proxy in the corresponding blanks.



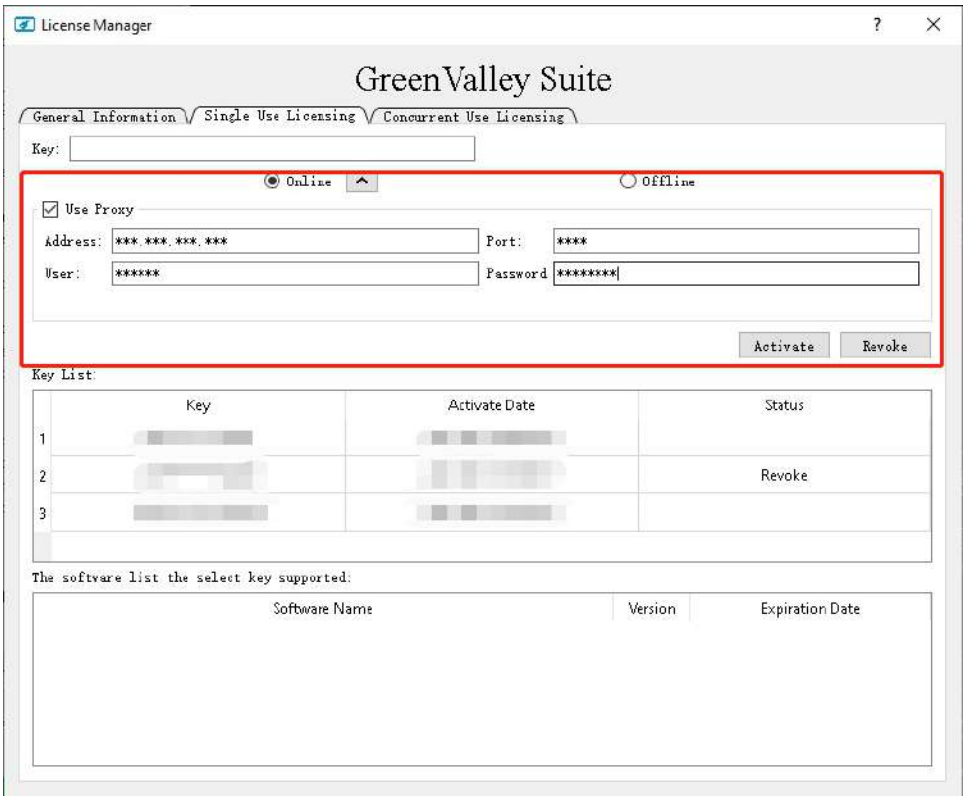
Offline Activatin/Update: Enter the key under Single Use License tab, select "Offline", and click"Generate request file"(.req). Use a computern with internet access, type https://user.bitanswer.cn in an internet browser, enter the license key to log in and click "Offline update". Upload .req file, and then a file named as "download.upd" will be generated. Download the "download.upd" file and click "Apply license file" in License manager page to activate/update the license.



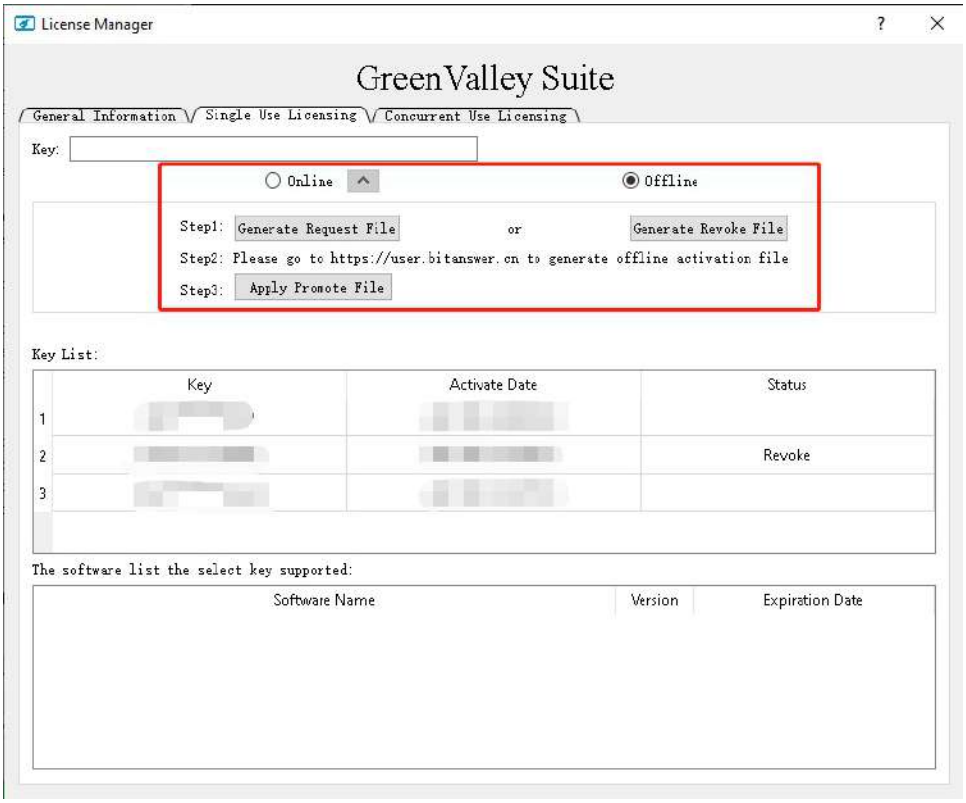
- Revoke

  When the user needs to unbind the authorization code and the machine, the authorization code can be moved out online or offline. After the authorization code is moved out, it can be used on this machine and other machines by reactivating it, and the authorization code can be re-entered to activate.

  Online migration: In the "Single-machine authorization" tab, enter the authorization code, select "Online", click "Move out", or select the authorization code in the Key list, click the right mouse button, and select "Move out key" in the pop-up menu . Click ^ to use the proxy to set the address, port, user name and password.

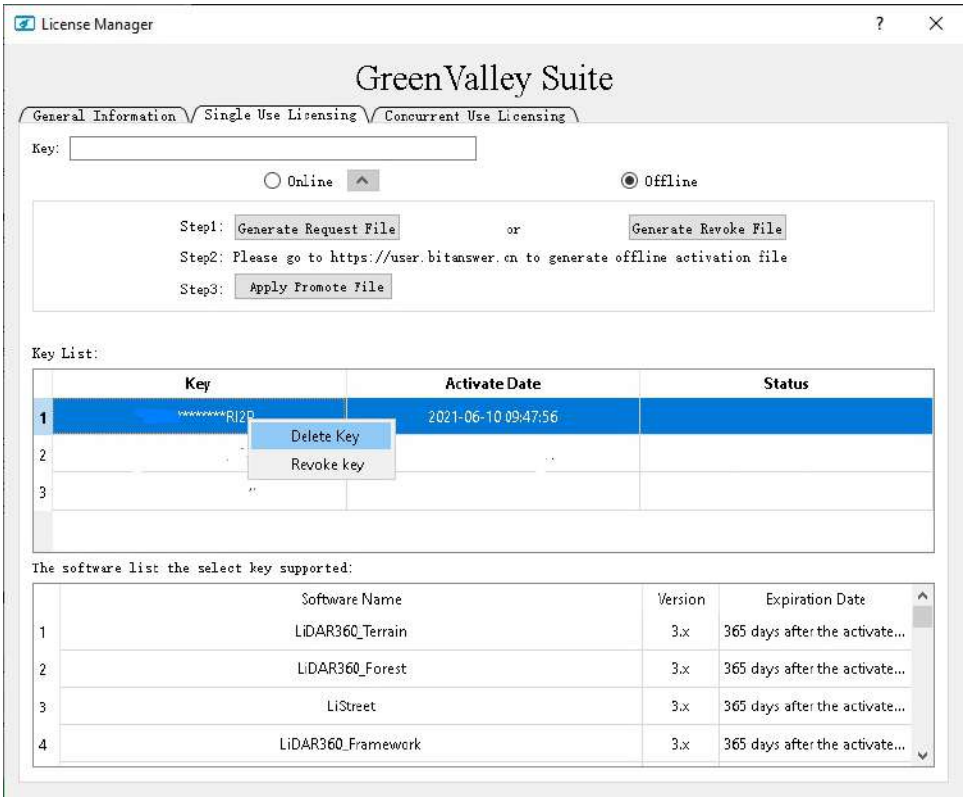


Revoke offline: Enter the authorization code, select "Offline", click "Generate Move Out File" to generate move out request file (.req). With the help of a computer that can be connected to the Internet, enter https://user.bitanswer.cn in the browser, enter the authorization code to log in, click Offline Upgrade, upload the request file (.req), and download the generated upgrade file (.upd). On the license management interface, click "Apply License File".



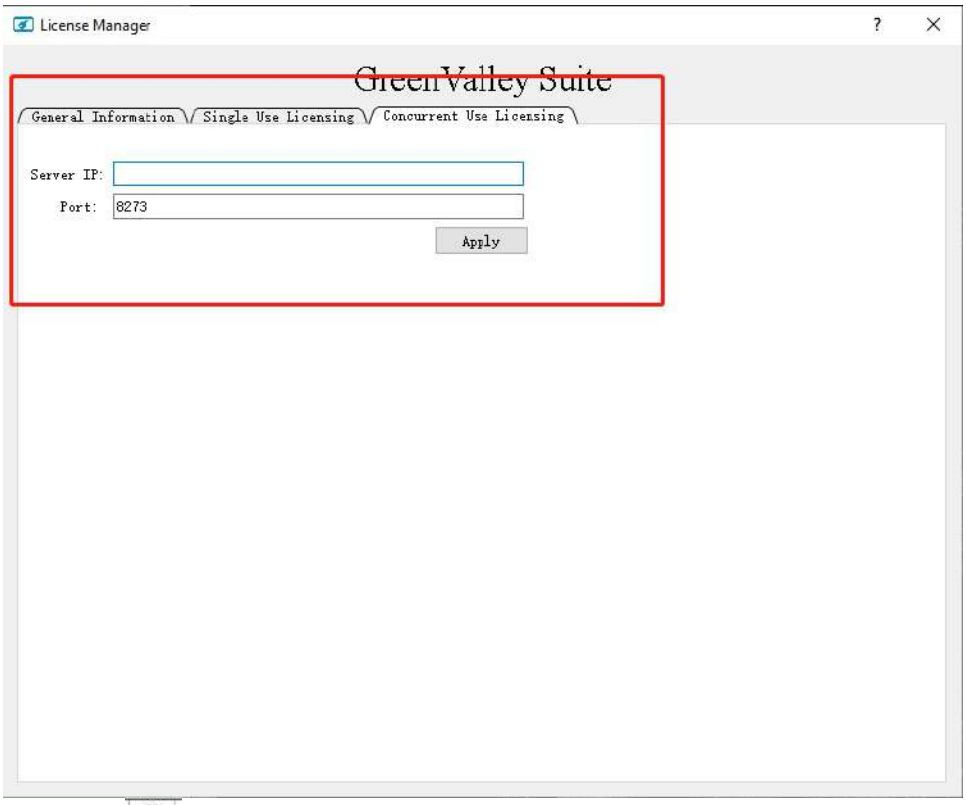
- Delete

When the user needs to delete the authorization information from the machine, click the right mouse button on the authorization code and select delete key. After the authorization code is deleted, it can be activated again for use on this machine, but cannot be used on other machines.



- Group authorization

Install the group authorization service tool on the group server in the local area network and add the group service extension module. In the group authorization management center, activate the authorization code online or offline. Other users in the local area network enter the server's IP address and port on the license activation interface The default is 8273, no need to modify, click "Apply".



6.Click the help button in the upper right corner of the license management interface to view the license management Help manual.

> Note:
>
> 1.If the software is already open when the authorization code is updated, please restart the software after the update.
>
> 2.If the authorization code has already been used on one of the machines, and now you want to use it on the other machine, you should first move the authorization code out on the first machine. If the activation code has been deleted, you should activate it on this machine first, and then move out.
>
> 3.Please contact info@lidar360.com to inquire and purchase a license key to activate LiDAR360 MLS.

Changing Programme Language

The software currently provides two languages: English, Chinese. Users can switch according to their needs. The switching steps are as follows:

1.Click **Display > Language > English, Chinese**.

2.Click "Yes" to restart the software and complete the software language switch. After selecting "Cancel", the software will not restart, it will be displayed in the set language the next time the software is started.

File

As shown in the figure below, the functions included in the file page are:

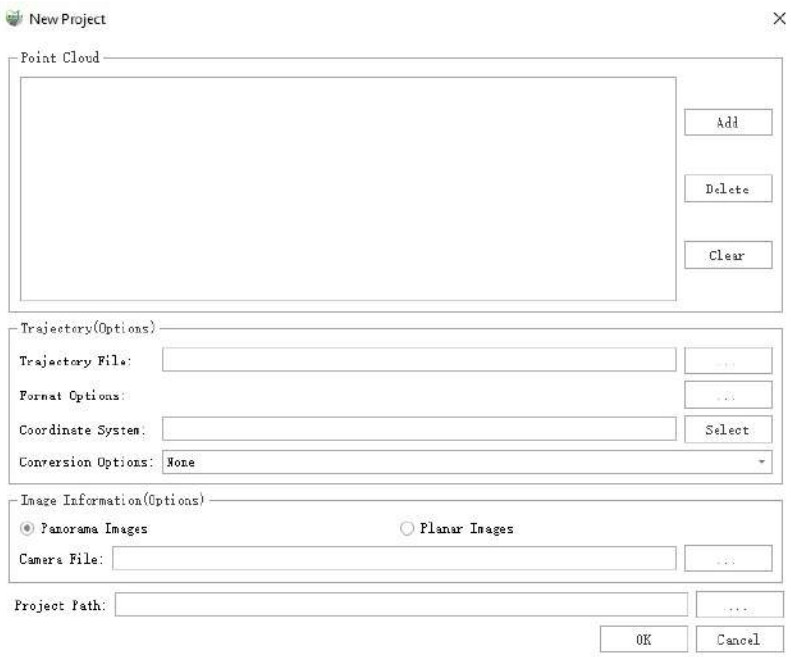
- New Project
- NewFromTemplate
- Open Project
- Modify Project
- Save Project
- Save As Project
- Close Project
- Option
- About
- License
- Help
- Exit

New Project

Function Description: Allows the user to create a new project in LiDAR360MLS. The project is created will be based on point cloud, track, image and other data.

Steps

After starting the software, click the **New Project** button to open the new project wizard configuration original data page:



> Before data processing, try to consolidate the collected data to a fixed directory on the local disk of the computer. It is recommended to use English and numeric characters for directory names, such as "LiDARData". The new project wizard mainly includes adding point cloud data, trajectory data (optional), panorama/plane image data (optional), and setting the project file path.

Set Point Cloud File Path

> Software can support ***.LiData**、 ***.las**、 ***.laz**、 ***.ply**、 ***.e57** format point cloud file.

- **Add**: Add point cloud data file, support multiple selection
- **Delete**: Delete the point cloud file path selected on the left
- **Clear**: Clear all point cloud file paths

Set Track File Path

> Software supports ***.traj**、 ***.pos**、 ***.txt**、 ***.csv**、 ***.asc**、 ***.xyz** 、 ***.pts** format trace files.

- **Track file**: The user can directly enter the track path in the edit box, or you can select the track file through the browse button on the right.
- **Track format selection**: The user can choose to convert the track file format, including:

- **None:** No format conversion
- **Original system and target system:** Select the time format of the input track file and the output time format, and select the measurement date for conversion
- **Multiplication and addition constants:** Linear transformation of time is performed using the formula, output time=a * input time + b
- **Heading angle adjustment:** Meridian convergence angle conversion for trajectory

- **Coordinate System**: If the point cloud data file has coordinate system information, the coordinate system information of the point cloud data file will be filled in. The coordinate system can also be set manually.
- **Conversion option**: The conversion option of track coordinate system, the user can choose whether to use seven parameters can be selected.

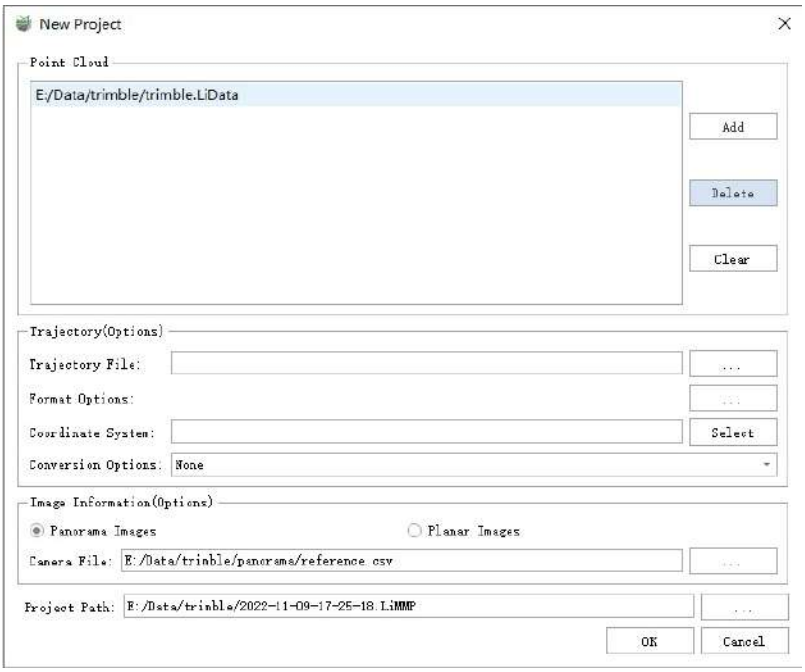
Set Camera File Path

- **Image information**: Supports panoramic image and planar images.

Panoramic Image

- **Camera file**: Load panoramic image file

> Software can support *.imglist, Leica Pegasus File* .csv, Trimble MX9 File *.csv, OrbitPos* .txt files



Select panoramic image

Planar Image

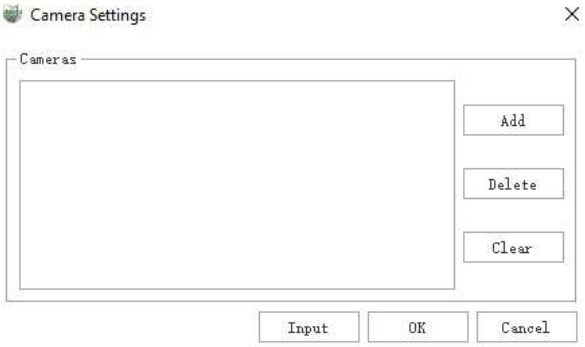
- **Camera file**: Fill in the flat image file. Only files with the *.imglist format are supported
- **Camera parameters**: The user can select their own calibration file(*.cal) to import as parameters, or can select the “Add” button in the camera parameters to manually add the calibration parameters of the plane camera. The user will need to fill in the camera ID, camera internal parameters, axial angle of the mounting structure and calibration parameters.
  - **Camera ID**: When the device has multiple cameras, the camera ID is accumulated and corresponds to the CameraName field in the *.imglist file

- **Camera internal parameter**: f is the length of focal length; cx, cy are the offset of the optical axis from the coordinate center of the projection plane; Wherein, k1, k2, k3 are the coefficients of Taylor series expansion, which are used to correct the radial distortion caused by the lens; p1 and p2 are used to correct the distortion caused by the tangential distortion of the lens; b1 and b2 are generally not required
- **Installation Structure (Axis)**: Angle value of camera rotation to IMU coordinate system around z, x, Y axes
- **Calibration**: X, Y, Z are the arm values of the camera, roll, pitch, heading are the camera placement angles

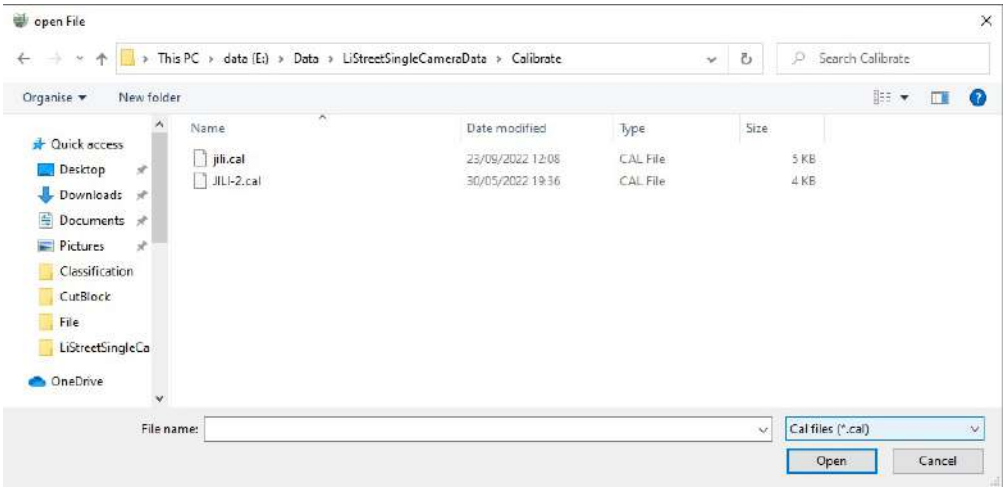
Steps

1.Select a planar image and add an image file.

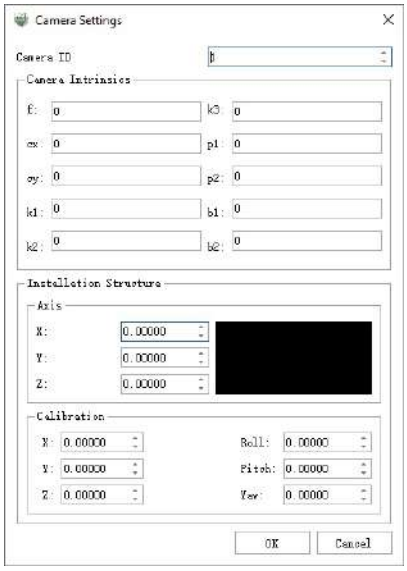
2.Click **Settings** to pop up the **Camera Settings** dialog box.



3.Calibration files(*.cal) can be imported directly or calibration parameters for flat cameras can be added manually.

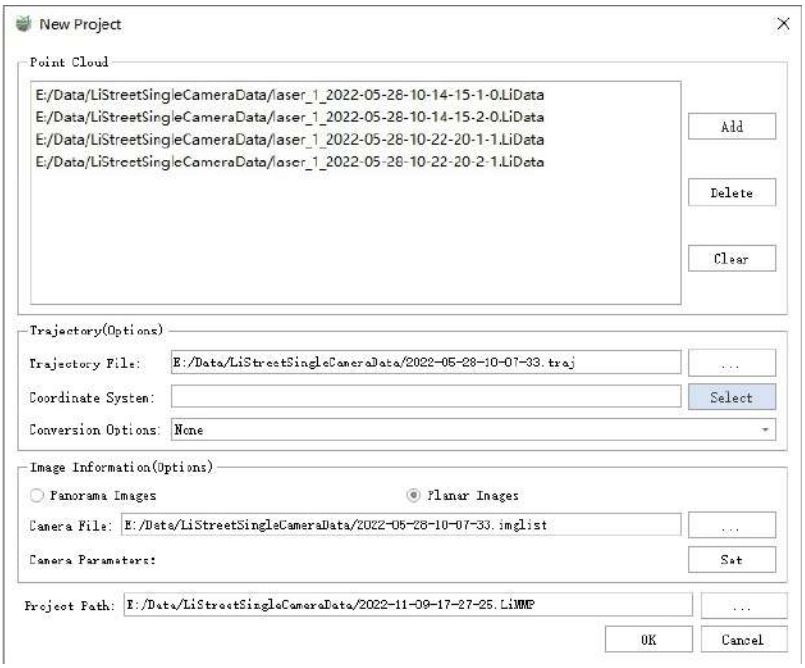


Import calibration file directly



Manually adding calibration parameters for flat cameras

4.Click the **OK** button when the setup is complete.



Select a plane image

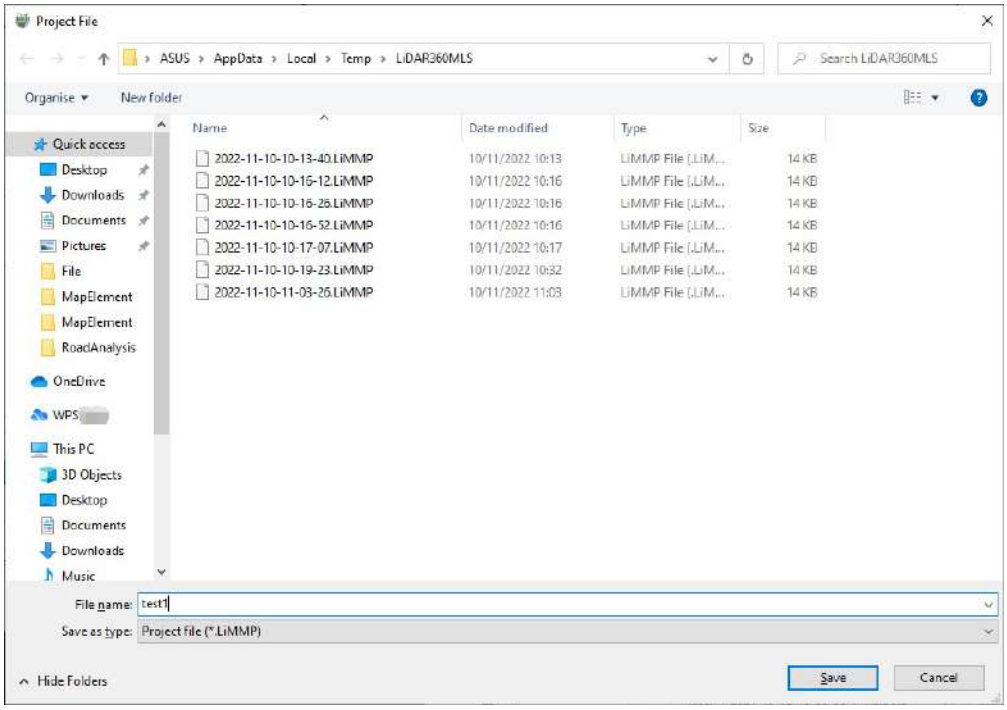
Set Project File Path

- **Project Path**: The storage path of the project file (*.LiMMP). By default, the current system time is used as the file name and stored in the same level directory of the input point cloud. Of course, users can also manually specify the storage path.

Drag Point Cloud

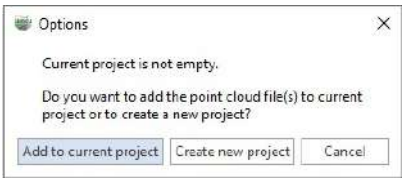
Function Description: Allows the user to can create or modify a project. Select several point cloud files with the mouse, drag them to the main window of the software, and then release the left button to complete relevant operations.

1.If the current project is empty, the project will be automatically created under the temporary path based on the current project. Subsequently, when saving the project a saving, the dialog box will pop up to set the formal saving path of the project. Once this setting is completed, all subsequent operations will be saved under the following set path.



Project files

2.If the current project is not empty, the option dialog box will pop up.



Options Dialog

- **Add to the current project**: The software will add the selected point cloud file to the current project, and this process will automatically filter out the point cloud files that have been loaded in the current project.
- **Create a new project**: The software will automatically create and open a new project based on the selected point cloud.
- **Cancel**: The software does not perform any operation and the dialog box automatically closes.

> Note: The point cloud cannot be dragged into the project after the cutting operation.

New from template

Function descriptions：Create a new LiDAR360MLS project based on the configured layer template.

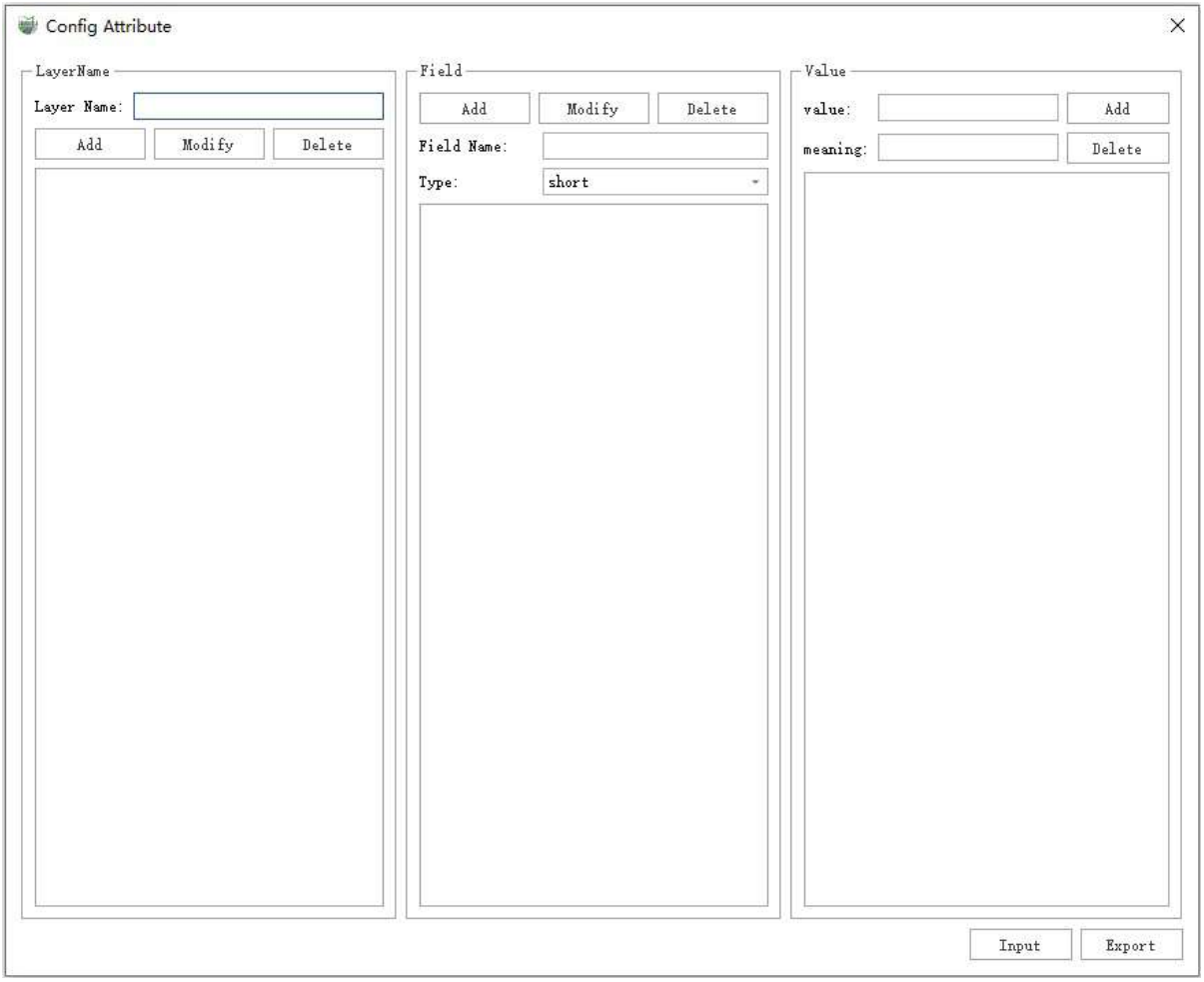
Steps

1.Configuration templates.

1.1 After starting the software, first configure a project according to the wizard by using the New Project button

1.2 Once the project has been successfully created, enter the software interface and in the bottom left corner, click on the **Layer Configuration** function

1.3 In the pop-up configuration layer dialog box, configure the template layers, fields and enumeration values as follows.



(1) **Parameter settings:**

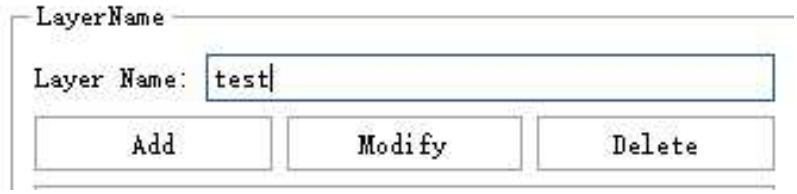
- **Layers**: Layer paging can modify the built-in layer name, add/modify/delete custom layers
- **Field**: Field paging allows you to add a new property field for the selected layer on the left layer paging and set the type of field value.
- **Value**: The Value sub-page allows you to set the value of the new field and also to note the meaning of the current value.
- **Save**: Save the configured layer as a template for subsequent new projects directly based on this template.

(2) **Operating instructions:**

(2.1) **Layer paging settings**: In the layer paging, **left mouse click** on the layer name in the list, you can modify and delete the layer of **currently selected**, but also for the addition of custom layers.

After selecting the operating layer, you can set the field as well as the field value in the field paging and value paging respectively.

- **Add layer**: Enter the name of the layer you want to create in the layer name edit box on the layer pane and click the Add button to complete the addition of the layer. The added layer will subsequently be hung in the directory tree as a custom layer. Once the layer has been added, the fields and field values can be set as described in the **Modify Fields and Modify Values** section below



- **Modify Layer**: Left mouse click to select the layer, enter the modified name in the layer name edit box and click the modify button to complete the layer name modification.

> Note:
>
> - Inherent layers cannot be modified by layer type

- **Delete layer**: Left mouse click on the selected layer, click on the delete button, the selected custom layer has been deleted

> Note.
>
> - Inherent layers cannot be deleted

(2.1) **Field paging settings**: After selecting the layer to be configured with fields in the layer paging, you can set the fields within the selected layer in the field paging, including adding fields, modifying fields and deleting fields.

> Notes.
>
> - Built-in layer fields cannot be deleted and modified, only fields can be added

Parameter description

- **Add**: Adds a new field to the selected layer.

> Note that.
>
> - The field name and type must be fully set before it can be successfully added.

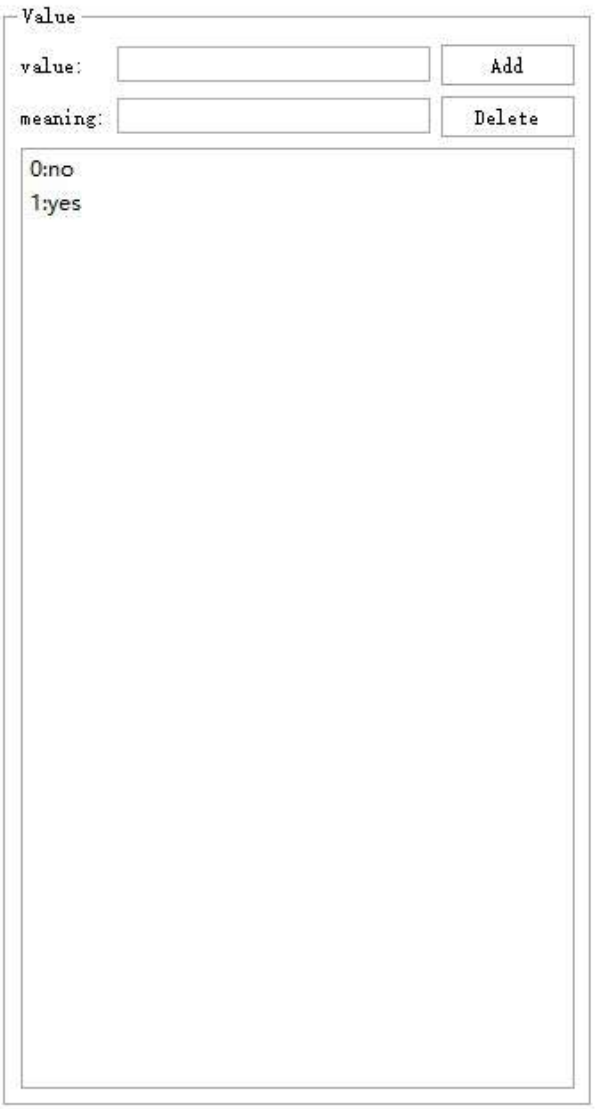
- **Modify**: Modifies an existing field in the selected layer.

> Caution.

> - This modifies the field itself (field name), if you want to modify the field value, refer to the field value setting later.

- **Delete**: Deletes an existing field in the selected layer.
- **Field name**: the name of the field.
- **Type**: one of the field properties that identifies the type of storage for the current field, the options are as follows.
  - short: identifies the current field as being stored as an integer type, with a small value range.
  - long: identifies the current field as being stored as an integer with a large value range.
  - float: identifies the current field as a decimal type with a small data precision, not suitable for storing coordinates and other values with high precision requirements.
  - double: identifies the current field as being stored as a decimal type with high data precision and can be used to store coordinate data.
  - text: Identifies the current field as being of the text type.
  - date: identifies the current field as being of the time type.

(2.3) **Field value setting**: After selecting a layer or field, you can set the value of the field and the corresponding explanatory notes in the value sub-page for configuring the enumerated value of a field.



Parameter Description

- **Value**: the value of the selected field
- **meaning**: the meaning of the selected field value
- **Add**: adds a new value

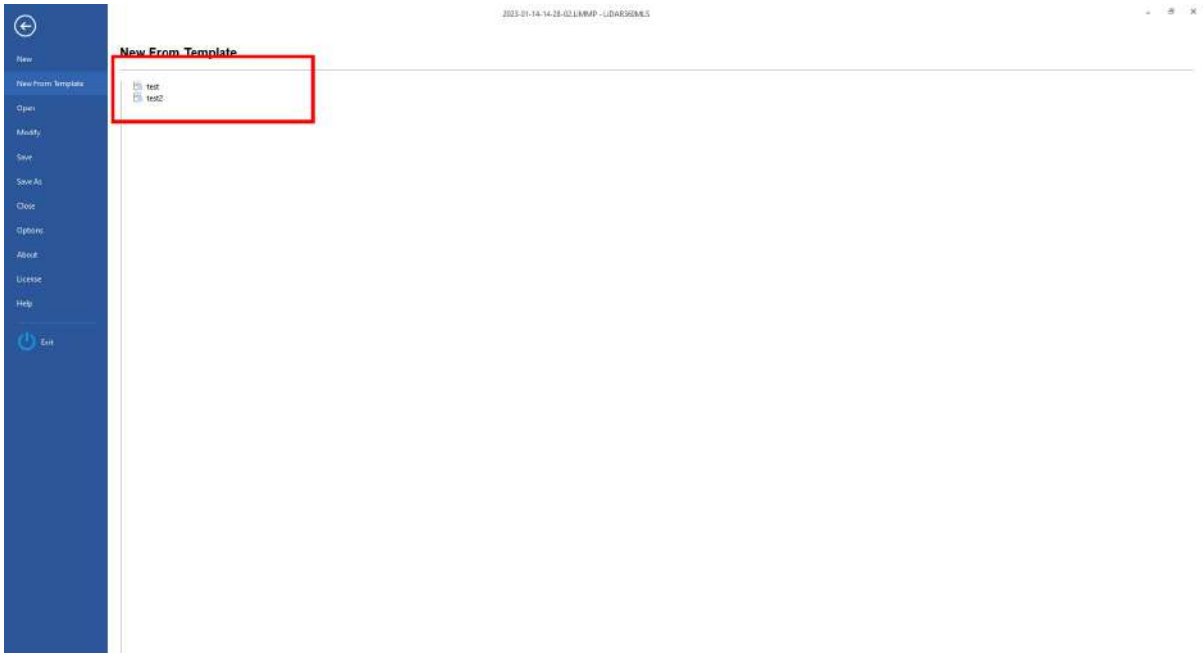
> Note that.
>
> - The value and meaning are both set before the addition is successful.
> - After the enumerated values have been configured subsequent new projects based on the current template, corresponding to fields with enumerated values, can be populated with fields directly based on the enumerated values and meanings, and the values are stored in the database.

- **Delete**: delete existing values

(3) **Save**: After completing the layer settings, click the Save button to customize a template name to save the current template, and after saving, it will be synchronized to the list of **Newly Created by Template**.

2.New from template.

When the template is set and saved, from the File menu page, switch to **New by Template**, click on the template you need to use, then follow the wizard to configure the point cloud, trajectory and image data for the new project, please refer to New Project for the specific configuration.



Once the project has been successfully created, you can see that the layers and properties of the directory tree are all set in the template and can be drawn directly using the corresponding functions.

Open Project

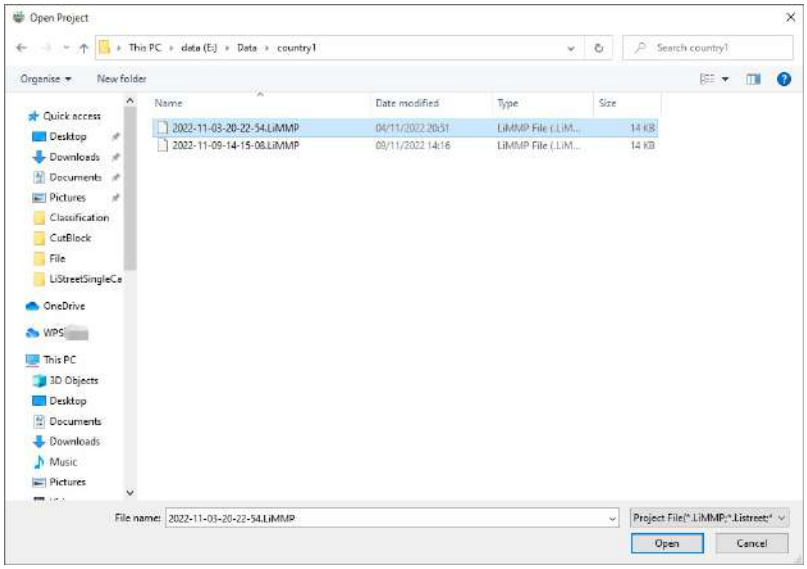
Function Description： Allows the user to launch the software. Click "Open", then select a recently saved project or open a project saved at a specific directory.

> The software can open ***. LiMMP**、 ***. Listreet**、 ***. LiGeo**, ***. mmprj** type project.

Steps

- **Recent Projects：** Select a recently saved project.

- **Browse：** Browse a project at a specific directory.



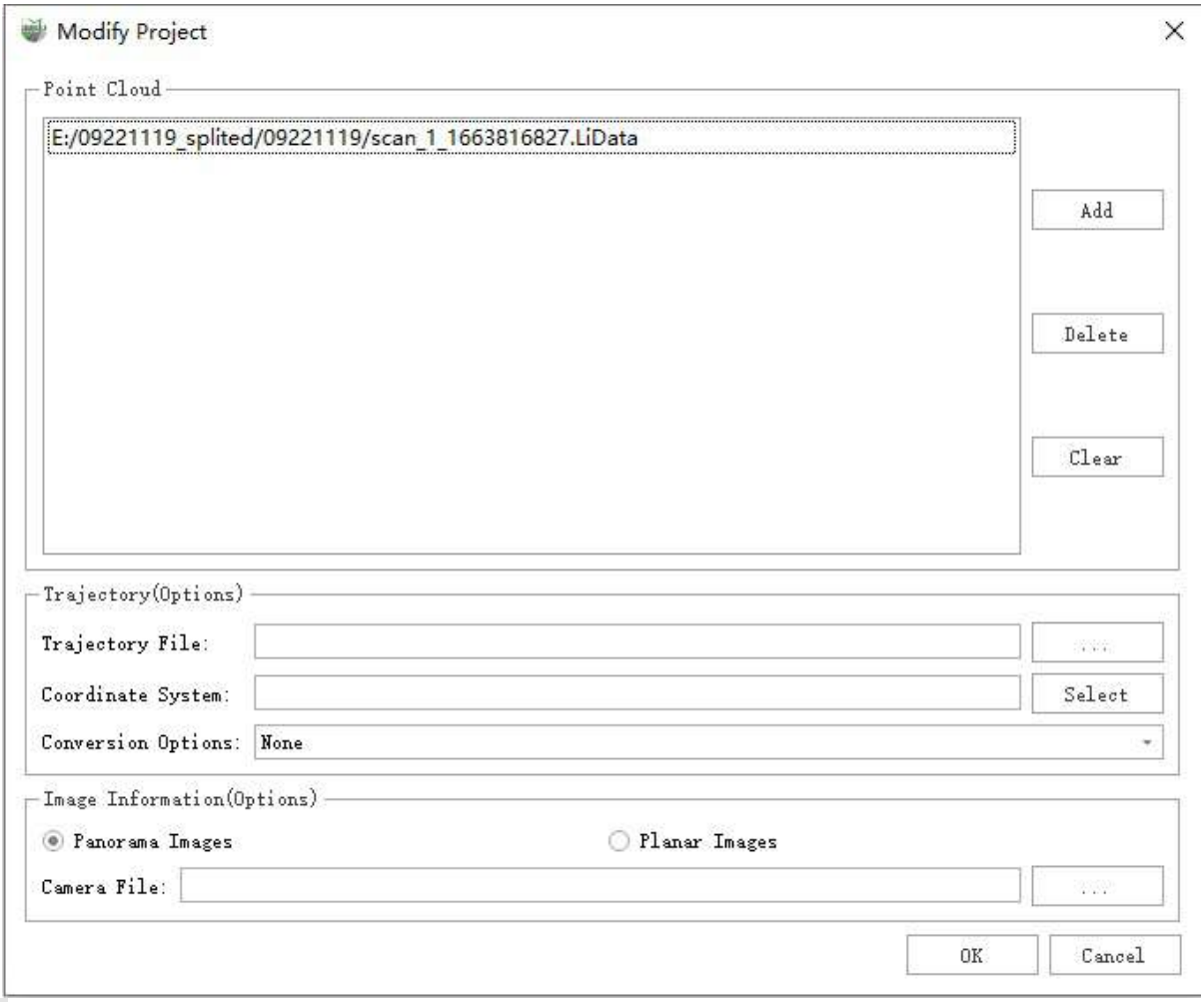
Modify Project

Function Description： Allows the user to modify a project. Click "Modify Project" to modify opened project, such as point cloud, trajectory or image.

Steps

1.Firstly, open a project that you want to modify.

2.Then, click  , to add, delete or remove point cloud in current project. You may also modify the image file by adding, deleting or adjusting the coordinate system of the trajectory.



Note：

- Currently, the software does not support projects that have been cut into blocks.
- Though you can clear all data on one click, you still have to add point cloud in project, or you are not able to save the modified project. The software will remind you "Please check the laser file path".

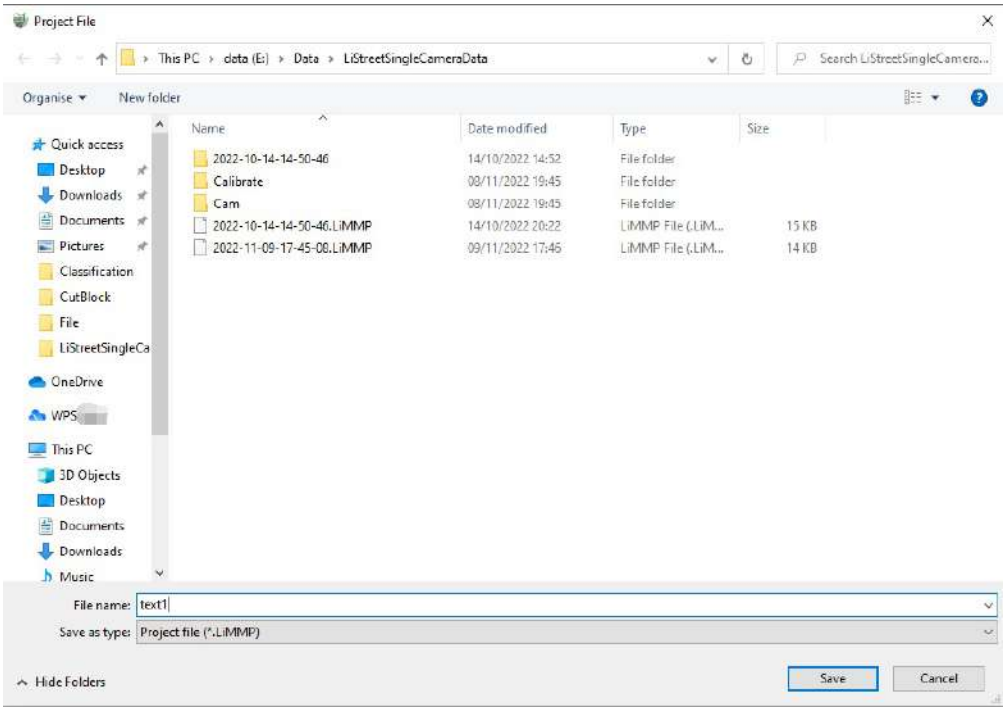

Warning
Please check the laser file path
Yes

Save Project

Function Description: Allows the user to save a project. Click "Save" and the software will automatically save the project file being edited. The shortcut key for this function is: Ctrl+S.

Save As

Function Description: Allows the user to save the project under a new name. Click the  button and enter the new name of the project file being edited, and save it.



Close Project

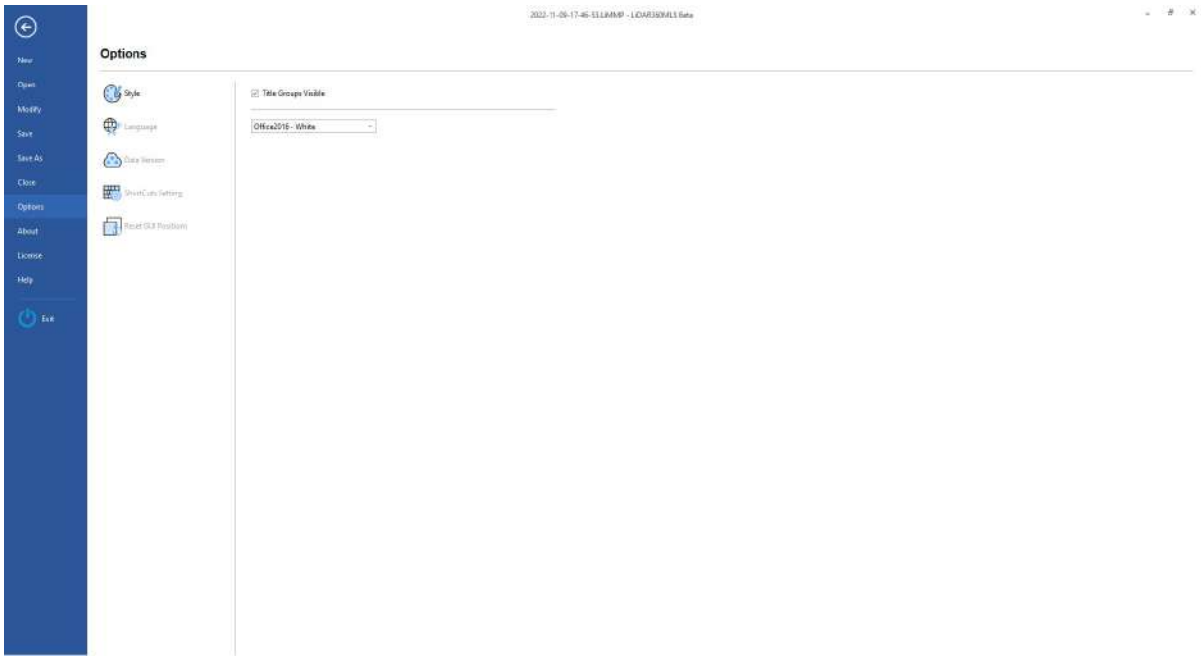
Function Description: Allows the user to close an opened project. The software will automatically save the project file being edited.

Options

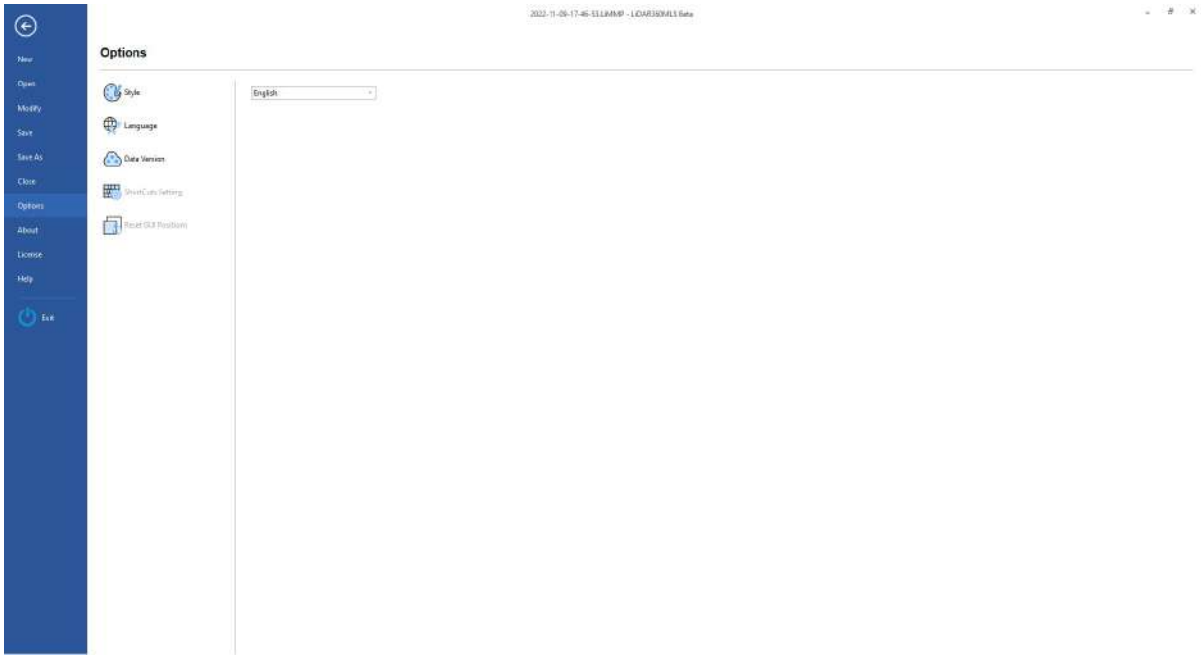
Function Description: Allows the user to view or adjust settings of the software.

Steps

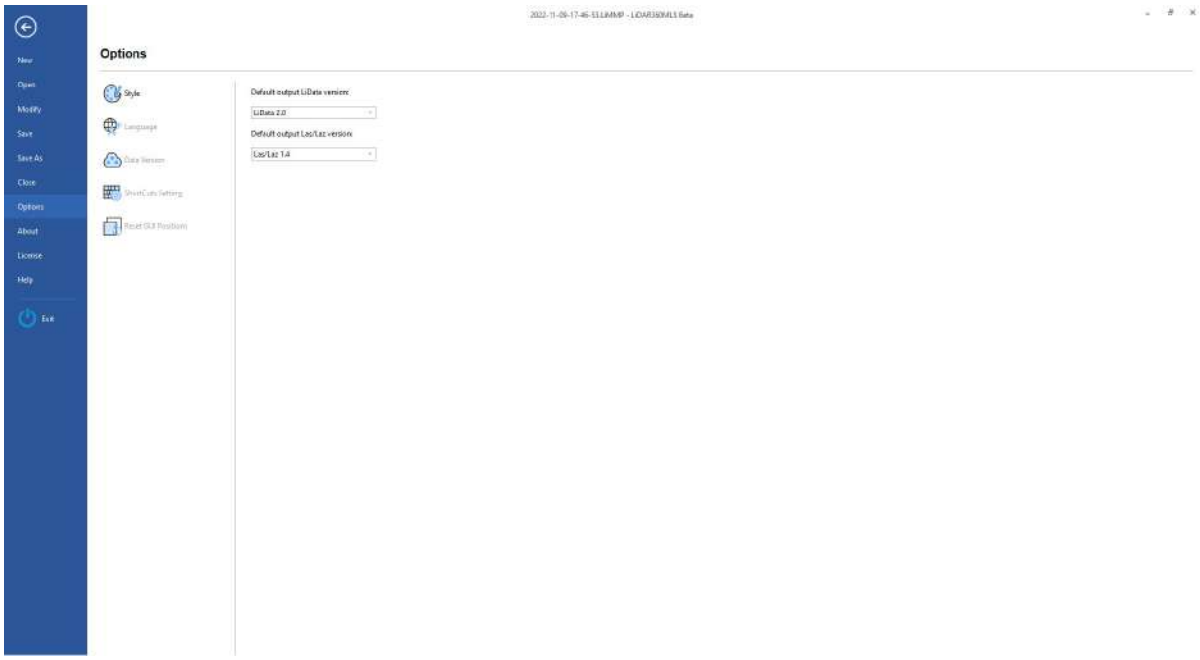
- **Style**: Set the display style of the software.



- **Language**: Set the language of the software.

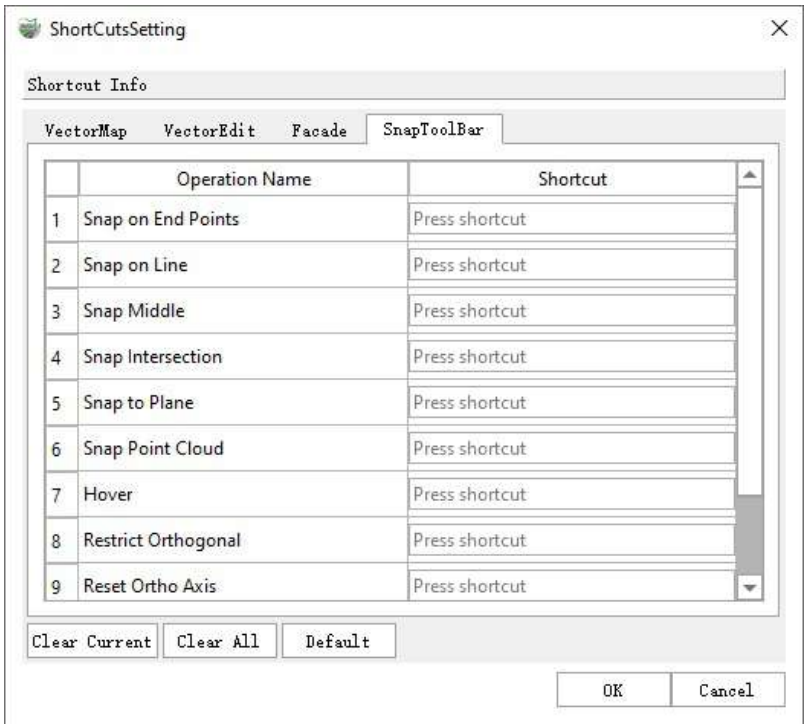


- **Data Version**: Set the data input and output to the data version under the *LAS*/LAZ format and the default point cloud storage format.



- **Shortcut Keys Settings**: Vector Editing, Elevation Measuring, Capturing four modules of the shortcut keys.
- **Steps**

1.Click the "Shortcut Setting"  button to pop up the Shortcut Setting dialog box.



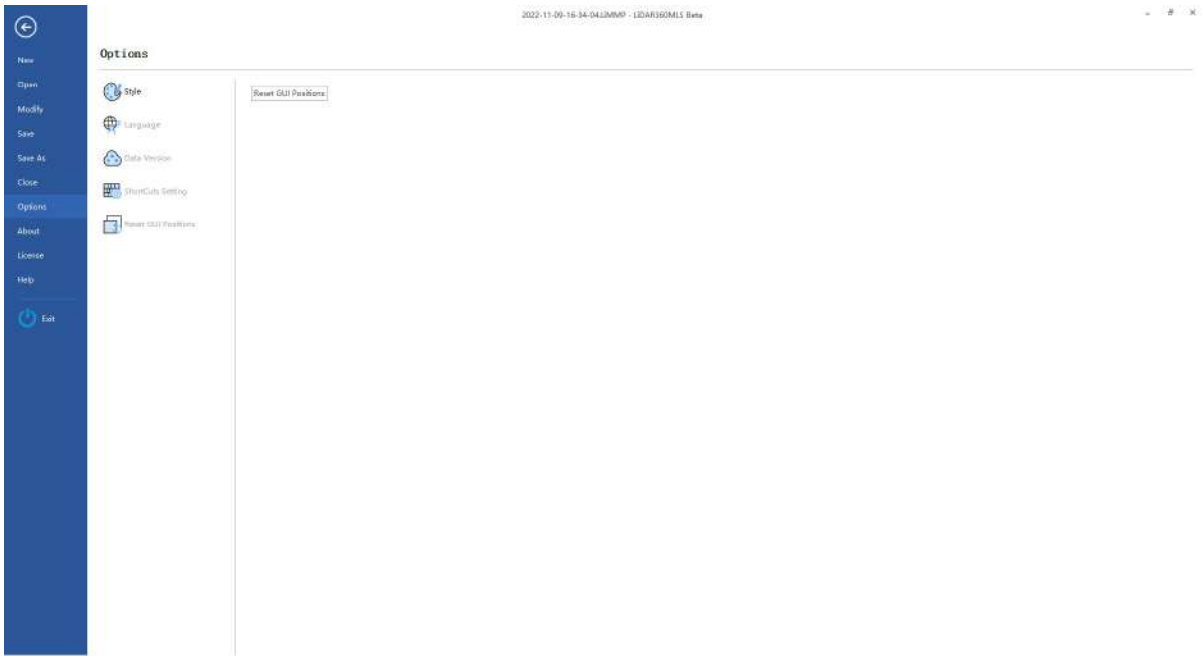
2.In the Shortcut Setting dialog box, you can freely combine the letters, numbers, Shift, Ctrl, Alt and other symbols of the keyboard, and click "OK" to take effect.

3.Click "Clear Current" to remove the shortcut keys for the selected function.

4.Click "Clear All" to remove all configured shortcuts.

5.Click on "Default Settings" to restore the default shortcut configuration of the software.

- **Reset GUI Positions：** The user can click "Restore GUI Positions" to restart the software and reset the toolbar. When the user drags the toolbar out of the interface and cannot be restored.



Software Default Shortcuts

1.Common Platform Shortcut Keys

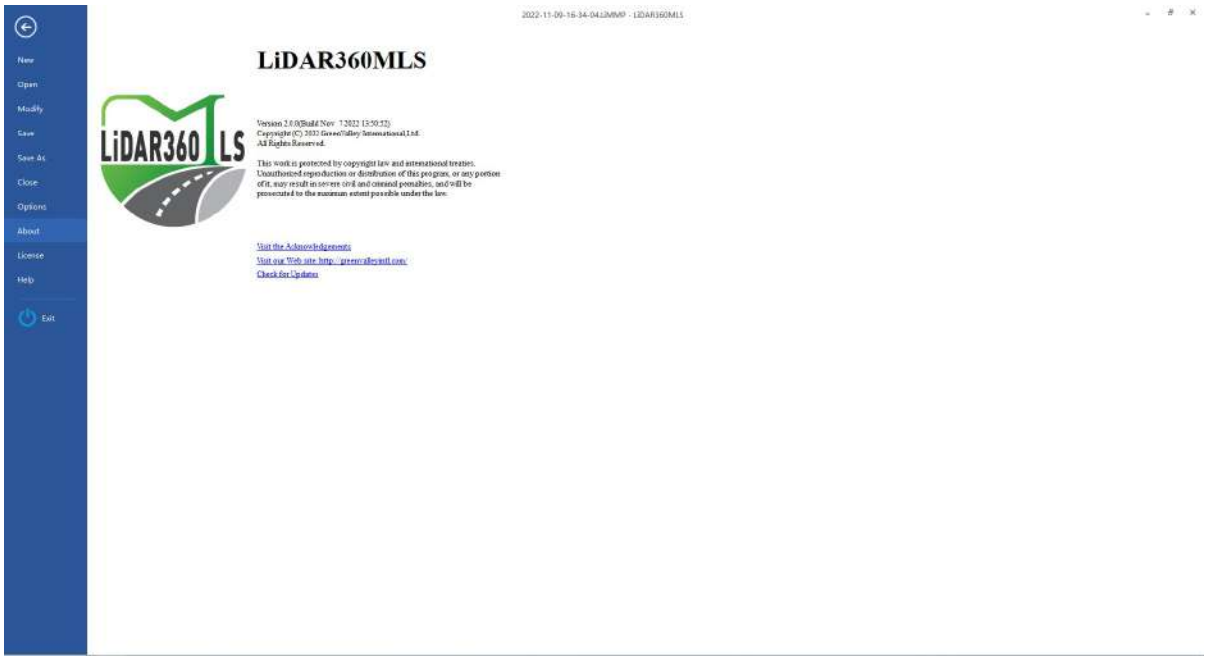
Function	Shortcut keys
Exit LiDAR360 MLS software	Alt+F4
Save LiDAR360 MLS project	Ctrl+S
Orthographic/Perspective Projection Toggle	F3
Shift the data up	↑
Shift the data down	↓
Shift the data left	←
Shift the data right	→
View zoom in	+
View zoom out	-
Rotation	Left mouse button
Shift	Right mouse button or press and hold wheel
zoom	Wheel
Global View	Space

2.Shortcut Keys for Common Functions

Function	Shortcut	Description
Line、Polygons	s/S	Short press S key, switch drawing mode, from drawing straight line to drawing arc, arc uses three-point mode, first point is the last point before short press s, second point is the end point of arc, third point controls radian
Line、Polygons	b/B	Short press B key, draw back node
Add traffic signs, facade measure moving rotation	q/Q;r/R	Rotate clockwise, counterclockwise

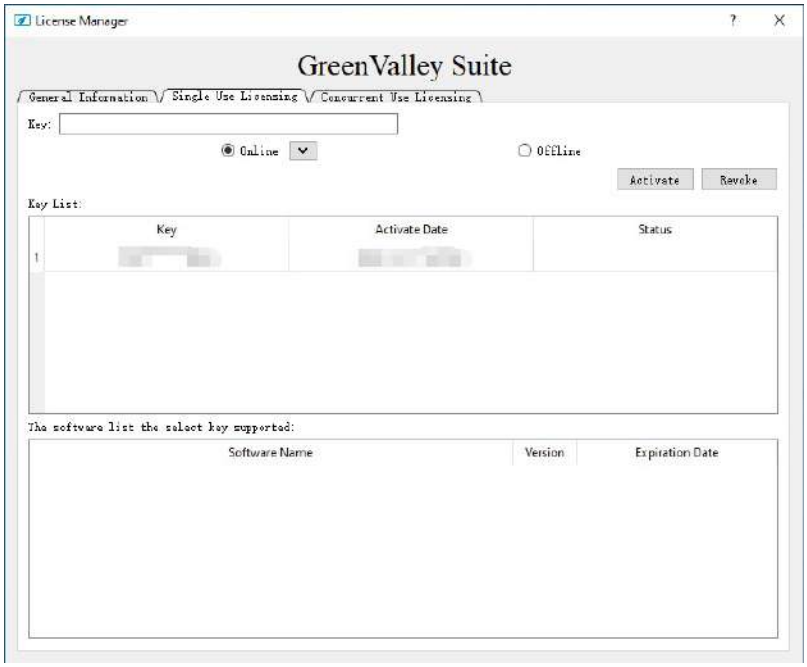
About

Function Description: Allows the user to view the software version, visit the official website and check for updates.



License

Function Description: Allows the user to view license details, activate or revoke a license key. Click "License" to activate or revoke a key.



Help

Function Description: Allows the user to browse for helpful information. Click "Help" and the software will automatically open the user manual.

Exit

Function Description: Allows the user to exit the software. Click "Exit" to exit the software.

Tools

The tool menu bar in the software mainly includes layer labels, cross-sections, roaming, and export tools. The following describes each tool separately.

Measurement

Measurement includes a range of measurement tools such as single-point, multi-point, density, length, area, angle, height, and volume measurement. In addition, the measurement tool can be selected to be used with the mouse hover tool to measure point clouds with obvious characteristics such as planes, ridges, corners, etc. They are introduced separately below.

- Single-point selection
- Multi-point selection
- Density measurement
- Length measurement
- Area measurement
- Angle measurement
- Height measurement
- Volume measurement
- Mouse Hover

Single point selection

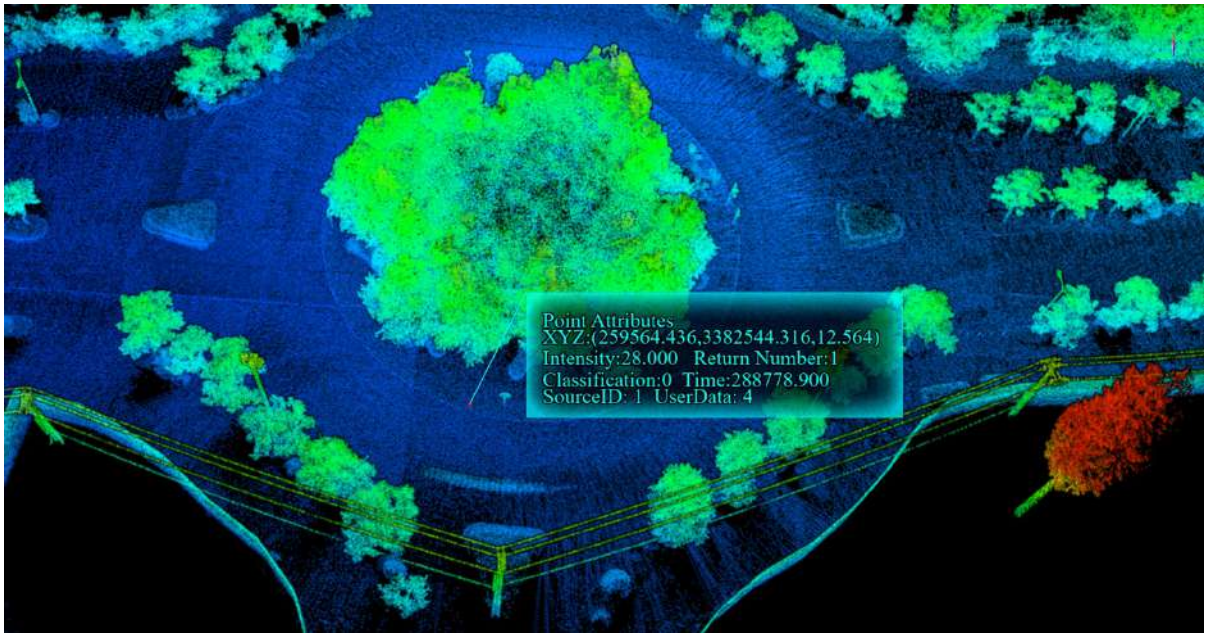
Function Description: The single point selection tool can interactively query the attribute information of a single point in the point cloud data, including the location, intensity, echo frequency, category, GPS time, etc. of the point.

Steps

1.Click the **Pick Point** button under the Measurement toolset.

2.In the 3D display window, use the left mouse button to click to select the point to be measured. A pop-up box will appear in the window to display the selected point's details.

The displayed information includes attributes such as location, intensity, echo frequency, category, GPS time, etc.



3.Click the right mouse button to cancel the selected point.

> Note: This function can only be used in the 3D view window.

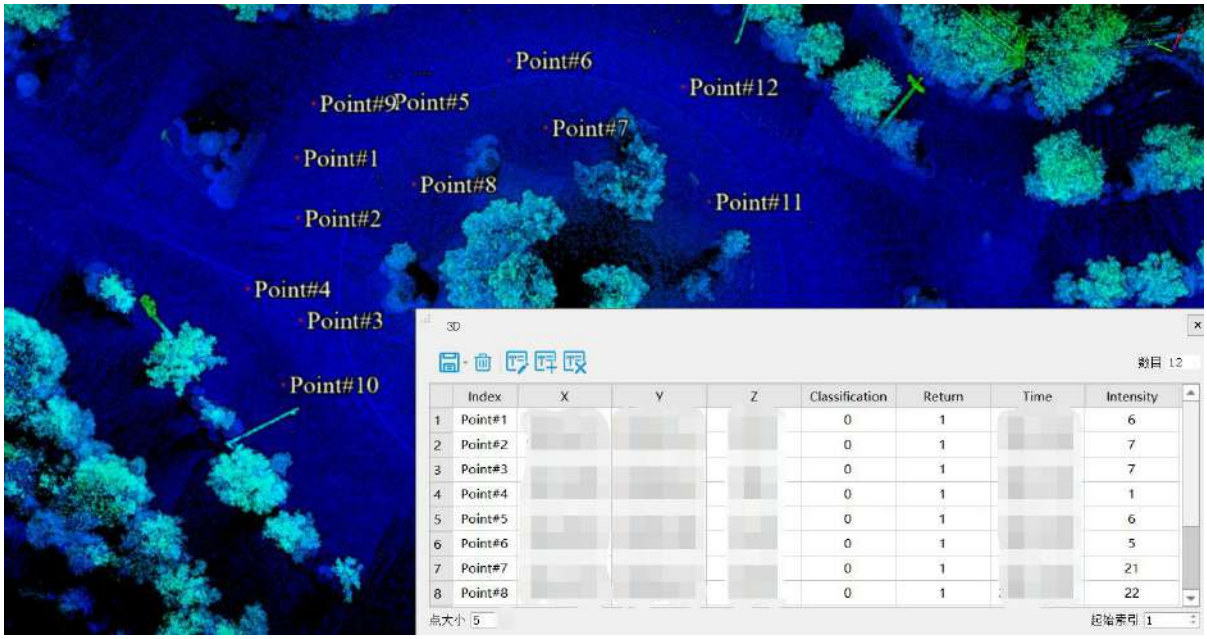
Multi-point selection

Function Description: The multi-point selection tool allows the user to select and interactively query the attribute information of multiple points, and also supports the export of selected points in txt format. The attributes that can be queried in point cloud data include the serial number, location, category, echo frequency, GPS time, intensity, etc. of the point.

Steps

1.Click on the **Multi-point selection** button under the Measurement toolset and use the left mouse button to click on the intended group of target points. The selected points will be marked in the form of dot labels in the scene, and the interface list will pop up to display the attribute information of the point set (as shown in the figure).

The point cloud attributes displayed in the list include the serial number, location, category, echo frequency, GPS time, intensity, etc. of the points. The edit box at the top right of the list displays the total number of selected points in real time.



2.The "Marker size" check box is used to set the dot size of the marker point in the scene, and the "Start Index" check box is used to set the starting index number of the marker point.

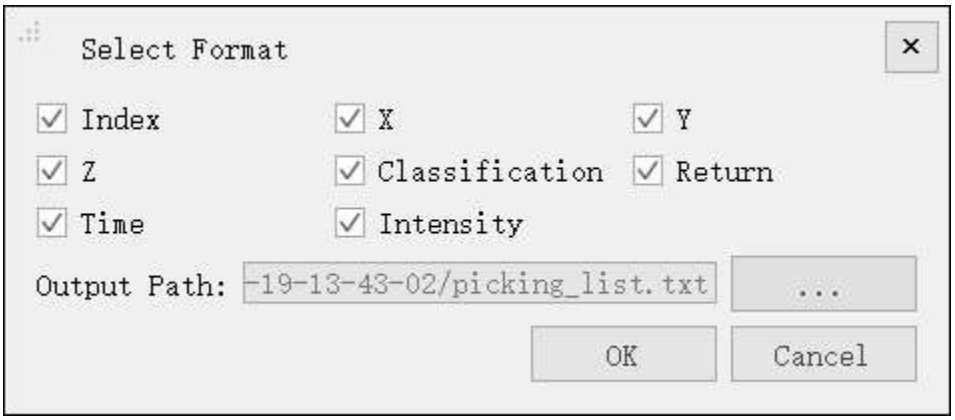
3.Click any row in the selection list and click the button deletes the point.

4.After pressing Start editing button , double-click the cell to modify the added field value.

5.After clicking the Add field button , the Add field dialog box pops up. The following types of custom fields are currently supported: integer, floating-point, text, date, and enumeration; after clicking the "OK" button, the new fields will be displayed in the list window.

6.Remove field button  is only available after the custom field is added. After the field is added, click this button and the Remove field dialog box will pop up. Users can only choose to remove the fields they added.

7.he selected point set is exported in *.txt* format. Click  button to drop-down menu, and the "Select method" dialog box will pop up (as shown in the figure). The user can click Save 3D point to save the coordinates and other attribute information as a *.txt* file.

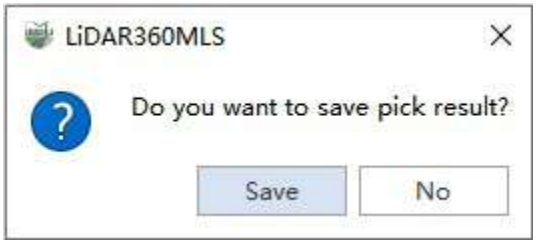


Parameter setting

- **X**: The X coordinate value of the point cloud data.
- **Y**: The Y coordinate value of the point cloud data.
- **Z**: The Z coordinate value of the point cloud data.
- **Category (Classification)**: The category attributes of point cloud data.
- **Echo (Return)**: The echo attribute of the point cloud data.
- **GPS Time (Time)**: The GPS time attribute of point cloud data.
- **Intensity**: The intensity attribute of point cloud data.
- **Index**: The index value of the selected point.
- **Output Path**: The output path of the exported file.

8.Click  to select the output path, check the attribute values you want to export, and click the "OK" button.

9.If the user does not save the selected point, when exiting the function, the software will pop up the following interface, click the **Save** button to save the point, and click **No** to exit the function.



> Note: This function can only be used in the 3D view window.

Density measurement

Function Description: Density is one of the indicators to measure the quality of point cloud data. The density measurement tool can be used to measure the density of point clouds and can measure the average number of points per square meter.

Steps

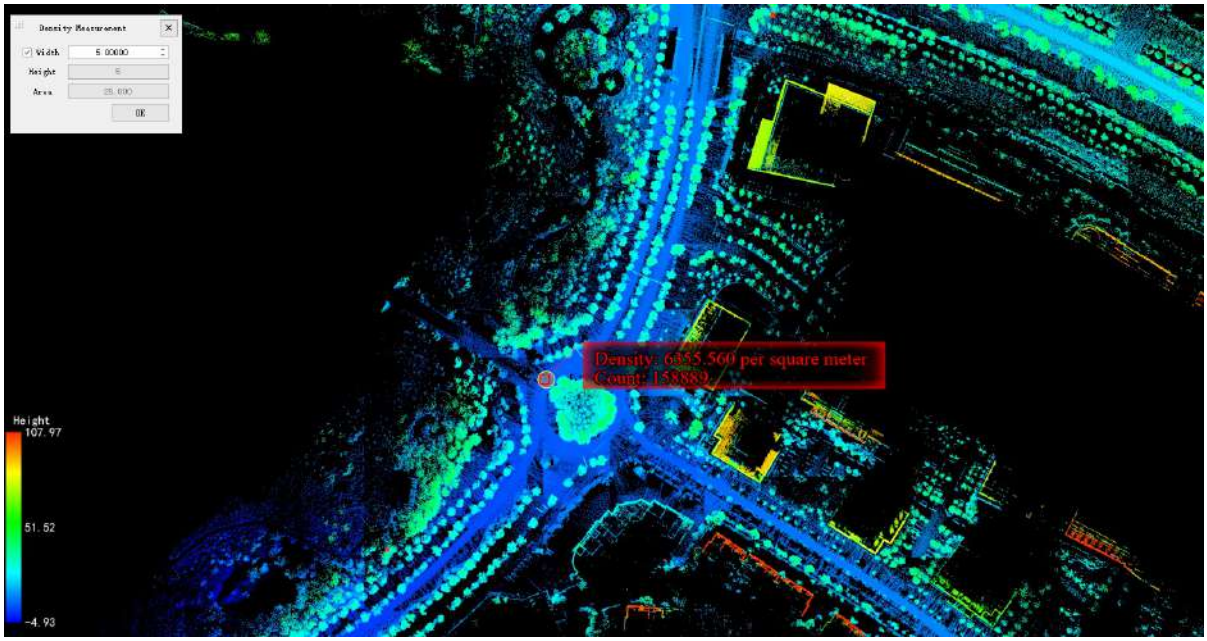
1.Click on the **Density** button in the Measurement toolset, open the point density measurement tool, and the 3D display window will automatically switch to orthogonal projection.

2.Click the left mouse button in the 3D display window to select the area where you want to measure the density, and the point density measurement settings dialog box will pop up.

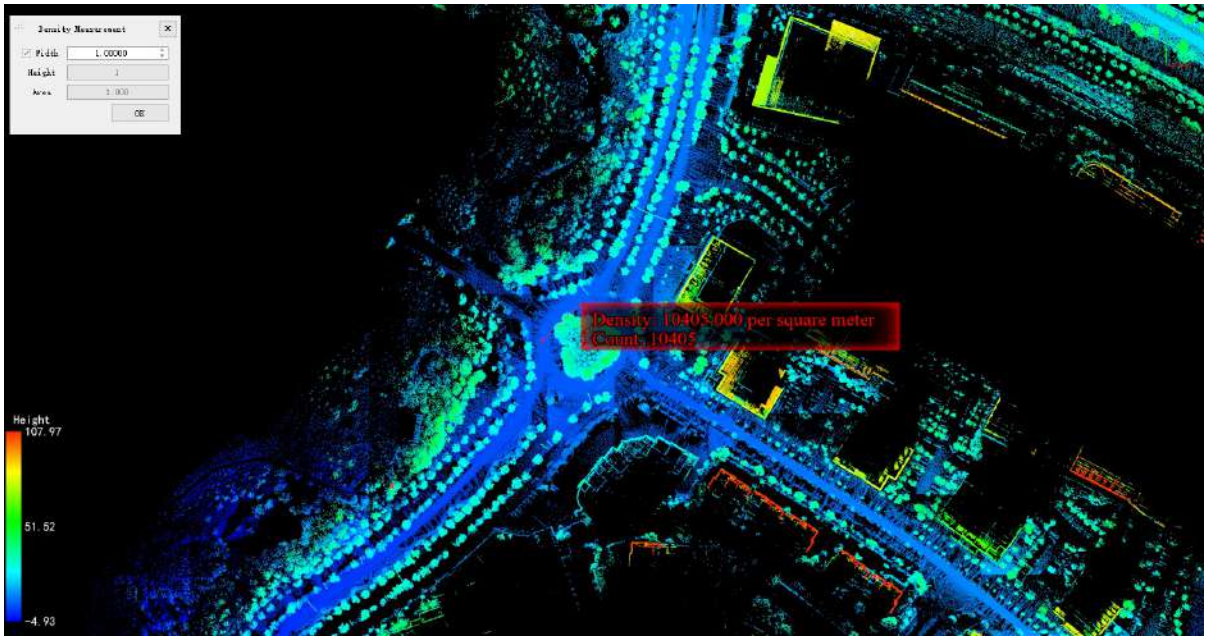
3.If "Width" is checked, you can enter the width value, the height value will be automatically set to the same as the width value, and the area is set to the product of width and height. Use the left mouse button to click to select the measurement area. If "Width" is not checked, the height value and width value will be set according to the size of the rectangle drawn, the area is set to the product of width and height, and the left mouse button can be clicked twice in a row to interactively draw the rectangle of the measurement area.

In the scene, the selected range is drawn with a solid red rectangle, and the measurement results are displayed in the form of a label (as shown in the figure).

Default width (width is 5)：



Modify the width (width is 1)：



Parameter setting

- **Width**: This parameter defines the width of the reference range for point density measurement.
- **Height**: This parameter defines the height of the reference range for point density measurement.
- **Area**: This parameter defines the area of the reference range for point density measurement.

> Note: This function can only be used in the 3D view window, and it is only valid under orthogonal projection.

Length measurement

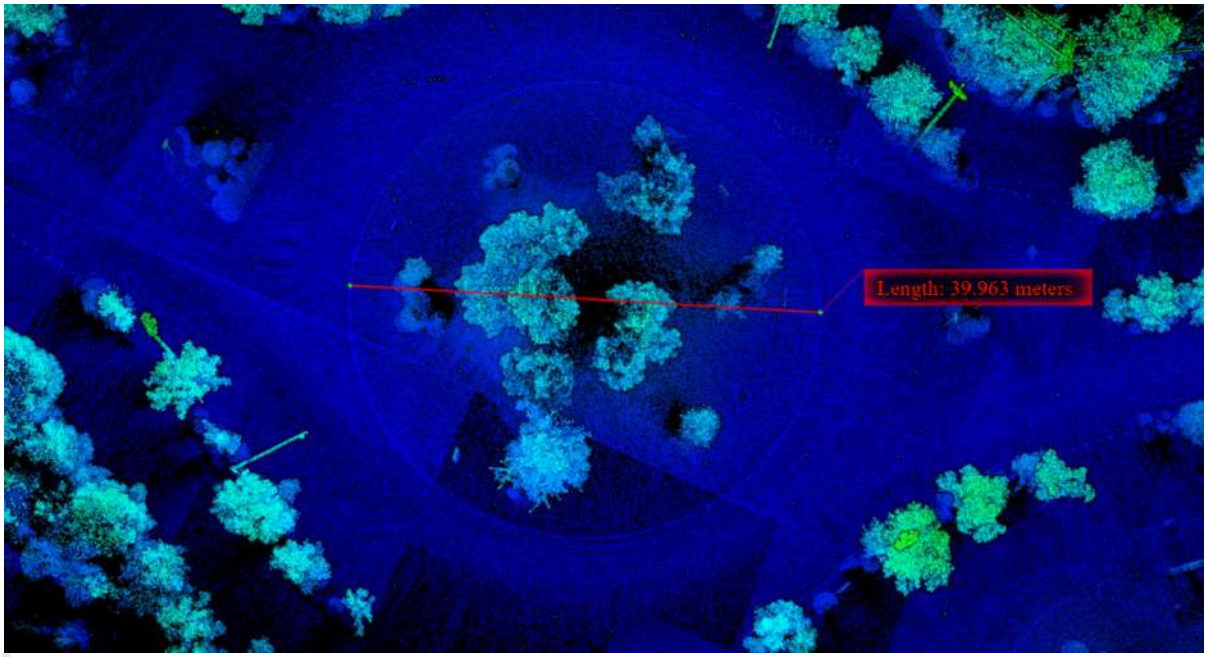
Function Description: The length measurement tool uses a mouse click to interactively query the distance information between multiple points. The measurement result represents the Euclidean distance of the point in the three-dimensional space.

Steps

1.Click on the **Length**  button in the Measurement toolset.

2.Use the left mouse button to click consecutively on the data points (minimum two points), and multiple line segments will be drawn in real time, and the measurement results will be displayed in the form of labels in the scene (as shown in the figure).

3.Click the right mouse button to fall back one point before completing the measurement.



> Note: This function can only be used in the 3D view window.

Area measurement

Function Description: The area measurement tool uses a mouse click to draw polygons interactively, and will query the projected area in the polygon area. For the area measurement of 3D data, the current window will automatically switch to orthogonal projection mode.

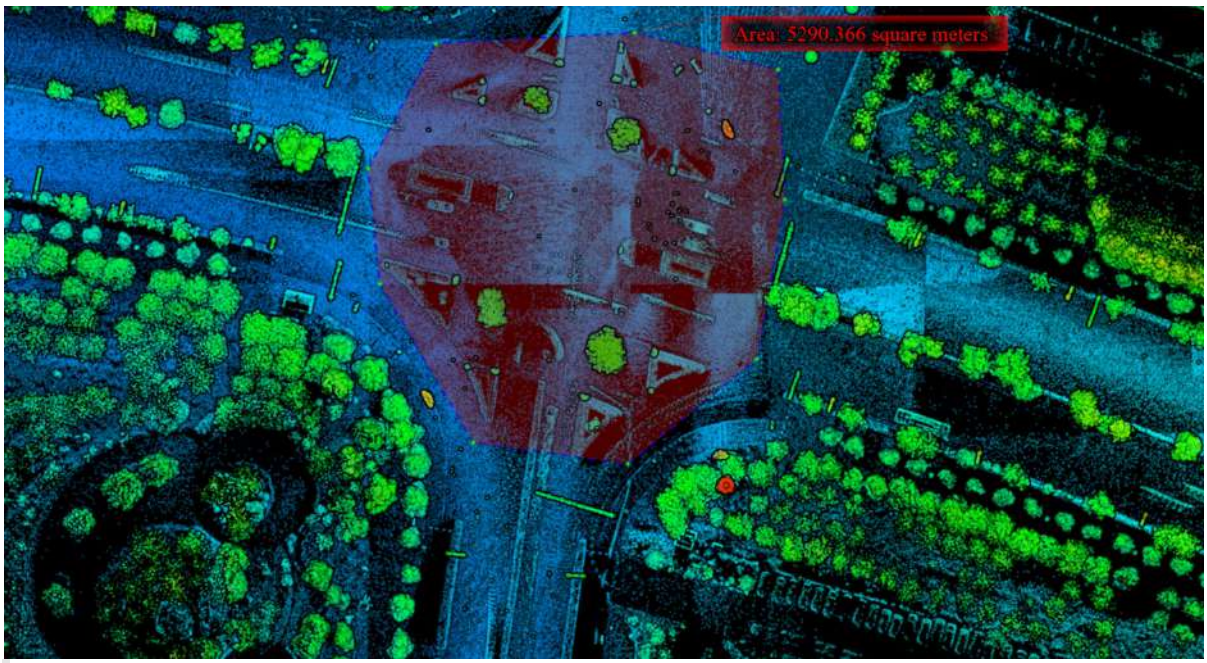
Steps

1.Click on the **Area**  button in the Measurement toolset.

2.Use the left mouse button to click consecutively on the data points(minimum three points) to select, polygons will be drawn in real time, and the measurement results will be displayed in real time in the form of labels (as shown in the figure).

3.Double-click the left mouse button on the last point to be selected to complete the measurement.

4.Click the right mouse button to fall back one point before completing the measurement.



> Note: This function is only for the area measurement of point cloud data. This function can only be used in orthogonal projection, and this function can only be used in the 3D view window.

Angle measurement

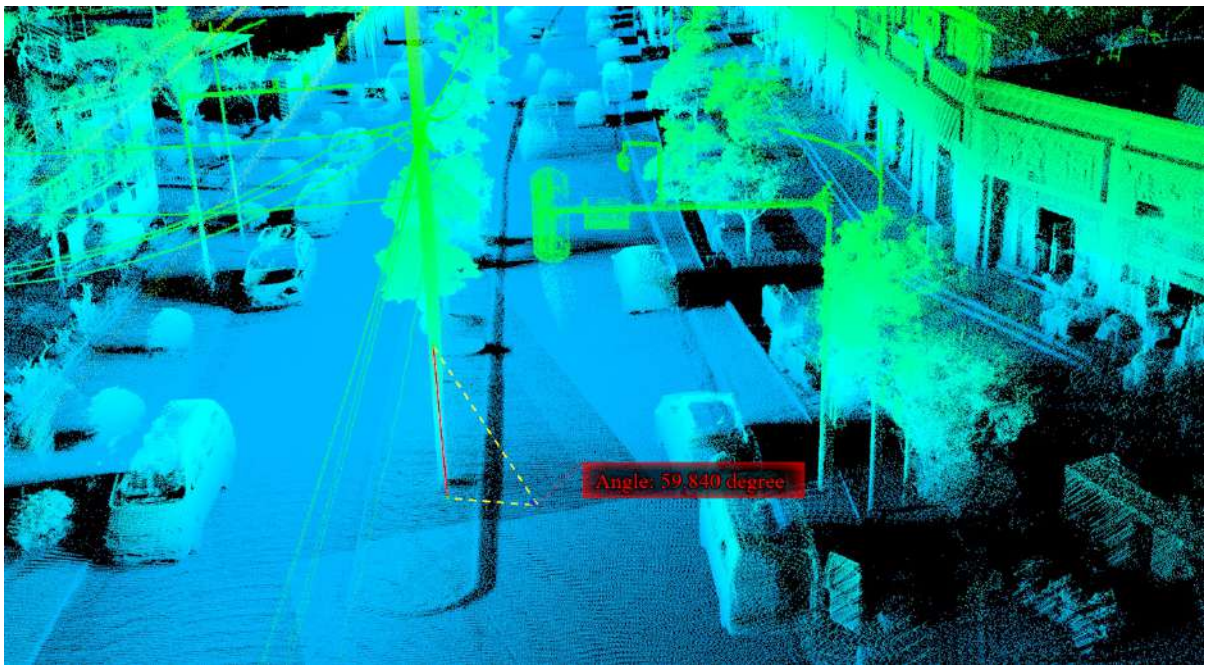
Function Description: The angle measurement tool allows the user to measure the angles between a set of selected points. In the 3D view, the pitch angle between the two points is queried, that is, the angle between the connecting line from starting point to the end point and the horizontal plane, and the projection angle of the three-point connecting line on the horizontal plane is queried in the 2D view.

Steps

1.Click on the **Angle**  button in the Measurement toolset.

2.In the 3D view window, click the data single point with the left mouse button and select the angle measurement reference point.

3.Double-click with the left mouse button to determine the measurement point, and the measurement angle will be drawn in real time, and the measurement results will be displayed in real time in the form of labels (as shown in the figure).



4.Click the right mouse button to fall back one point before completing the measurement.

> Note: This function can only be used in the 3D view window.

Height measurement

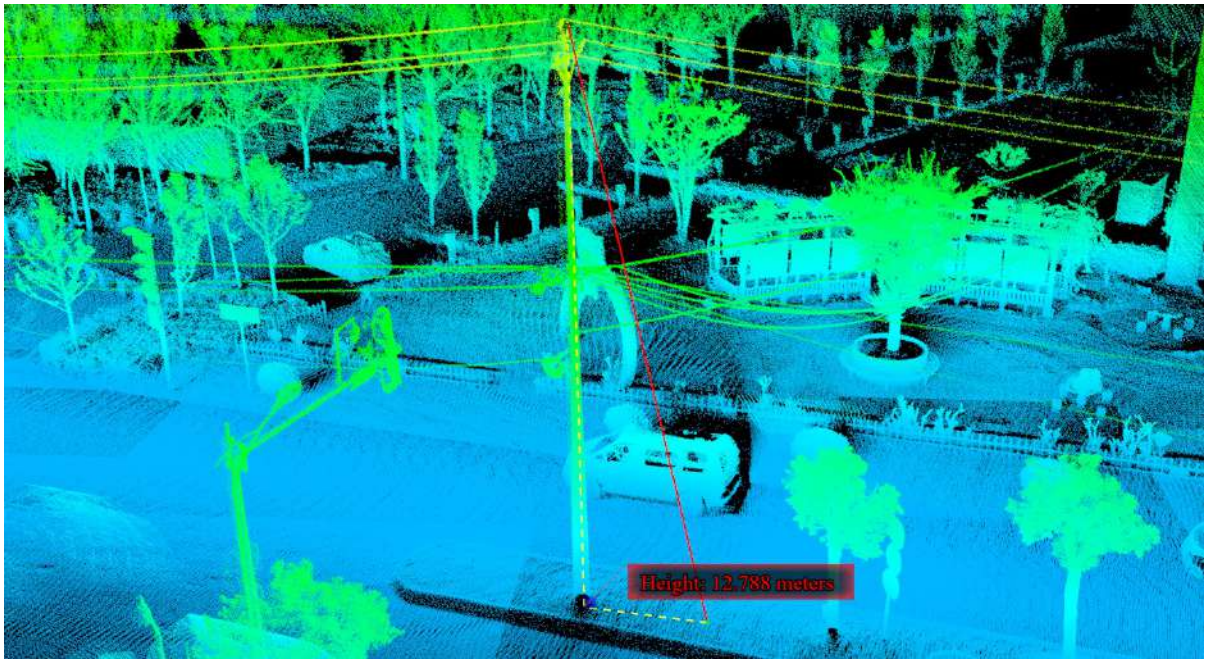
Function Description: The height measurement tool interactively selects the measurement point with a mouse click and queries the relative height between the two points.

Steps

1.Click on the **Height** button in the Measurement toolset.

2.Click the data single point with the left mouse button to select the height measurement reference point.

3.Double-click the left mouse button to select the measurement point, and the relative height between the two measurement points will be drawn in real time, and the measurement results will be displayed in real time in the form of labels (as shown in the figure).



4.Click the right mouse button to fall back one point before completing the measurement.

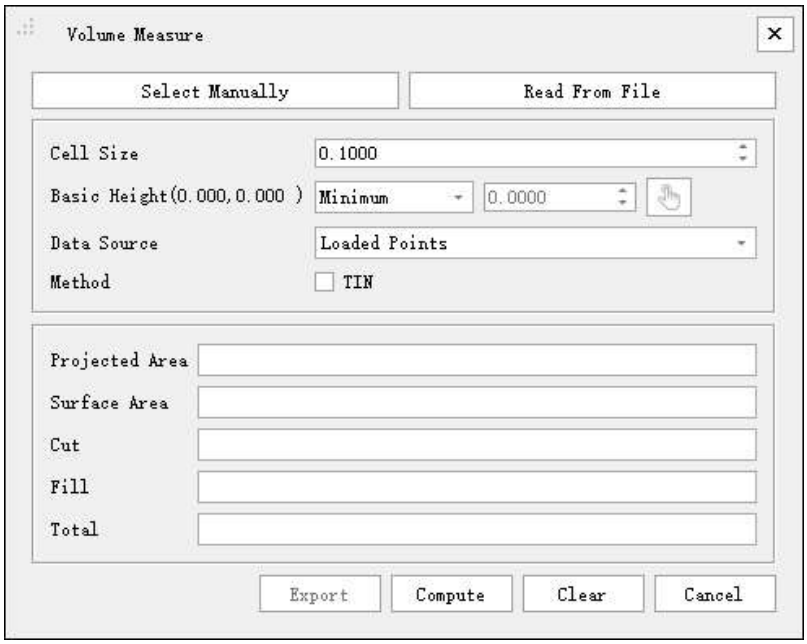
> Note: This function can only be used in the 3D view window.The fallback function can only be used before double-clicking the mouse.

Volumetric measurement

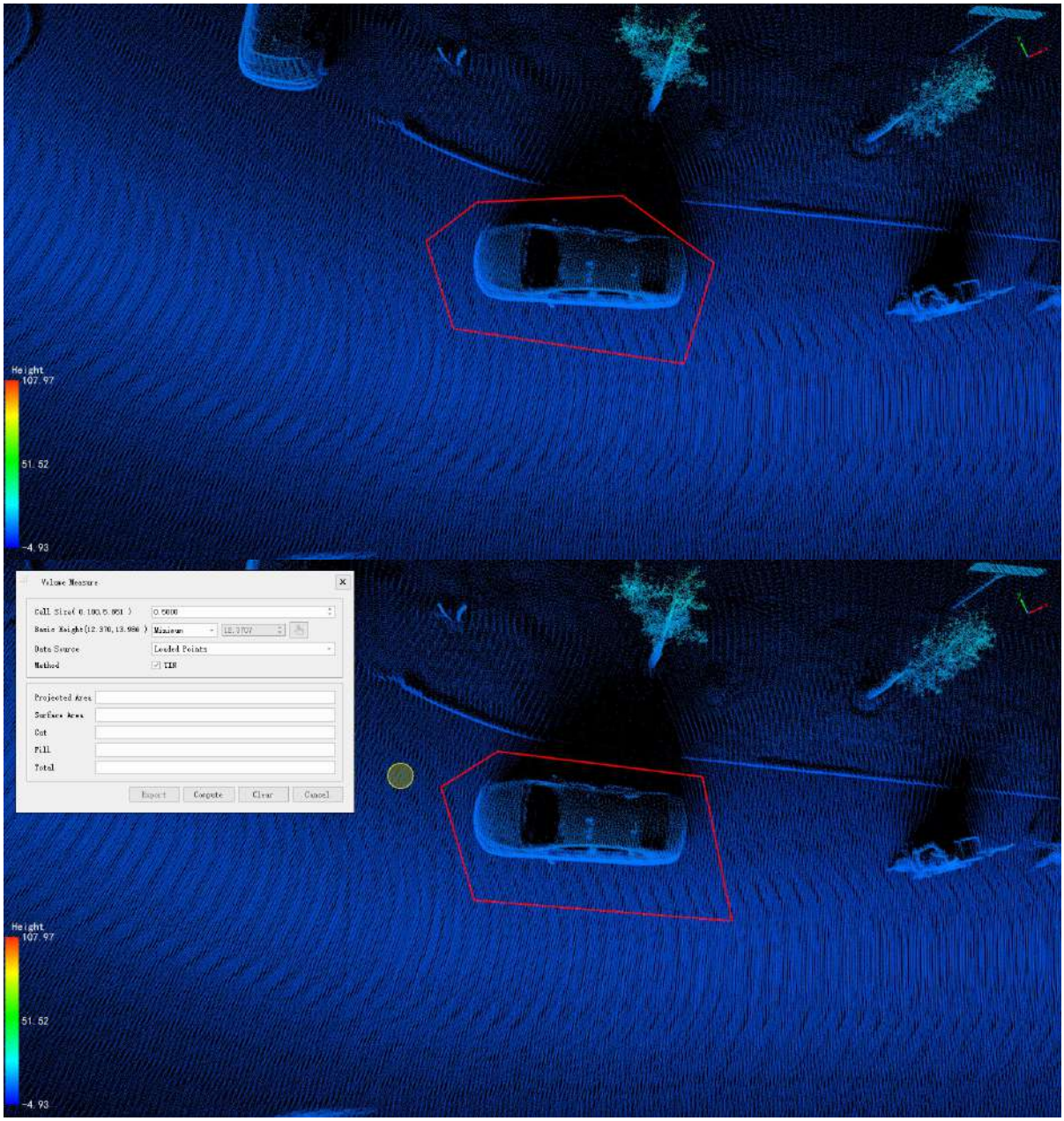
Function Description: The volume measurement tool selects the measurement reference plane by clicking with the mouse, and calculates the amount of fill and cut relative to a certain height. It is generally used in coal pile volume measurement, ship hull volume measurement, etc.

Steps

1.Click on the **Volume** button in the Measurement toolset, the volume measurement dialog box pops up.



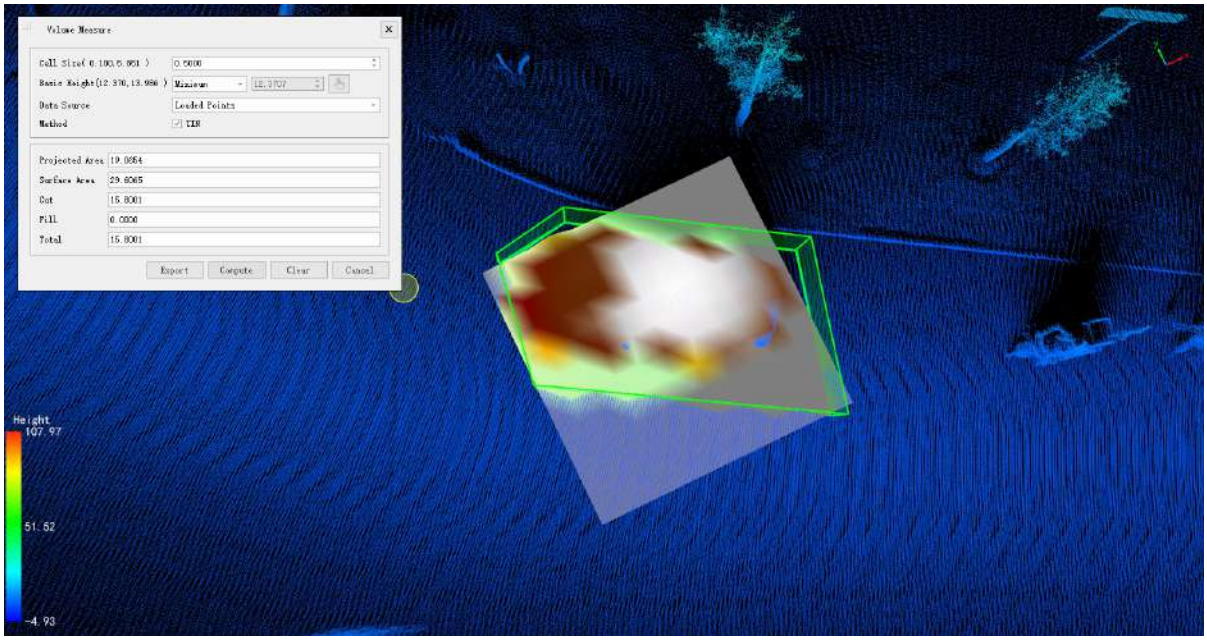
2.Before volumetric measurement, generally adjust the window to the top view, and then use the left mouse button to continuously click on the area where the point cloud data is loaded in the window loaded with point cloud data (the reference plane used to generate the volumetric measurement, select at least three points), double- click the mouse to end the selection, and the selected area will be drawn with a solid red line, and the dialog box will appear, as shown in the figure.



3.Set the cell size.

4.Set the datum for volume measurement, and the calculation method of the datum includes the minimum value, the fitting plane, and custom.

5.Click the calculate button to generate the cut, fill, and fill and cut results. At the same time, the space occupied by the current measurement data will be plotted, as shown in the figure.



Parameter setting

- **Cell Size**: This parameter defines the minimum calculation unit size for volume measurement. The smaller the value, the more accurate the calculation.
- **Datum**: This parameter defines the datum of the volume measurement fill and cut.
  - **Minimum Value (default)**: Take the minimum Z value within the selected point range as the height value of the plane.
  - **Fitting Plane**: Fit the best plane according to the selected point.
  - **Custom**: The height specified by the user is used as the height value of the datum for volume measurement.
- **Data Source**: This parameter defines the type of data source for a volume measurement.
  - **Loaded Points (default)**: Using points loaded into the scene within the specified range, the speed is relatively fast, and changes in data loading in the scene have an impact on the calculation results.
  - **All Points**: Using points in the point cloud file within the specified range, the speed is relatively slow and the calculation results are stable.
- **Method**: This parameter defines whether an irregular triangular network (TIN) is constructed during volume measurement. The construction of an irregular triangular network can make the measured value more accurate, but at the same time it will reduce the calculation speed. Users can choose whether to use this method for calculation according to their needs.

6.Click the export button to export the volume measurement results in the format of *.pdf* or *.txt*.

> Note: This function can only be used in the 3D view window.

Mouse Hover

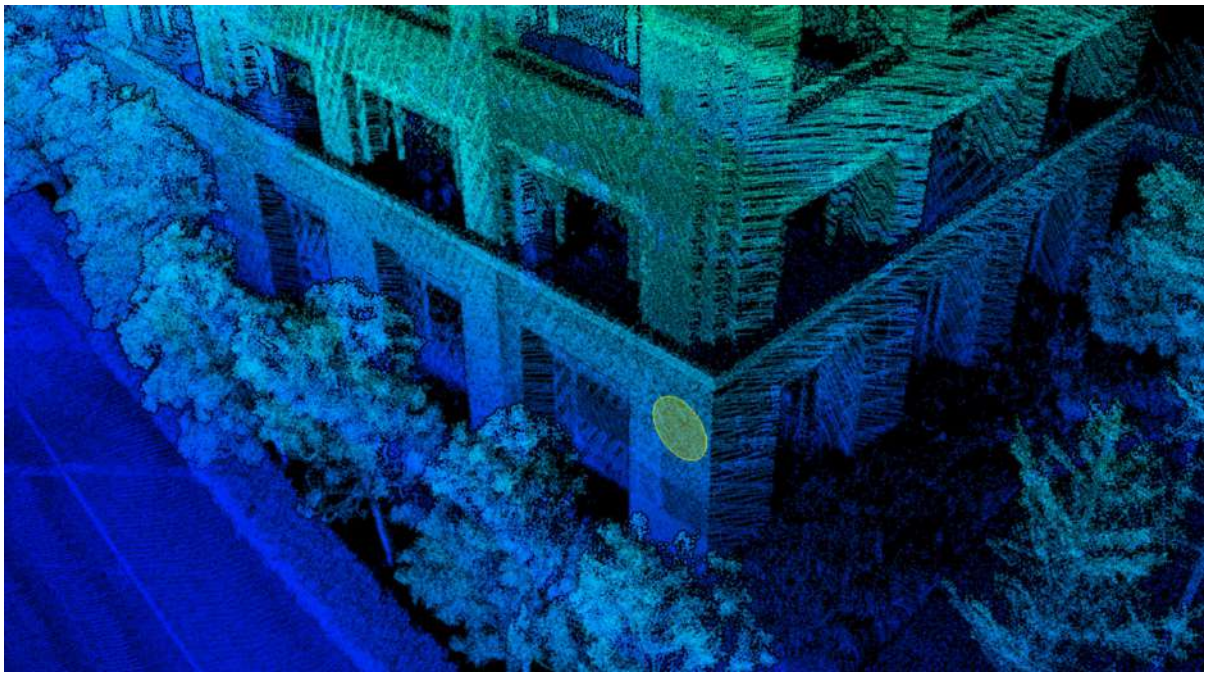
Function Description: The mouse hover query tool takes the point cloud within a certain pixel range for feature fitting at the position where the mouse is currently gliding, which is used to assist in measuring planes, ridges, corners and other local features.

Steps

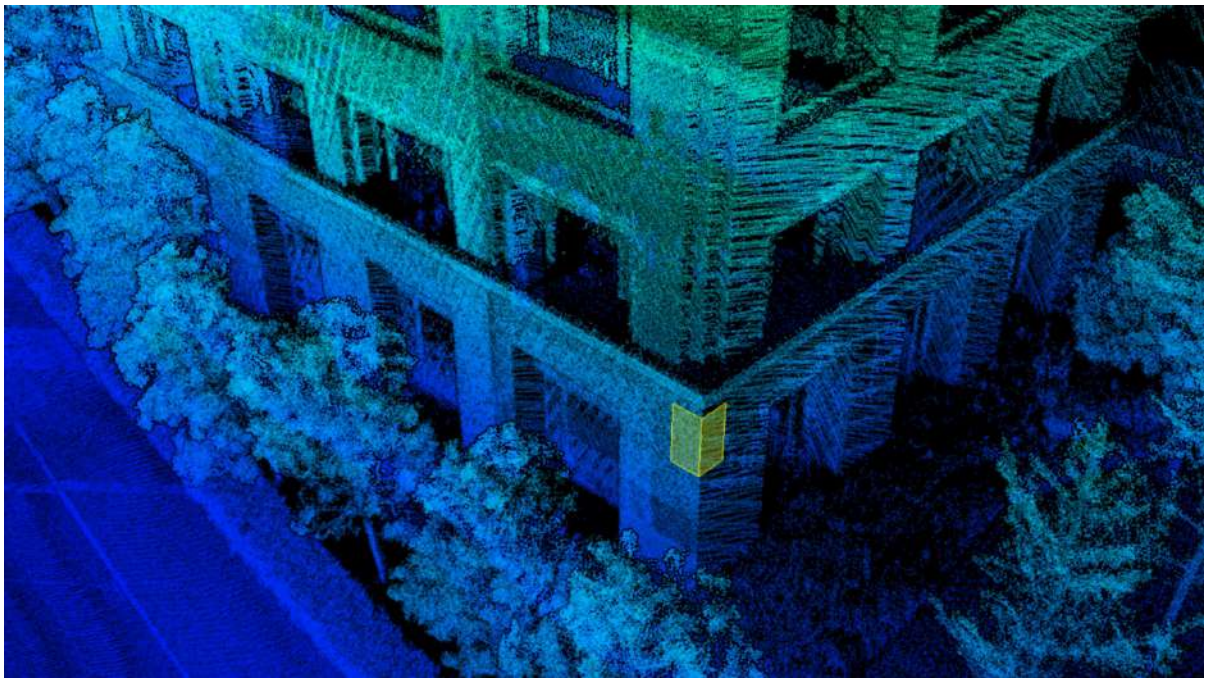
1.Click the **Use mouse hover query** check box in the **Mouse hover** to turn on the hover tool.

2.Adjust the **search radius** in the **mouse hover** item to modify the pixel threshold.

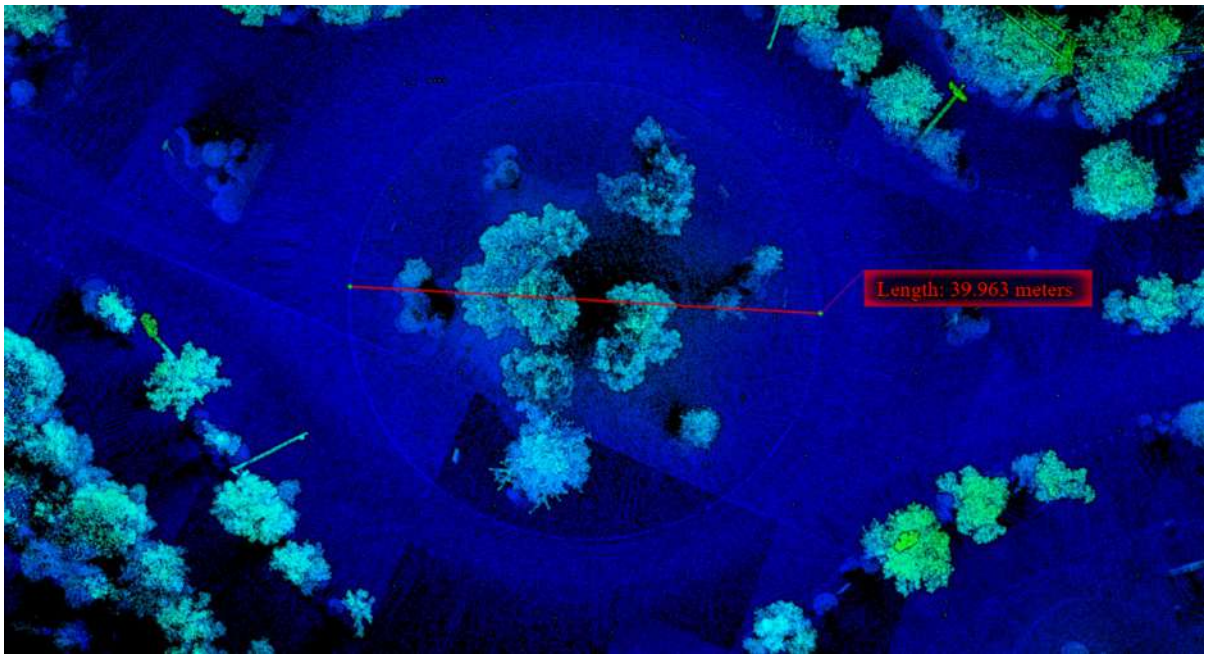
3.The effect of hovering the mouse to detect the plane is shown in the figure.



4.The effect of hovering the mouse to detect the ridge line is shown in the figure.



5.The effect of hovering the mouse to detect corners is shown in the figure.



Select

Using the Select toolset, select and save the point cloud object of the current window.

- Polygon Selection
- Rectangle Selection
- Sphere Selection
- Circle Selection
- Line Above Selection
- Line Below Selection
- Plane
- Subtract
- Plane Above
- Plane Below
- Plane Distance Setting
- InCut
- OutCut
- Save and Cancel
- Cross Section

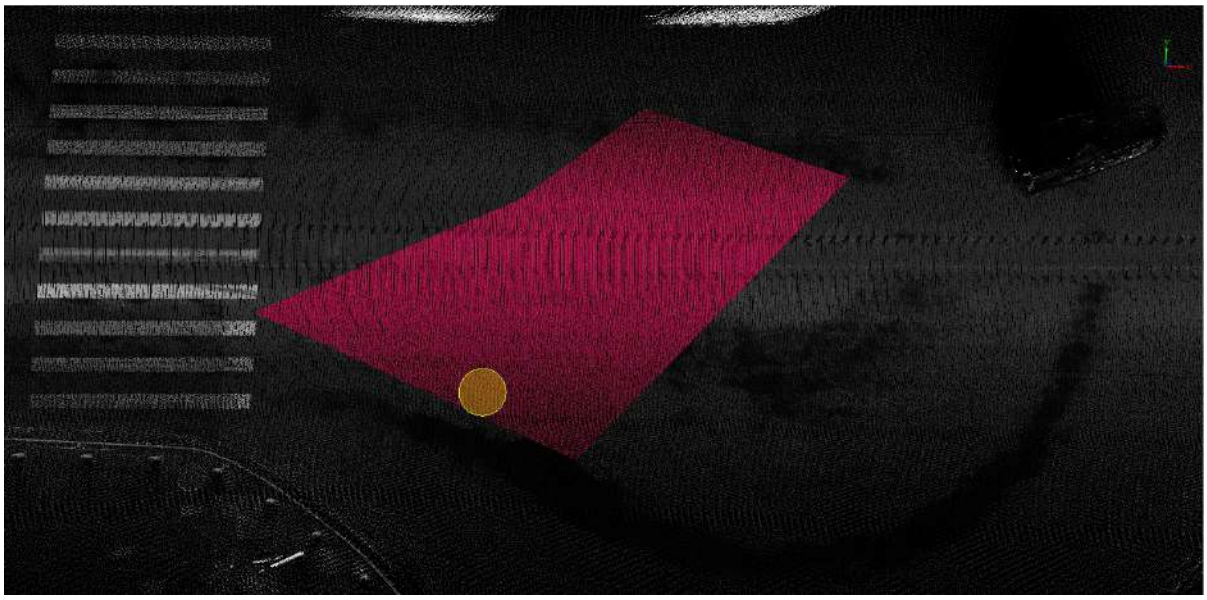
Polygon Selection

Function Description: This tool allows the user to select the point cloud by polygon.

Steps

1.After clicking the **Polygon Selection** button, the button is in the selected state and the function is activated.

2.In the window, left-click in turn to select the polygon vertex position, the program automatically forms a closed polygon, left-click the last vertex position to end the vertex selection, identify the point cloud in the selected polygon area for highlighting (red) display.



3.When the position of the polygon's vertex is misselected, right-click to cancel the last selected vertex, and you can cancel multiple times in a row.

4.After the first selection is over, the next selection can be made on the basis of the first selection.

5.Each time the selection area is selected, the corresponding plus selection area or minus selection area will be selected according to the reverse selection state.

> Note: This function is only for point cloud data.

Rectangle Selection

Function Description: This tool allows the user to select the point cloud by rectangular.

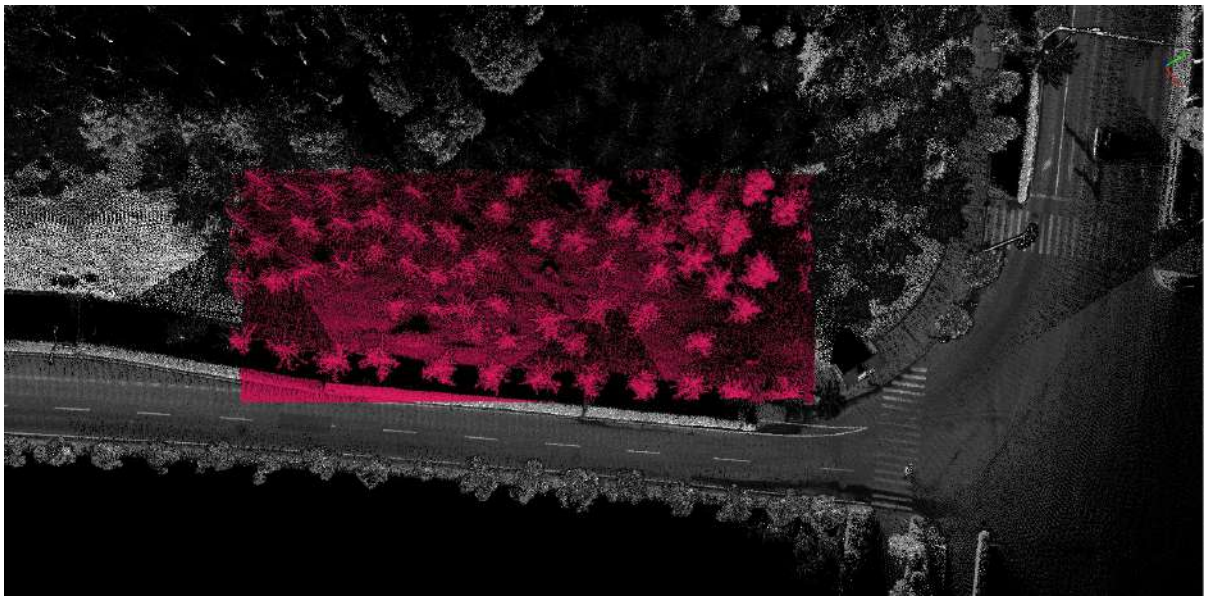
Steps

1.After clicking **Rectangle Selection** button, the button is in the selected state and the function is activated.

2.Left-click in the window, select the initial vertex position of the rectangle, and move the mouse. The current position of the mouse is regarded as another diagonal vertex of the rectangle, and the rectangular area is previewed in real time.

3.When the initial vertex position of the rectangle is selected incorrectly, right-click to cancel the selection of the vertex, and return to Step 2 to reselect.

4.Double-click the position of the other corner of the rectangle to end the selection, and identify the point cloud in the selected area for highlighting (red) display, as shown in the figure.



5.After the first selection is over, the next selection can be made on the basis of the first selection.

6.Each time the selection area is selected according to the reverse selection state, the corresponding plus selection area or minus selection area will be selected.

> Note: This function is only for point cloud data.

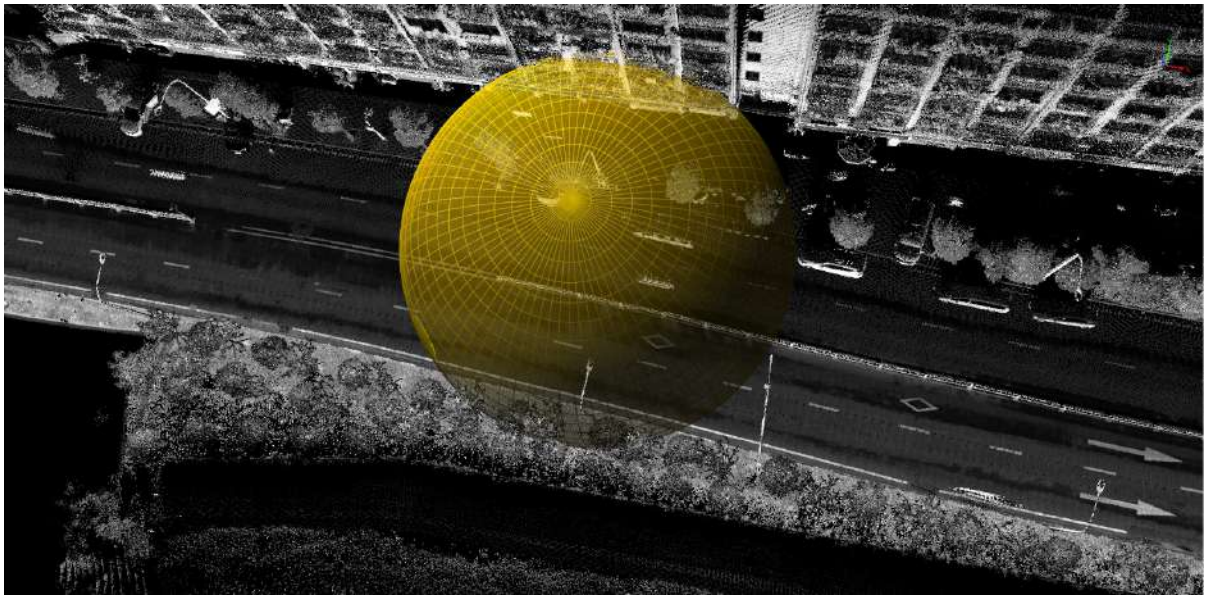
Sphere Selection

Function Description: This tool allows the user to select the point cloud by spherical.

Steps

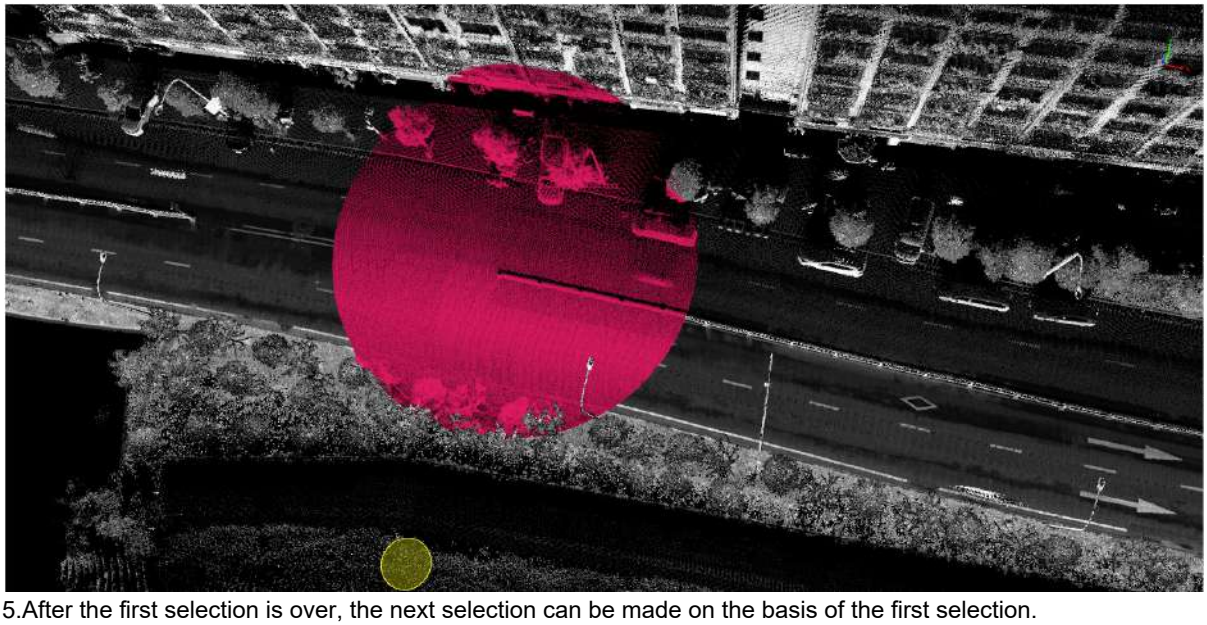
1.After clicking **Sphere Selection** button, the button is in the selected state and the function is activated.

2.Left-click in the window to select the position of the center point of the sphere, move the mouse, the mouse position is judged as the boundary point of the current ball, and the spherical area is previewed.



3.When the initial vertex position of the rectangle is selected incorrectly, right-click to cancel the selection of the vertex, and return to Step 2 to reselect.

4.Double-click the position of the other corner of the rectangle to end the selection, and identify the point cloud in the selected area for highlighting (red) display, as shown in the figure.



5.After the first selection is over, the next selection can be made on the basis of the first selection.

6.Each time the selection area is selected according to the reverse selection state, the corresponding plus selection area or minus selection area will be selected.

> Note: This function is only for point cloud data.

Circle Selection

Function Description: This tool allows the user to select the point cloud by a circle.

Steps

1.After clicking **Circle Selection** button, the button is in the selected state and the function is activated.

2.Left-click in the window to select the position of the center point of the circle, move the mouse, the mouse position is judged as the boundary point of the current circle, and the circular area is previewed.

3.When the position of the center point of the circle is selected incorrectly, right-click to deselect the center point, and return to the state of Step 2 to re-select the center point.

4.Double-click the circular boundary point with the left button to end the selection, and identify the point cloud in the selected area for highlighting (red) display.

5.After the first selection is over, the next selection can be made on the basis of the first selection.

6.Each time the selection area is selected according to the reverse selection state, the corresponding plus selection area or minus selection area will be selected.

> Note: This function is only for point cloud data.

Line Above Selection

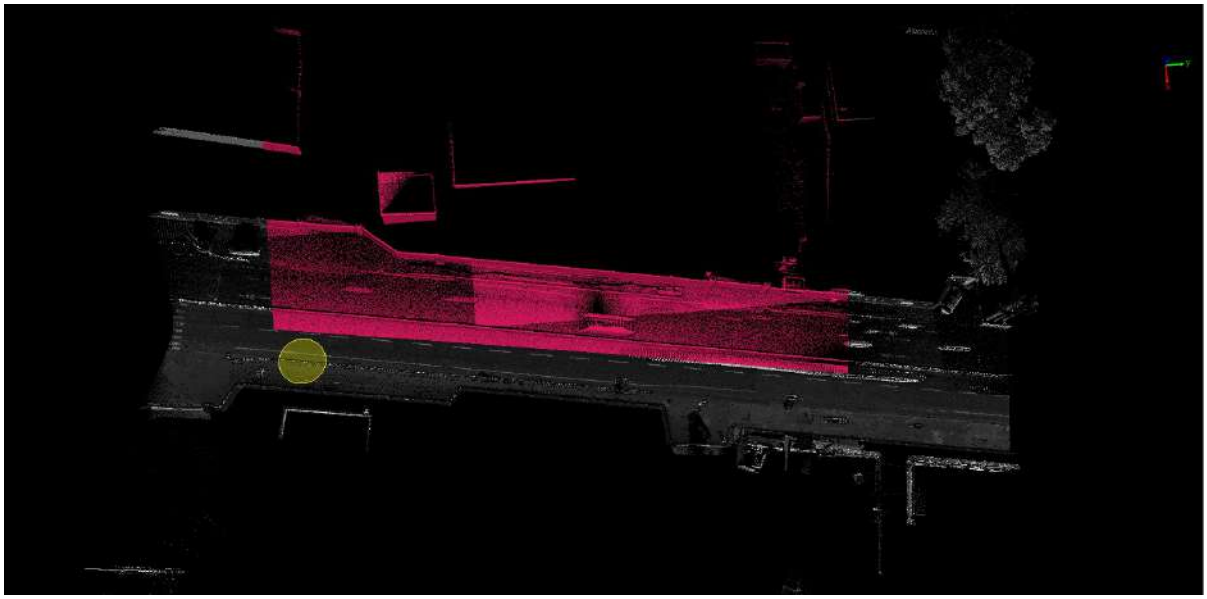
Function Description: This tool allows the user to select the point cloud according to the online method.

Steps

1.After clicking **Line Above Selection**  button, the function is activated and selection mode will be on.

2.Left-click in the window to select the starting point, move the mouse, and preview the online area.

3.Finally, double-click the left button to end the selection, and identify the point clouds in the selected area for highlighting (red) display.



4.After the first selection is over, the next selection can be made on the basis of the first selection.

5.Each time the selection area is selected according to the reverse selection, the corresponding plus selection area or minus selection area will be selected.

> Note: This function is only for point cloud data.

Line Below Selection

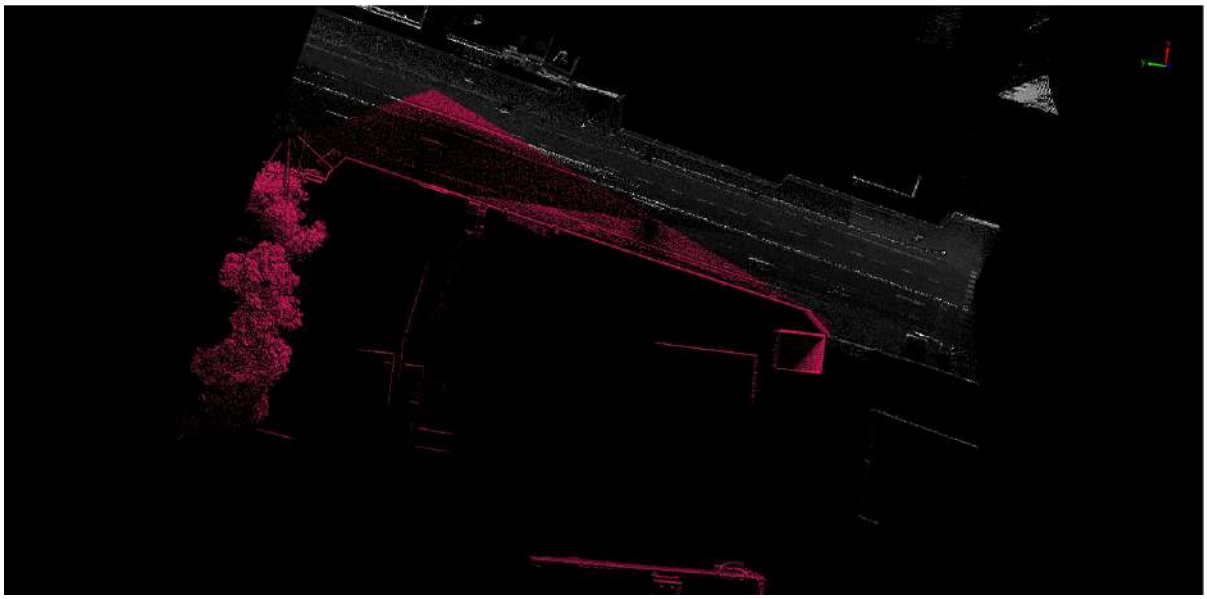
Function Description: This tool allows the user to select the point cloud according to the offline method.

Steps

1.After clicking **Line Below Selection**  button, the button is in selection mode and the function is activated.

2.Left-click in the window to select the starting point, move the mouse, and preview the offline area.

3.Finally, double-click the left button to end the selection, and identify the point clouds in the selected area for highlighting (red) display.



4.After the first selection is over, the next selection can be made on the basis of the first selection.

5.Each time the selection area is selected according to the reverse selection state, the corresponding plus selection area or minus selection area will be selected.

> Note: This function is only for point cloud data.

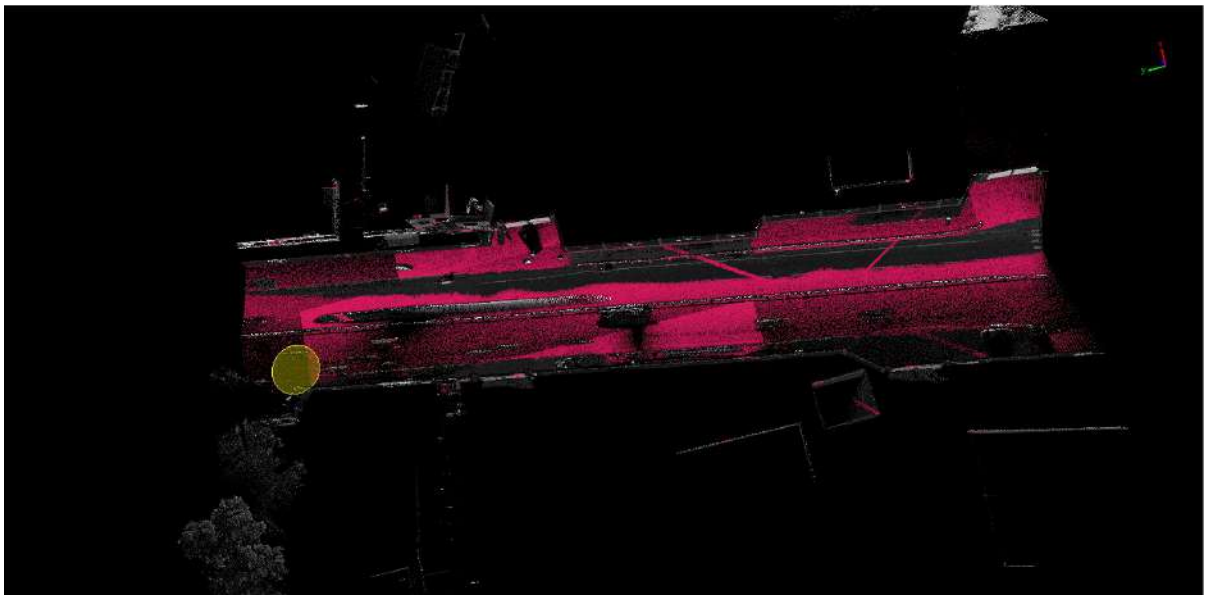
Plane

Function Description: This tool allows the user to select the point cloud by plane.

Steps

1.After clicking **Plane** button, selection mode is on and the function is activated.

2.Identify the point clouds in the selected area for highlighting (red) display, as shown in the figure.



3.After the first selection is over, the next selection can be made on the basis of the first selection.

4.Each time the selection area is selected according to the reverse selection state, the corresponding plus selection area or minus selection area will be selected.

> Note: This function is only for point cloud data.

Subtract

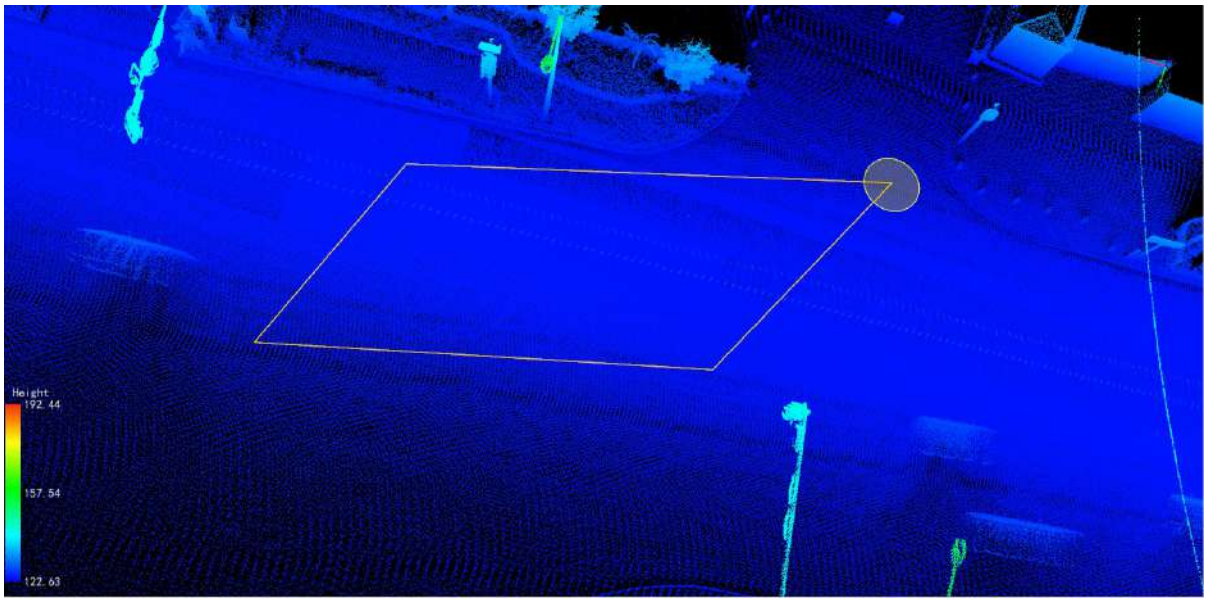
Function Description: This tool allows the user to deselect an existing selection and indicates the current selection state, and is used to control which the selected areas are added or subtracted. This function cooperates with Polygon Selection, Rectangle Selection, Sphere Selection, Circle Selection, Line Above Selection, Line Below Selection, Plane, Plane Above, Plane Below.

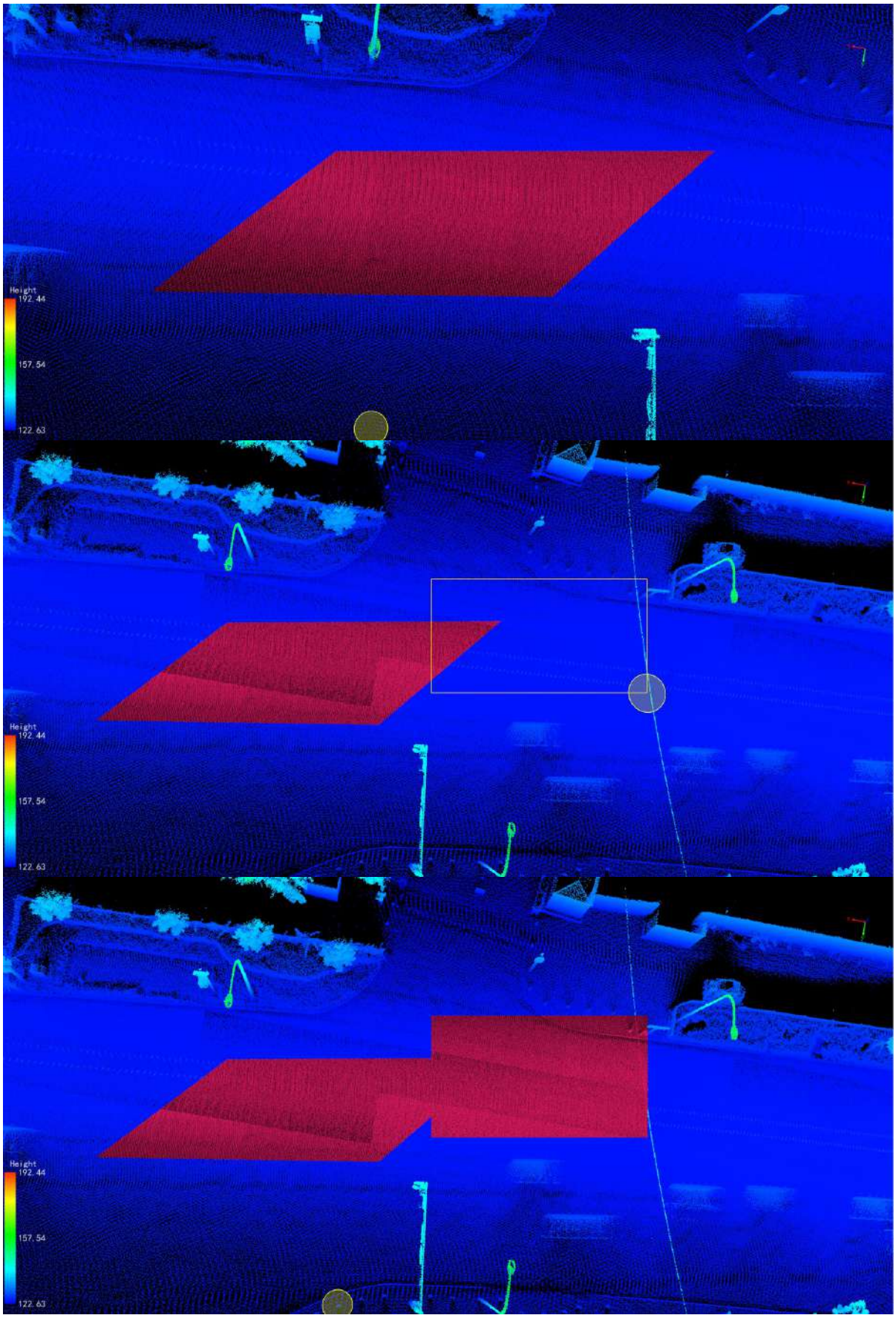
Steps

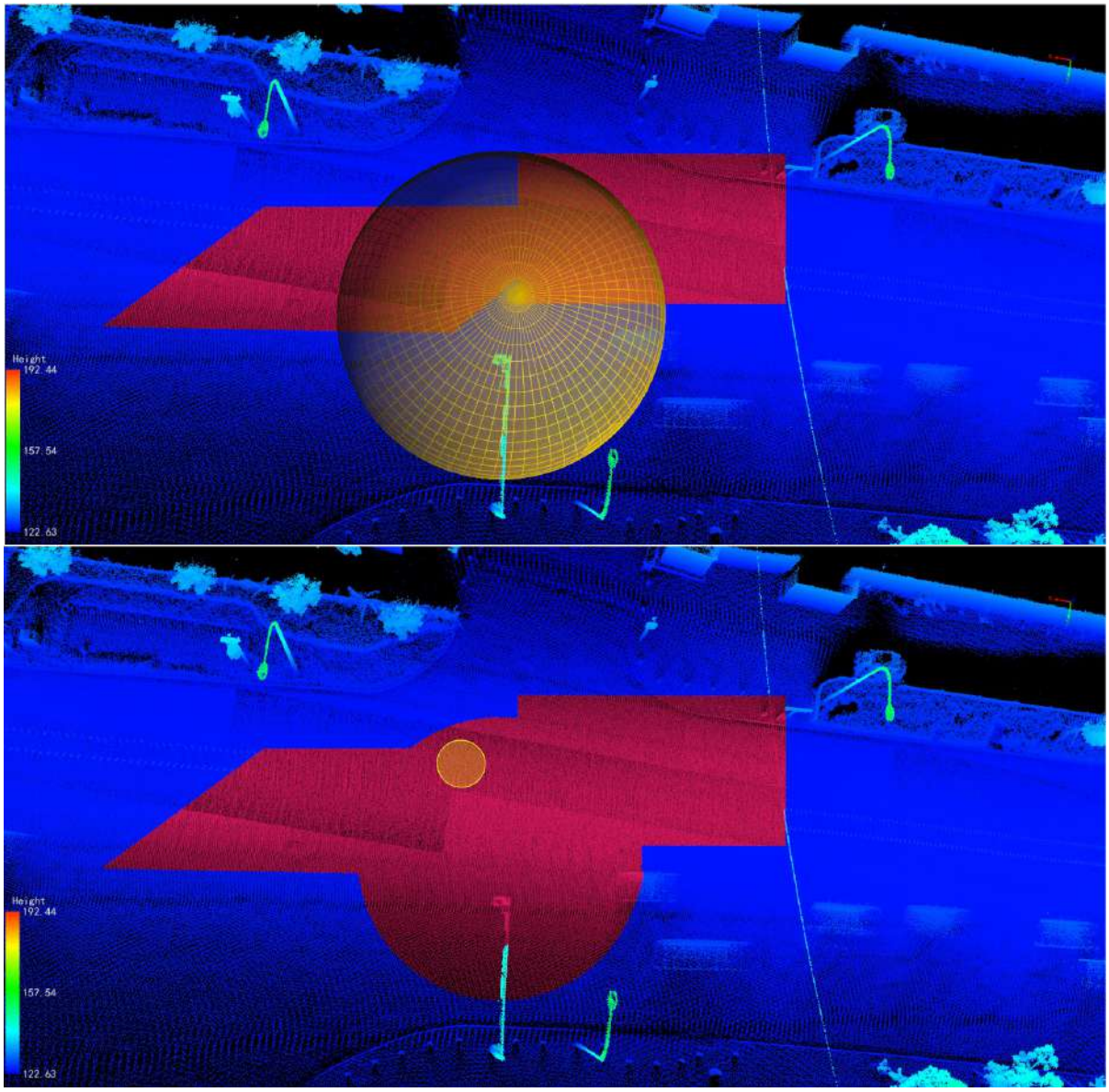
1.After clicking **Subtract** button, the button is in the selected state and the function is activated.

> Note: This function cannot work alone, when a certain selection tool (Polygon Selection, Rectangle Selection, Sphere Selection, Circle Selection, Line Above Selection, Line Below Selection, Plane, Plane Above, Plane Below. This function can only be activated when it is in the active state.

2.If you do not press and select this function, when the button is in the unselected state, the selection result is in the plus selection state, and the area selected by the current selection tool is added as part of the selection area, as shown in the figure.

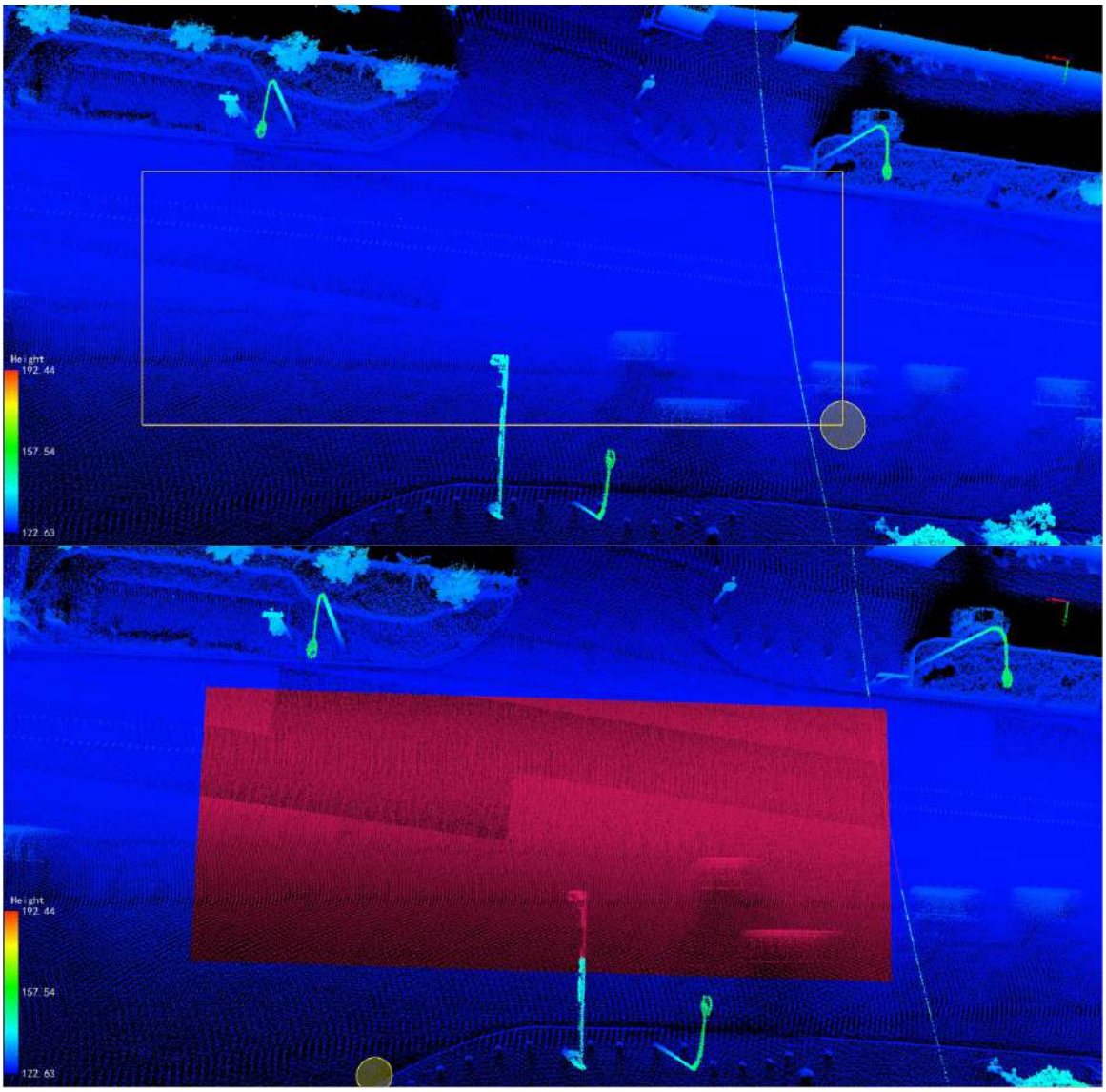



Height
192.44
157.54
122.63
Height
192.44
157.54
122.63
Height
192.44
157.54
122.63

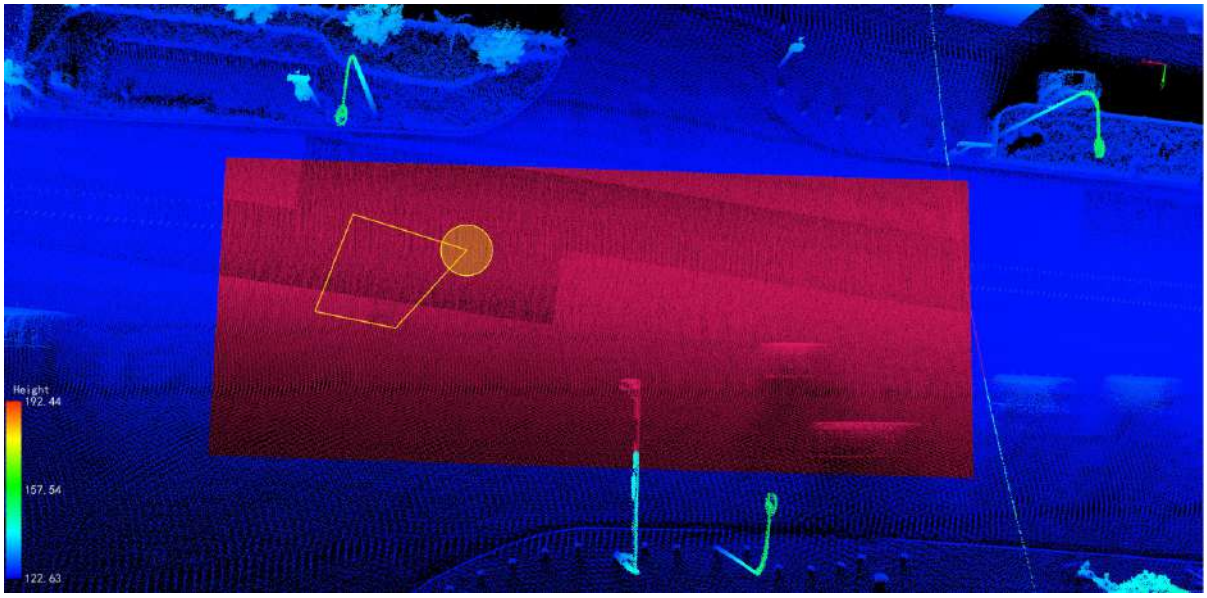


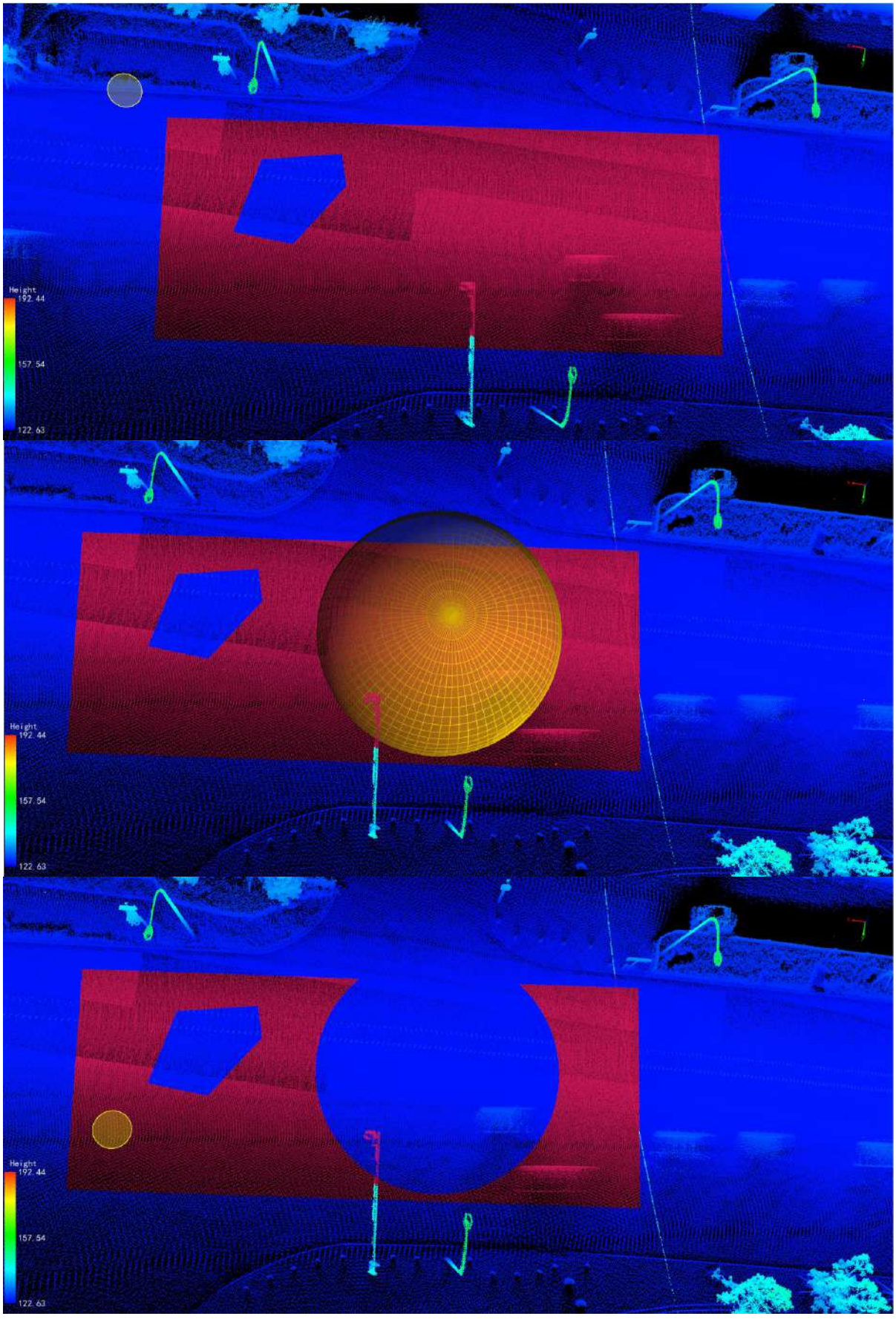
3.Press and select this function. When the button is in the selected state, the selection result is in the minus state, and the selected area is subtracted from the common part of the currently newly selected area as the final selection result.

- 3.1 Select an initial selection area in the plus selection state.



- 3.2 Check the deselect function, and select polygons and spheres in turn in the deselect mode. The selected area will be deselected from the original area, as shown in the figure.





Parameter setting

- **Shortcut key**: Ctrl + z, cancel the result of the last selection.

> Note: This function is only for point cloud data.

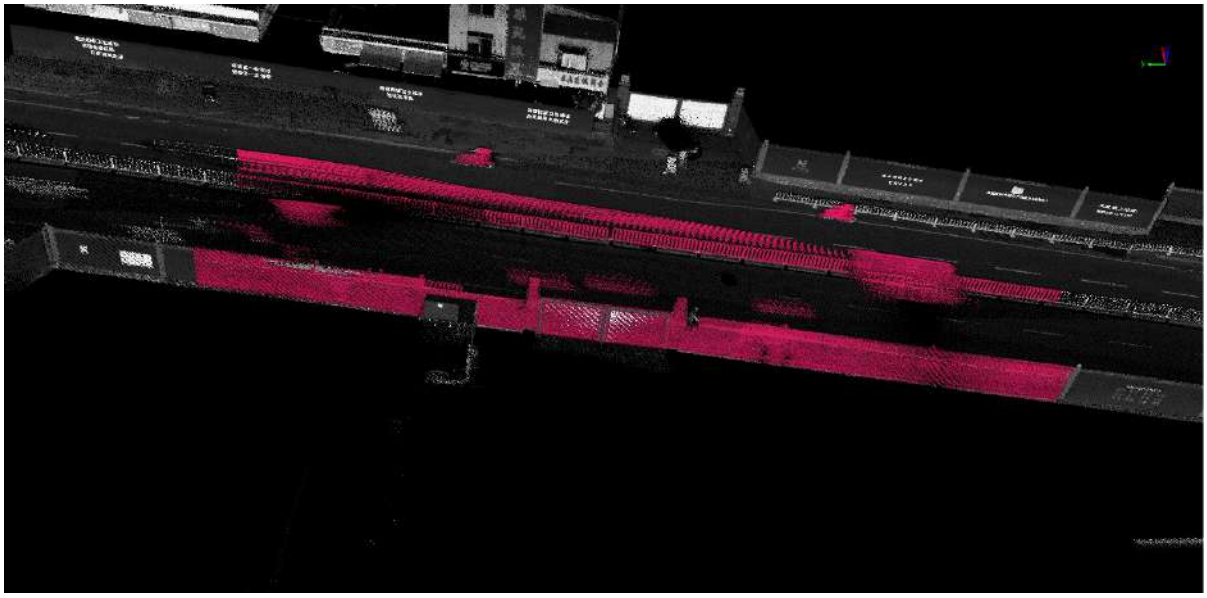
Plane Above

Function Description: This tool allows the user to draw a plane, and select the point clouds above the plane.

Steps

1.Click **Plane Above**  button , the selection mode is on and the function is activated.

2.Identify the point clouds in the selected area for highlighting (red) display, as shown in the figure.



3.After the plane is drawn, when there is no point cloud on the plane, there will be a pop-up prompt：

4.After the first selection is over, the next selection can be made on the basis of the first selection.

5.Each time the selection area is selected according to the reverse selection state, the corresponding plus selection area or minus selection area will be selected.

> Note: This function is only for point cloud data.

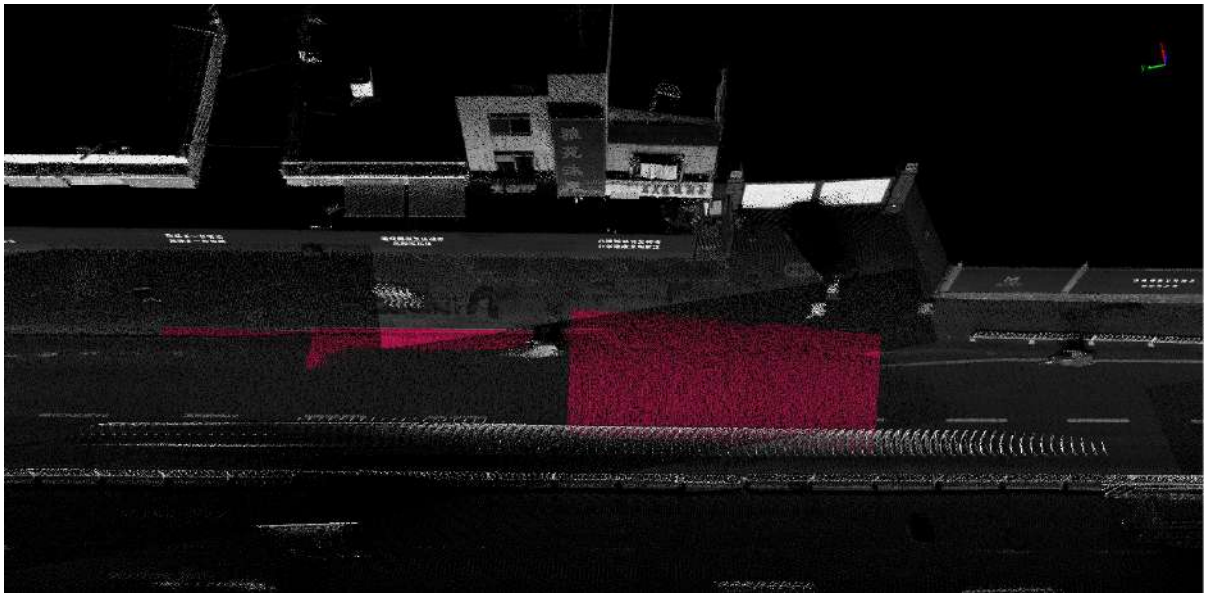
Plane Below

Function Description: This tool allows the user to draw a plane, and select the point cloud below the plane.

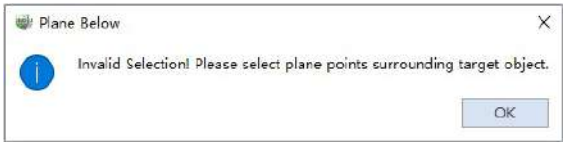
Steps

1.Click **Plane Below**  button, the selection mode is on and the function is activated

2.Identify the point clouds in the selected area for highlighting (red) display, as shown in the figure.



3.After the plane is drawn, when there is no point cloud under the plane, there will be a pop-up prompt.



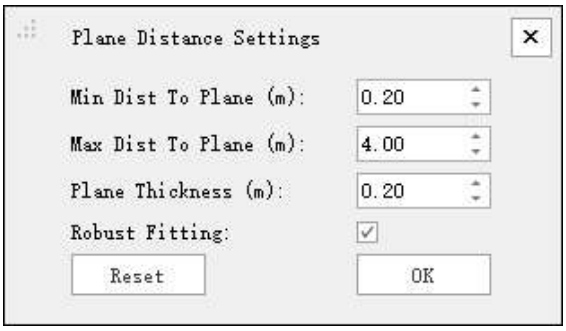
4.After the first selection is over, the next selection can be made on the basis of the first selection.

5.Each time the selection area is selected according to the reverse selection state, the corresponding plus selection area or minus selection area will be selected.

> Note: This function is only for point cloud data.

Plane Distance Setting

Function Description: This tool allows the user to set the plane distance.



Description

- **Min Dist To Plane (m)**: Set the minimum distance from the plane.
- **Max Dist To Plane (m)**: Set the maximum distance from the plane.
- **Plane Thickness (m)**: Set the parameters of plane thickness.
- **Robust Fitting**: The effect is better, but the speed is slower.
- **Reset**: Click this button to restore the default values of all parameters.

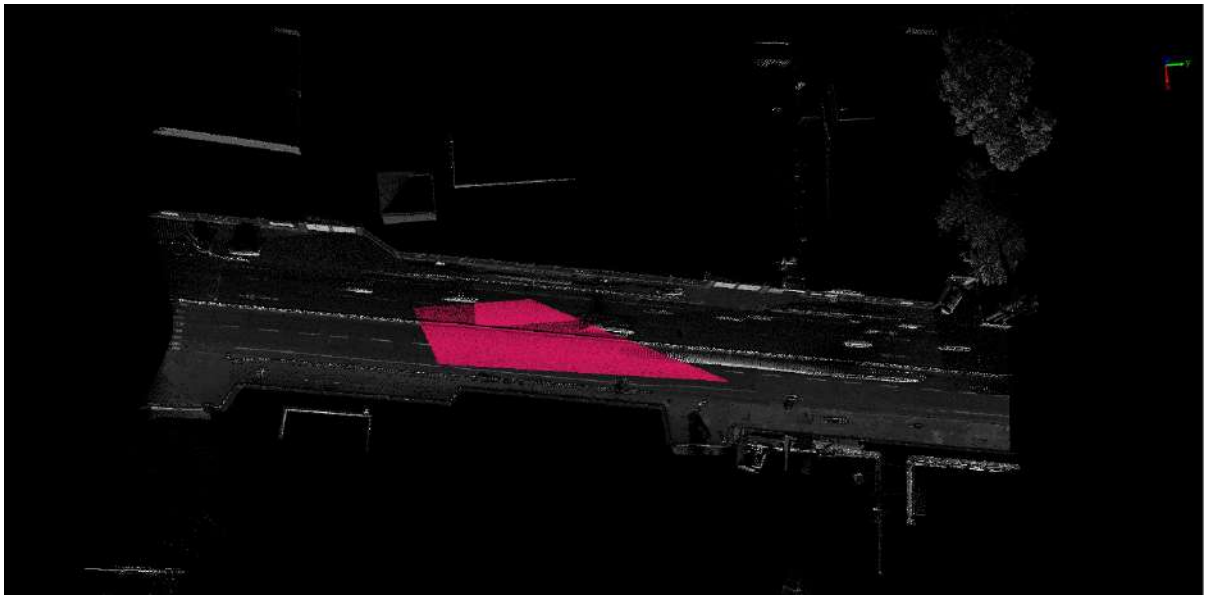
InCut

Function Description: This tool allows the user to crop all point cloud data in the window, select the point cloud in the area based off the currently selected area, and hides the point cloud outside the area.

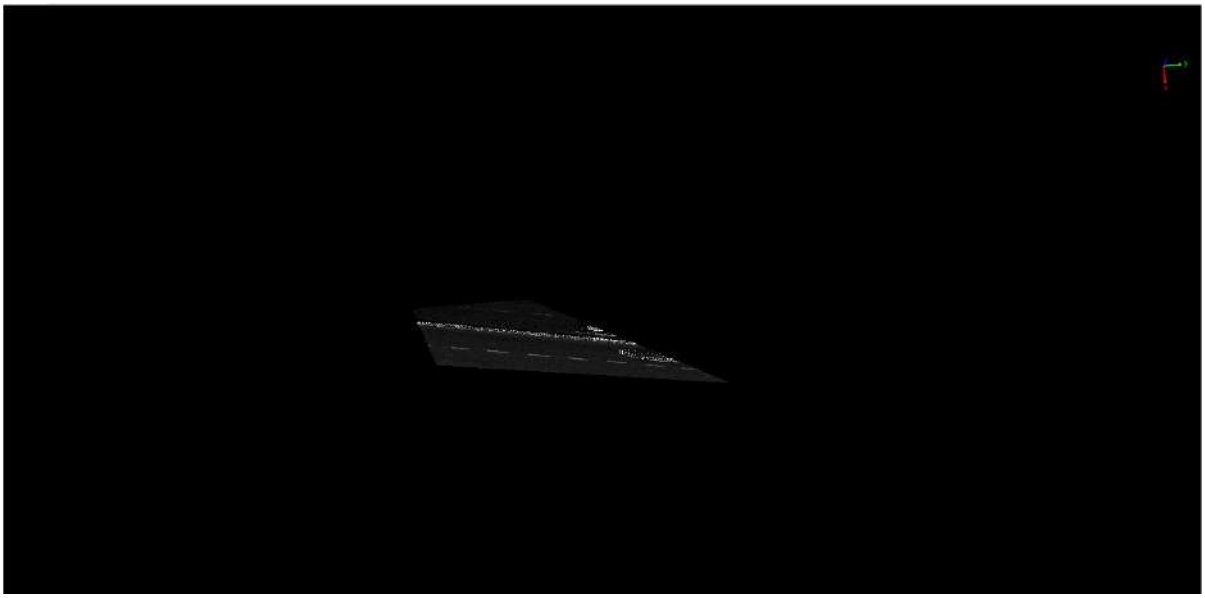
Steps

Click **InCut** button:

1.First select the area, refer to the selection tools (polygon selection, rectangle selection, spherical selection, etc.) and reverse selection to form the required selection area, and identify the point clouds in the selected area for highlighting (red) display.



2.Click this function to form the effect of internal cutting, as shown in the figure.



3.After one cutting, multiple selections and cut (external cut) can be performed on the result of point cloud data (polygon selection, rectangle selection, spherical selection, etc.).

Parameter setting

Shortcut key: Ctrl + Z, cancel the result of the last crop, and the selection area corresponding to the crop will also be cancelled.

> Note: This function is only for point cloud data.

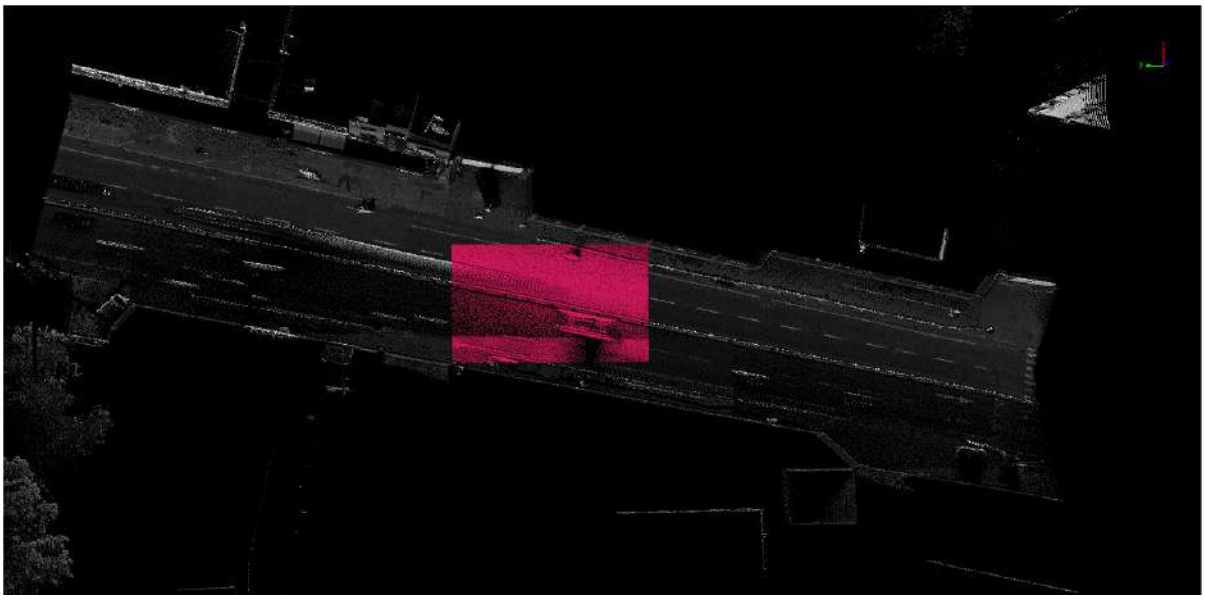
OutCut

Function Description: This tool allows the user to crop all point cloud data in the window and select the point cloud in the area based off the currently selected area, and hides the point cloud outside the area.

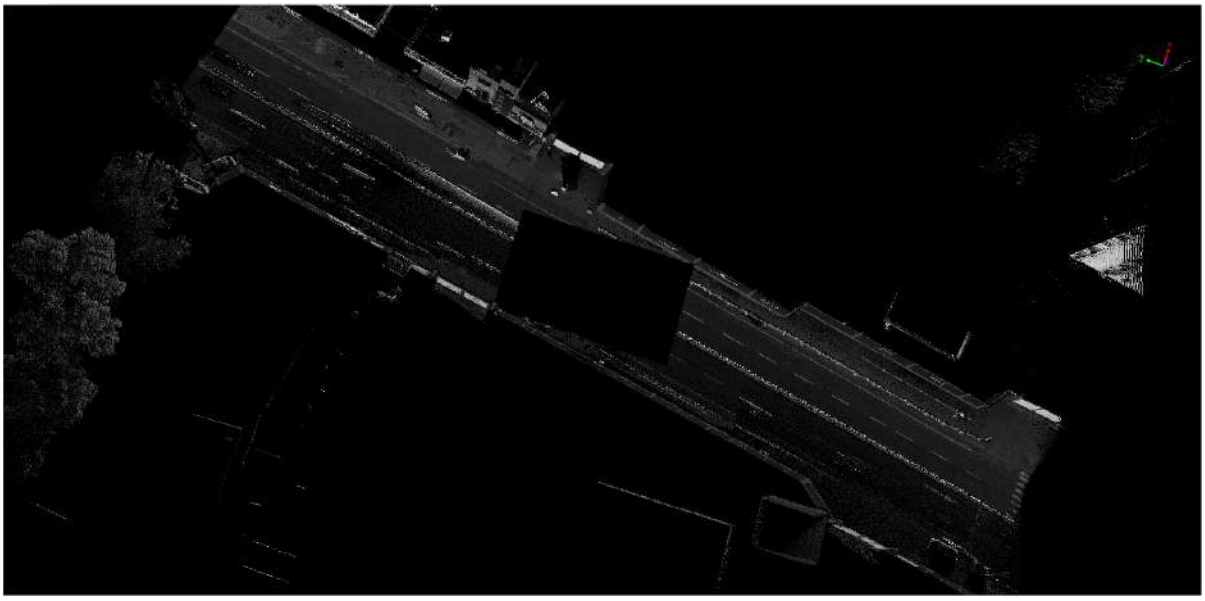
Steps

Click **OutCut**  button:

1.First select the area, refer to the selection tools (polygon selection, rectangle selection, spherical selection) and reverse selection, etc.) to form the required selection area, and identify the point clouds in the selected area for highlighting (red) display.



2.Click this function to form the effect of outer cropping, as shown in the figure.



3.After one cutting, multiple selections and cut (including internal cutting) can be performed on the result of point cloud data.

Parameter setting

Shortcut key: Ctrl + Z, cancel the result of the last crop, and the selection area corresponding to the crop will also be cancelled.

> Note: This function is only for point cloud data.

Save and Cancel

Save

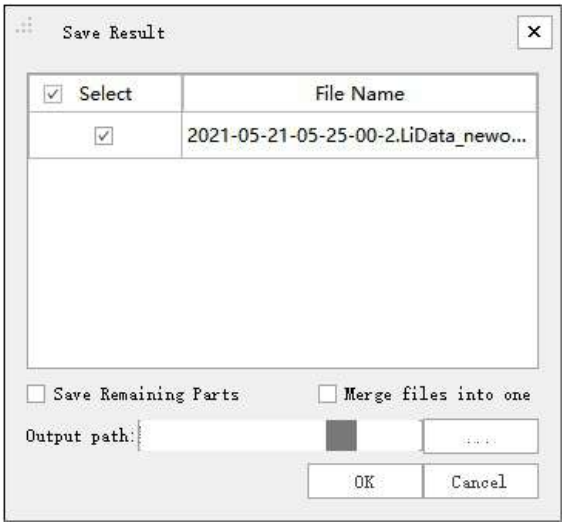
Function Description: This tool allows the user to save the resulting point cloud after cropping into a new point cloud file.

Steps

Click **Save**  After the button:

1.First perform the cutting operation to form the cutting result, refer to the inner cutting and outer cutting.

2.In the case of cropping results, this function is activated. Click this function to pop up the save settings interface, as shown in the figure.



3.Select the original point cloud data wanted to participate in cropping and saving.

4.Check whether to merge all cropped point clouds and save them to a point cloud file as needed.

5.Select the directory where the cropped file is saved, and the system will form a new cropped file name based on the original file name and the current system time. The naming rule is Original file name CutResult time information ".LiData".

6.After saving, the original selection and cropping operations will be automatically cancelled, and the user will be asked if he wants to add the cropped data to the current project, as shown in the figure.

7.The user selects yes or no according to his needs, and if he selects yes, the corresponding cropped save file will be loaded into the project.

Cancel

Click **Cancel** <img src="../../../../images/Icon/Tools/Tools/Select_Cancel.png" alt="InputFile(s)" width="30" style = "vertical-align:middle"/button, cancel the result of the last cropping, and the selection area corresponding to the crop will also be cancelled.

Parameter setting

- Shortcut key: Ctrl + Z, cancel the result of the last crop, and the selection area corresponding to the crop will also be cancelled.

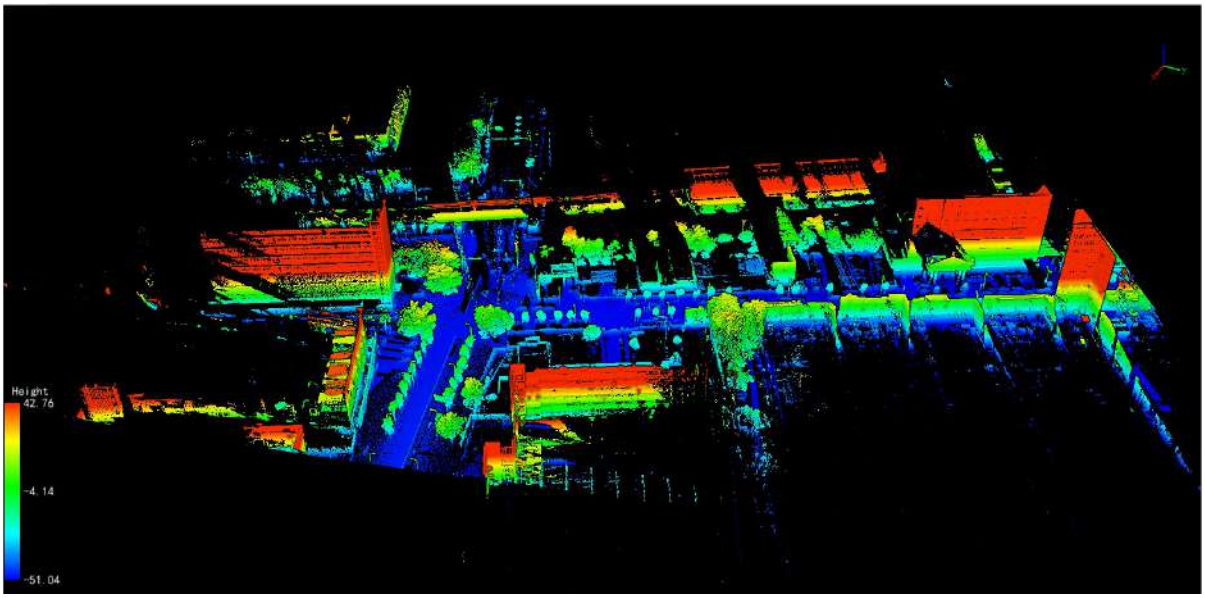
Cross Section

Function Description: This tool allows the user to crop data based on a bounded box.

Steps

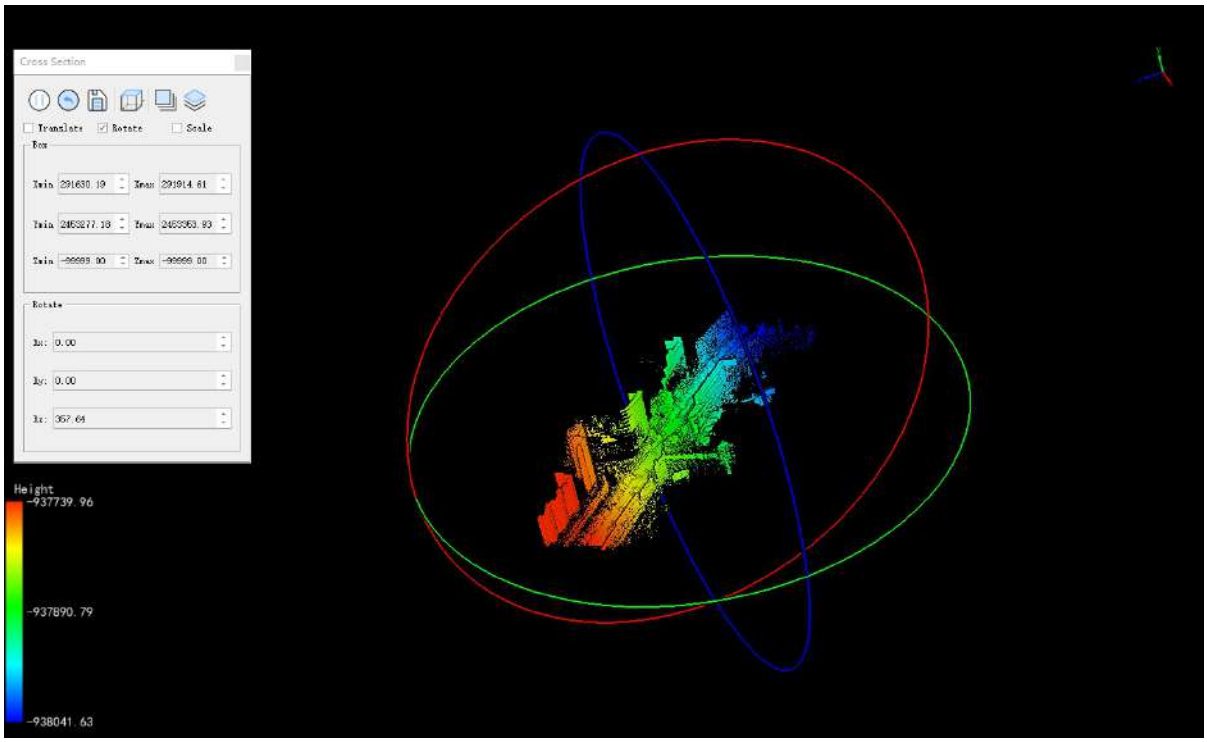
1.Open the LiDAR360 MLS software and load the corresponding project file.

2.The point cloud display mode is set to elevation +EDL.



The point cloud is displayed as elevation+EDL mode

3.Click **Cross Section**  button, the 3D view window pops up the interface shown in the figure below. The three options in the upper left corner of the window (**Translate, Rotate, Scale**) control whether the enclosing box can be translated, rotated, and scaled separately, and the corresponding options can be checked to make it effective. The mouse clicks (does not pop up) the corner of the enclosing box (green square) in the view and adjusts the position. You can adjust the zoom of the enclosing box X, Y, and Z (the **Zoom** option must be checked).The mouse clicks (does not bounce) on the 6 faces of the enclosing box and adjusts the position, you can adjust the position of the enclosing box X, Y, and Z (the **Translate** option must be checked); the mouse clicks (does not bounce) on the spherical surface composed of red and green curves in the view and adjusts the position, you can adjust the direction of the enclosing box X, Y, and Z (the **Rotate** option must be checked).



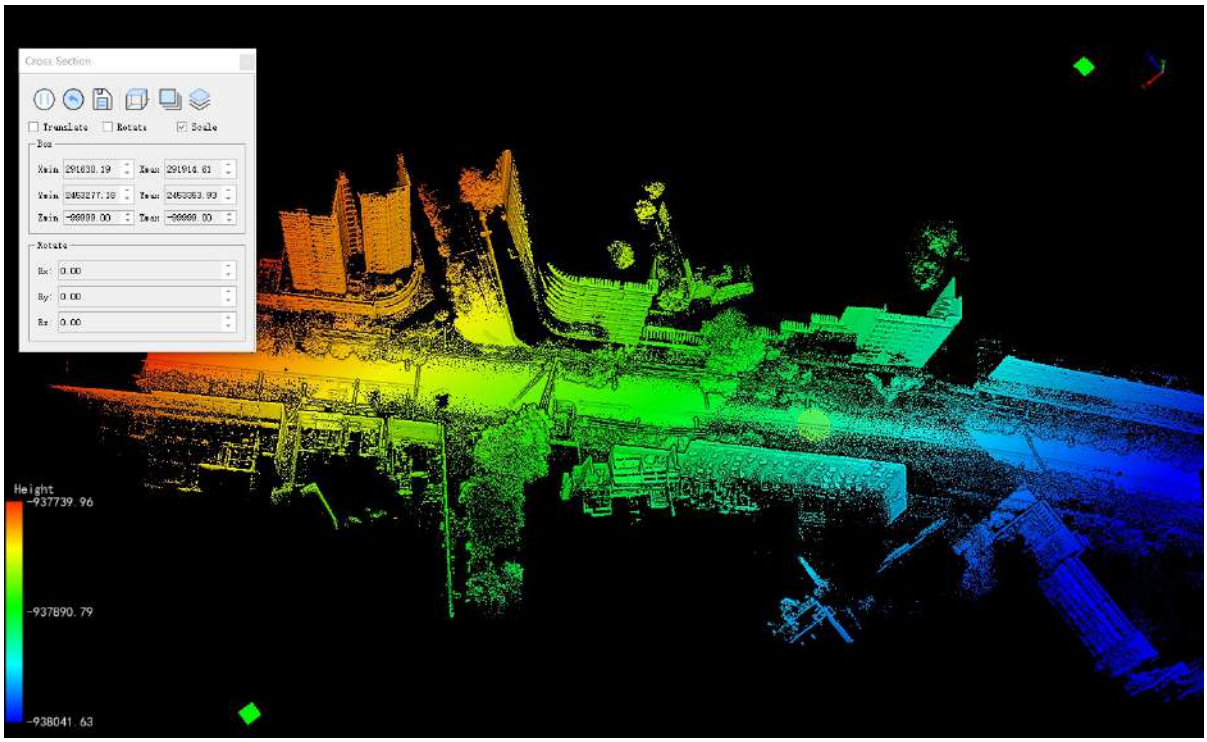
Adjust the enclosure

Parameter setting

- **Translate**: Control whether the enclosing box can be moved.
- **Rotate (Rotate)**: Control whether the enclosing box can be rotated.
- **Scale**: Controls whether the enclosing box can be scaled.
- **Enclosing Box Range (Box)**: The maximum and minimum values of the X, Y, and Z directions of the enclosing box can be precisely adjusted.
- **Surround Box Rotation (Rotate)**: The angle between the X, Y, and Z directions of the enclosing box can be precisely adjusted.

4.(Optional) Click  button to make enclosing box return to its initial state.

5.The picture below is shown the cropping according to the cross-section perpendicular to the Z-axis, only retaining the renderings of higher-elevation objects such as buildings.



Cut according to the z axis

6.Click  button to save the cropped point cloud.

7.Click  button to pause the cross-selection.

8.Click  button sets the view mode selection and displays each default view of the currently activated view.

Roaming

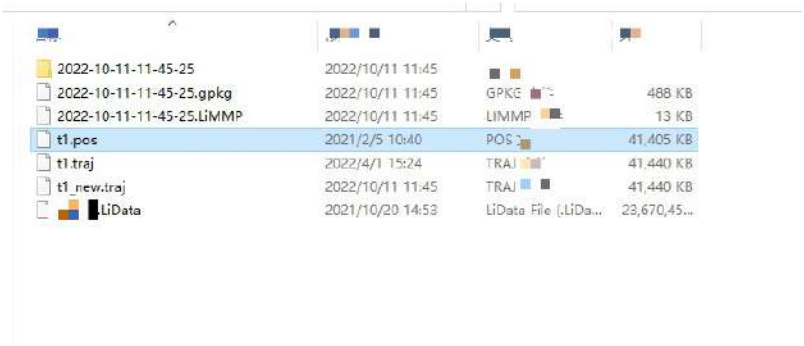
The Roaming toolset primarily allows the user to track roaming and camera roaming. Just switch the top menu bar to **tool**.

Trajectory Roaming

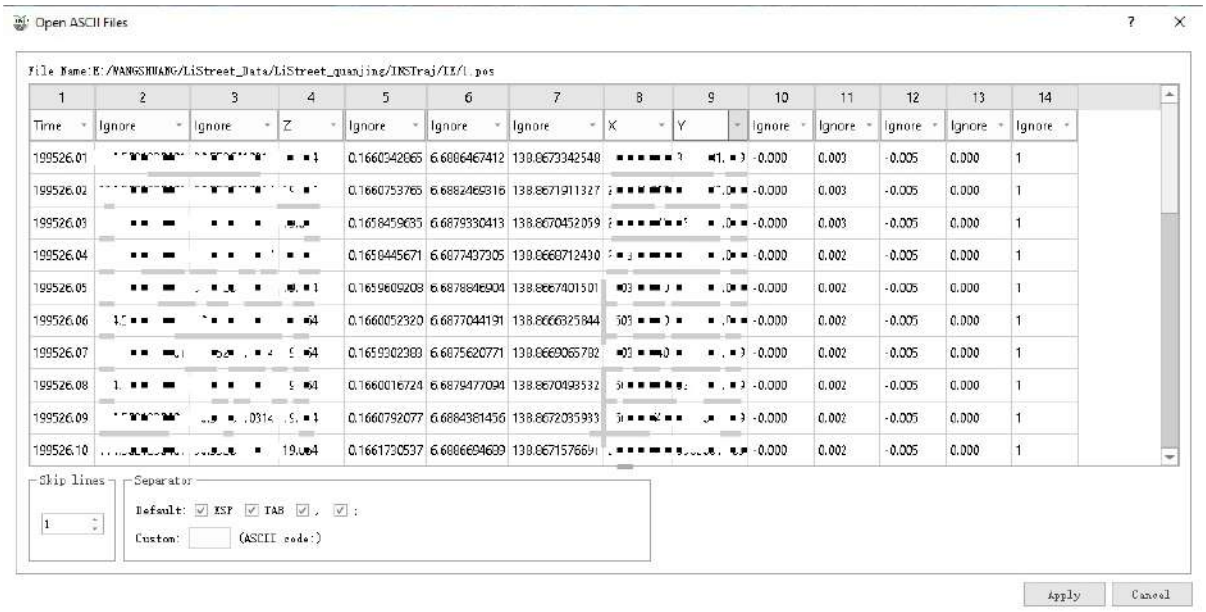
Function Description: Based on the roaming of the data trajectory, this tool allows the user to control the camera and move it according to the trajectory points, imitating the process of the vehicle collecting data in the real scene.

Steps

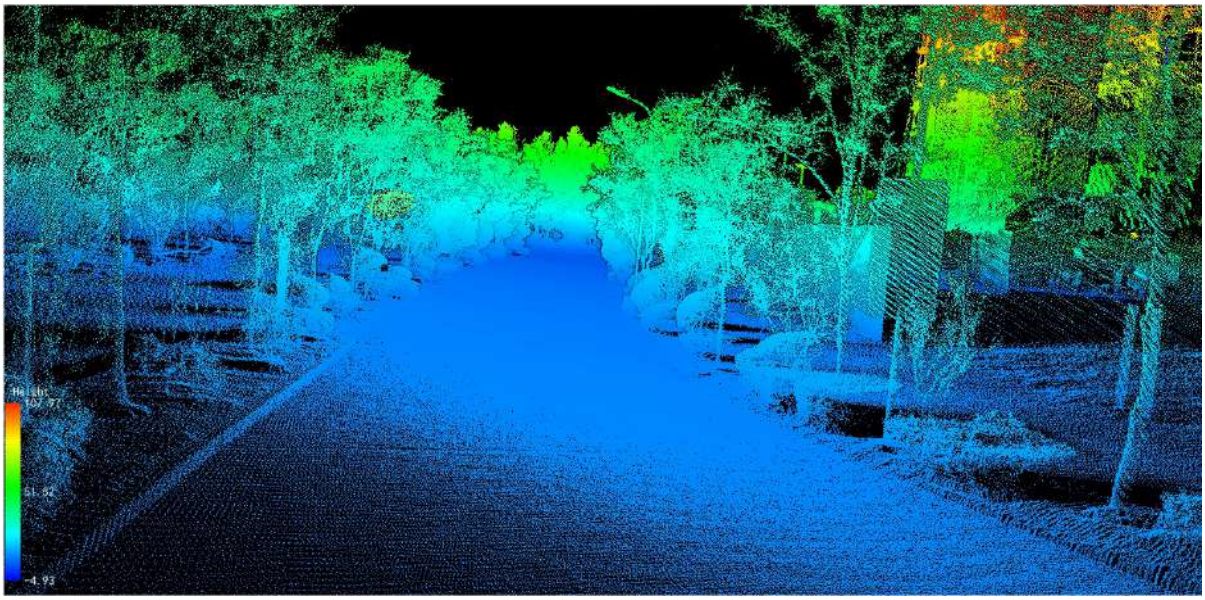
1.Click **Trajectory Roaming** button, the track selection interface pops up, which supports the import of track files in multiple formats, and generally selects the solved by default *.pos* format is enough.



2.According to the track file, select the track point to be loaded and the column corresponding to the XYZ coordinates, and click OK.



3.After the track is loaded, the software roams according to the loaded track points. As shown in the figure below, the roaming viewing angle is the driving viewing angle of the vehicle.



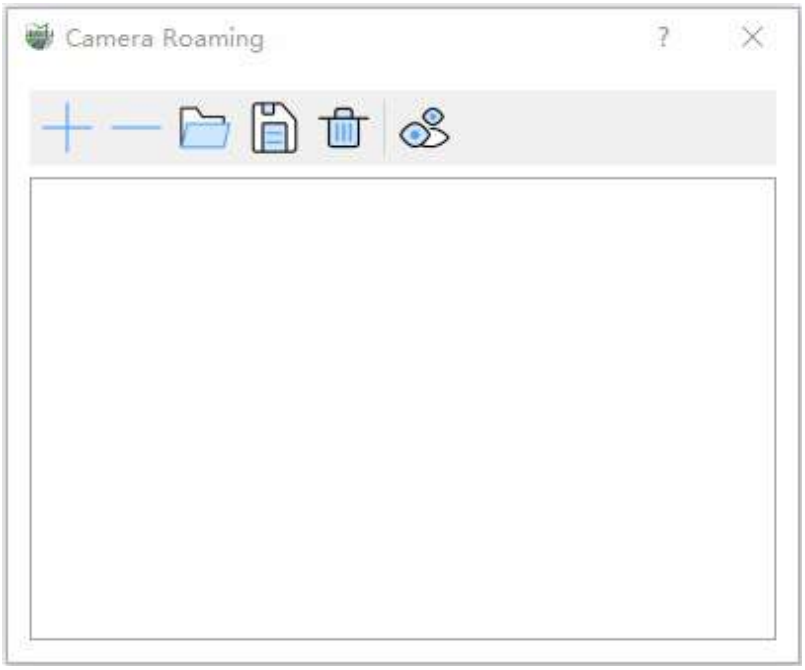
4.The user can click **Trajectory Roaming** button again to end the track roaming operation.

Camera Roaming

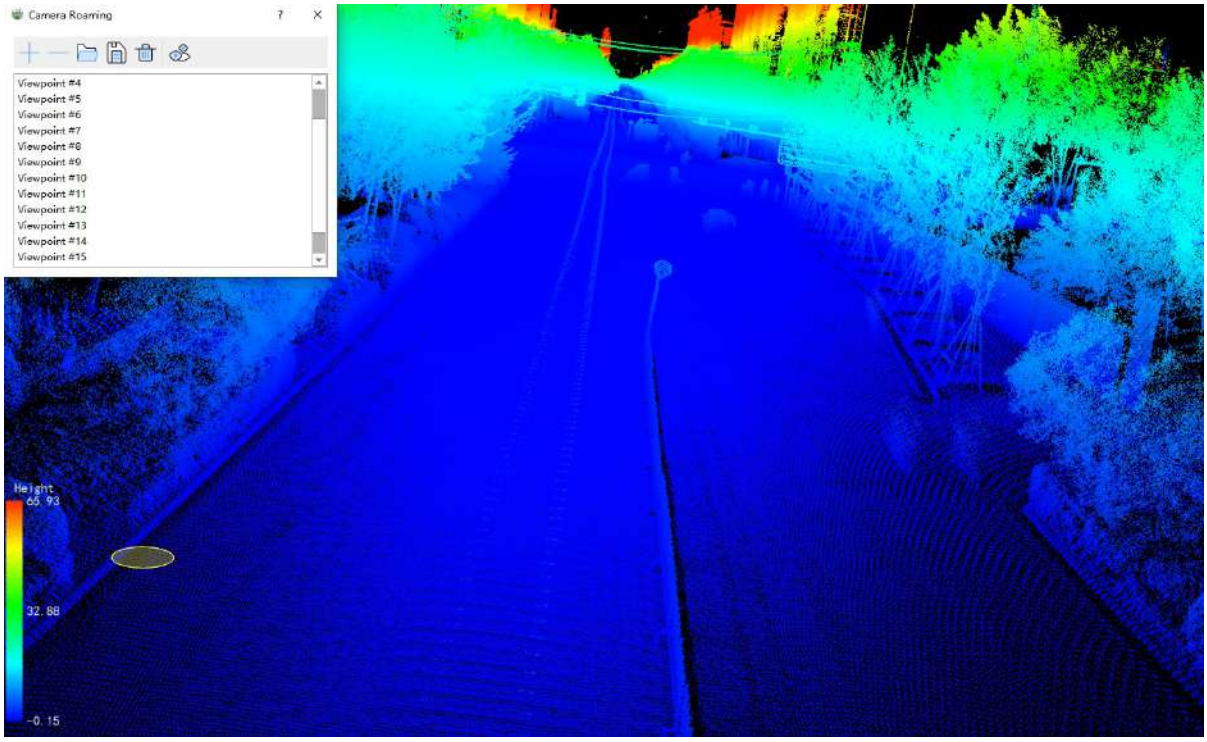
Function Description: Based on the roaming of user-defined viewpoints, This tool allows the user to control the camera can be and move it according to the viewpoints, as well as imitating the process people observing data in real scenes is realized, which enhances the immersion and presentation of the scene.

Steps

1.Click **Camera Roaming** button, the camera roaming dialog box pops up.

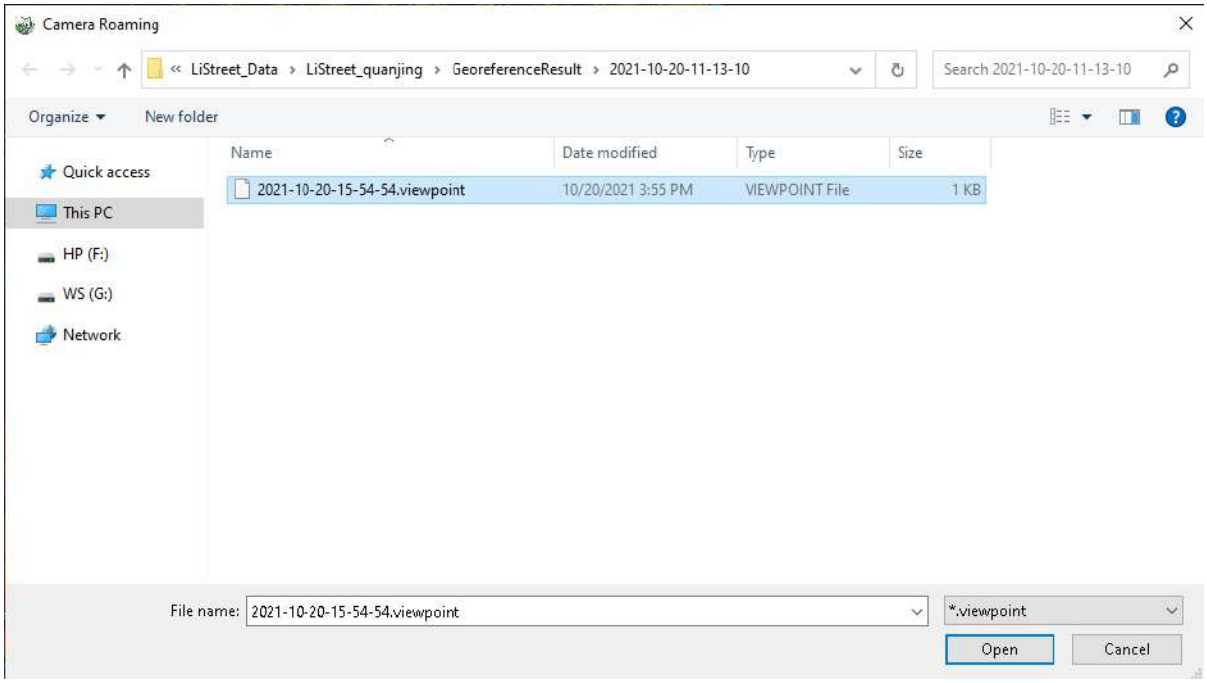


2.By adjusting the appropriate viewing angle, the user clicks **Add Current Viewpoint** button, add the current viewpoint position to the roaming path, as the keyframe of the roaming, the new viewpoint will appear in the viewpoint list box.

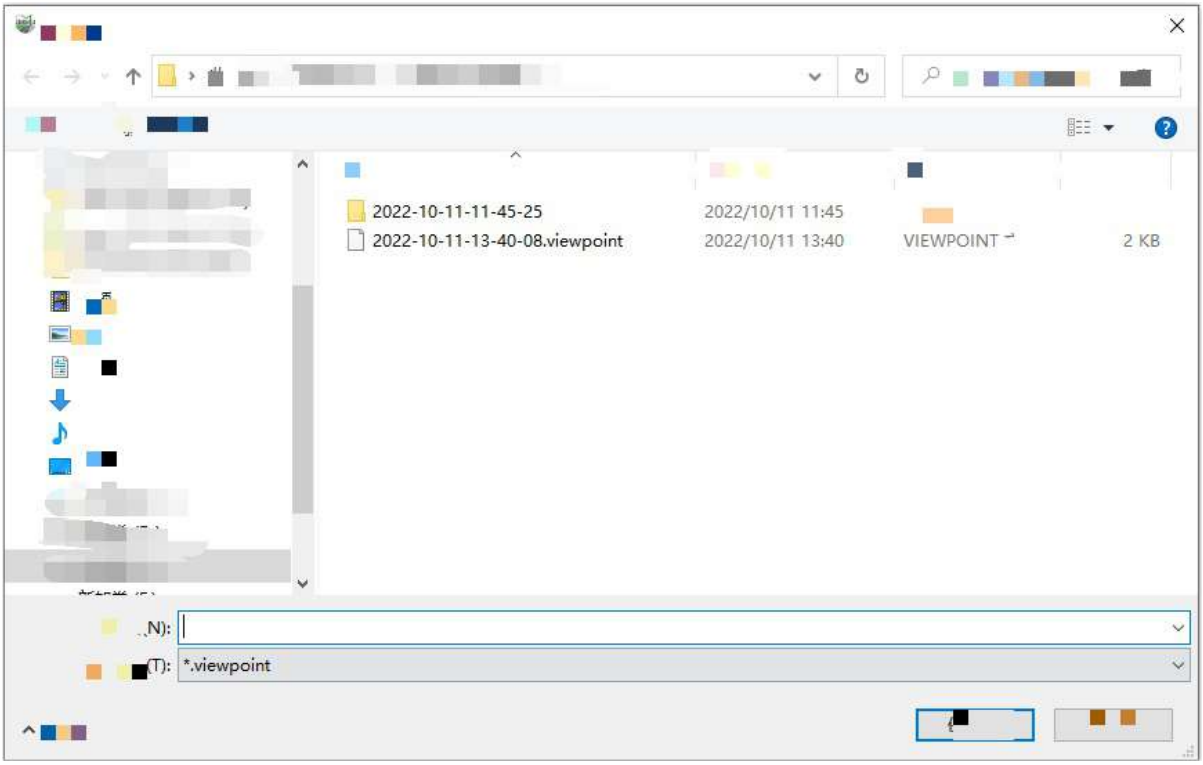


3.Users can also click **Remove Selected Viewpoint (optional)** button to delete unwanted or problematic viewpoints.

4.In the case of an existing viewpoint file, the user can also click **Load ViewpointS (optional)** button, load keyframe information through the viewpoint list file, and display it in the viewpoint list box.

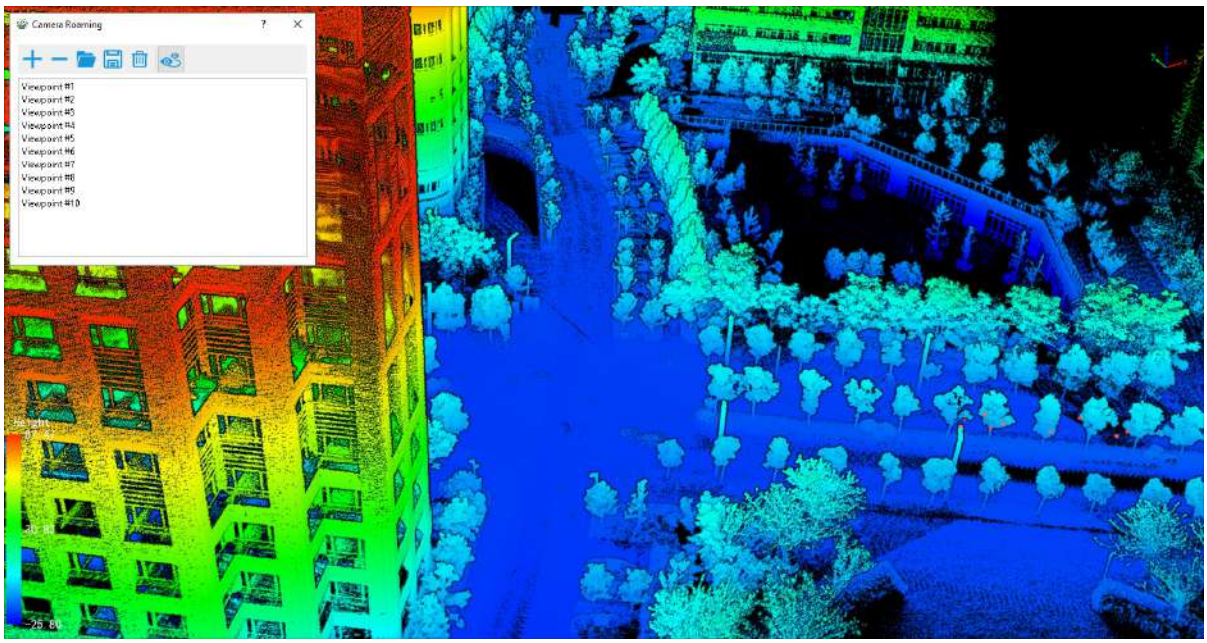


5.After adjusting the viewpoint, the user can click **Save Viewpoints (optional)** button, to save all viewpoints in the viewpoints list box as keyframe information in a viewpoints list file.



6.Users can click **Clear Viewpoints (optional)** button, clear all viewpoints in the viewpoints list box.

7.After confirming all viewpoints, the user clicks **Roaming** button, at this time, the three-dimensional scene observation camera will be set to the position and posture corresponding to the viewpoint, for the user to preview the scene information corresponding to the viewpoint.



8.The user clicks **Camera Roaming** button again in the camera roaming box to stop roaming.

Import Shape Vector

Function Description: This tool allows the user to import shp files and add them to the software, which will be introduced in detail below. The dialog box layer for importing vectors is shown in the figure below.

Description

- **Add**: After clicking the button, a dialog box for selecting files will pop up, select the shp file, and the table in the dialog box will display a list of exported shp files.
- **Delete**: Delete the shp file that has been added.
- **Clear**: Clear all added shp files and clear the table.
- **Coordinate Conversion (optional)**: If coordinate conversion is checked, you can select the coordinate system of the current shp file and select the target coordinate system to convert the currently added shp format vector to the target coordinate system and add it to the software.

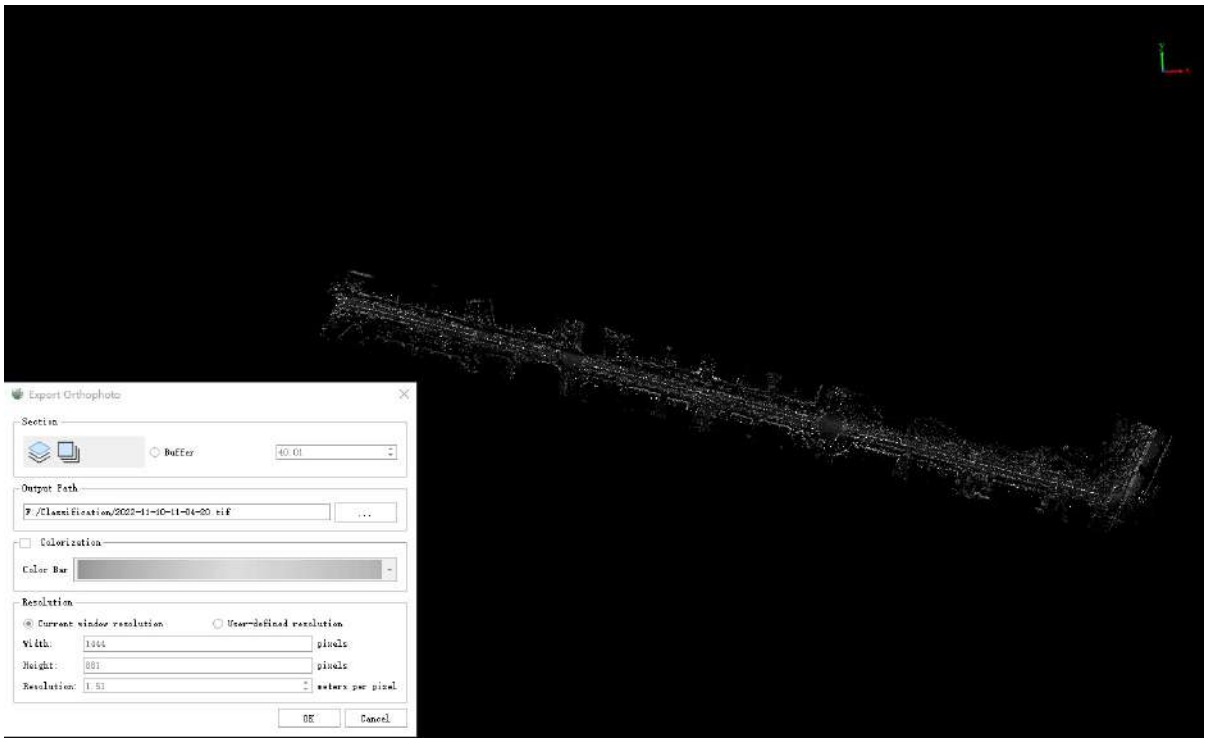
Export tool

Function Description: This tool allows the user to export engineering data into a variety of formats, as input to third- party software, for data display or further data processing. Switch the top menu to the Export Option in Tools.

- Orthophoto
- Point Cloud Conversion
- Projection
- Export Vector

Orthophoto

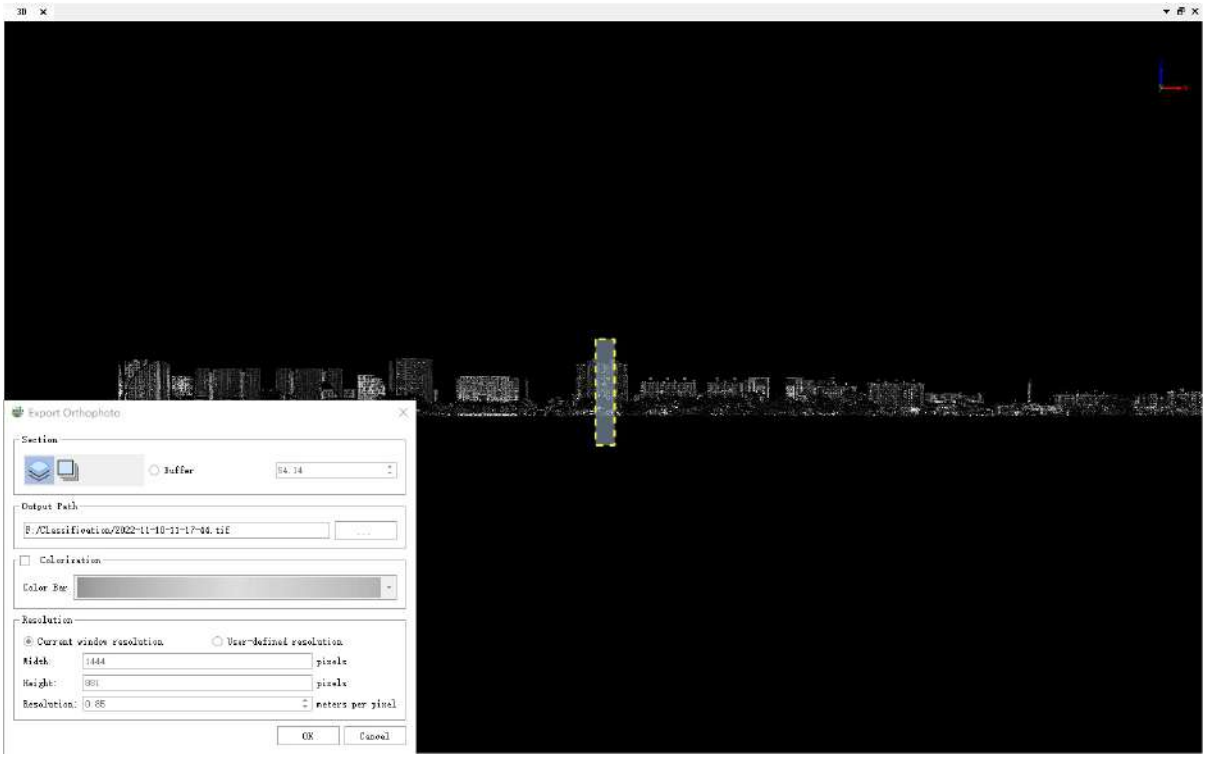
1.Click **Orthophoto** button, the export orthophoto image dialog box will pop up, as shown in the figure, the export orthophoto image can be combined with the horizontal cross-section tool and the vertical cross-section tool, and the exported orthophoto image can be opened and measured in other GIS software.



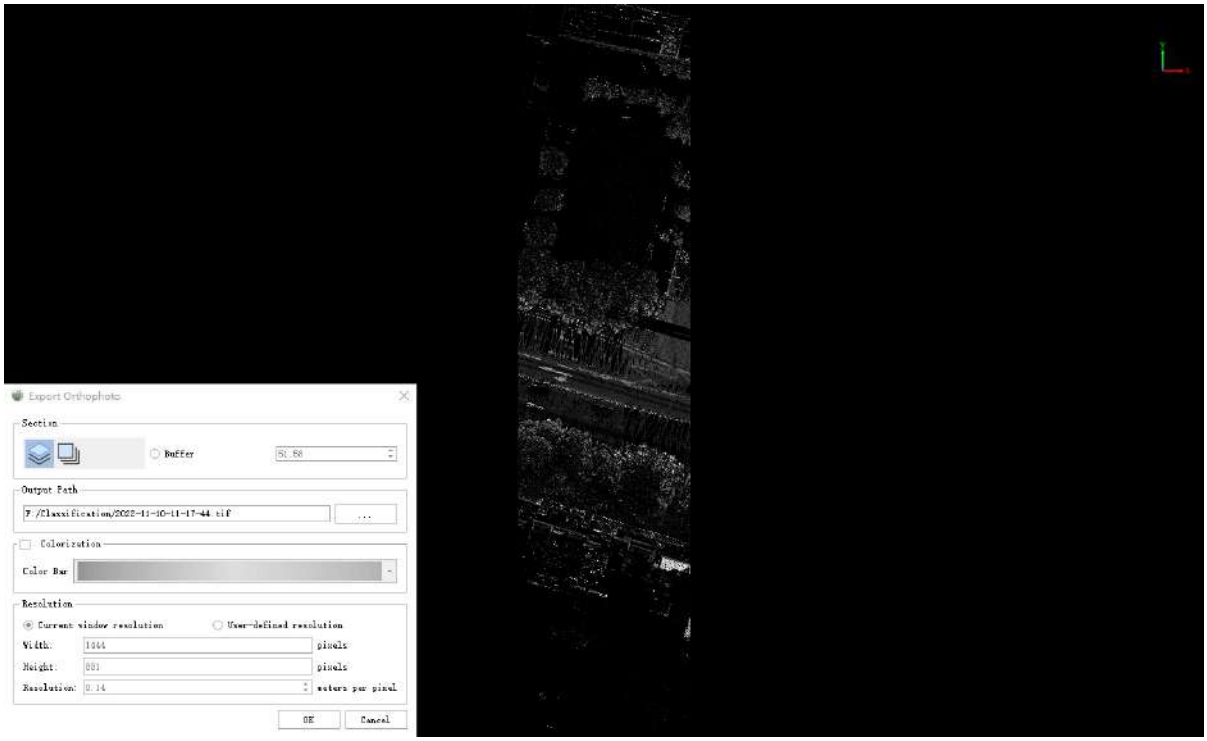
Horizontal Section

1.Click **Horizontal Section** button, the 3D scene automatically switches to the front view.

2.The mouse interactively draws a rectangular box, and ends with a double-click to slice the cross-section, as shown in the figure.



3.The 3D scene automatically switches to the top view, as shown in the figure.

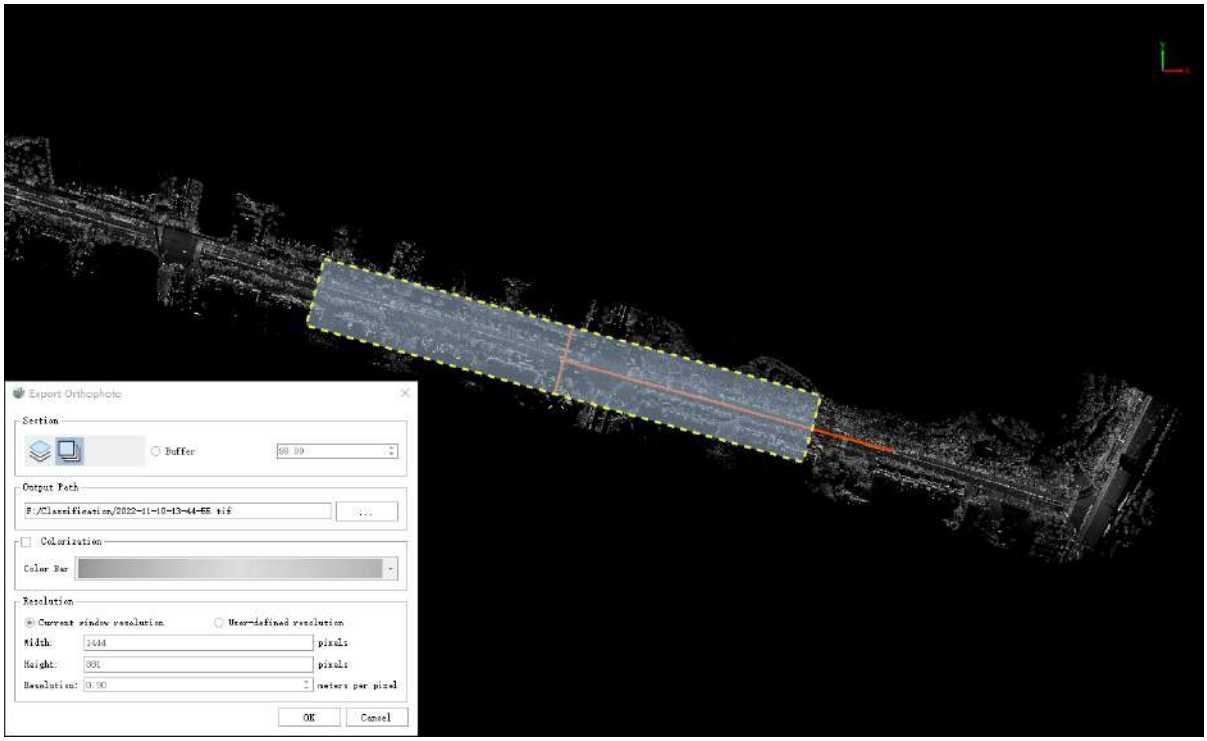


4.If you need to fix the cross-section buffer, you can click the Buffer radio box and adjust the value of the buffer.

Vertical Section

1.Click **Vertical Section** button, the 3D scene automatically switches to the top view.

2.The mouse draws a rectangular box, the first two points determine the axial direction, the third point determines the size of the buffer, and the cross-section slicing is performed at the end of the double-click, as shown in the figure.



3.The 3D scene automatically switches to the front view, as shown in the figure.

4.If you need to fix the cross-section buffer, you can click the Buffer radio box and adjust the value of the buffer.

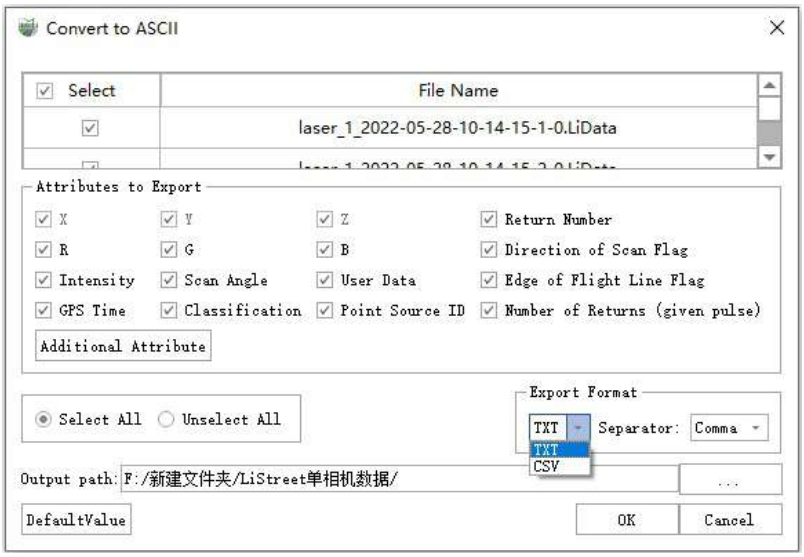
Point Cloud Conversion

Function Description: This tool allows the user to export of a variety of point cloud data, including the conversion of point clouds into point clouds in ASCII, las, E57, and ply formats, which will be described in detail below.

Convert To ASCII

Click **Convert To ASCII** button.

Function Description: The Convert to ASCII tool can convert LiData point clouds to ASCII format, a text format that can be easily viewed in a text editor.
Usage：Click Point Cloud Conversion -> Convert To ASCII.



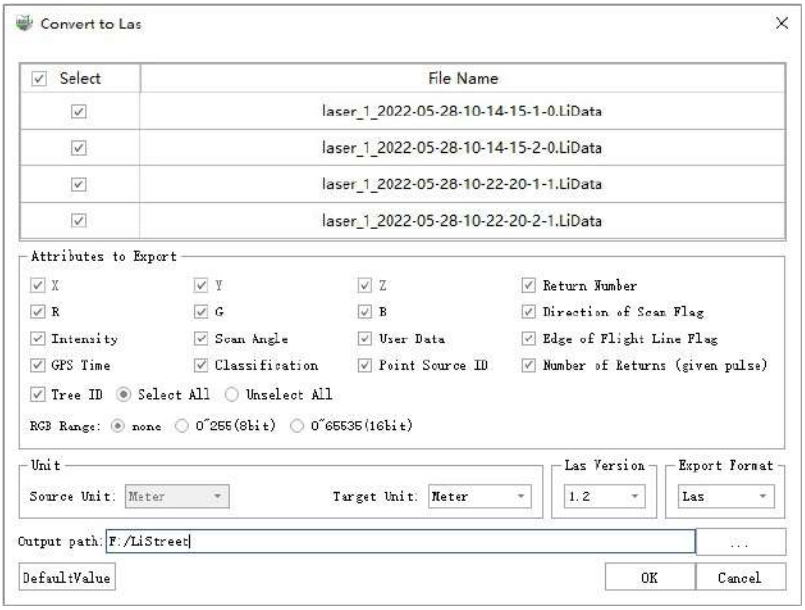
Parameter setting

- **Input Data**: Select the LiData file you want to export. The input file can be a single point cloud data file or multiple data files.
- **Attributes to Export**: Support basic attribute types and additional attributes.
- **Export Format**: The software supports two suffix formats, txt and csv, and the separator supports commas, spaces, and TABS.
- **Output path**: The path of the output folder, and the converted new file is generated after the function is performed.

Convert To Las

Click **Convert To Las** button.

Function Description: The Convert to Las tool can convert LiData point clouds to Las format, which is the standard lidar point cloud data format.
Usage：Click Point Cloud Conversion -> Convert To Las.



> Convert To Laz format as above。

Parameter setting

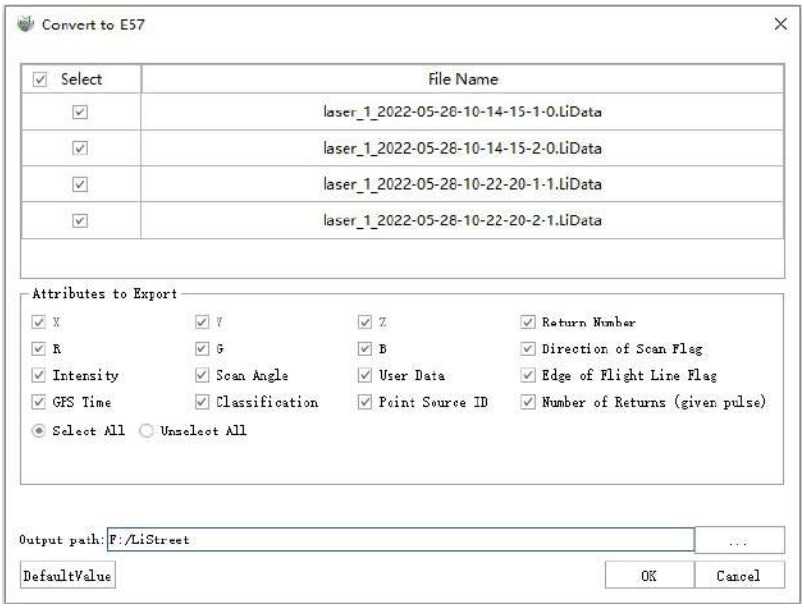
- **Input Data**: Select the LiData file you want to export. The input file can be a single point cloud data file or multiple data files.
- **Attributes to Export**: Support all Las attribute types.
- **RGB Range**: There are three RGB ranges to choose from.
  - **none**: Output according to the original RGB range of the selected file.
  - **0~255 (8 bits)**: Map the RGB range to the 0~255 output. If the color range of the file selected by the user is 0~1 or 0~65535, it can be mapped to 0~255 output at this time.
  - **0~65535 (16 bits)**: Map the RGB range to the output of 0~65535. If the color range of the file selected by the user is 0~1 or 0~255, it can be mapped to 0~65535 output at this time.
- **Source Unit**: The unit of the LiData file to be exported. The point cloud unit supported in the software is meters, so this parameter defaults to meters and cannot be modified.
- **Target Unit**: The unit exported as Las data, you can choose meters, decimeters, centimeters, millimeters, feet, and inches.
- **Las Version (default value 1.4)**: The version exported as LAS supports 1.2~1.4, refer to LiData format.When converting from a high version to a low version, out-of-range attribute fields will be set to zero.
- **Output path**: The path of the output folder, and the converted new file is generated after the function is executed.

Convert To E57

Click **Convert To E57** 

button.

Function Description: The Convert to E57 tool can convert LiData point clouds to E57 format.
Usage：Click Point cloud Conversion -> Convert To E57.



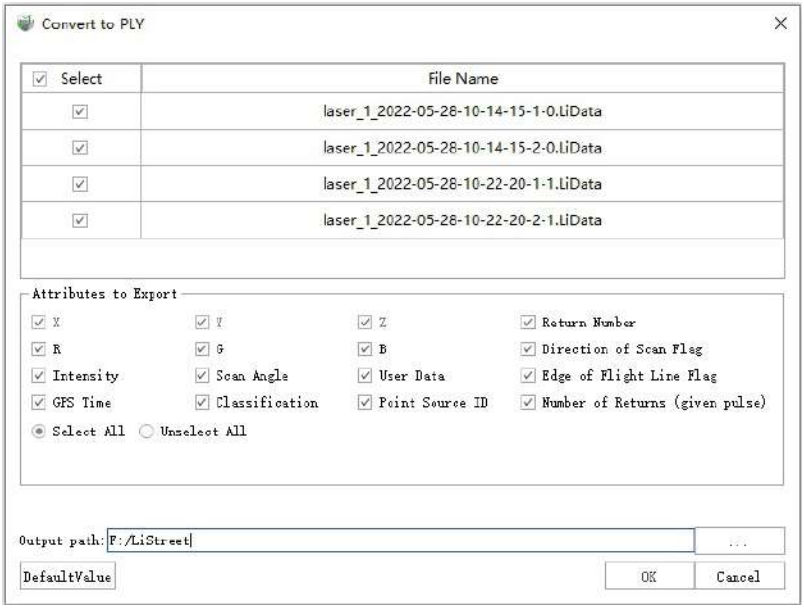
Parameter setting

- **Input Data**: Select the LiData file you want to export. The input file can be a single point cloud data file or multiple data files.
- **Attributes to Export**: Support basic attribute types and additional attributes.
- **Output path**: The path of the output folder, and the converted new file is generated after the function is executed.

Convert To Ply

Click **Convert To Ply** button.

Function Description: The Convert to Ply tool can convert LiData point clouds to Ply format.
Usage：Click Point cloud Conversion -> Convert To Ply.



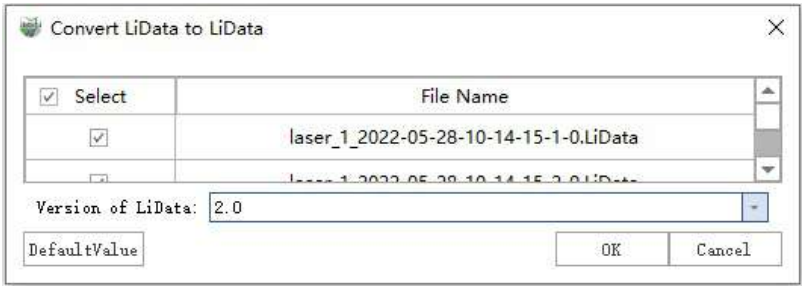
Parameter setting

- **Input Data**: Select the LiData file you want to export. The input file can be a single point cloud data file or multiple data files.
- **Attributes to Export**: Support basic attribute types and additional attributes.
- **Output path**: The path of the output folder, and the converted new file is generated after the function is executed.

Version Conversion

Click the **Convert LiData to LiData**  button.

Function Description：Conversion between point cloud versions.
Usage：Click Point Cloud Conversion -> Convert LiData to LiData.



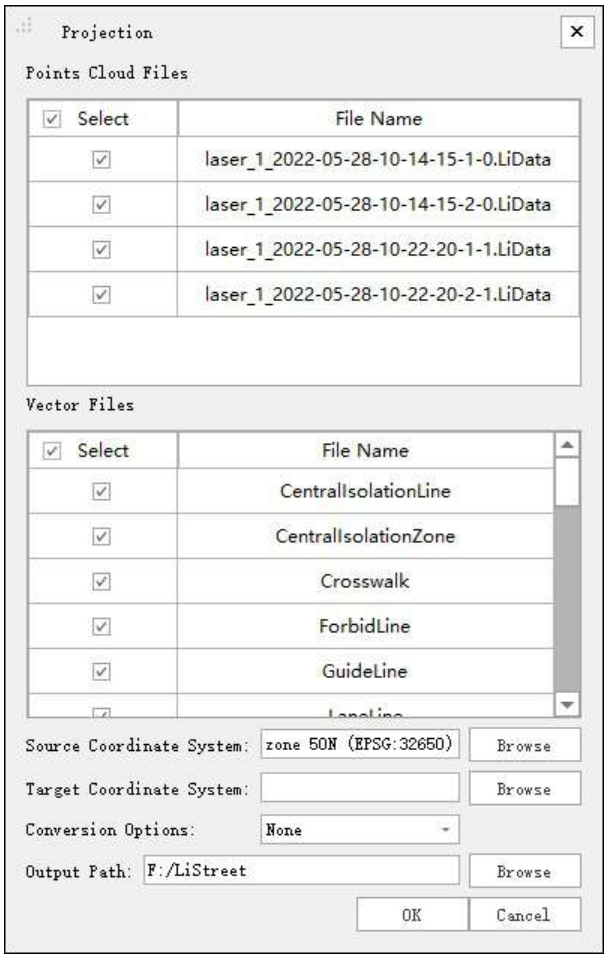
Parameter setting

- **Input Data**：Select the LiData file to export. The input file can be a single point cloud data file or multiple data files.
- **Version of LiData**：Supported versions.

Projection

Function Description: This tool allows the user to project the coordinate system of the point cloud and vector data in the software. The dialog box layer of the projection function is shown in the figure below.

1.Click **Projection**  button.



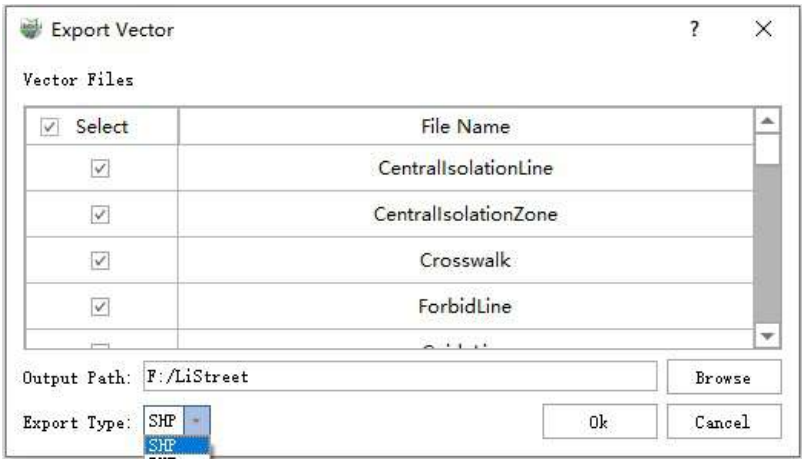
Description

- **Points Cloud Files**: The dialog box initializes and displays all point cloud files in the current software, and all are checked by default. Users can manually uncheck them.
- **Vector Files**: The dialog box initializes and displays all layer files in the current software, and all are checked by default. Users can manually uncheck them.
- **Source Coordinate System**: The software automatically reads the coordinate system of the point cloud and vector files and displays them in the output box. Users can also click the browse button to manually select.
- **Target Coordinate System**: Click the Browse button on the right to select the projection coordinate system to be exported.
- **Conversion Options**: Select the method of projection conversion.
- **Output Path**: Select the output path after the point cloud and vector are converted, and the software automatically defaults to the path where the current point cloud is located.

Export Vector

Function Description: This tool allows the user to select the layer to be exported, select the output directory and the exported vector format, and click the ok button to export. The dialog box layer for exporting vectors is shown in the figure below.

1.Click **Export Vector**  button.



Description

- **Vector Files**: Initialize the table to display all layers, and check all layers for export by default.Users can check/uncheck the layer by themselves.
- **Output Path**: Initialized to the path where the point cloud is located, the user can also click the browse button to change the output path.
- **Export Type**: Currently, vector files in shape file and dxf formats are supported for export.

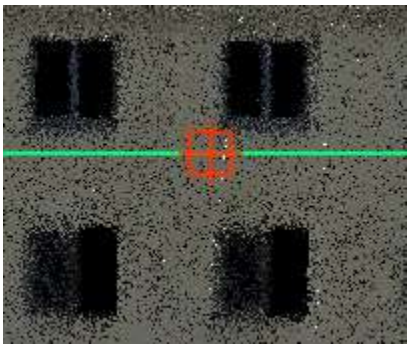
Catch

Function Description: This tool allows the user to capture the required scene requirements.

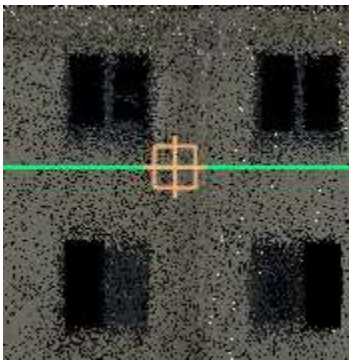
Steps

The software capture functions are shown in the figure above, in the order from left to right: Snap on End Points, Snap on line, Snap middle, Snap intersection, Snap Plane, Snap Point Cloud, Hover, Restrict Orthogonal, Pick Orthogonal Axis, Reset orthogonal Axis. The capture description is as follows:

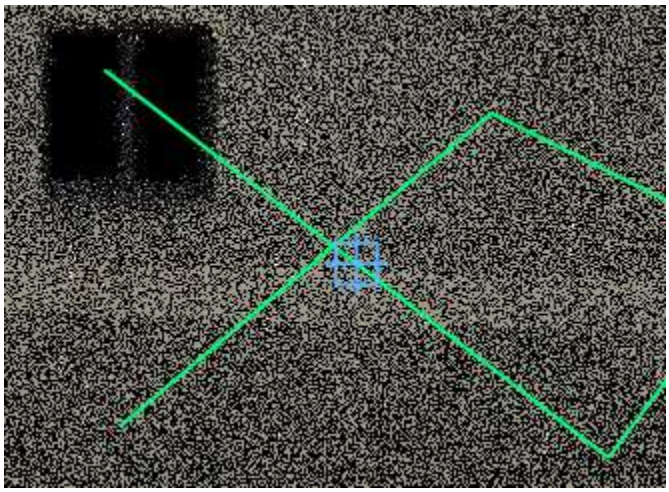
1.Snap on End Points: Capture each node of the vector data, and the state of the cross wire captured by the node is red.



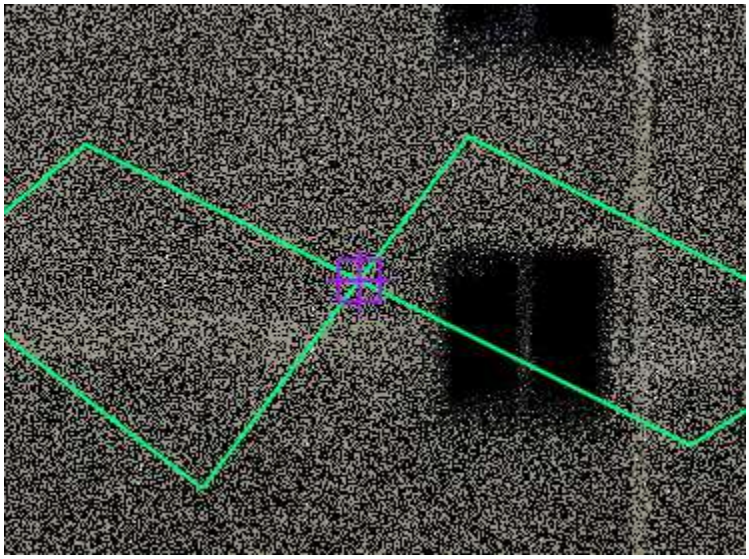
2.Snap on line: Capture the edges of vector data, such as any position on the edges of lines and polygons, the state of the line captured to the cross wire is orange.



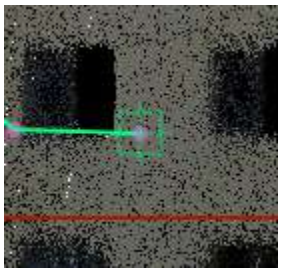
3.Snap middle: Capture the midpoint position of the vector line, and the state of the cross wire captured by the midpoint is blue.



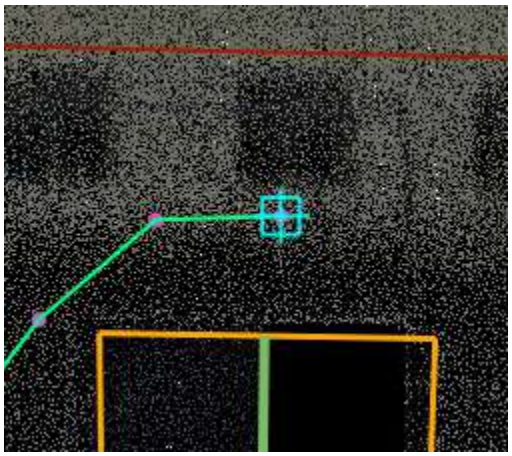
4.Snap intersection: Capture the intersection of the cross line, and the state of the cross wire captured at the intersection is purple.



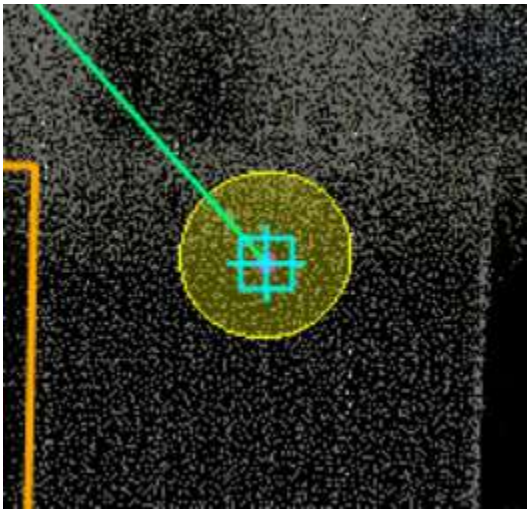
5.Snap Plane: Capture any point on the reference plane, and the state of the cross wire captured by the plane is green.

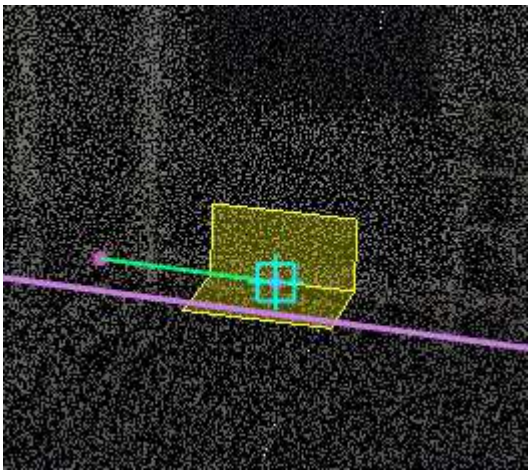


6.Snap Point Cloud: Capture the real point cloud coordinate points on the point cloud, and the state of the cross wire captured by the point cloud is light blue.



7.Hover: On the basis of turning on point cloud capture, turn on the point cloud fitting function to better find points on the plane or at the edges and corners. The point cloud fits a yellow circle on the plane and forms an icon structure of two vertical planes at the edges and corners.





8.Restrict Orthogonal: After the orthogonality mode is turned on, during the drawing process, the lines are drawn in parallel or vertical modes. And perpendicular to the parallel direction is the screen direction.

9.Pick Orthogonal Axis: In the orthogonal mode, click the Select orthogonal axis button to first determine that any vector line is an orthogonal reference line, and the subsequent vector lines drawn are all orthogonal to it.

10.Reset orthogonal Axis: If the orthogonal axis is set, if you need to cancel or reselect the orthogonal axis, you need to use the Reset orthogonal axis button to cancel the current axis first.

> Note: 8-10 cannot be used in 3D mode.

Classification

As shown in the figure below, the functions contained in the Classification page are：

- Classification
- Extract

Classification

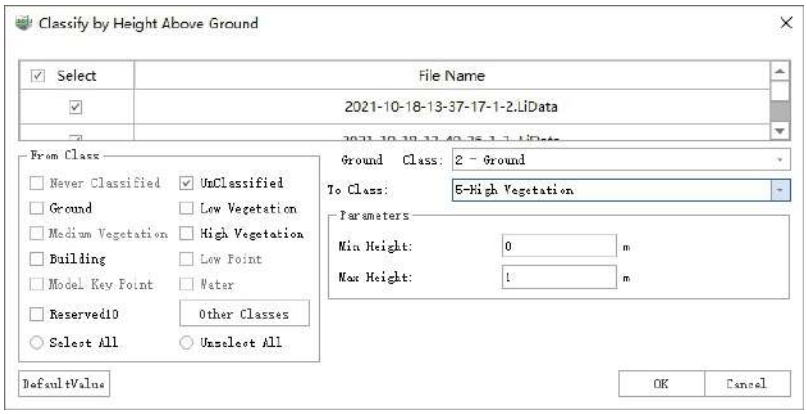
- Classify Air Points
- Classify Below Surface Points
- Classify by Height Above Ground
- Classify Low Points
- Classify Isolated Points
- Classify Closeby Points
- Classify Ground Points
- Classify by Attribute
- Classify by Polygons
- Classify by Polylines
- Classify by Cluster Size
- Classify by Deeplearning
- Classify by Manual Editing

Classify by Height Above Ground

Function Description: This function classifies the points on the surface of the terrain with a certain height, which can quickly classify vegetation at different heights.

Steps

1.Click **Classify by Height Above Ground** button, pop up the **Classify by Height Above Ground dialog box**.



Parameter description：

- **Input Data**: The input file can be a single point cloud data or a point cloud dataset.
- **From Class**: Source class(es).
- **Ground Class**: The default ground point is 2-class.
- **To Class**: Target class.
- **Minimum Height (meters)**: The default is “0”, the minimum height difference of the area to be classified above the ground points.
- **Maximum Height (meters)**: The default is “1”, the maximum height difference of the area to be classified above the ground points.
- **DefaultValue**: Click this button to set all parameters as default.

> Note: This function need to contain the ground point in the cloud.

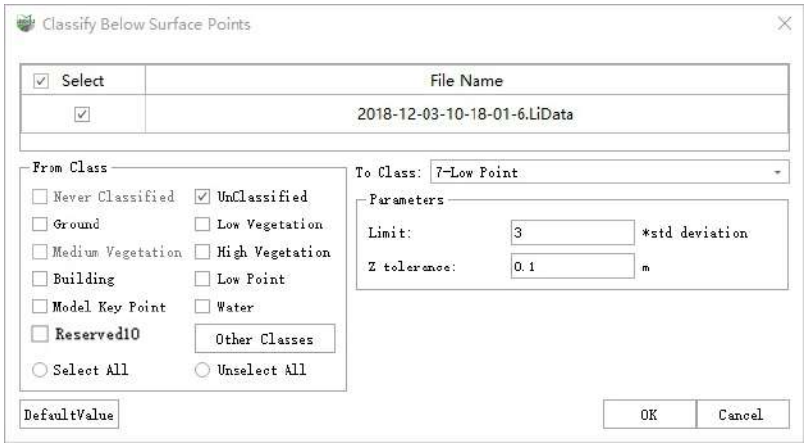
Classify Below Surface Points

Function Description: This function classifies points in the initial category that are below the elevation of the surrounding neighborhood.For example, when the starting category is ground, this method can be used to classify points lower than the surface elevation to be the lower-than-the-surface point.

- The main algorithm idea of this function is:
  1. Search for a certain number of nearest points for the current point in initial class.
  2. Fit the plane with the nearest point.
  3. Calculate the absolute value of the height difference between the current point and the plane. If the value is less than the set Z tolerance, it is not categorized. If it is greater than the tolerance, go to the next step.
  4. Calculate whether the difference between the current point elevation and the average value of the neighboring points is greater than the Limit of the standard deviation. If it is greater than, then it is classified as the target category; otherwise, it is not classified.

Steps

1.Click **Classify Below Surface Points** button, pop up the **Classify Below Surface Points dialog box**



Parameter description：

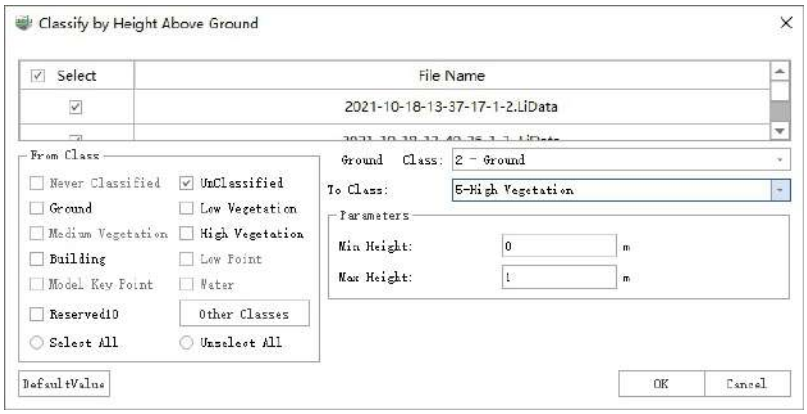
- **Input Data**: The input file can be a single point cloud data or a point cloud dataset.
- **From Class**: Source class(es).
- **To Class**: Target class.
- **Limit**: The default is “3".The multiple of the mean squared error of the neighboring point fitting plane of the unclassified points. The larger the value, the less points will be classified into the target class.
- **Z tolerance (meters)**: The default is “0.1”, The threshold of height difference. The point to fit plane distance less than this value is not classified. The larger the value, the less points will be classified into target class.
- **DefaultValue**: Click this button to set all parameters as default.

Classify by Height Above Ground

Function Description: This function classifies the points on the surface of the terrain with a certain height, which can quickly classify vegetation at different heights.

Steps

1.Click **Classify by Height Above Ground** button, pop up the **Classify by Height Above Ground dialog box**.



Parameter description：

- **Input Data**: The input file can be a single point cloud data or a point cloud dataset.
- **From Class**: Source class(es).
- **Ground Class**: The default ground point is 2-class.
- **To Class**: Target class.
- **Minimum Height (meters)**: The default is “0”, the minimum height difference of the area to be classified above the ground points.
- **Maximum Height (meters)**: The default is “1”, the maximum height difference of the area to be classified above the ground points.
- **DefaultValue**: Click this button to set all parameters as default.

> Note: This function need to contain the ground point in the cloud.

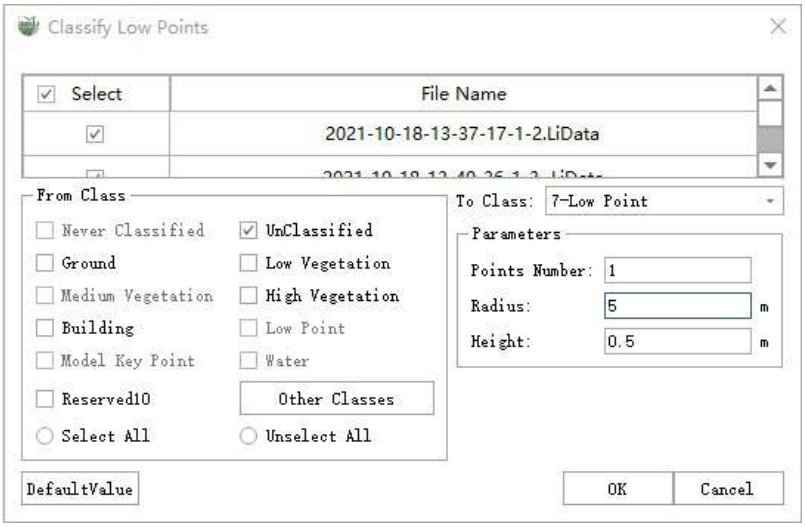
Classify Low Points

Function Description: The low point refers to the noise point that is lower than the actual terrain. The existence of the low point will affect the extraction of the ground point, because the progressive triangulation filtering algorithm is based on the seed points represented by the lowest point of the gridded point cloud. Therefore, filtering out the low point is a preprocessing operation, which directly affects the filtering effects, the establishment of a digital model, and the generation quality of contour lines.

- The distribution of low points is divided into individual points or clusters. The algorithmic flow of this function is:
  1. Traverse the point cloud and search for points to be classified within a certain range of radius of the current single point or point cluster.
  2. Calculate the maximum height difference between the current point and the neighboring point, which is compared with the threshold.
  3. If the value is greater than the threshold, the current point is considered to be a low point, otherwise, it is not classified as a low point.

Steps

1.Click **Classify Low Points**  button, pop up the **Classify Low Points dialog box**



Parameter description:

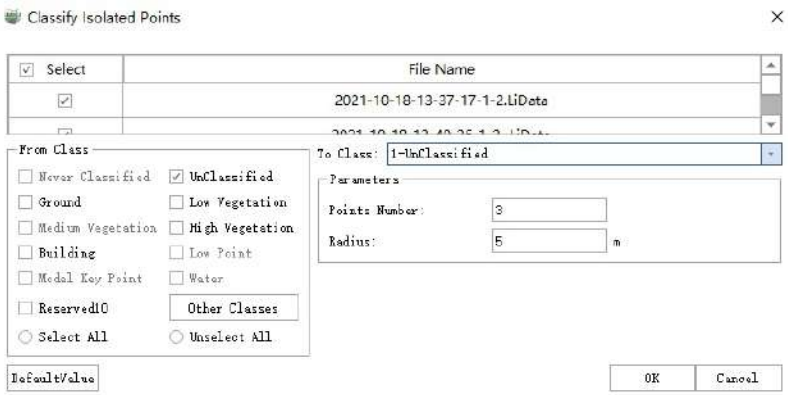
- **Input Data**: The input file can be a single point cloud data or a point cloud dataset.
- **From Class**: Source class(es).
- **To Class**: Target class.
- **Points Number**: The default is “1”. When the number of points is set to 1, the single low point is classified. If it is greater than 1, cluster low points are classified.
- **Radius (meters)**: The default is “5”, the radius threshold between the unclassified point and the neighboring point.
- **Height (meters)**: The default is “5”, the height difference threshold between the unclassified point and the neighboring point.
- **DefaultValue**: Click this button to set all parameters as default.

Classify Isolated Points

Function Description: This function classifies points in a certain area of point cloud, which is generally used to find outliers in the air or below the ground.

Steps

1.Click **Classify Isolated Points**  button, pop up the **Classify Isolated Points dialog box**.



Parameter description：

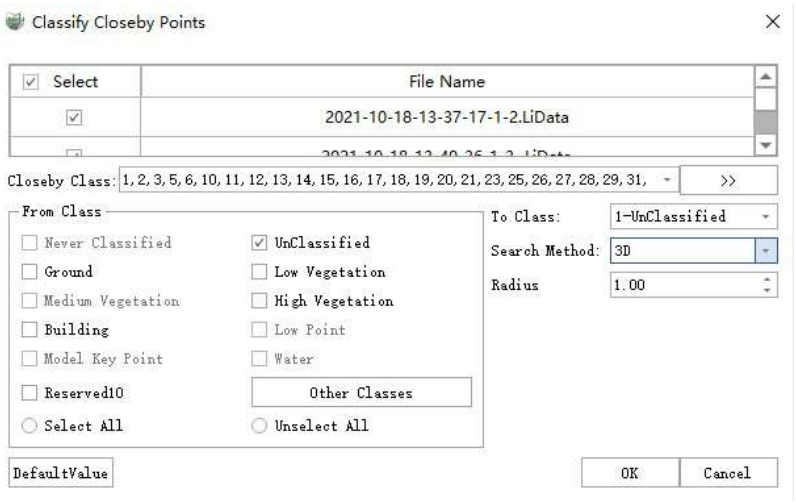
- **Input Data**: The input file can be a single point cloud data or a point cloud dataset.
- **From Class**: Source class(es).
- **To Class**: Target class.
- **Points Number**: The default is “3". If the number of points in the neighboring radius are less than or equal to the value, the point is considered as an isolated point.
- **Radius (meters)**: The default is “5”. Neighboring search radius.
- **DefaultValue**: Click this button to set all parameters as default.

Classify Closeby Points

Function Description: This function is used to classify those points close to the certain class(es) of points. For each points in the source class, find the points in its specified 2D or 3D neighborhood area, and determine whether these points meet certain conditions (i.e., belong to a specified class). Those points meet the requirement will be classified as the target class.

Steps

1.Click **Classify Closeby Points** button, pop up the **Classify Closeby Points dialog box**.



Parameter description：

- **Input Data**: The input data could be one point cloud file or a set of point cloud files.
- **Closeby Class**: The points of this class which is close to the source points will be classified.
- **From Class**: The points close to this class will be classified.
- **To Class**: Target class of the classification.
- **Search Method**: The method for searching in the neighborhood. Support 2D or 3D neighborhood.
- **Radius**: Neighborhood search radius.
- **DefaultValue**: Click this button to set all parameters as default.

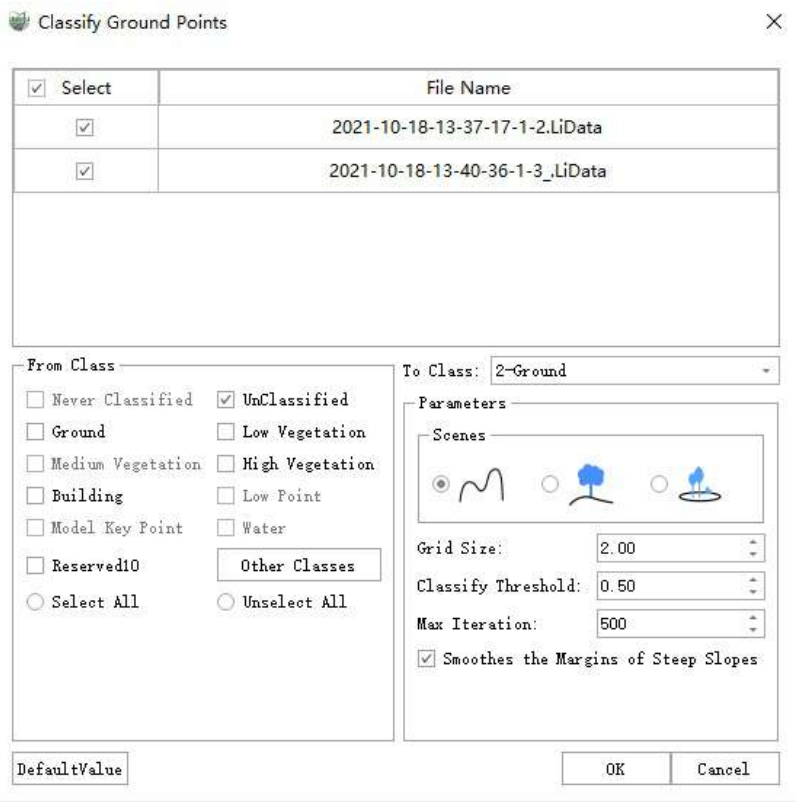
Classify Ground Points

Function Description: Ground point classification is the basic operation of point cloud data processing. This function uses a ground point filtering algorithm based on cloth simulation (Cloth Simulation Filter, IPTD (Zhang et al.,2016)).

- This algorithm can be divided into the following steps：
  1. Turn the point cloud upside down in the Z direction.The point cloud is gridded with a certain resolution in the XY direction, and the nodes of the grid are used as a simulated fabric.
  2. Set the initial height of the fabric to the maximum point cloud Z and start the iteration.During each iteration, the fabric will “sink” to the point cloud with a certain gravity, and the positional relationship between the fabric node and the point cloud is calculated at the same time.Nodes that have fallen on the point cloud will not be movable during the next iteration.Immovable nodes will slow down the sinking speed of surrounding nodes according to the stiffness value R.
  3. After a certain number of iterations, calculate the positional relationship between each point and the fabric; points whose distance from the fabric in the Z direction is less than a certain threshold value will be divided into target categories.

Steps

1.Click **Classify Ground Points** button, pop up the **Classify Ground Points dialog box**.



Parameter description：

- **Input Data**: The input file can be a single point cloud data or a point cloud dataset.
- **From Class**: Source class(es).
- **To Class**: Target class.

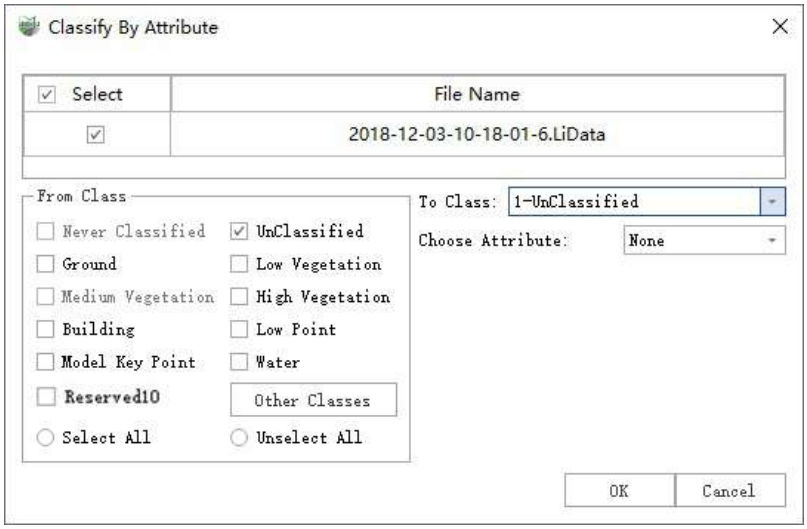
- **Scenes (Optional)**: For different terrain scenes, you can choose steep terrain, gentle terrain, and flat terrain.Check different terrain features to correspond to different default parameters
- **Grid Size (meters)**: The default is “1.0”, the resolution of the fabric node.1.0 is suitable for most point clouds.This value can be appropriately reduced for data with large terrain ups and downs.
- **Classify Threshold (meters)**: The default is “0.5”. After the iteration is completed, points with a distance of less than this threshold in the Z direction from the fabric will be divided into target categories.
- **Max Iteration**: The default is “500”, and the iteration is completed when the algorithm reaches the maximum number of iterations or all fabric nodes are immovable.
- **Smoothes the Margins of Steep Slopes**: When the fabric is located on a steep slope, due to internal constraints between the fabric nodes, it cannot be well matched with the ground, and the algorithm may produce large errors.Checking this item can eliminate the impact of steep slopes to a certain extent.If the scene does not contain steep slopes, you can uncheck it.
- **DefaultValue**: Click this button to set all parameters as default.

Classify by Attribute

Function Description: This function classifies the point cloud into another class according to their attributes. Currently available classification attributes include Absolute Elevation, Intensity, GPS Time, Scan Angle, and Return Number. In addition, the function can restore all classes and/or reclassify the point cloud for undesirable classification results.

Steps

1.Click **Classify by Attribute** button, pop up the **Classify by Attribute dialog box**.



Parameter description：

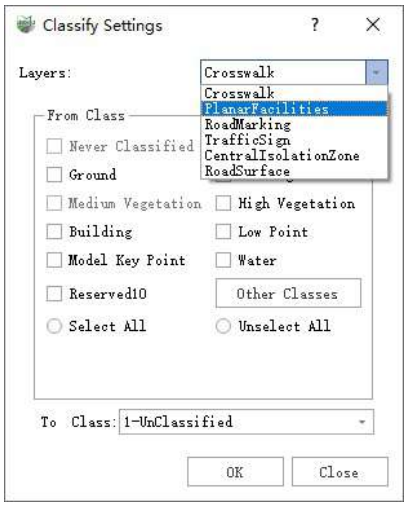
- **Input Data**: The input file can be a single point cloud data or a point cloud dataset.
- **From Class**: Source class(es).
- **To Class**: Target class.
- **Choose Attribute**: The selected attribute will be used to classify the point cloud.
  - **None (default)**: The default setting will change all points in the From Class option to To Class.
  - **Elevation**: Classified by elevation range. If the elevation value of a point is between the specified range, it will be assigned to the To Class.
  - **Intensity**: Classified by intensity range. If the intensity value of a point is between the specified range, it will be assigned to the To Class.
  - **Time**: Classified by GPS time. If the GPS time value of a point is between the specified range, it will be assigned to the To Class.
  - **Angle**: Classified by scan angle. If the scan angle value of a point is between the specified range, it will be divided into the To Class.
  - **Return**: Classified by the return number. If the return number at a point is between the specified range, it will be assigned to the To Class.

Classify by Polygons

Function Description: Classify point clouds according to vector polygons, and point clouds within the polygons will participate in the classification.

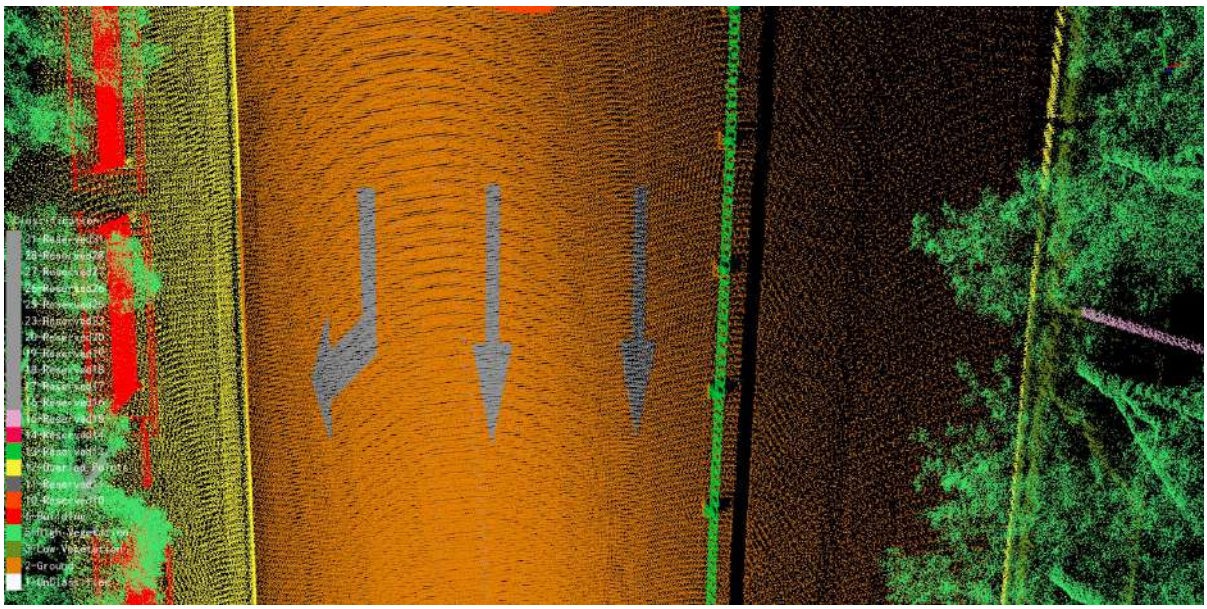
Steps

1.Click **Classify by Polygons**  Button, pop up the **Classify Settings dialog box**.

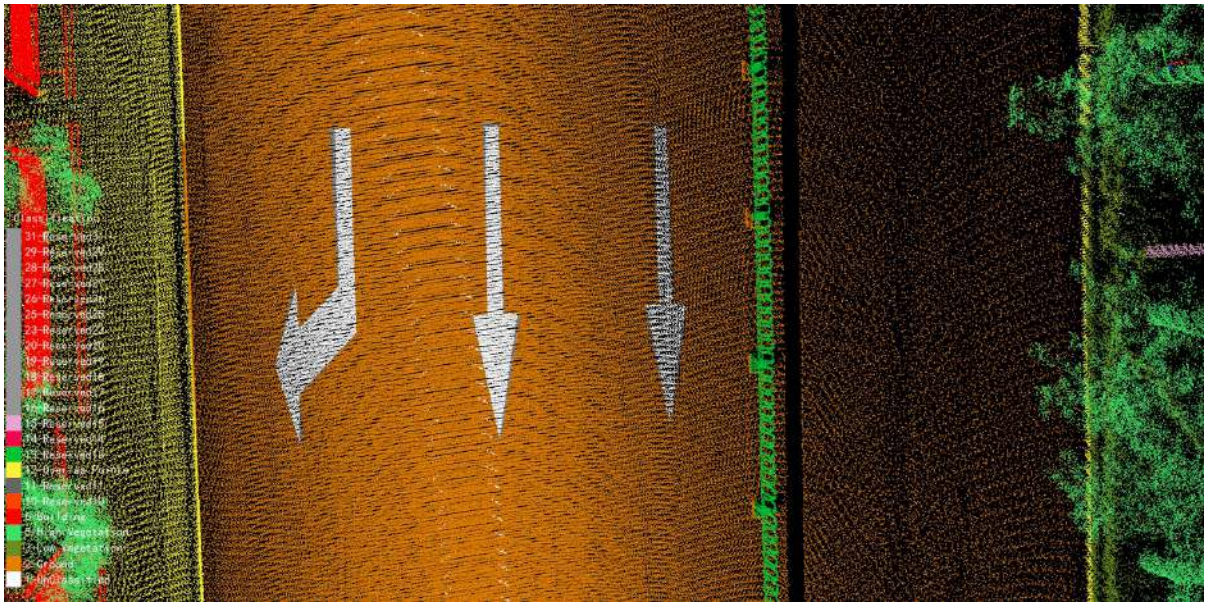


Parameter description：

- **Layers**: Classify point clouds based on the vector polygons of a certain layer.
- **From Class**: The category to be classified of the point cloud, only the point cloud of the set source category will participate in the classification.
- **To Class**: The target category of the classification, the point cloud will be assigned to this category.



Before Classify by Polygons



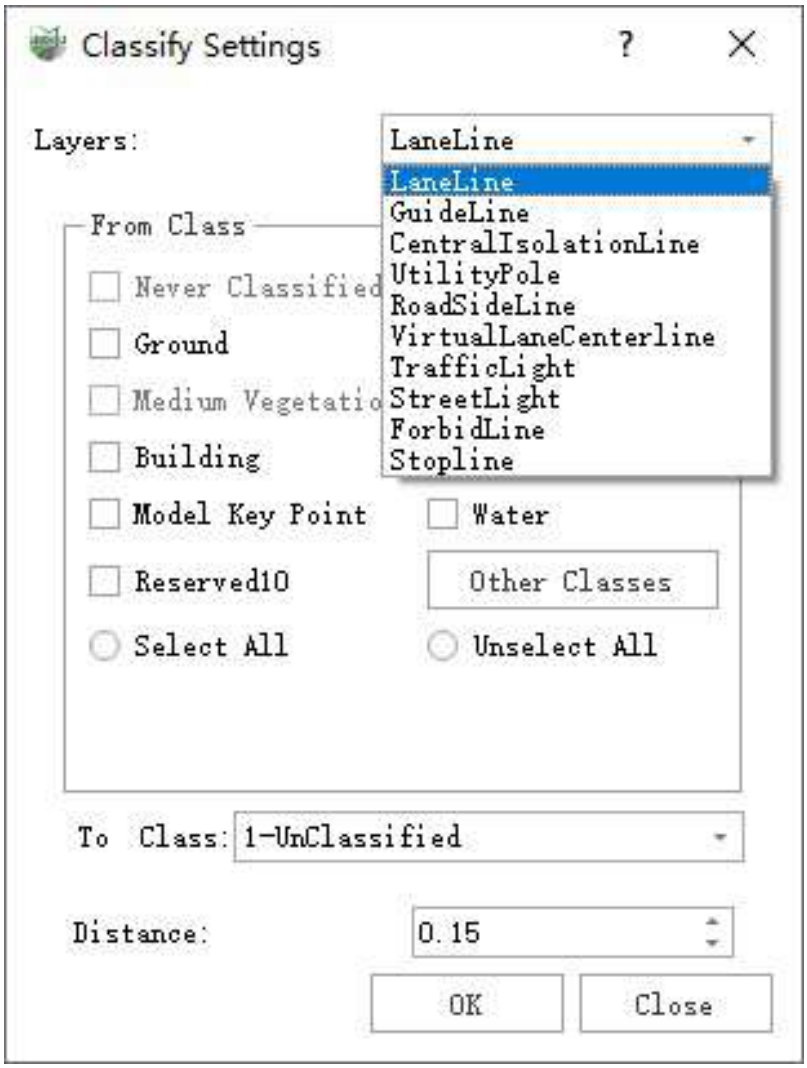
Effect diagram after Classify by Polygons

Classify by Polylines

Function Description: Classify point clouds based on vector lines.

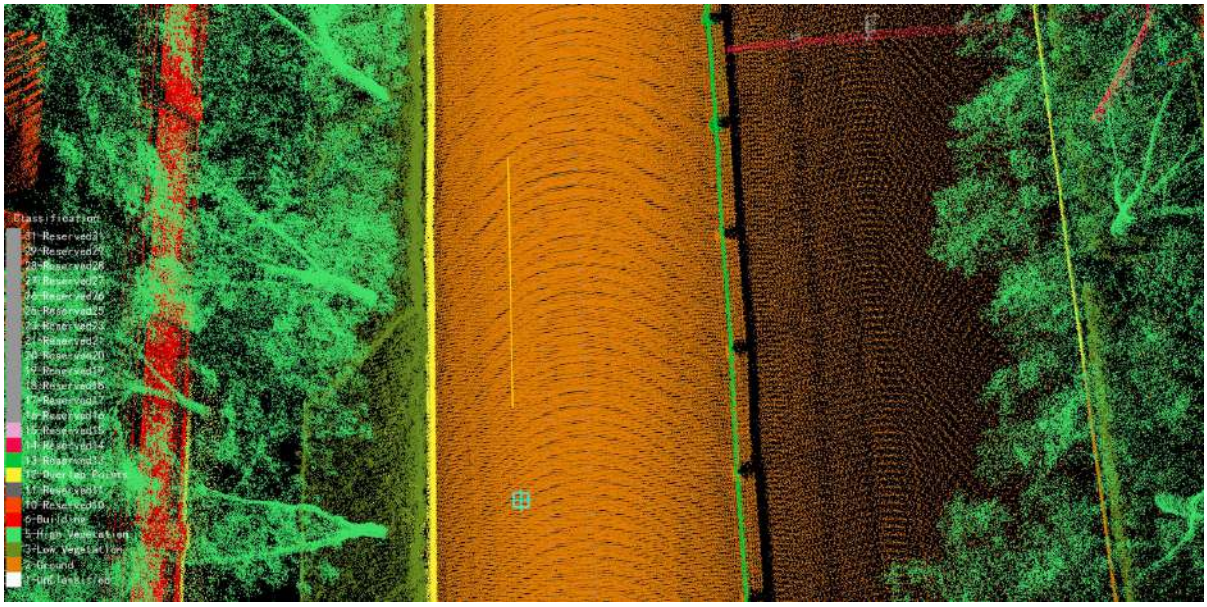
Steps

1.Click **Classify by Polylines** Button, pop up the **Classify Settings dialog box**.

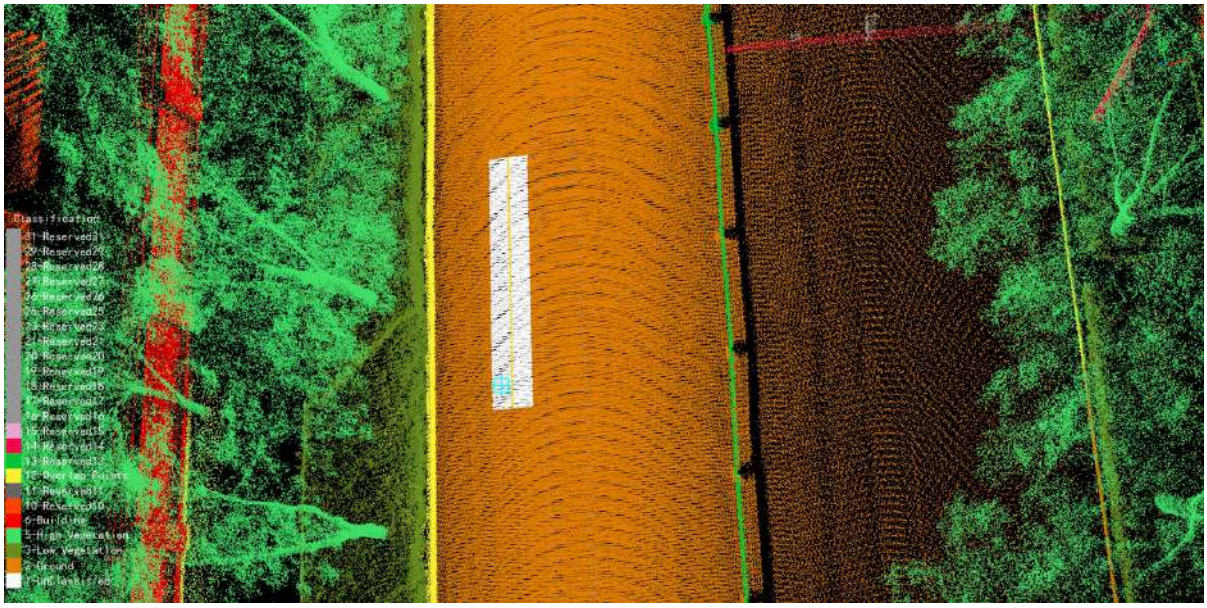


Parameter description:

- **Layers**: Classify point clouds based on the vector lines of a certain layer.
- **From Class**: The category to be classified of the point cloud, only the point cloud of the set source category will participate in the classification.
- **To Class**: The target category of the classification, the point cloud will be assigned to this category.
- **Distance**: Points within the width range set on both sides of the vector line will participate in the classification.



Before Classify by Polylines



Effect diagram after Classify by Polylines

Classify by Cluster Size

Function Description: For a given input category, cluster and calculate the length, width and height of each cluster. If the size or number of points of a cluster is less than the set threshold value, the cluster is set as the target category.

> This function **is not recommended** for large areas, surfaces, or categories with too many points. **It is recommended** to use relatively independent small objects such as poles and trash cans, and **It is recommended** to filter out the ground in advance to prevent the various targets from being separated.

Parameter dialog box

Parameter dialog box

Parameter description:

- **From Class**: Displays the input categories participating in the cluster, and you can check/uncheck the extraction category as needed.

  > Checking options is proportional to the time-consuming, that is, the more you check, the longer the entire processing process takes. Please check according to the actual situation, and it is recommended to check the category of small objects (please refer to the description below the function description for details).

- **Select all/Unselect all**: Click the button to check all source categories/uncheck all.
- **To Class**: After calculation, the eligible points are classified from the original category to the target category.
- **Max Height, Max Length, Max Width**: After clustering, if the length, width and height of the current cluster are less than the set maximum height, maximum length, and maximum width, the points of the current cluster will be classified as the target category.**Required**.

  > Height refers to the height of the physical Z-axis direction.

- **Points Num**: After clustering, if the current option is checked and the number of points in the current cluster is less than the set number of points, the points of the current cluster will be classified as the target category (regardless of whether the length, width and height are within the threshold value),**Optional**.

  > The priority of this option is higher than the length, width and height limit.

- **Global**: Function operation mode, if the global option is checked, all selected files will be processed.
- **Select By Polygon**: Click the mouse to build a polygon. If the select by polygon option is checked, the point clouds within the polygon range will only be processed.

  > Multiple polygons can be framed at once.

- **OK**: After the parameters are set, click the OK button to start automatic detection.
- **Cancel**: Exit function.

Steps

1.In the **Classification** panel, click **Classify by Cluster Size** button in the Classification drop-down box.

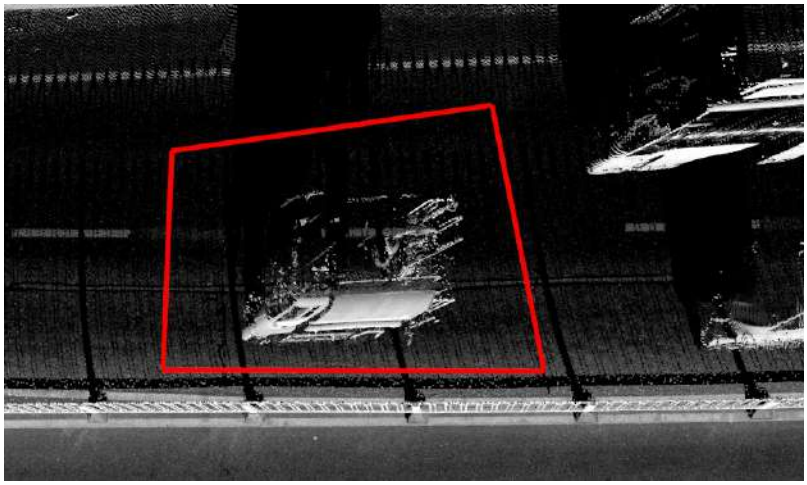
2.Pop up the **Parameter Settings dialog box**.

3.Parameter settings, please refer to the **Parameter dialog box for details for setting reference**.

4.Select the operating mode and perform the calculation.

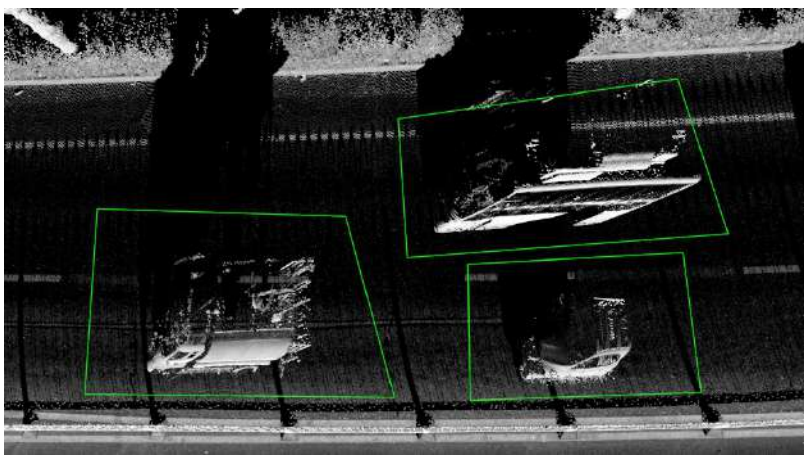
Select by Polygon

1.Click the left mouse button to select the polygon, and double-click to end the creation of a polygon.



Select by Polygon

2.Repeat step 1 until the desired range.



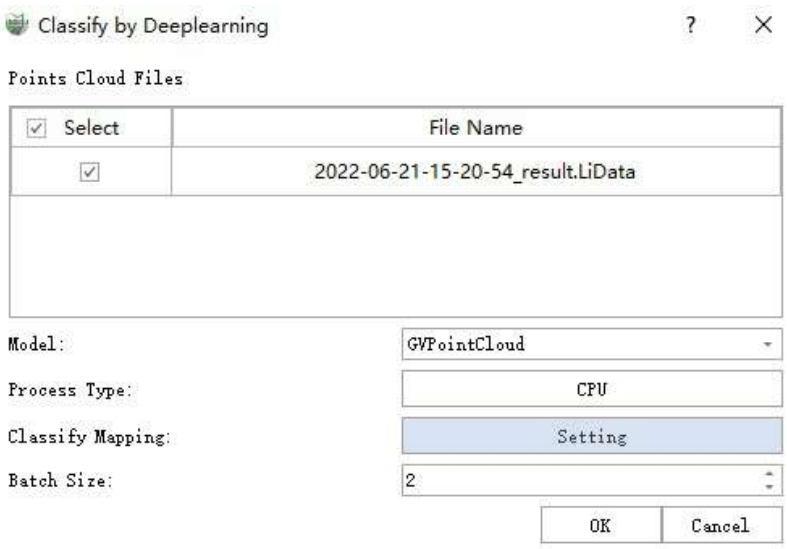
Select by Polygon

Classify by Deeplearning

Function Description: Using deep learning methods to classify point cloud data.

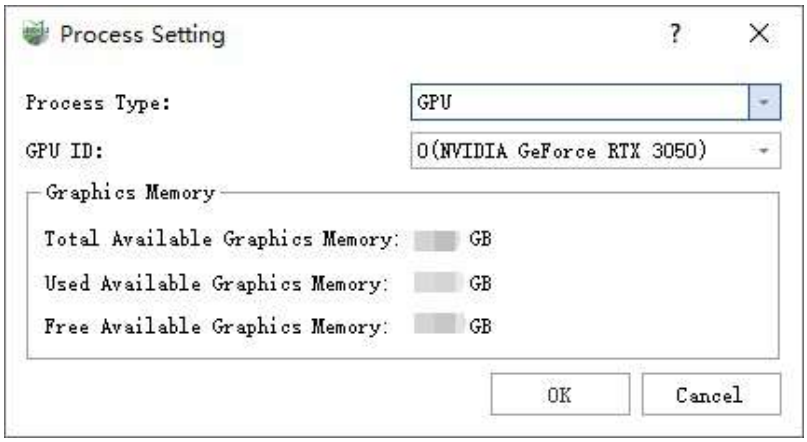
Steps

1.Click **Classify by Deeplearning** Button, pop up the **Classify by Deeplearning dialog box**

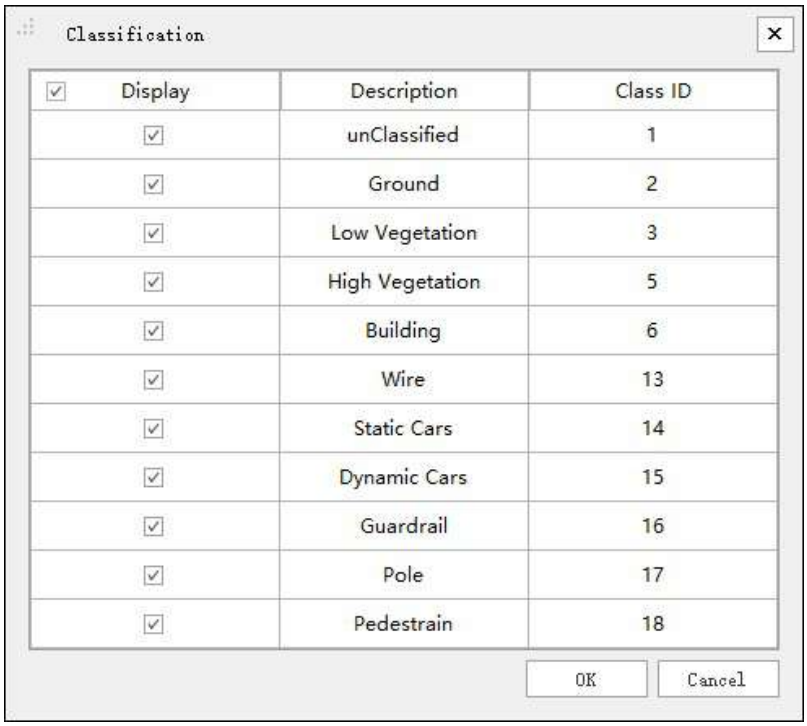


Parameter description:

- **Input Data**: For data that needs to be classified, you can check one or more point cloud data.
- **Model**: The model used for deep learning classification. The software provides two models by default. Among them, the GVPointCloud model is suitable for data classification of domestic urban scenes, and the NPM3D model is suitable for classification of foreign urban scenes.
- **Process Type**: There are two processing modes: GPU and CPU, the default is CPU mode, if the computer graphics card performance is high, you can switch to GPU mode on your own, GPU efficiency will be about twice as high as the CPU, and if the computer has more than one graphics card, you can choose the right graphics card on your own, here it is highly recommended to use NVIDIA graphics card for GPU mode classification.

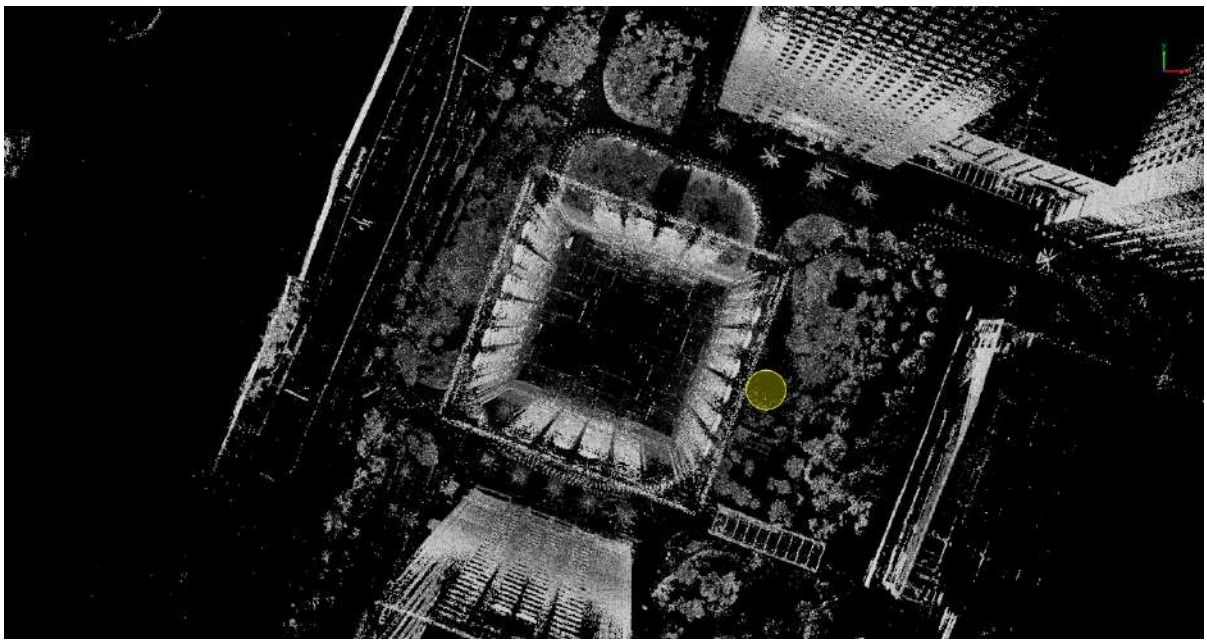


- **Classify Mapping**: Map the category and map the category to the corresponding category label.

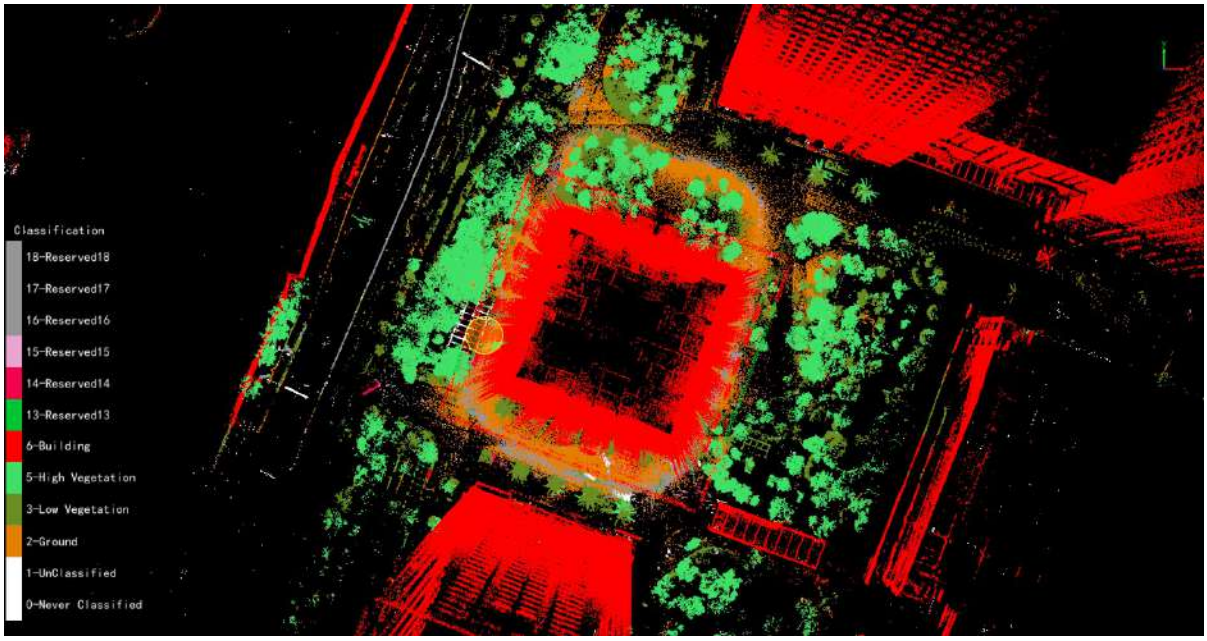


Here you can set your own categories and labels according to user needs, such as: The corresponding class ID for unClassified is 1.

- **Batch Size**: Indicates the number of point cloud samples processed each time, the higher the computer memory and display memory, the larger the set value, and the faster the overall processing speed. Software already set default value is set according to the performance of the computer. It is recommended that the user change the value by himself not more than 8.



Before Classify by Deeplearning



After Classify by Deeplearning

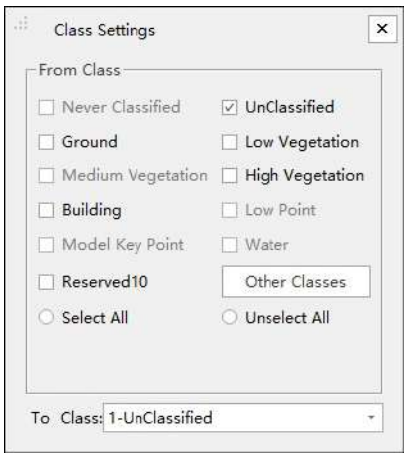
Classify by Manual Editing

Class Settings

Function Description: This function is mainly to determine the source category and the target category.

Steps

1.Click **Class Settings** button, pop up the **Class Settings dialog box**.



Parameter description：

- **From Class**: Source class(es).
- **To Class**: Target class.

Classify Selection

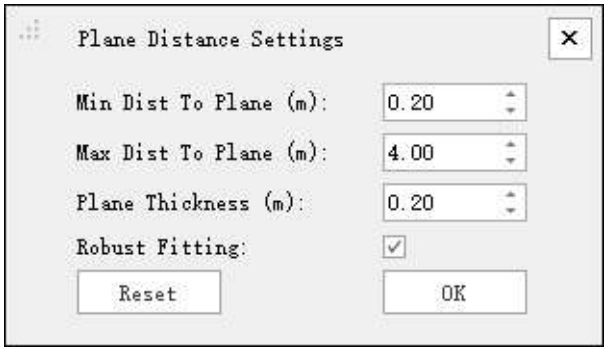
Function description : Select the area through the box selection, you can classify the selected area.

Steps

1.Click **Classify Selection** button, pop up the **Edit dialog box**.

Parameter description：

- **Polygon**: Select a polygon area.
- **Rectangle**: Select a rectangular area.
- **Spherical**: Select a spherical area.
- **circle**: Select a circle area.
- **Online**: Select an online area.
- **offline**: Select an offline area.
- **Plane**: Select a plane area.
- **On the surface**: By clicking on at least three points to select objects above the plane, this tool can effectively select moving objects above the ground.
- **Under the plane**: Select objects under the plane by selecting at least 3 plane points; it can select moving objects on the ground/road.
- **Plane distance setting**: Set the selected area on the plane.
- Click Plane Distance Settings to pop up the **Plane Distance Settings dialog box**



- **Min Dist To Plane (m)**: Set the minimum distance to the plane.
- **Max Dist To Plane (m)**: Set the maximum distance to the plane.
- **Plane Thickness (m)**: Set the plane thickness.

- **Robust Fitting**: the effect is better and the speed is slower.
- **Reset**: Click this button to restore the default values of all parameters.

-  **Unselect**: Reverse the selection in the area.
-  **cancel**: Deselect.
-  **Classification**: Classify the selected point cloud area.
-  **exit**: Exit editing mode.

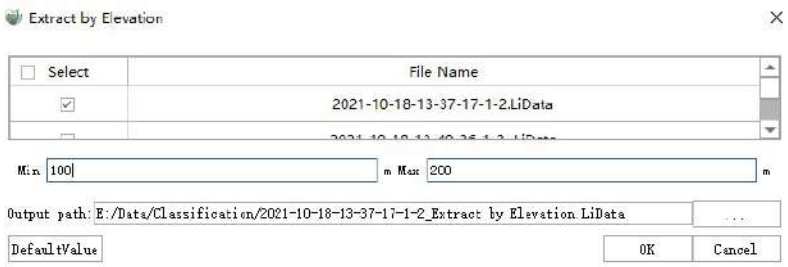
Extract

Extract by Elevation

Function Description: Extract by Elevation tool can extract the point cloud data in the user-defined elevation range to a file. This function supports multiple file operations.

Steps

1.Click **Extract by Elevation** button, pop up the **Extract by Elevation dialog box**.



Parameter description：

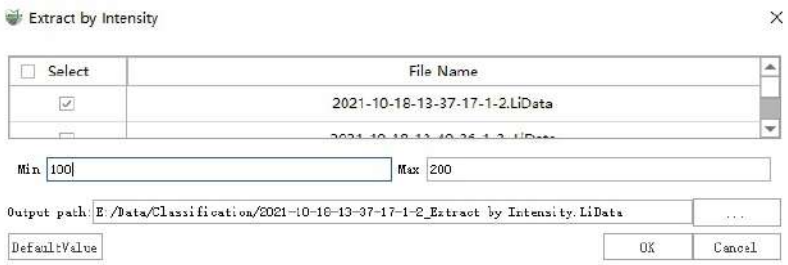
- **Input Data**: Input one or more point cloud data files. File Format: *.LiData.
- **Min (meters)**: The default is “100”. Users need to enter the minimum elevation value of the point cloud data to be extracted.
- **Max (meters)**: The default is “200”. Users need to enter the maximum elevation value of the point cloud data to be extracted.
- **Output path**: Path of the output folder. After the function being executed, the extracted new file(s) will be generated.

Extract by Intensity

Function Description: Extract by Intensity tool can extract all the point cloud data within the user-defined intensity range. This function supports multiple file operations.

Steps

1.Click **Extract by Intensity** button, pop up the **Extract by Intensity dialog box**.



Parameter description：

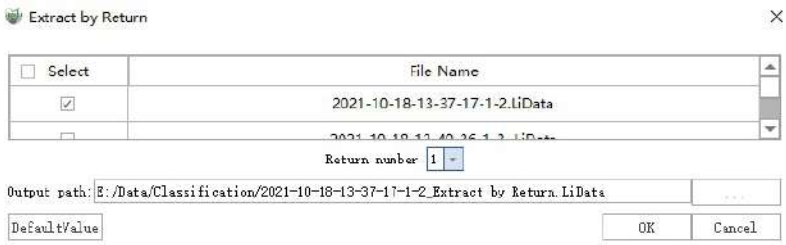
- **Input Data**: Input one or more point cloud data files. File Format: *.LiData.
- **Min (meters)**: The default is “100”. Users need to enter the minimum intensity value of the point cloud data to be extracted.
- **Max (meters)**: The default is “200”. Users need to enter the maximum intensity value of the point cloud data to be extracted.
- **Output path**: Path of the output folder. After the function being executed, the extracted new file(s) will be generated.

Extract by Return

Function Description: Extract by Return tool can extract all the point cloud data of the user-defined return number, and saved the data in a file. This function supports multiple file operations.

Steps

1.Click **Extract by Return** Button, pop up **Extract by Return dialog box**.



Parameter description：

- **Input Data**: Input one or more point cloud data files. File Format: *.LiData.
- **Return Number**: The user needs to select the number of echoes of the point cloud data to be extracted.Click the drop-down menu of the number of echoes and select the number of echoes to be extracted, including: 1-7 echoes.
- **Output path**: Path of the output folder. After the function being executed, the extracted new file(s) will be generated.

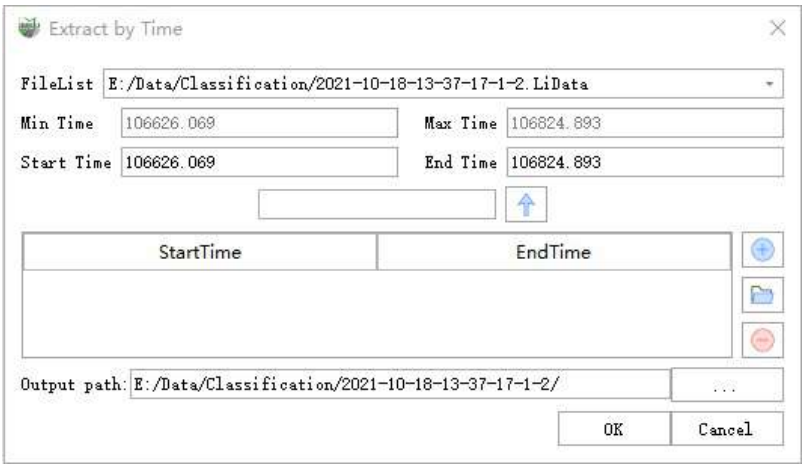
  > Note: IIf the data does not have the return number selected by the user, the extraction will not be executed.

Extract by Time

Function Description: Extract by Time tool can extract all the point cloud data within the user-defined time range, and save them in a file.

Steps

1.Click **Extract by Time** button, pop up the **Extract by Time dialog box**.



Parameter description:

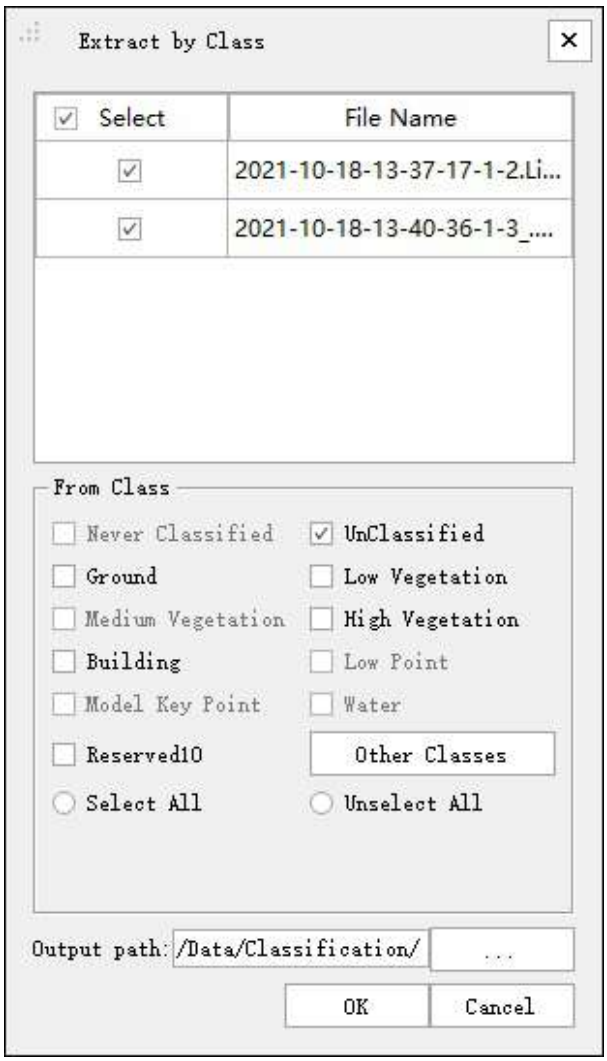
- **File List**: Select the file to be processed from the drop-down list.
- **Min Time**: Displays the minimum time value in the point cloud file selected by the user. This value does not require user settings.
- **Max Time**: Displays the maximum time value in the point cloud file selected by the user. This value does not require user settings.
- **Start Time (default value is "min Time")**: Input the minimum time value of the point cloud data to be extracted.
- **End Time (default value is "max Time")**: Input the maximum time value of the point cloud data to be extracted. This value must be larger than the start time.
- : If you want to extract the point cloud at a specified interval, enter the interval value in the text box, and then click this button, the values of the start time and end time will increase at the set interval.
- : Add the input time range to the range list. All point cloud data in the time range will be extracted into one file.
- : Users can click this button to load external GPS range file. The time range in the file needs to be between the min Time and the max Time.
- : Users select a row in the time range list and click this button to remove the row from the list.
- **Output path**: Path of the output folder. After the function being executed, the extracted new file will be generated.

Extract by Class

Function Description:Extract by Class tool can extract all the point cloud data of the user-selected class, and save the data in one file. This function supports multiple file operations.

Steps

1.Click **Extract by Class** button, pop up the **Extract by Class dialog box**.



Parameter description:

- **Input Data**: Input one or more point cloud data files. File Format: *.LiData.
- **From Class**: Users need to select the class to be extracted from the drop-down list. The unavailable status in the drop-down list represent the corresponding category that does not exist in the file.
- **Output path**: Path of the output folder. After the function being executed, the extracted new file(s) will be generated.

Profile

In the orthogonal projection top view mode, the profile view supports the user to arbitrarily define a rectangular area, and displays the side view of the area in an independent profile window. The profile window provides a lot of small tools, which are convenient for users to observe, measure, and even change data attributes at specific angles, such as modifying the category of point cloud data.

Profile and Measuring Tool

The main window of the profile supports all 3D measurement tools on the **Measure** page.

The sub-window of the profile only supports the measurement tools on the **Profile** page.

> Note: If users switch the main window to measurement tools, the profile tools will be activate. And to reuse the profile tools, it is necessary to reclick the profile tools.

Profile and Select Tool

The main window of the profile supports all point cloud selection (including cropping) tools in the main menu. The sub-window of the section only supports the selection tool on the **Profile** page.

> Note: If users switch the main window to point cloud selection tools interface, theprofile tools will be activate. To reuse the profile tools, it is necessary to reclick the profile tools.

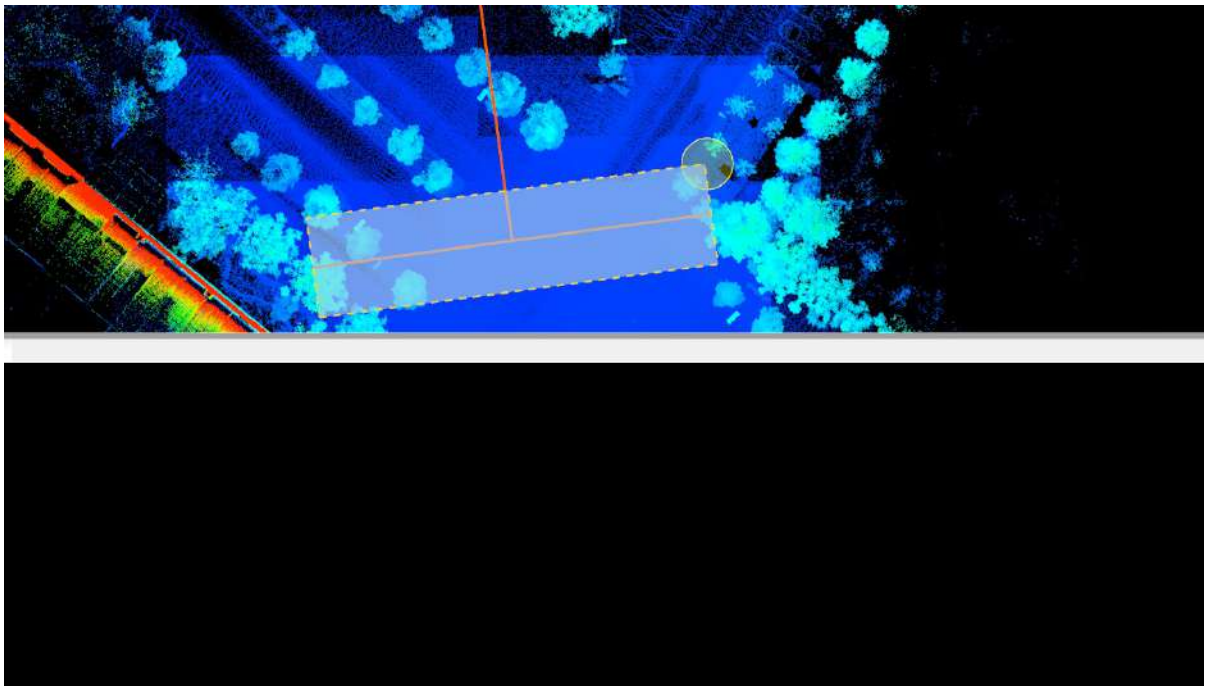
Select Profile Region

Function Description： Support the user to draw a profile area in the main view window, and display the data in the section window.

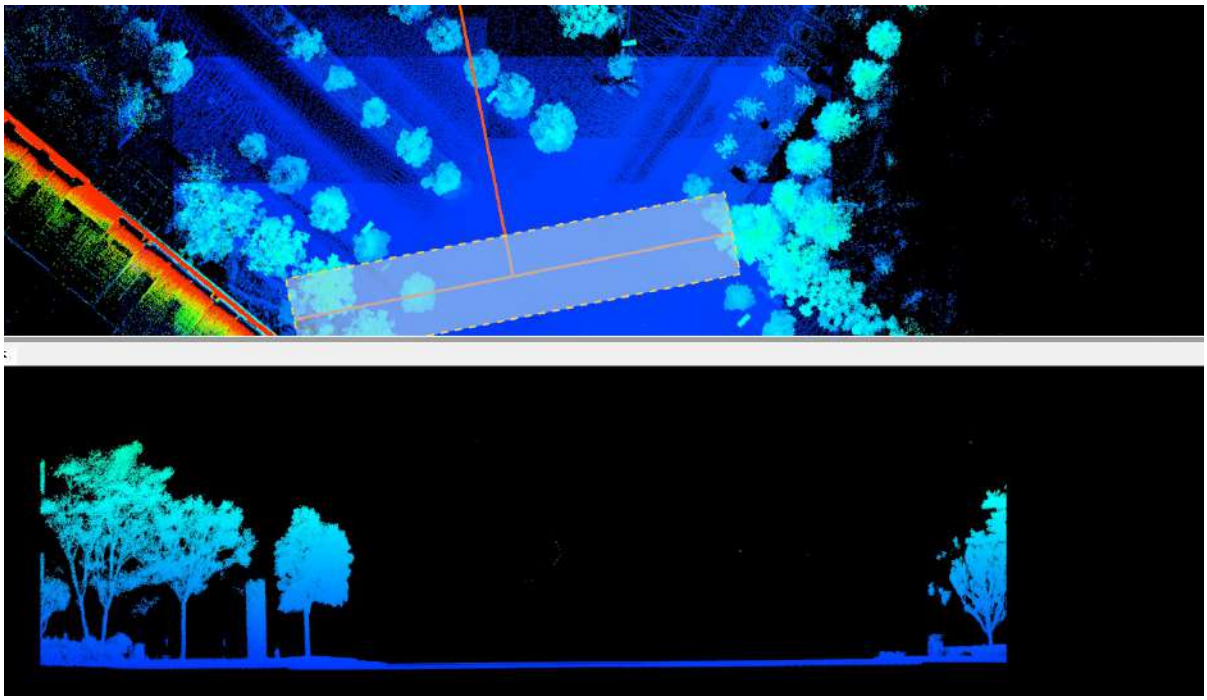
Steps

1.Move the mouse to 3D window to zoom and pan the scene to the target area.

2.Select the first point by left-clicking, move the mouse to select the second point. The profile direction is done.



3.Move the mouse to select the profile width, double left-clicking to finish profile area. The point cloud of the currently selected profile area will be displayed in the Profile window.



After the profile area is selected, the selection area may need to be adjusted according to different application scenarios.

LiDAR360 MLS provides a variety of adjustment methods for the selection area.

Fixed Buffer

Move Selected Section

Rotate Selected Section

Expand Selected Section

> Note: The software supports mixed use of tools mentioned above.

Fixed Buffer

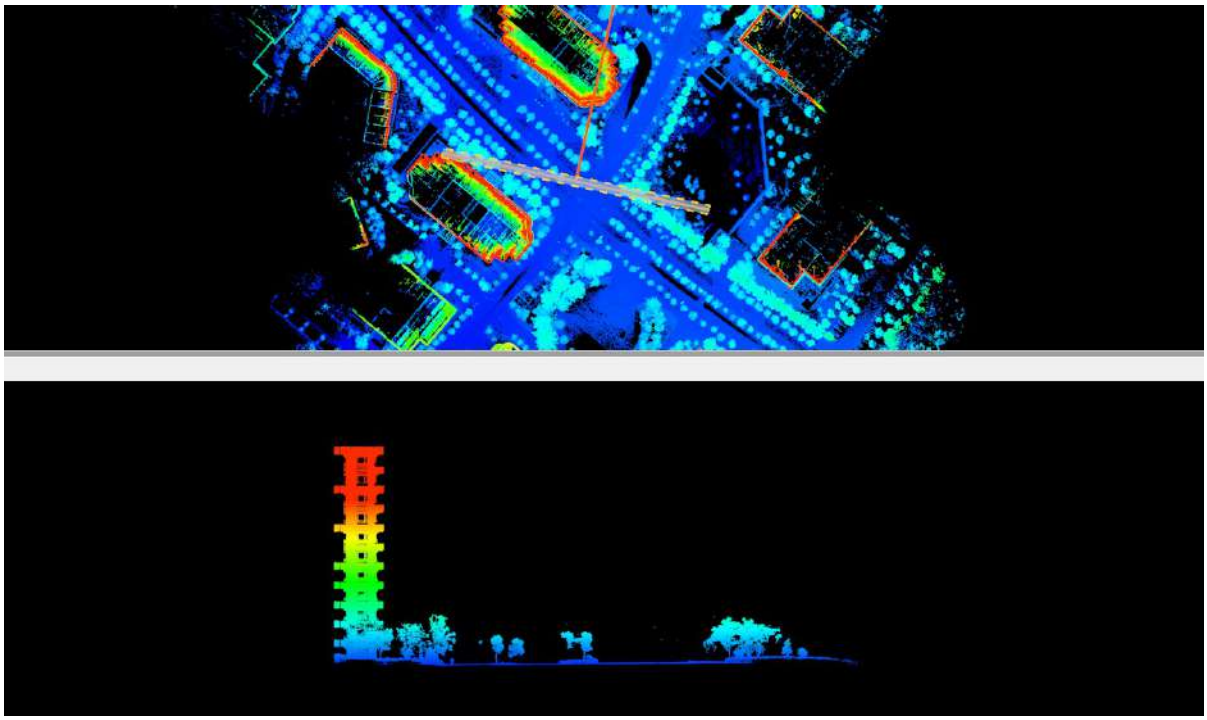
Function Description: Fixed rectangular width is supported to use.This function can help users to fix the size of the buffer area.

Steps

1.Set the **buffer value**, for example, set 2 meters as fixed buffer.

2.Click the **Buffer** button.

3.Select the first point by left-clicking, move the mouse to select the second point. The profile direction is done.



Measure Tools

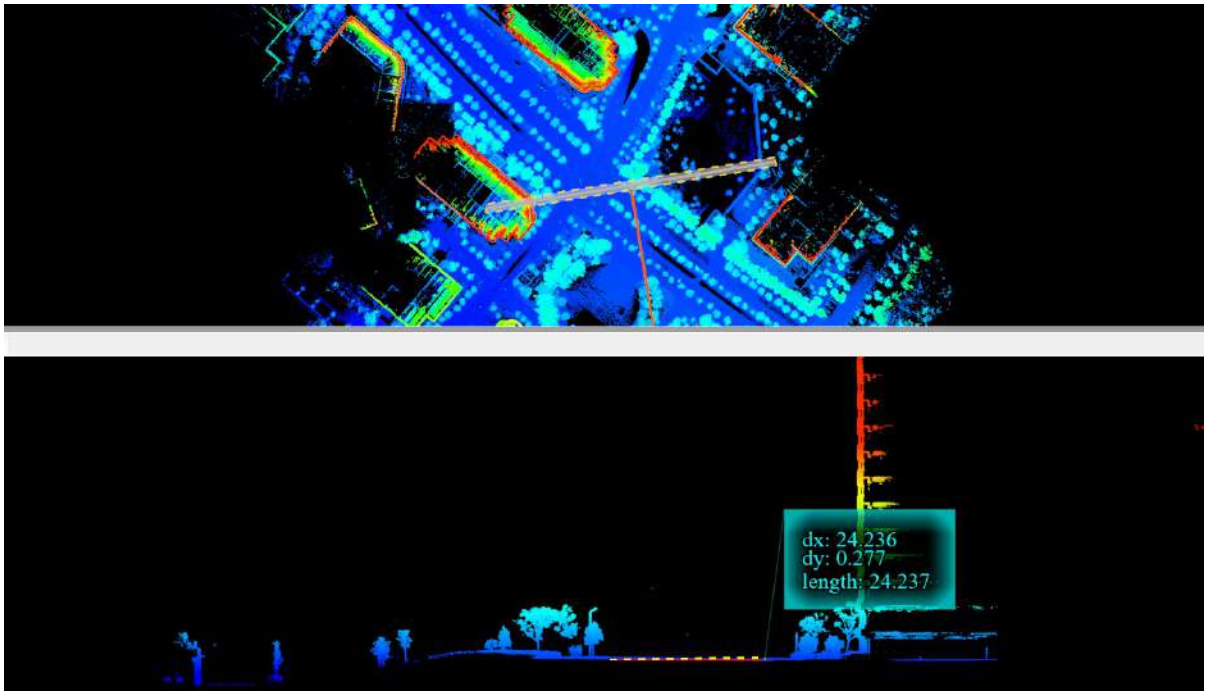
Function Description： Orthogonal projection is as default for profile window, and is convenient to calculate the horizontal and vertical distance.

Steps

1.Click **Measure**  button via **Profile** mode to start profile measurements.

2.Left-click to select the first point.

3.Select the second point by double-clicking to finish calculating the distance between two points.



Move Selected Section

Function Description: Used to move up (move down) the current profile to create a new profile with the same size and right above (below) the current profile. According to the direction and width of the current selection area, automatically move forward or backward by an offset.

Steps

1.Move the mouse to determine the width of the selection area, then double-click to confirm.

2.(optional) Click  to move up the profile.

3.(optional) Click  to move down the profile.

Rotate Selected Section

Function Description： To rotate the profile area based on the angle that users enter in the rotate settings window.

Steps

1.Click 

edit box, adjust the angle of each rotation, defaulting to "5".

2.Click  button to rotate the scene of the section window.

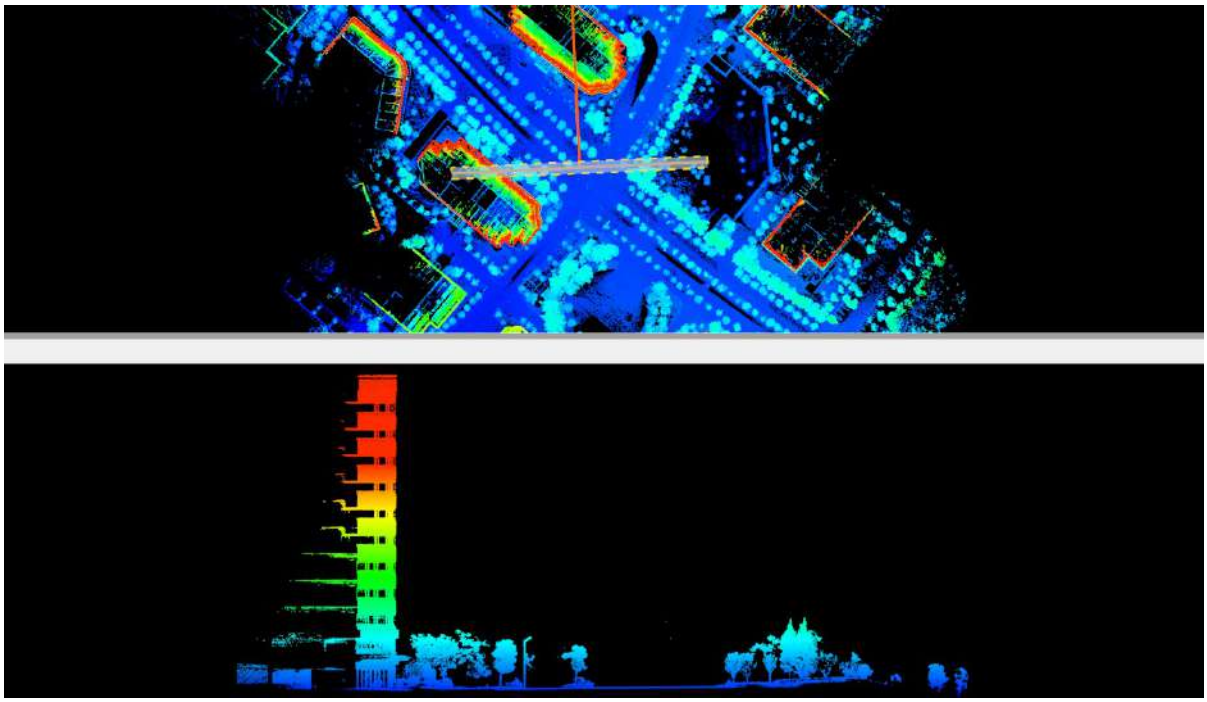
Expand Selected Section

Function Description：This tool can be used to expand the width of profile area by the entered value.

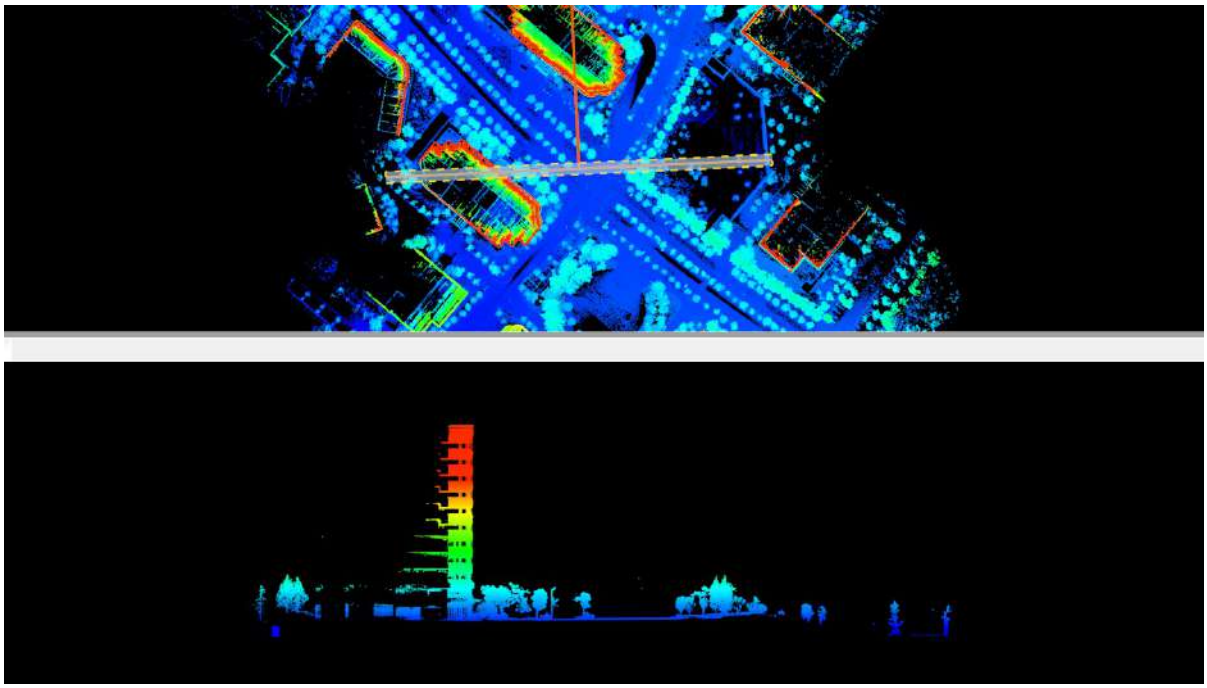
Steps

1.Click **Expand** button to expand current selected area.

Before expanding：



After expanding:



2.Click **Extent**  button to change the expand length, defaulting to "5".

Switch View

Function Description： These tool controls the view mode and rotation, where in addition to allowing the user to switch between front view, back view, left view, and right view, it also provides interactive rotation mode to enable viewing the point cloud from other angles:

Steps

1.By default, profile view shows the front view.

2.(optional) Click  button to switch to front view.

3.(optional) Click  button to switch to back view.

4.(optional) Click  button to switch to left view

5.(optional) Click  button to switch to right view.

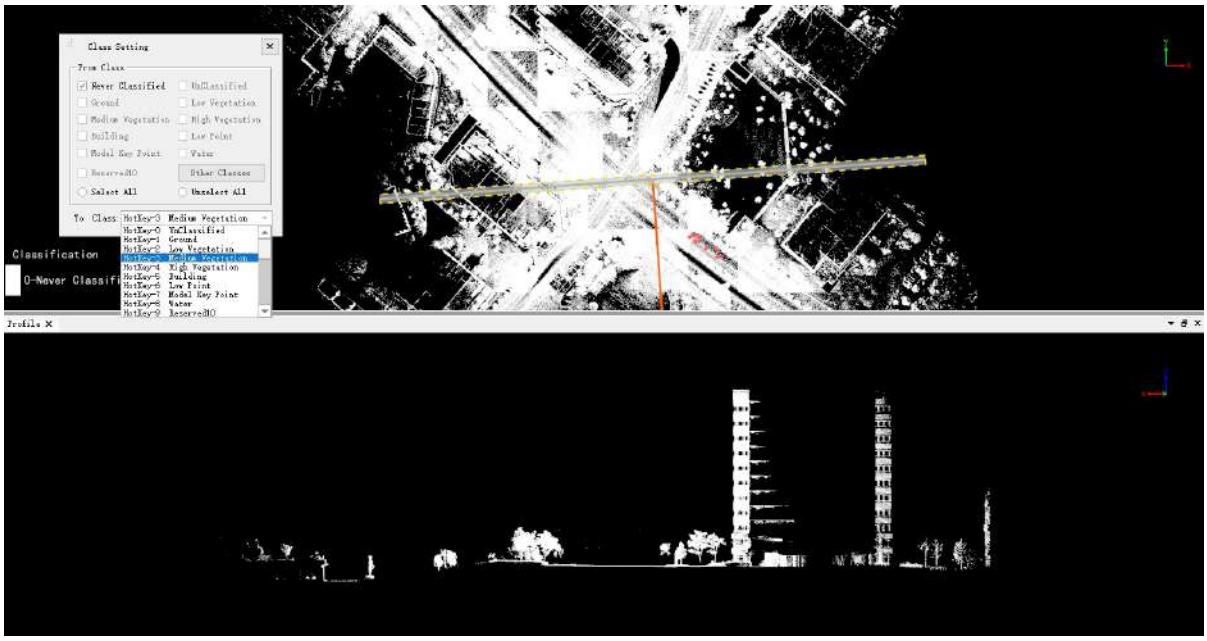
6.(optional) Click  button to start viewing at any angle. By default, the section scene cannot be rotated. After activating this button, you can observe the section point cloud at any angle.

Manual Classification

Function Description：A set of tools supported by profile window that can be used for manual classification.

Steps

1.Click the  button to set modified classification and targeted classification.



The category that needs to be modified can be determined by checking it. As long as it is selected by the selection tool, it will become the target category of the setting. For example, if you only want to turn unclassified points into vegetation points, you only need to check the unclassified points and uncheck the rest of the categories, and then set the target category to low vegetation points in the drop-down menu. In this way, in the next selection operation, all the selected unclassified points will automatically become low vegetation points, until the next time the initial category is re-checked or the target category is re-selected.

2.Choose a suitable selection tool to modify the point cloud category：

Polygon Selection  ：It is suggested for complexed polygons using Even-odd Rules. For example, a tree or a building.

Rectangle Selection  ：In some cases where the shape is relatively simple, or the requirements are not particularly strict, you can directly use the rectangular selection. Compared with polygon selection, the sides of the rectangle selection can only be parallel to the sides of the window. In other words, only rectangles aligned with the coordinate axis of the window are supported.

Circle Selection  ：In some cases, circular selection is also very convenient, such as choosing a circular pool or a flower bed.

Above the Polyline Select  ：The point cloud above the line will be selected.

Below the Polyline Select  : The point cloud below the line will be selected.

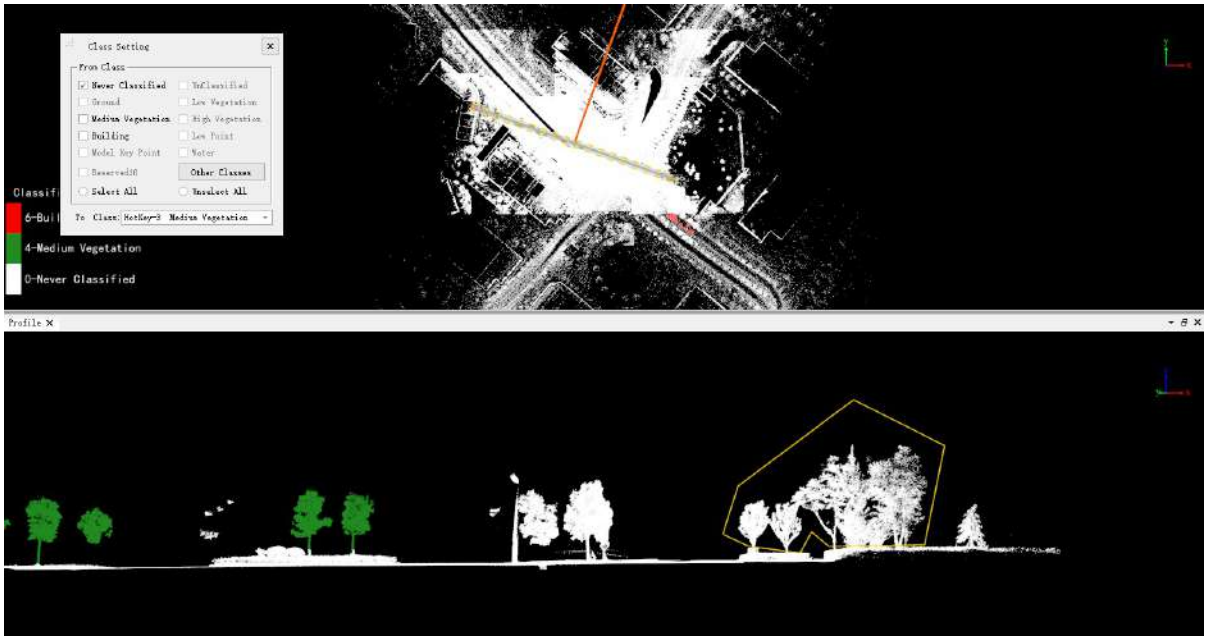
3.Adjust the display mode to display by category, and you can clearly see the point cloud selected in the previous step.

4.(optional) Use Ctrl+Z to undo the previous steps. Or by clicking  button to clear all the unsaved steps.

5.Click  to save the profile.

Example (Use the polygon selection tool to modify the point cloud category):

Draw a polygon (double-click the left mouse button to end the selection), then click to save the result



The result of changing the category will be displayed in the main 3D point cloud window in real time.

> All operations of point cloud classification are temporary, and must be saved after modification to take effect.

Panorama Measure Tools

The panorama menu mainly includes sequence frame display, panorama setting, panorama measurement, and result panel and configuration window. The panorama measurement can be divided into two ways: measurement based on point cloud depth interpolation and measurement based on image forward intersection.

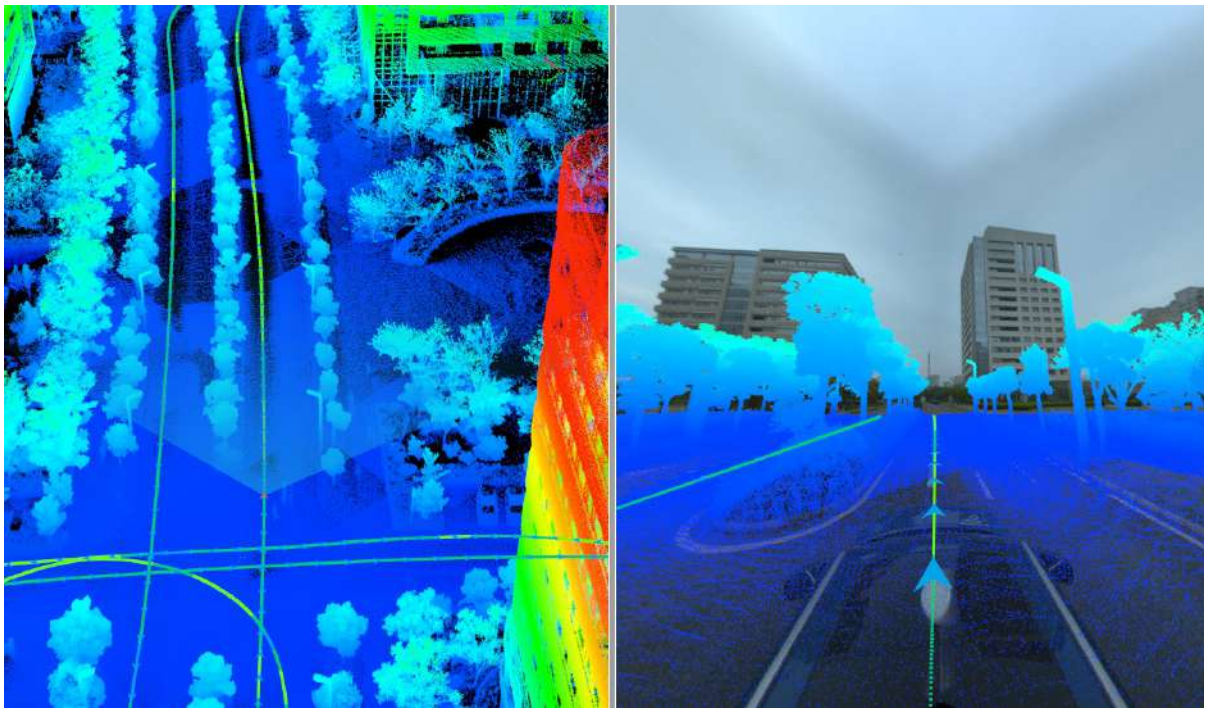
The panorama menu is displayed only after image data is added when a new project is created. Before measurement, switch the top menu of the software to the panorama page.

Point Cloud and Panoramic Image Roaming

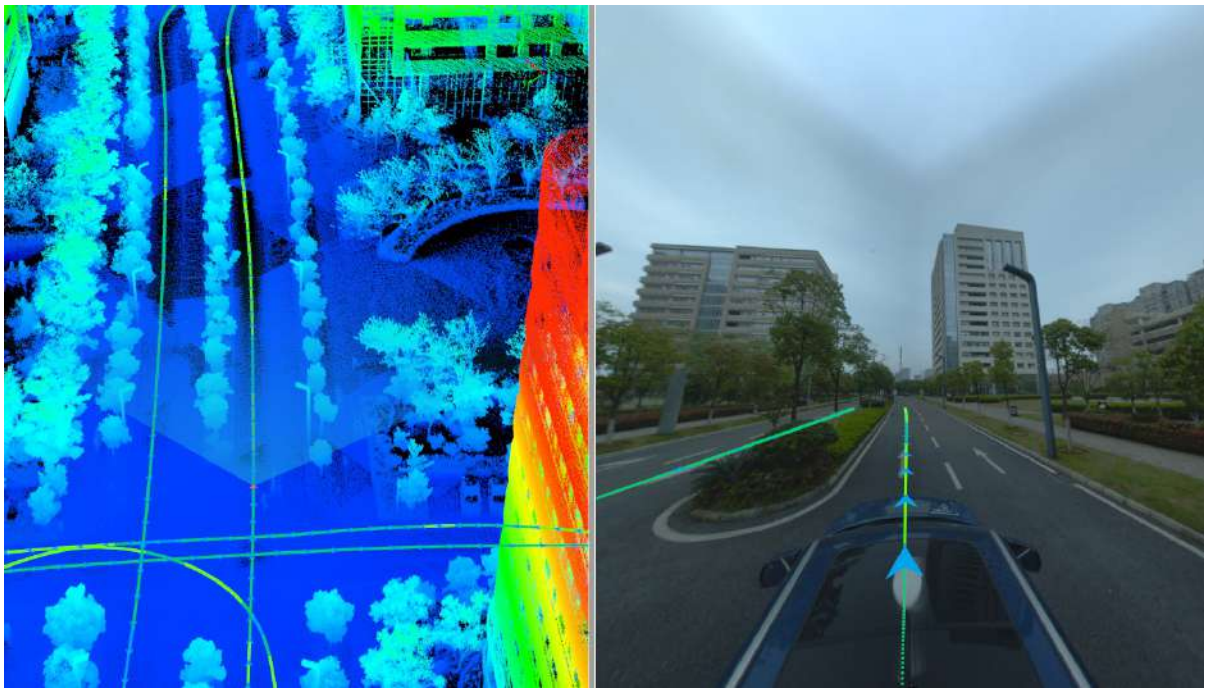
Function Description： The software supports point cloud and panoramic image roaming. The original camera file must be added when creating a new project before the menu bar will be activated.

Steps

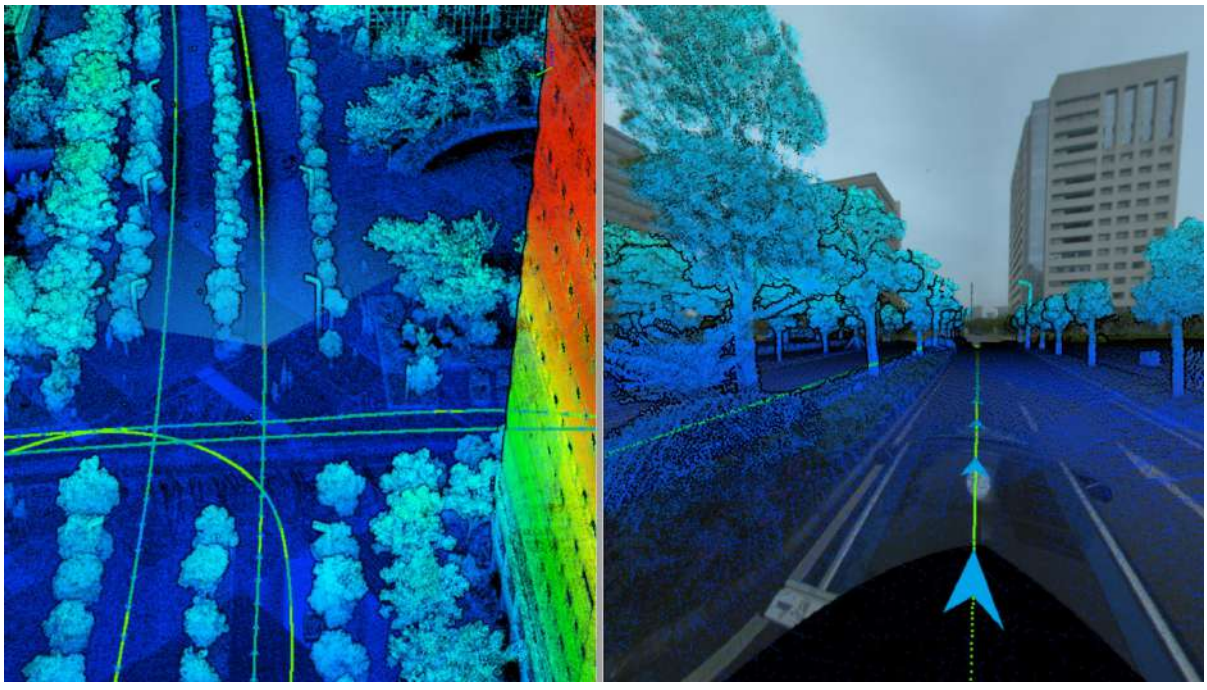
1.After a new project is created, a 3D window and a panoramic window will be opened. The point cloud data will be displayed in the 3D window and the panoramic window at the same time, and the panoramic image will only be displayed in the panoramic window. You can choose to show or hide the corresponding data through the check box in front of the data name in the project management window.



2.Uncheck the menu bar ☐ Show Point Cloud , only the image data that you can browse in the panoramic window.



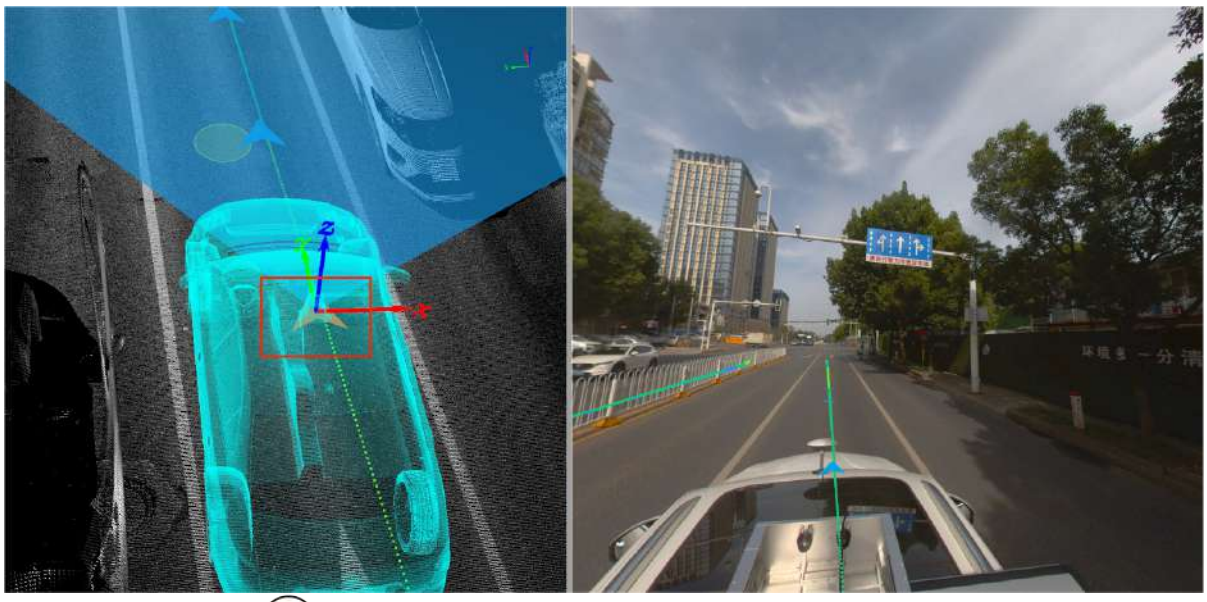
3.Click the color bar tool on the left side of the window to switch the display mode of the point cloud, including by elevation, intensity, category, RGB, echo times, time, mixed display, combination display, display by selected color and EDL display, EDL can be used in conjunction with other display methods to enhance the display of the contour feature information of point cloud features. The following figure shows the effect of point cloud display by elevation + EDL.



> Note:
>
> - The display modes of elevation, intensity, RGB, etc. on the toolbar affect all point clouds in the project. If you need to give a separate display effect to a certain point cloud, click the right mouse button on the data name and select **View mode** > By elevation/intensity/category, etc.

> - The EDL display effect is related to the window. If you need to display the EDL effect in a certain window, click the mouse anywhere in the window to activate the window, and then click the EDL button.

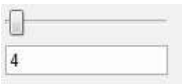
4.Click the **Select Frame** button to select the image exposure position on the trajectory in the 3D window or the panoramic window (the default is the blue triangle arrow symbol, and the orange after clicking) to jump to the selected image position.



5.Click the **Next Frame** button to jump to the next frame of the current image. Click the **Previous Frame** button to jump to the previous frame of the current image.

> Note: Shortcut of next frame and previous frame:

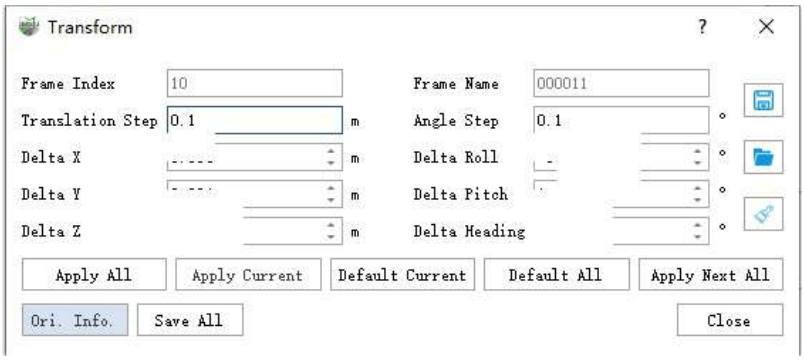
Shortcut key	Function
PageUp	Next Frame
PageDown	Previous Frame

6.Drag the slider 

to switch between different image positions or directly input the number of image frames.

7.Click the **Auto Roaming** button to roam the point cloud and image from the first-person perspective in the panoramic window, and click the button again to stop the automatic roaming. If the perspective of the panoramic window changes, click the space bar on the keyboard to restore the default forward direction.

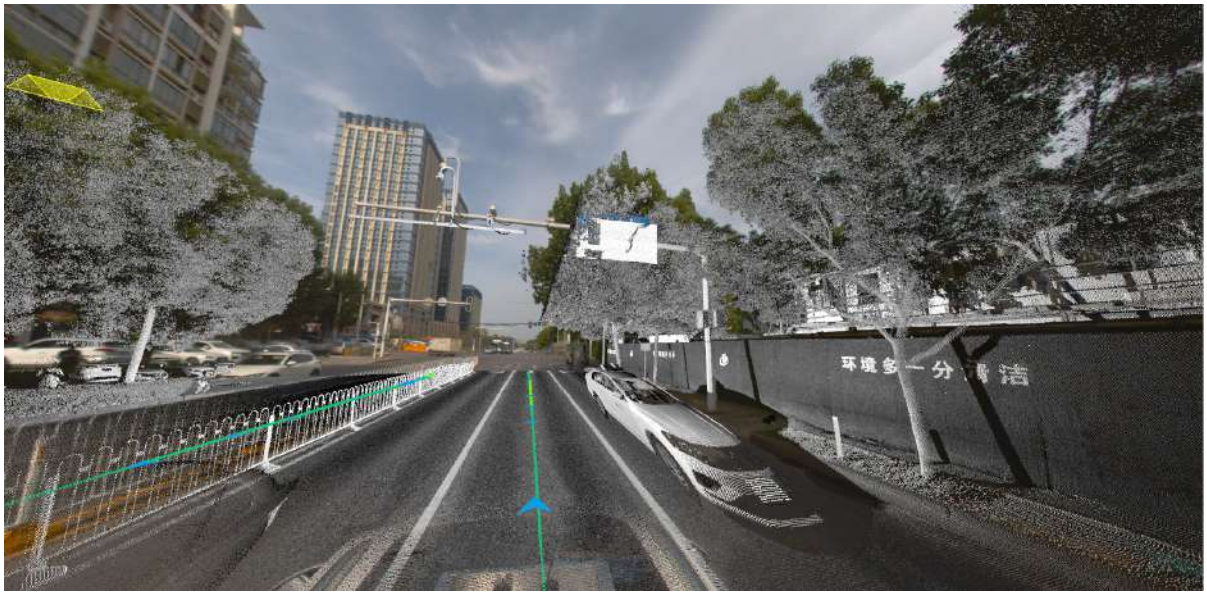
8.Click **Transform** button, the transformation is mainly to adjust the pose of the image in real time and superimpose the point cloud perfectly.



Description

- **Apply to All**: Apply the image pose set in the current dialog to all frames, including panorama images before and after the current frame.
- **Apply to Current**: Apply the pose set in the current dialog to the image of the current frame.
- **Restore Current Default**: Restore the pose of the panoramic image of the current frame to the original default value.
- **Restore All Defaults**: Restore all frame panorama images to their original default values.
- **Apply to All subsequent**: Apply the pose in the current dialog to all image frames after the current frame.
- **Bearing Information**: Displays the bearing information of the current frame.
- **Save All**: Save the current pose information to the imglist file and update the display.
- **Close**: Close the dialog.
- **Save**: Save the current pose information to a txt file.
- **Open**: Read the pose in the txt file and display it on the dialog.
- **Clear**: Clear the pose data in the dialog.

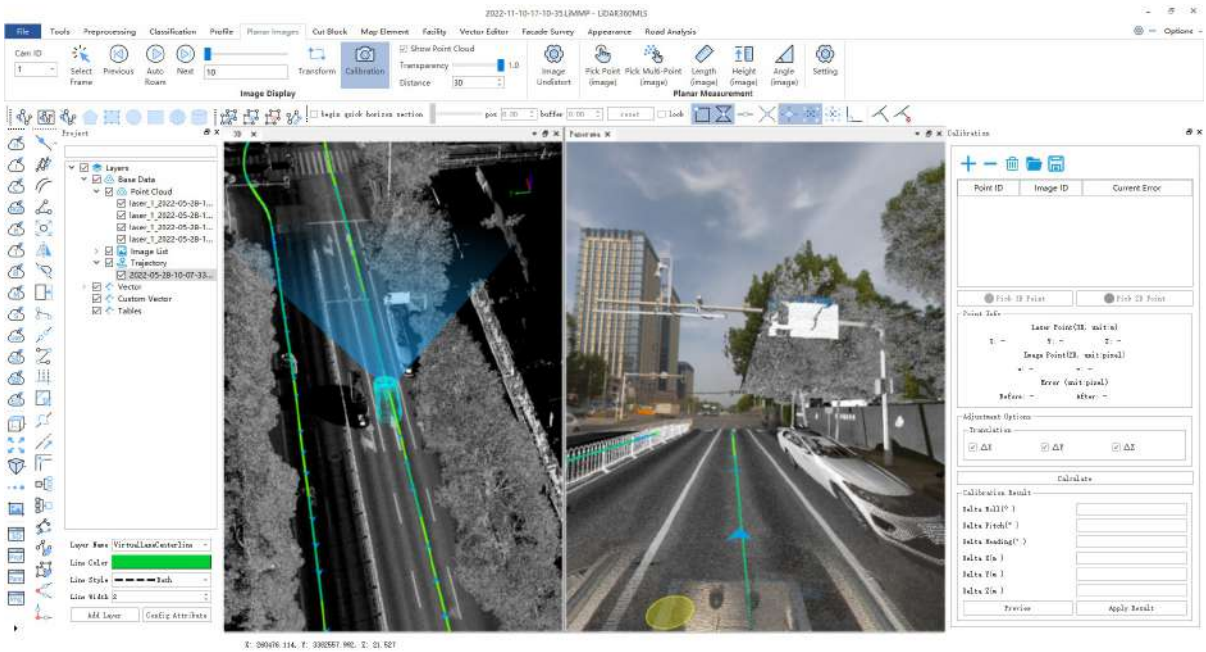
9.Due to the placement error between the panoramic image camera and the laser, there is still a certain level of deviation between the plane image and the point cloud data, which cannot be superimposed perfectly, as shown in the following figure:



LiDAR360 MLS provides an image calibration module to estimate the placement error between the plane camera and the laser, thereby improving data bias and improving accuracy.

> Note: Set the point cloud display depth by adjusting the menu bar Panorama->Panorama->Radius, so as to hide the distant point cloud and facilitate the observation of the area of interest. This little trick can also be used in the calibration point selection process below.

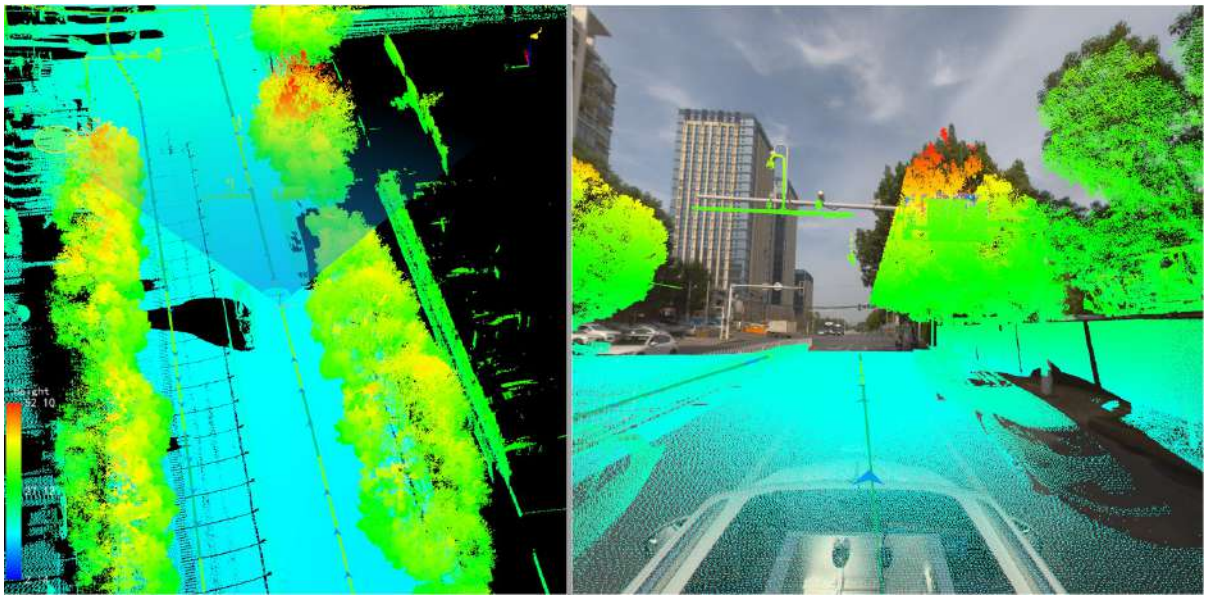
1.Switch to the menu page, and click **Calibration**  button, the calibration window will pop up on the right side of the user interface.



2.Add calibration point. It is recommended to select camera exposure points in multiple directions of the scene, and then select multiple point pairs at each exposure point, at least 4 point pairs need to be added (for example, 4 point pairs are selected at the four exposure moments of the south, south, north,west, that is, a total of 16 point pairs are selected).

The specific steps are:

- Click the *Select Frame* button, click the triangle arrow in the 3D view window, and the image window will automatically switch to the corresponding image frame.

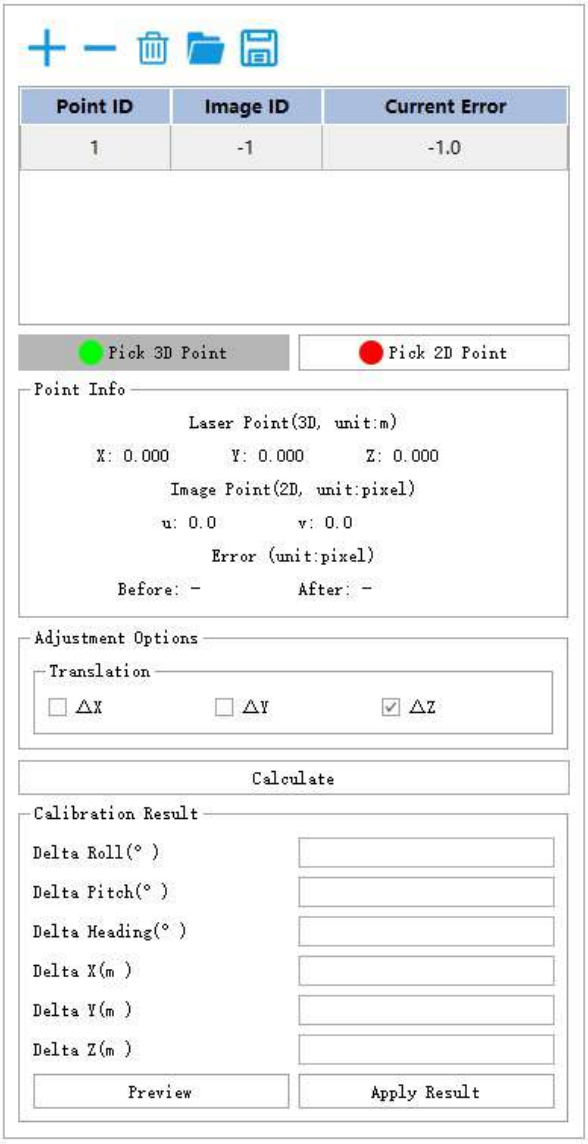


- Click the *Add Point* button at the top of the calibration window to add a row to the point list.

- Click the *Pick 2D Points* button to select image points in the image window. The top of the picture is a magnifying glass, which can enlarge the position of the mouse.

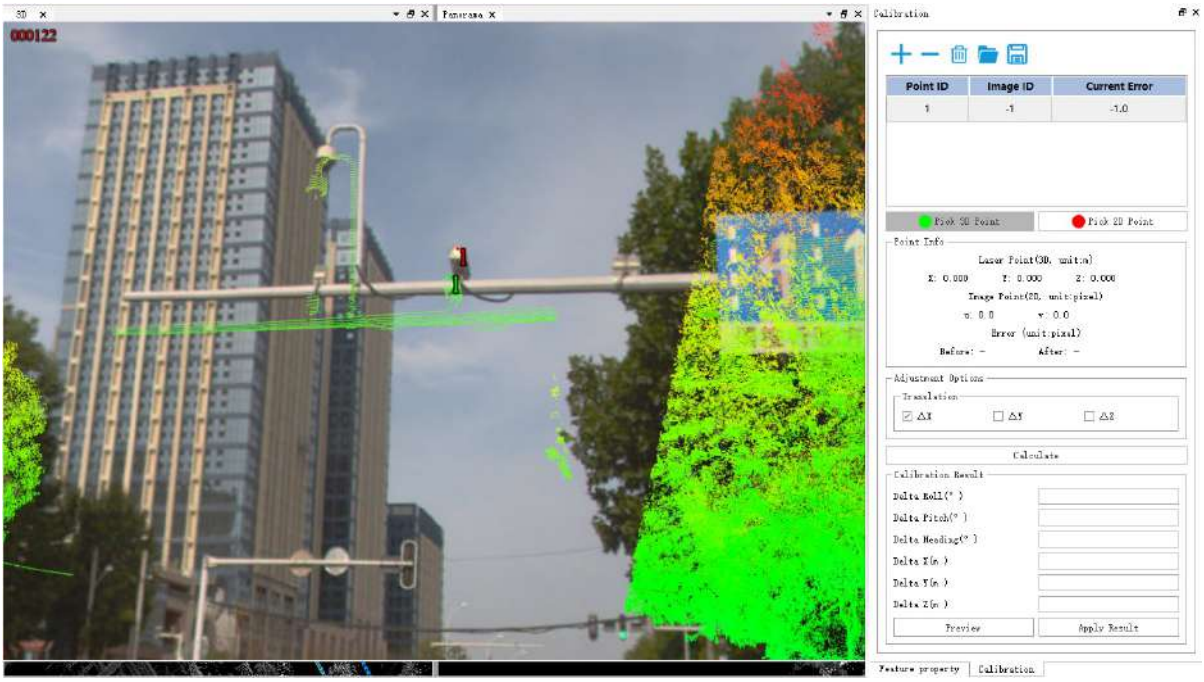
- Click the *Pick 3D Point* button to select the laser point corresponding to the image point in the image window.
- Repeat the above three steps until enough point pairs are selected on the current image frame (four point pairs are recommended).
- Select the corresponding image frames in the other three directions, and repeat the above steps until all points are selected (a total of 13 point pairs are selected in this example).

3.Click the *Calculate* button to start the calibration calculation.

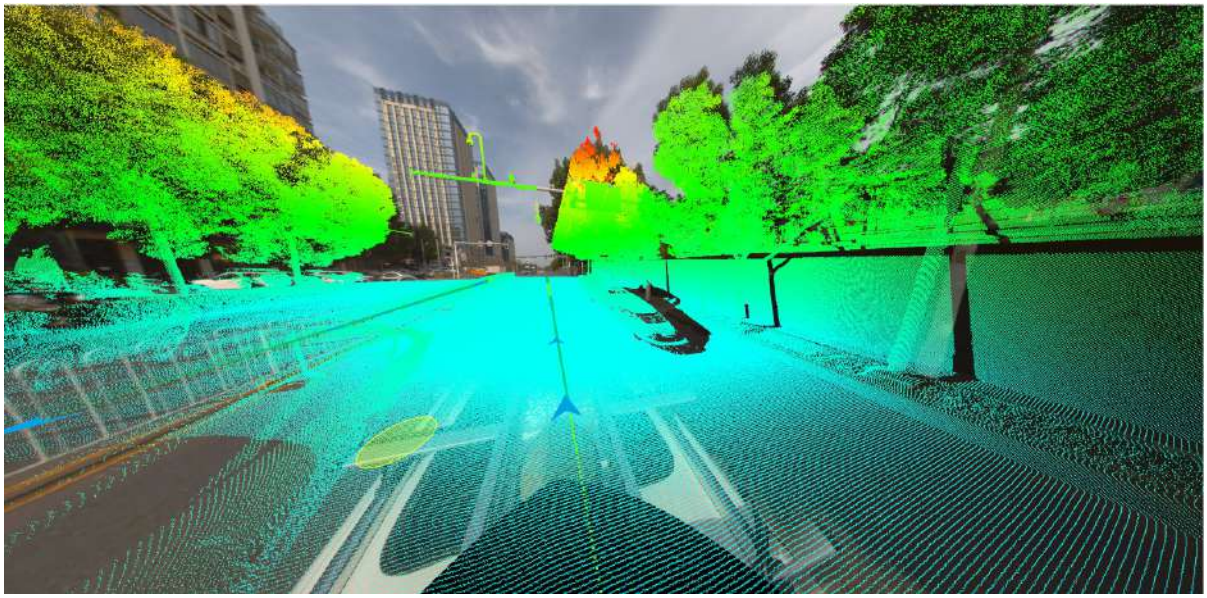


4.Click the *Preview* button to preview the calibration result.

Before preview:



After preview:



5.Apply the result. Check whether the preview results meet the requirements, and click the *Apply* button to apply the calculated calibration parameters to the data.

> Note:
>
> 1.There are two ways to control the display/hide of the laser point cloud: click the Show Point Cloud button on the menu bar; adjust the transparency scroll bar. By controlling the display/hide of the point cloud at the right time, the efficiency of point selection can be effectively improved.
>
> 2.In general, it is more convenient to set the point cloud display mode to display by intensity.

Panorama Measurements Based on Point Cloud Depth Interpolation

The principle of estimating the location information of measuring points is based on the point cloud data depth value and interpolation algorithm within a certain range around the measuring point.

The main functions are listed below:

- Pick Point
- Pick Multi-Point
- Length Measurement
- Height Measurement
- Angle Measurement

Pick Point (Depth Interpolation)

Function Description： This tool is applicable to panoramic data, in which the attributes that can be queried contain position information.

Steps

1.Click **Pick Point(image)**  button via **Panorama Measurement** mode.

2.In the panoramic display window (*Panorama*), use the left mouse button to click to select the point to be measured, and a pop-up box will appear in the click complete window to display the detailed information of the selected point.

3.Right click mouse button to cancel the selected point.

Multi Pick Point (Depth Interpolation)

Function Description： The multi-point selection tool allows the user to select and query the attribute information of multiple points in the image, and supports the export of the selected point set in .txt format. The attributes that can be queried for the data include the serial number, position, image ID, and pixel position of the point.

Steps

1.Click the **Pick Multi-Point (image)** button in **panoramic measurement**, and use the left mouse button to click the single points in the scene in turn. The selected points are marked in the scene in the form of point labels, and at the same time, the pop-up interface list displays the attribute information of the point set (as shown in the figure).

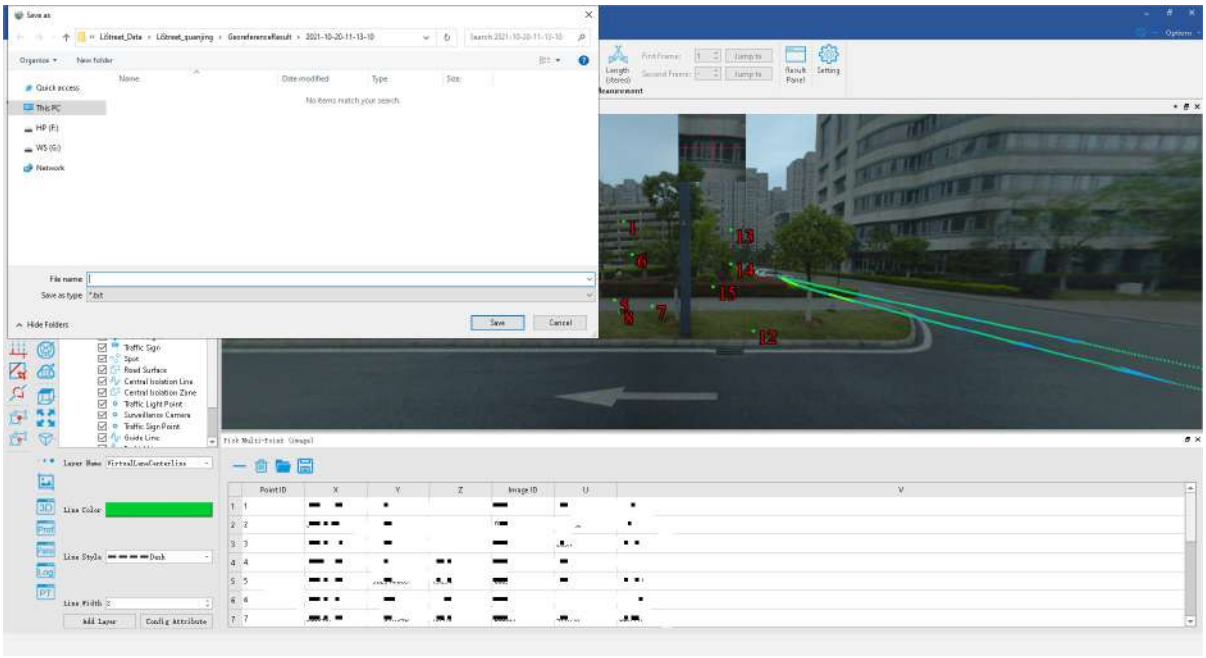
The point cloud attributes displayed in the list include point number, location, image ID, and pixel location.

Parameter settings

- **X**: The X coordinate value of the point cloud data.
- **Y**: The Y coordinate value of point cloud data.
- **Z**: The Z coordinate value of the point cloud data.
- **Image ID**: The ID number of the image.
- **U**: The abscissa of the pixel.
- **V**: The vertical coordinate of the pixel.

2.Click any row in the selection list and click the button to delete the point.

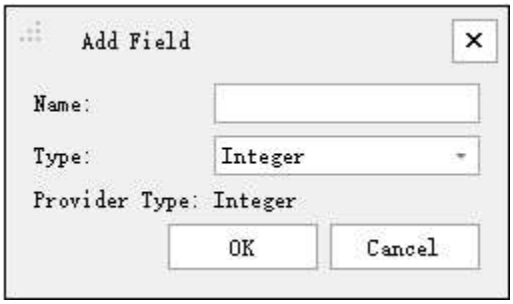
3.The selected point set is exported in *.txt* format. Click the button to save the coordinates and other attribute information as a *.txt* file.



4.If a point has been created by user, click  to open the *.txt* file saved before.

5.At the same time, if the user does not need the multiple points currently selected, or if the user has saved multiple points and needs to continue to click multiple points, he can directly click the  button to clear the interface.

6.To add a user-defined field, click to add field directly  .



7.To modify the value of a user-defined field, you can directly click Modify Field Value  .

8.To delete a field, click the button directly  .

> Please note: This function can only be used in the panoramic view window.

Length Measurement (Depth Interpolation)

Function Description： Length measurement tools are available for panoramic data and allows the user to query the distance information between two points by clicking the mouse. The measurement result represents the Euclidean distance of the point in the three-dimensional space.

Steps

1.Click **Length(image)**  button via **Panorama Measurement** mode.

2.Left click in the Panoramic display window (*Panorama*) to select the starting point of length measurement.

3.Click the left mouse button to select the next measurement point.

4.Repeat the previous step until all measurement points are selected, double-click the left mouse button to complete the measurement, and the interface pop-up box displays the measurement length information.

5.Click the right mouse button before completing the measurement to cancel the last measurement point.

Height Measurement (Depth Interpolation)

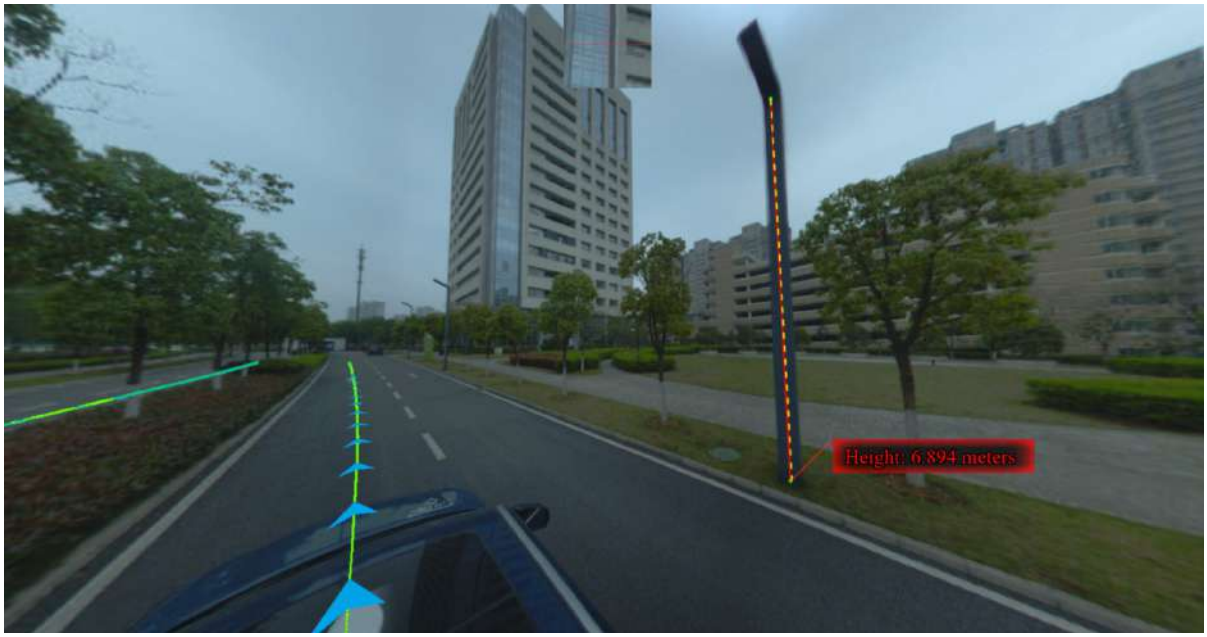
Function Description： This tool is applicable to point cloud data and works on panorama data. It allows the user to calculate the relative height difference between two points in panorama window.

Steps

1.Click **Height(image)**  button via **Panorama Measurement** mode.

2.Select the starting point of height measurement by left-clicking in panorama window.

3.Double-click the left mouse button to select the end point of the height measurement to complete the measurement. The interface pop-up box displays height information.



4.Right-click to go back to the previous point during the measurement.

Angle Measurement (Depth Interpolation)

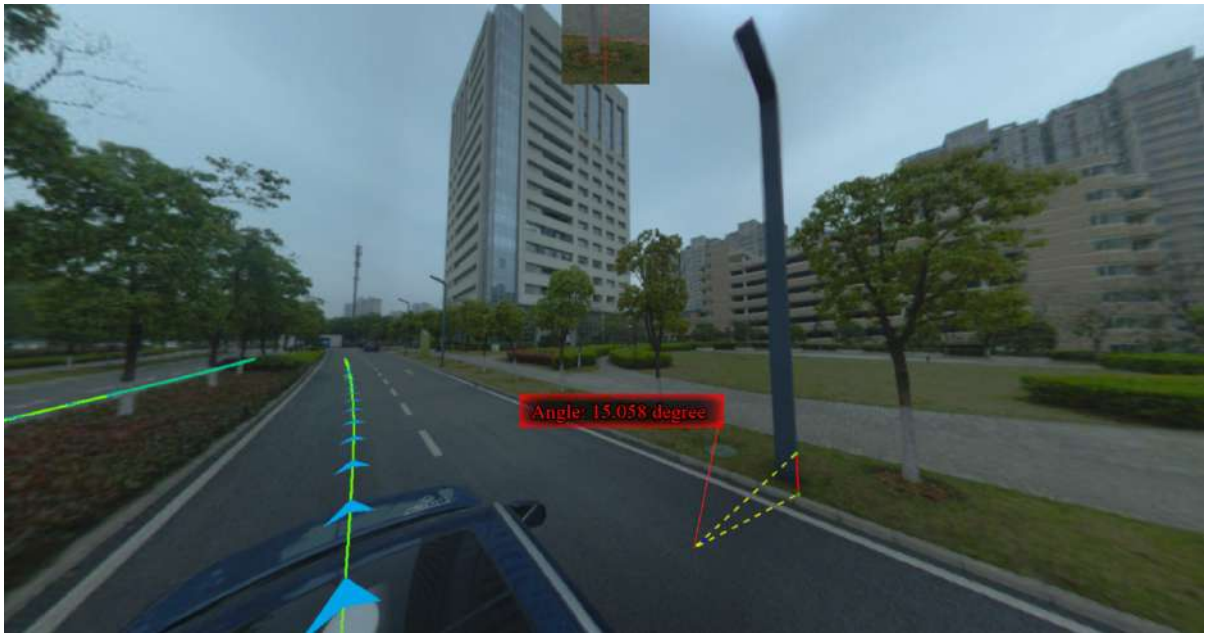
Function Description： The Angle measurement tool works on panorama data and allows the user to select the measurement point, and calculates the angle of pitch between two points in Panorama view.

Steps

1.Click **Angle(image)**  button via **Panorama Measurement** mode.

2.Select the starting point of angle measurement by left-clicking in (Panorama) view.

3.Select the measurement end point by double-clicking. The pitch angle between the reference point and the measurement point will be rendered in the scene and the measurement result is displayed in a label as follows.



Panorama Measurements Based on Forward Intersection

By using the measuring point to select the tie-points on the two-frame image, and combining with the forward intersection algorithm, the measurement point position information is obtained.

The main functions are listed below

- Pick Point
- Length Measurement

Pick Point (Forward Intersection)

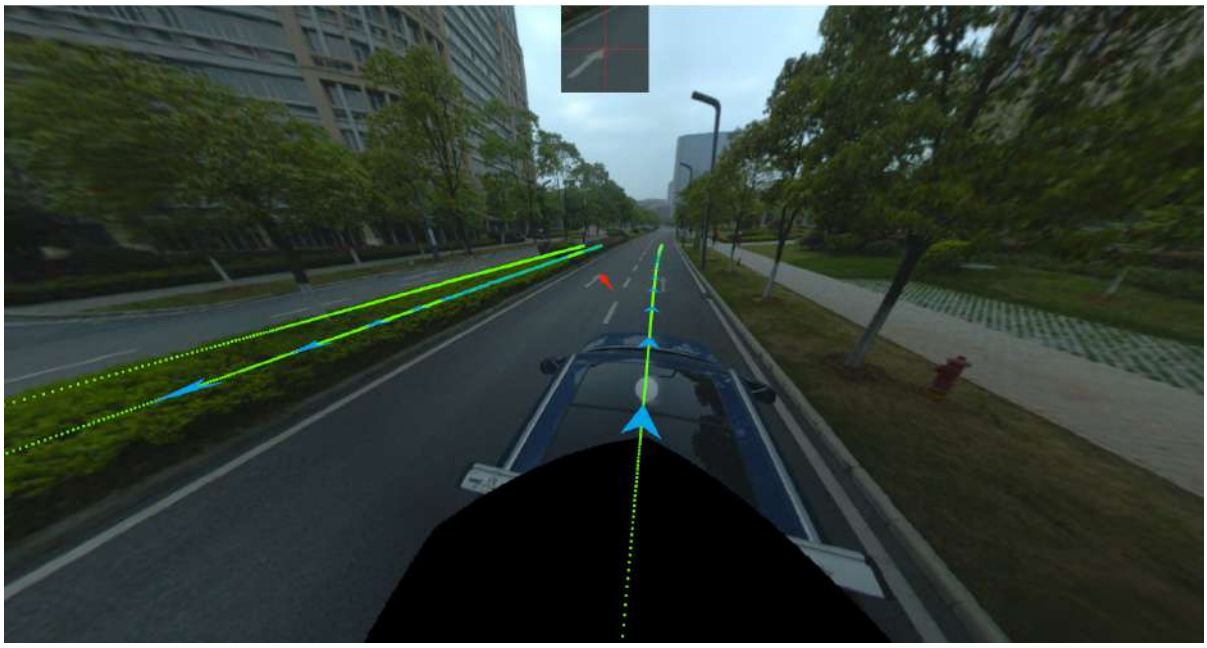
Function Description: This tool is applicable to panoramic data and allows the user to query attributes that contain position information.

Steps

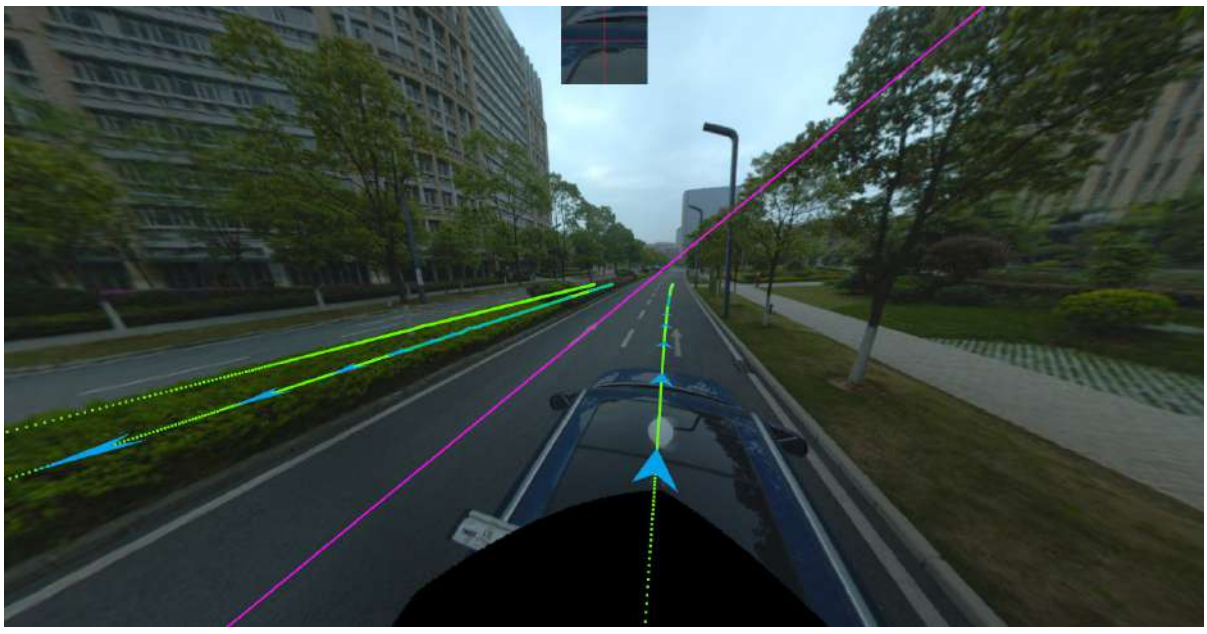
1.Click Pick Point(stereo)  button via Panorama Measurement mode.

2.Select the first corresponding point by left-clicking in panorama window.

You can use the small window of the image crosshairs for auxiliary selection:

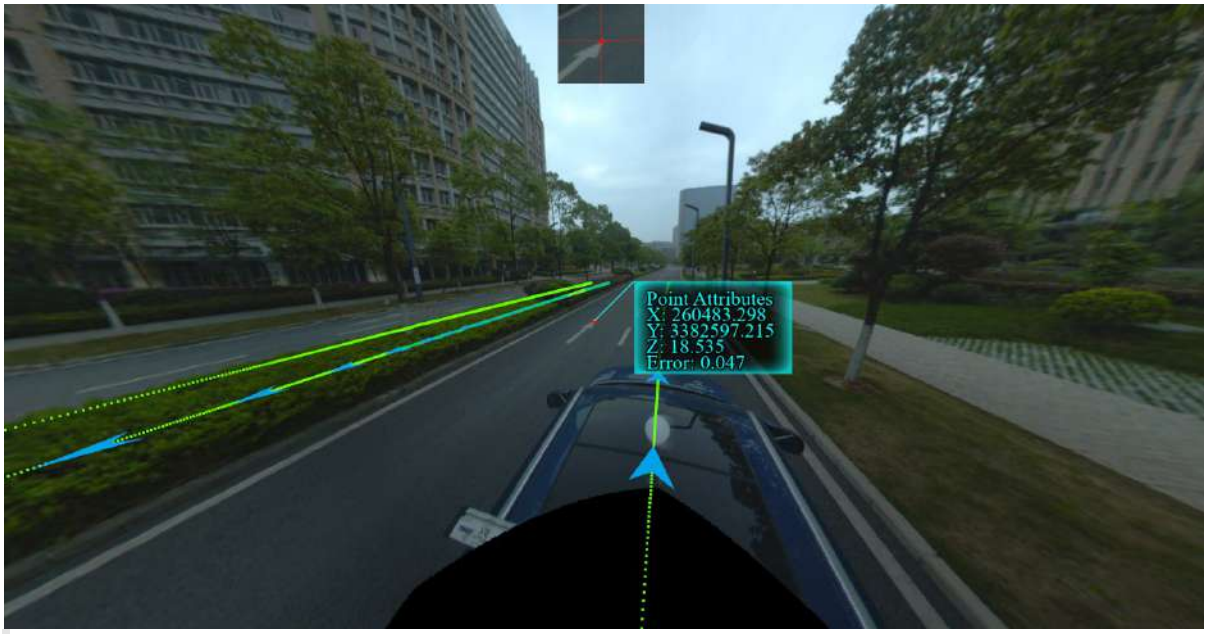


The panorama window will switch to the second frame image automatically and the auxiliary line will display：



3.The image will update automatically in panorama window. Select the second corresponding point by left-clicking, and the auxiliary line shown on the image helps to select the point.

Select the corresponding point on the second frame image:



Note:

- The auxiliary line displayed in step 3 is actually the epipolar line generated by the camera corresponding to the first frame image and the second frame image. If the placement error between the panoramic camera and the laser has been corrected (in this case, we think the image pose is more accurate), the second point with the same name we selected is theoretically near the line, so this line can be used to assist in locating the point of with the same name.
- You can switch the first frame of image to be measured before selecting the first point with the same name, and switch to the second frame of image to be measured after selecting the first measurement point. The specific switching method is to edit the frame numbers after the **first frame** and **second frame** respectively and click the **jump** button or press the *Enter* key on the keyboard.

Length Measurement (Forward Intersection)

Function Description： Length measurement tools are available for panoramic data and allows the user to query the distance information between multiple points. The measurement result represents the Euclidean distance of the point in the three-dimensional space.

Steps

1.Click **Length(stereo)**  button via Panorama Measurement mode.

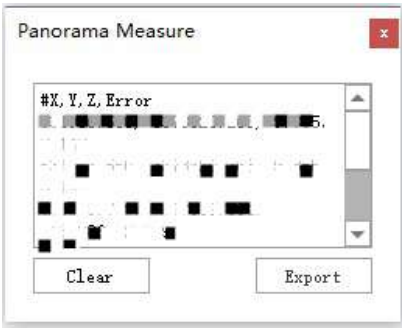
2.Left-click to select the starting point in the scene, the operation is the same as the Pick Point step.

3.Select the end point, the operation is the same as the Pick Point step.

4.Right-click to go back to the previous point during the measurement.

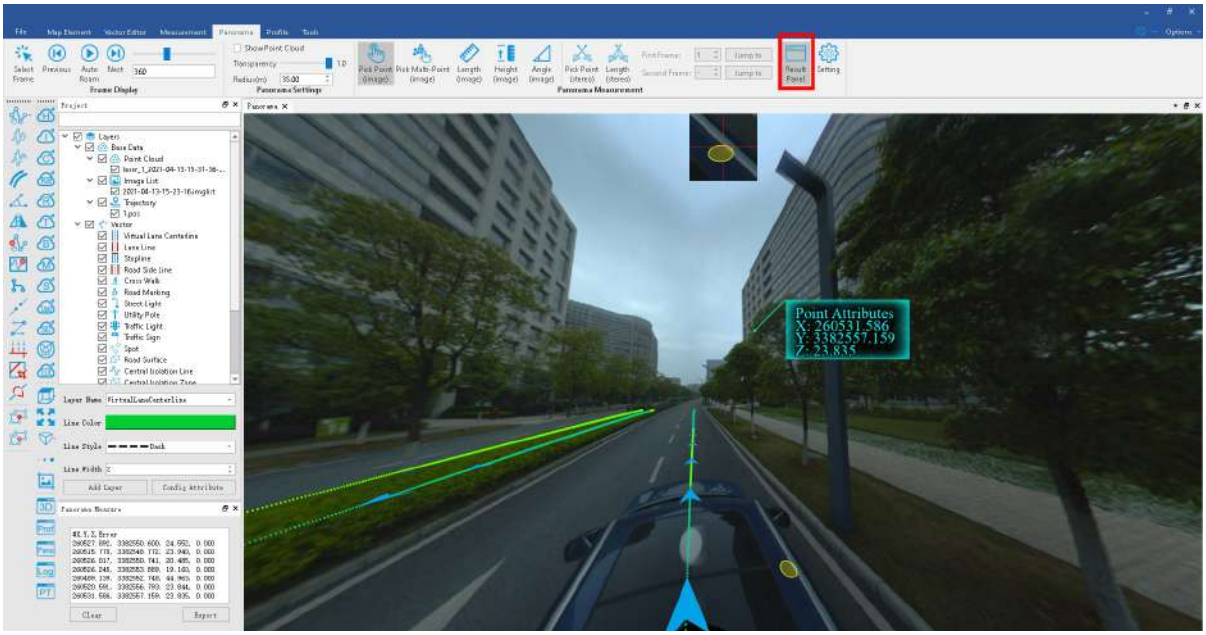
Result Panel

Function Description： The result of panorama measurements can be recorded in Result Panel and it is supported to export measured points list.



Steps

1.Click Result Panel  button via Panorama Measurement mode. The Result Panel will pop up from the right corner.



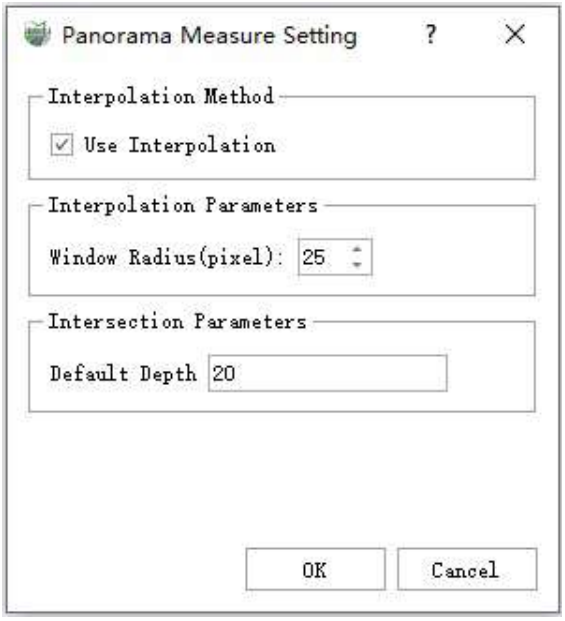
2.Click Clear button to clear Result Panel.

3.Click Export button to export measured points list.

Dialog Setting

Function Description： The Dialog Setting is allows the user to set parameters for running the function of panorama measurements.

1.Click **Setting** to pop up the dialog box:



Parameters Description

- **Interpolation Method**: This function is set by default. It is available to select the complex interpolation algorithm when using the function of Panorama Measurements based on Panorama Measurements Based on Point Cloud Depth Interpolation. If this function is unselected, nearest neighbor algorithm is used to calculate the point location information.
- **Interpolation Parameters**: This is the windows radius to be used when running interpolation algorithm. For example, when the windows radius is N, all the depth value around the measured points in the square area whose length value is 2N + 1 (the unit is pixel) will be read. If the point density is not high and measurements can not perform with default windows radius, users can increase the value of windows radius.
- **Intersection Parameters**: This is the assumed depth of measured points. It is recommended in the second frame image when using the function of Panorama Measurements Based on Forward Intersection.

Planar Image

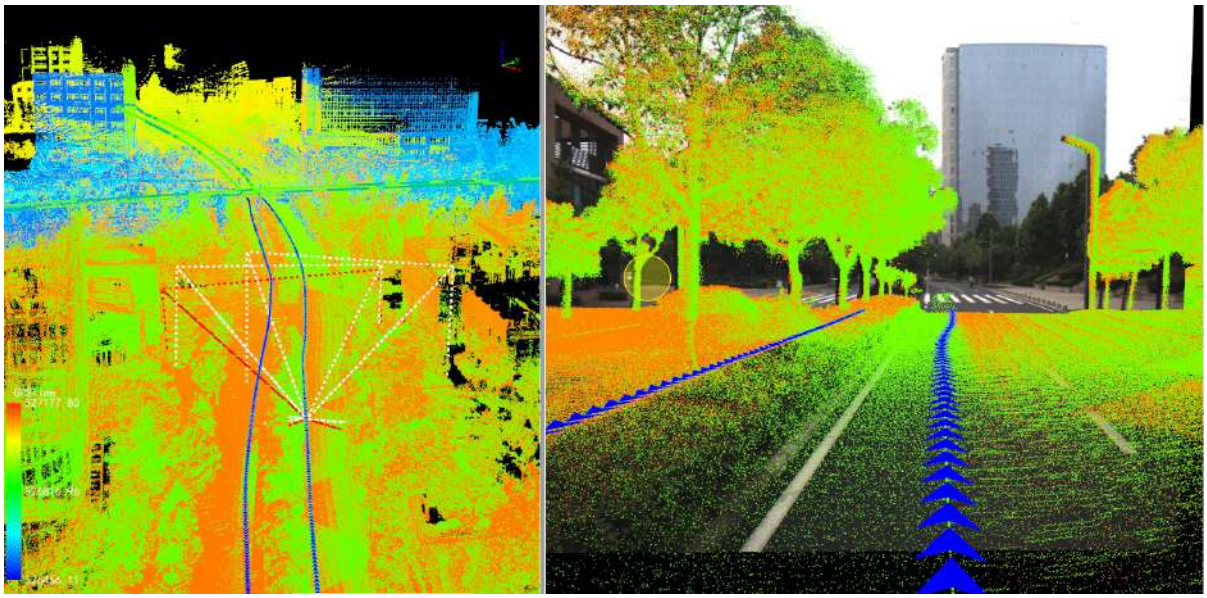
The planar image menu mainly includes display, image measurement, image distortion removal and configuration windows. The planar image menu is displayed only after the planar image data is added when a new project is created. Before measuring, switch the top menu of the software to the planar image page.

Point Cloud and Image Roaming

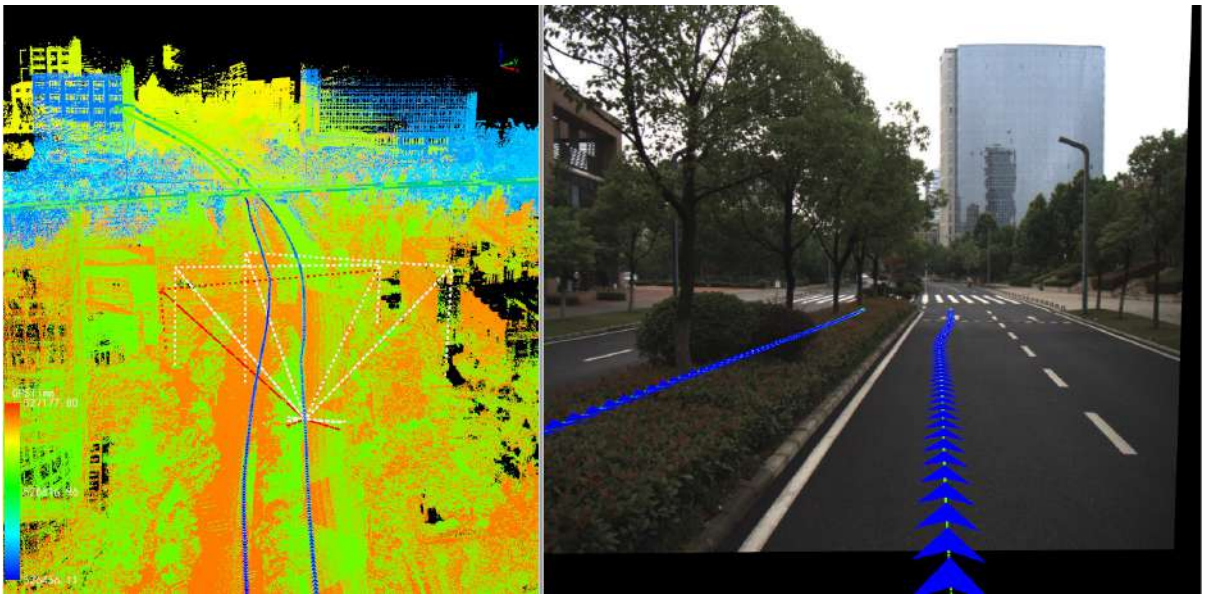
Function Description: The software supports point cloud and image roaming. The original image camera file must be added when creating a new project, and then the menu bar will be activated.

Steps

1.After creating a new project, a 3D window and an image window will be opened, and the point cloud data will be displayed in both the 3D window and the image window, and the image will only be displayed in the image window. The user can choose to show or hide the corresponding data through the check box in front of the data name in the project management window.

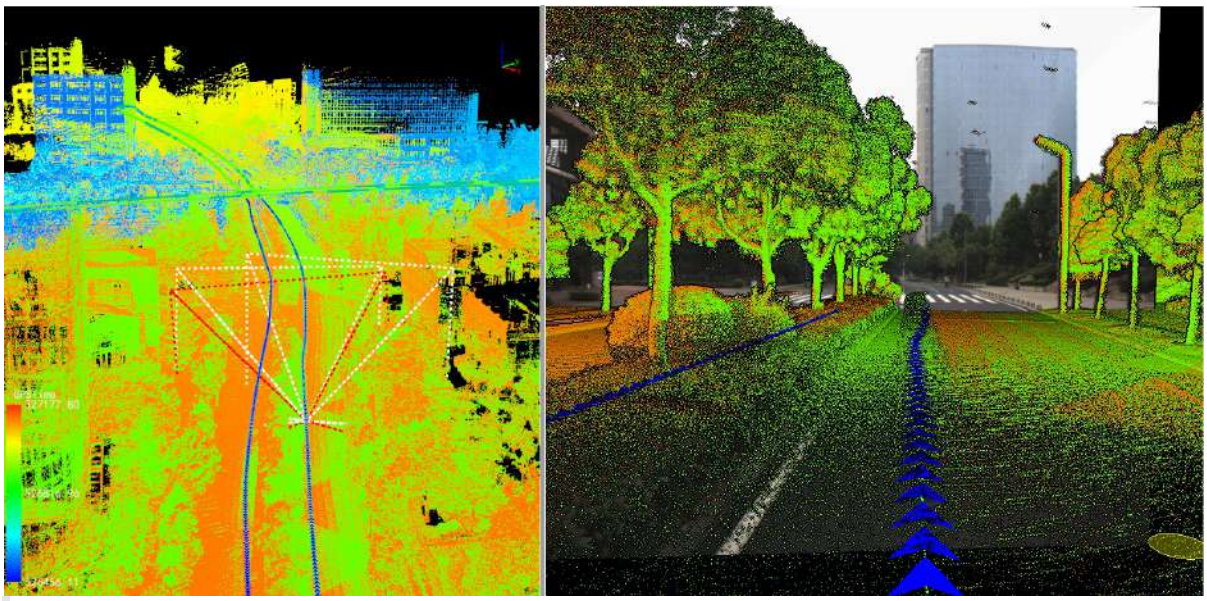


2.In the menu bar uncheck ☐ Show Point Cloud , the user can browse only the image data in the image



3.Click the color bar tool on the left side of the window to switch the display mode of the point cloud, including by elevation, intensity, category, RGB, number of echoes, time, mixed display, combined display, display by selected color and EDL display, EDL can be used in conjunction with other display methods to enhance and

display the contour feature information of point cloud objects. The following picture shows the effect of point cloud displayed by elevation + EDL.

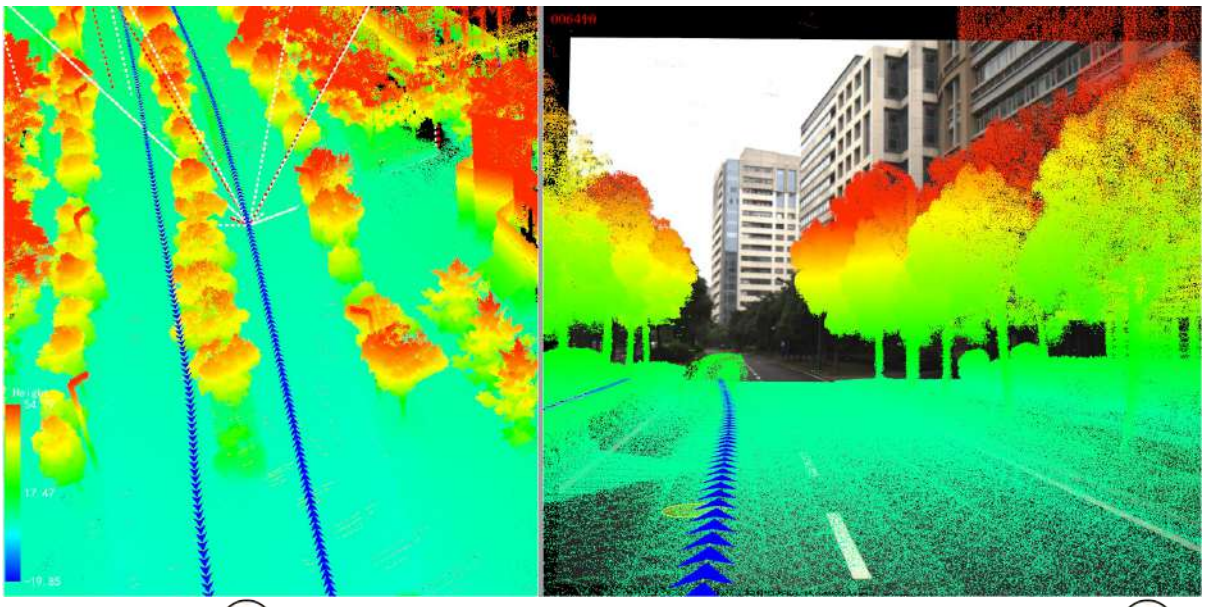


Note:

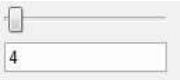
1. Display modes such as elevation, intensity, RGB, etc. on the toolbar work on all point clouds in the project. If you need to give a separate display effect to a point cloud, right-click on the data name and select **View Mode** > By Elevation/Intensity/Category etc.
2. The EDL display effect is related to the window. If you need to display the EDL effect in a window, click the mouse anywhere in the window to activate the window, and then click the EDL button.

4.Camera ID: Pull down to switch to display the corresponding camera image.

5.Click **Select Frame** button, and the user can select the image exposure position on the track in the 3D window or the image window, and jump to the selected image position.

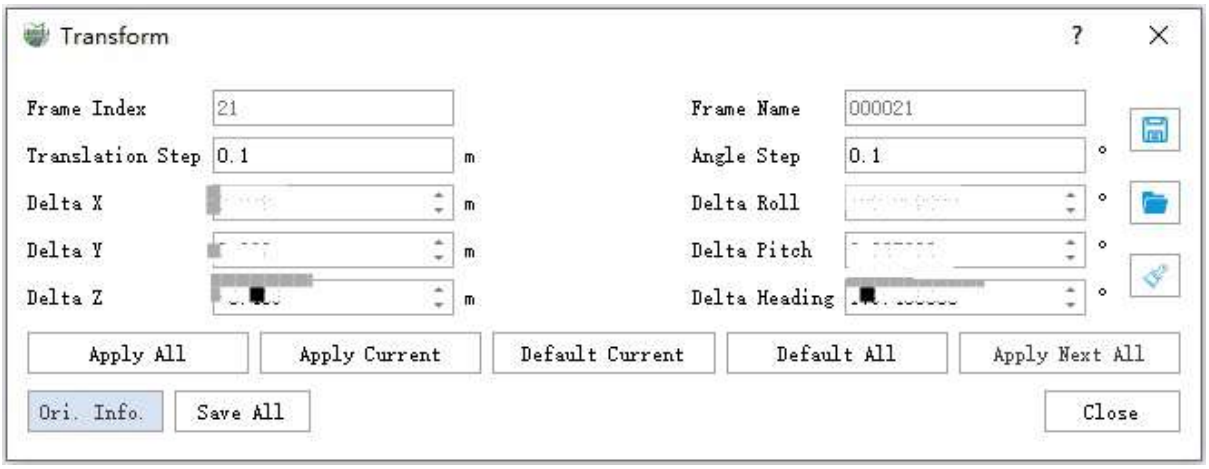


6.Click **Next Frame** button jumps to the next frame of the current image. Click on **Previous Frame** button jumps to the previous frame of the current image.

7.Drag the slider  to switch different image positions or directly input the number of image frames.

8.Click **Auto Roam**  button to roam the point cloud and image from the first-person perspective in the image window. Click this button again to stop the automatic roaming. If the viewing angle of the image window changes, you can click the space bar on the keyboard to restore the default forward direction.

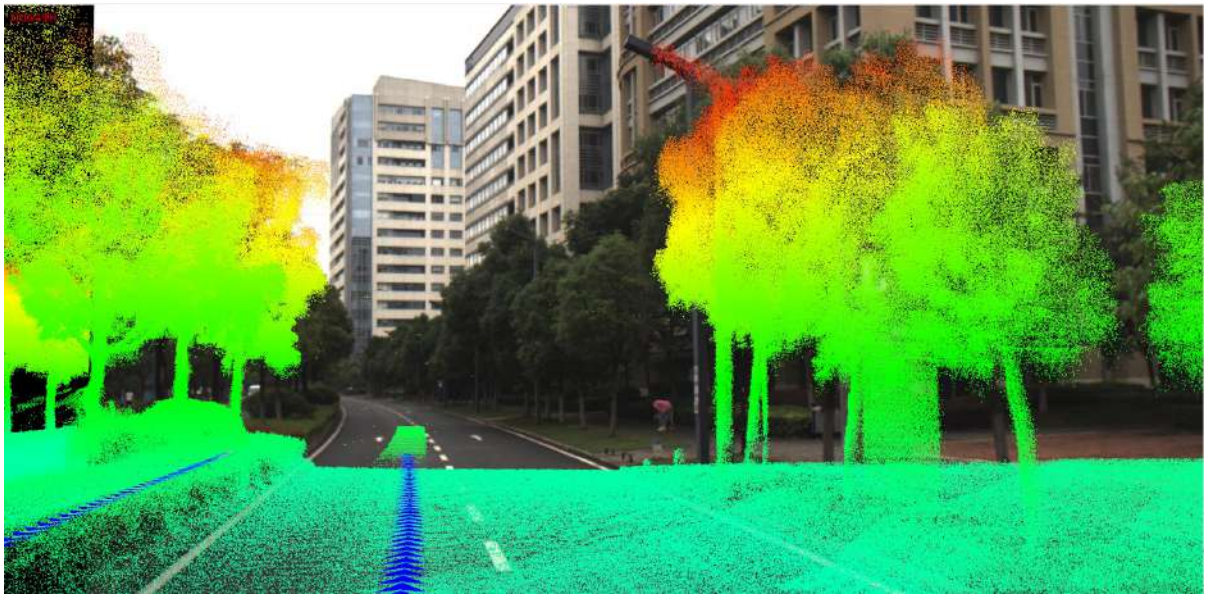
9.Click **Transform**  button, the transformation is mainly to adjust the pose of the image in real time and superimpose the point cloud perfectly.



Illustrate

- **Apply to All**: Apply the image pose set in the current dialog to all frames, including images before and after the current frame.
- **Apply to Current**: Apply the pose set in the current dialog to the image of the current frame.
- **Restore current default**: Restore the pose of the image of the current frame to the original default value.
- **Restore All Defaults**: Restore all frame images to their original default values. Restores the pose of the image in the current frame to the original default value.
- **Apply to All Subsequent**: Applies the pose in the current dialog to all image frames after the current frame.
- **Bearing Information**: Displays the bearing information of the current frame.
- **Save All**: Save the current pose information to the imglist file and update the display.
- **Close**: Close the dialog.
- **Save**: Save the current pose information to a txt file.
- **Open**: Read the pose in the txt file and display it on the dialog.
- **Clear**: Clear the pose data in the dialog.

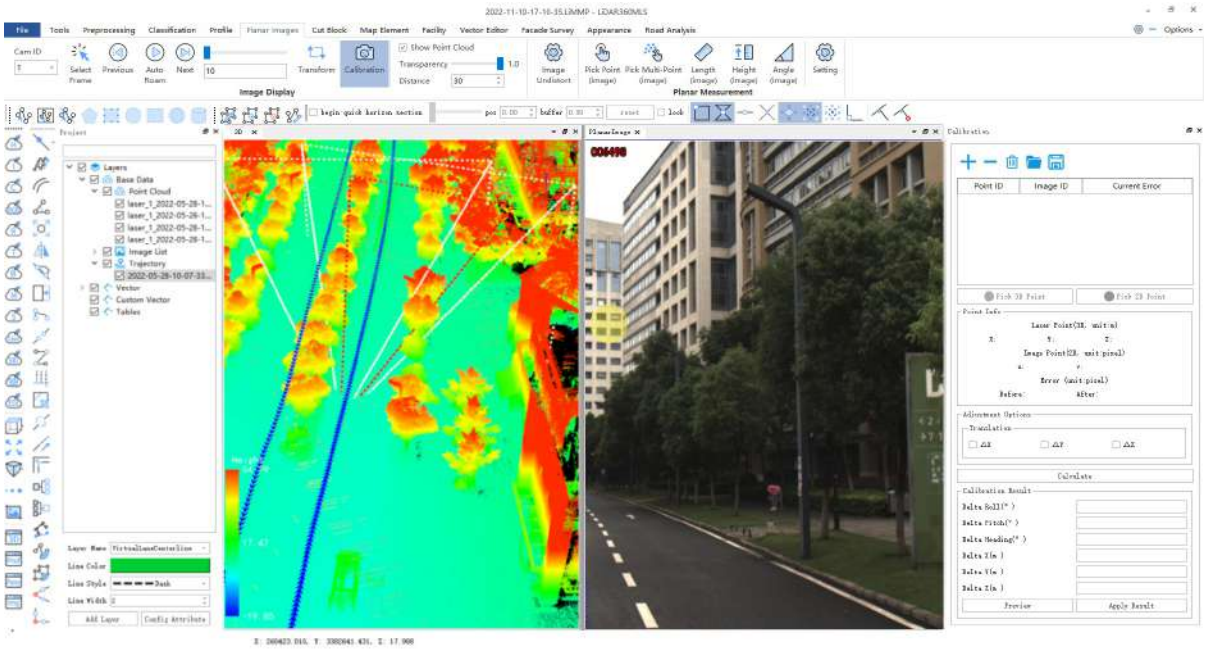
10.Due to the placement error between the image camera and the laser, there is still a certain deviation between the plane image and the point cloud data, which cannot be superimposed perfectly, as shown in the following figure:



LiDAR360 MLS provides an image calibration module to estimate the placement error between the plane camera and the laser, thereby improving data bias and improving accuracy.

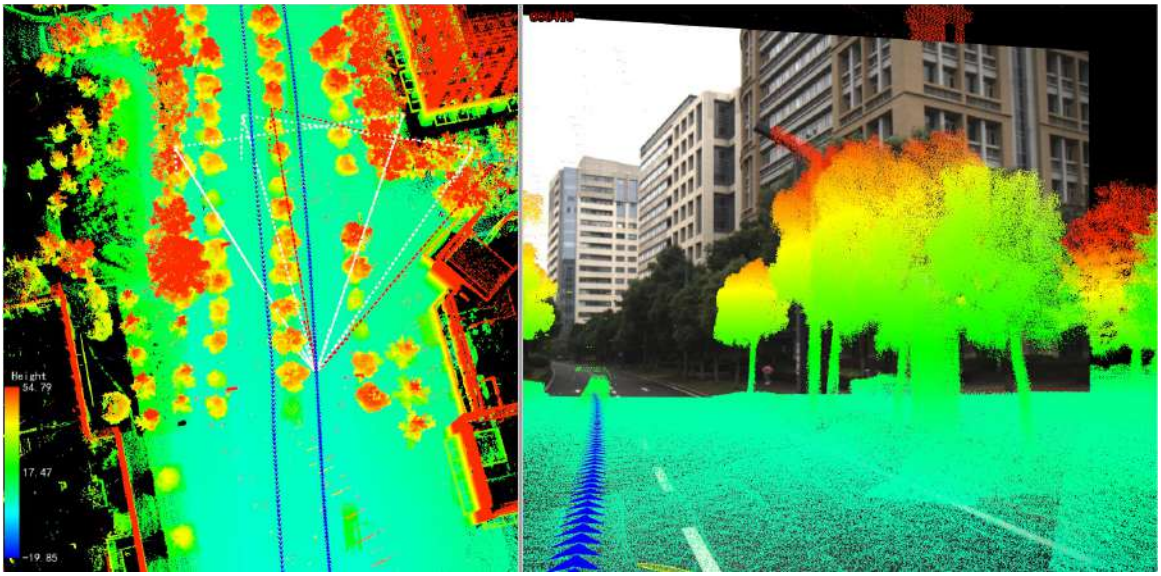
> Note: Set the point cloud display depth by adjusting the menu bar Image->Image Display->Distance, so as to hide the distant point cloud and facilitate the observation of the area of interest. This little trick can also be used in the calibration point selection process below.

1.Switch to the menu page, and click **Calibrate** button, the calibration window will pop up on the right side of the user interface.

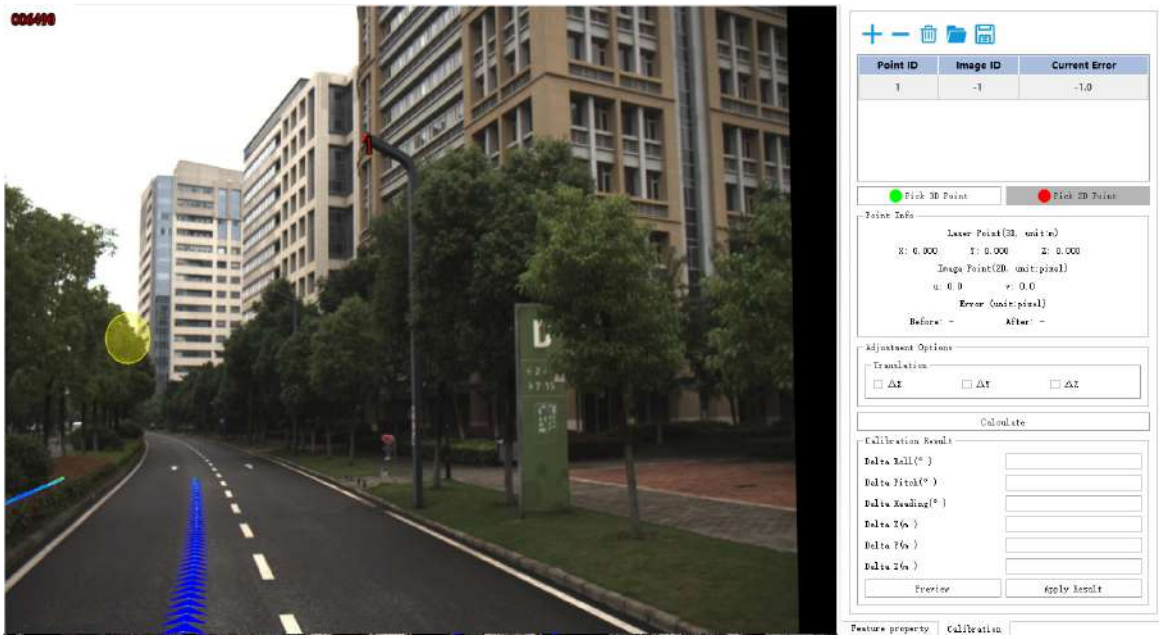


2.Add calibration point. It is recommended to select camera exposure points in multiple directions of the scene, and then select multiple point pairs at each exposure point, at least 4 point pairs need to be added (for example, 4 point pairs are selected at the four exposure moments of the east, south, west and north, that is, a total of 16 point pairs are selected). The specific steps are:

- Click the *Select Frame* button, click the triangle arrow in the 3D view window, and the image window will automatically switch to the corresponding image frame.



- Click the *Add Point* button at the top of the calibration window to add a row to the point list.
- Click the *Pick 2D Point* button to select image points in the image window. The top of the picture is a magnifying glass, which can enlarge the position of the mouse.

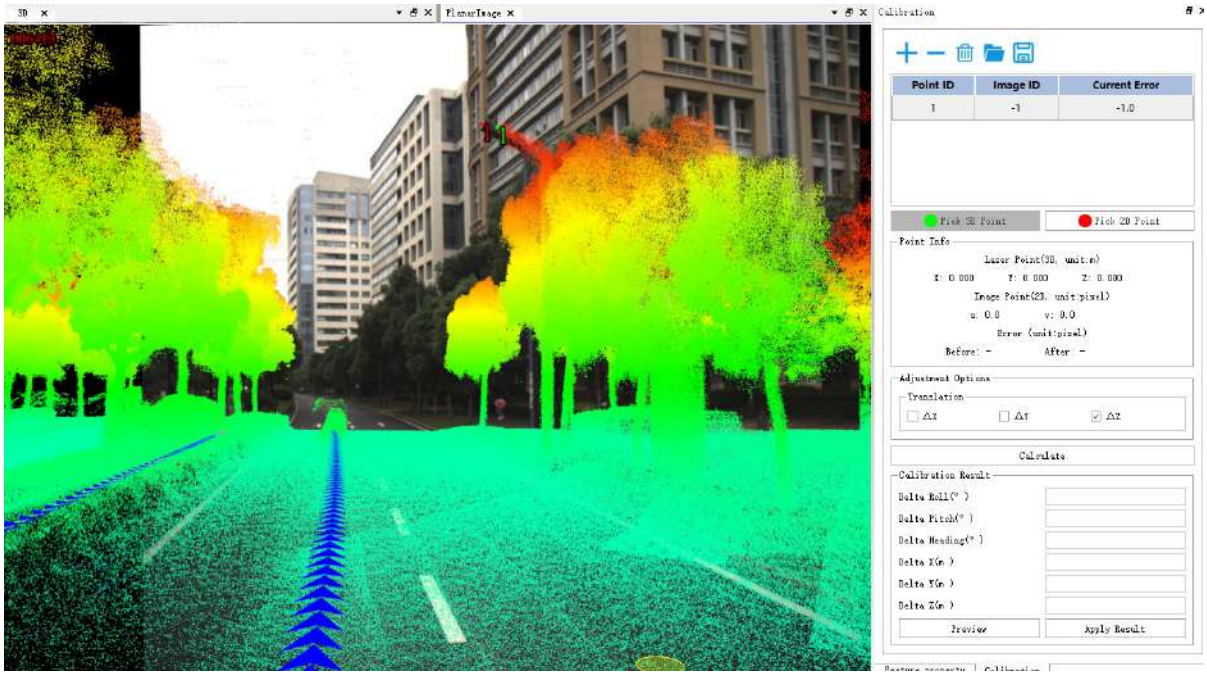


- Click the *Pick 3D Point* button to select the laser point corresponding to the image point in the image window.
- Repeat the above three steps until enough point pairs are selected on the current image frame (four point pairs are recommended).
- Select the corresponding image frames in the other three directions, and repeat the above steps until all points are selected (a total of 13 point pairs are selected in this example).

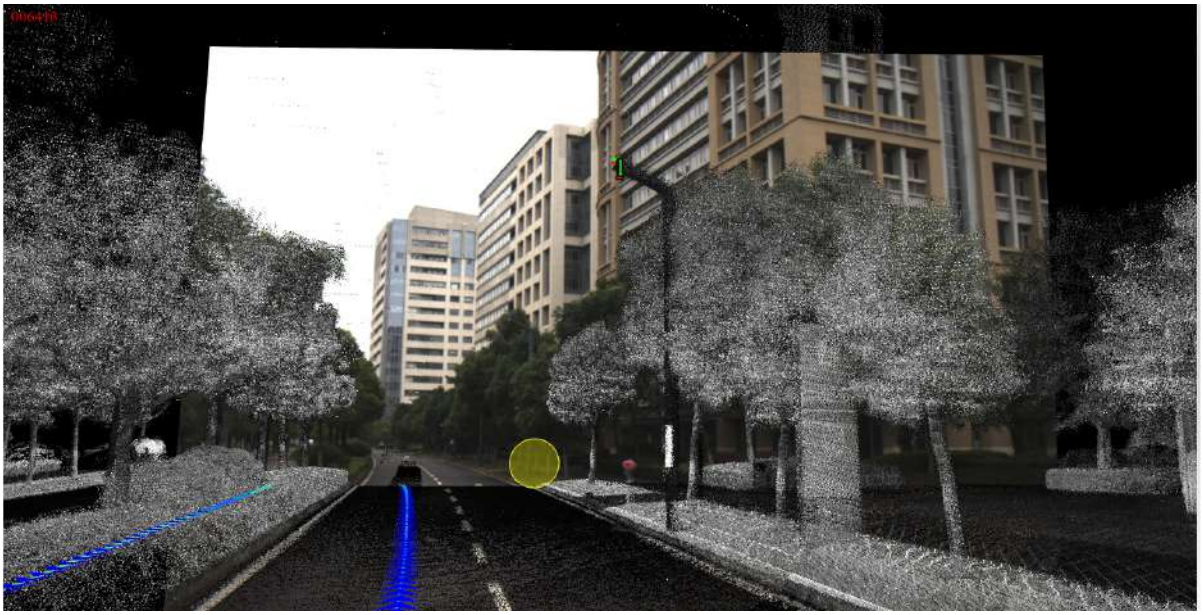
3.Click the *Calculate* button to start the calibration calculation.

4.Click the *Preview* button to preview the calibration result.

Before preview:



After preview:



5.Apply the result. Check whether the preview results meet the requirements, and click the *Apply* button to apply the calculated calibration parameters to the data.

> Note:
>
> 1.There are two ways to control the display/hide of the laser point cloud: click the *Show Point Cloud* button on the menu bar; adjust the transparency scroll bar. By controlling the display/hide of the point cloud at the right time, the efficiency of point selection can be effectively improved.
>
> 2.In general, it is easier to select points by setting the point cloud display mode to display by intensity.

Image Measurement Based on Point Cloud Depth Interpolation

The principle of estimating the location information of measuring points is based on the point cloud data depth value and interpolation algorithm within a certain range around the measured point.

The main functions are listed below:

- Pick Point
- Pick Multi-Point
- Length Measurement
- Height Measurement
- Angle Measurement

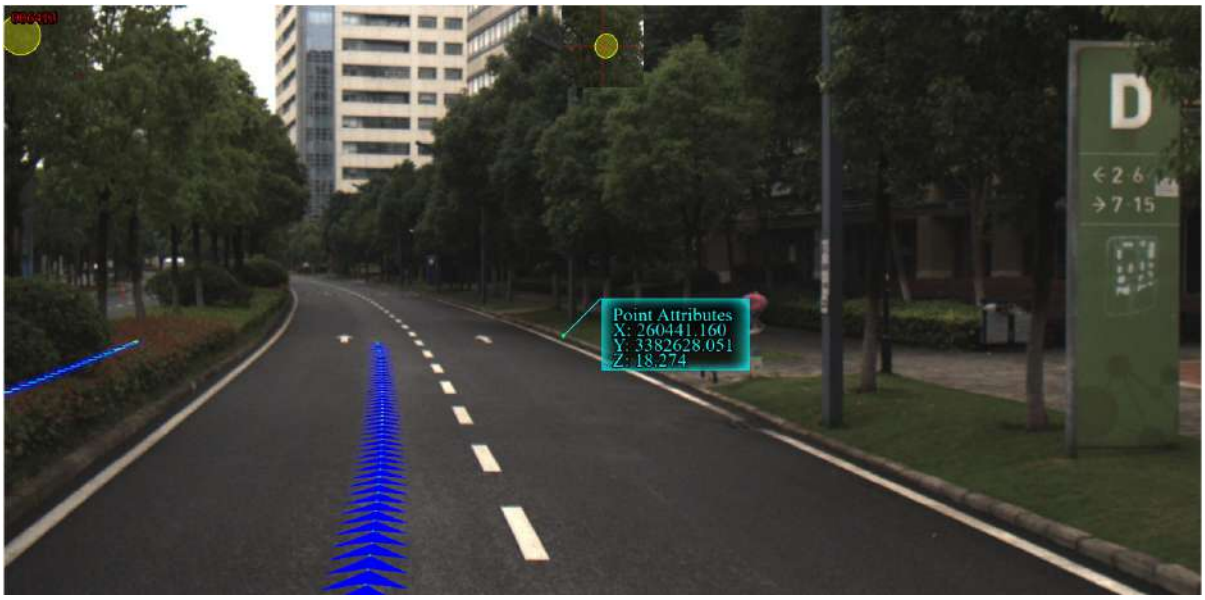
Pick Point (Depth Interpolation)

Function Description： This tool is applicable to panoramic data, where the attributes that can be queried contain position information.

Steps

1.Click **Pick Point(image)**  button via **Planar Measurement** mode.

2.In the image display window, use the left mouse button to click to select the point to be measured, and a pop-up box will appear in the click complete window to display the detailed information of the selected point.



3.Press the right click mouse button to cancel the selected point.

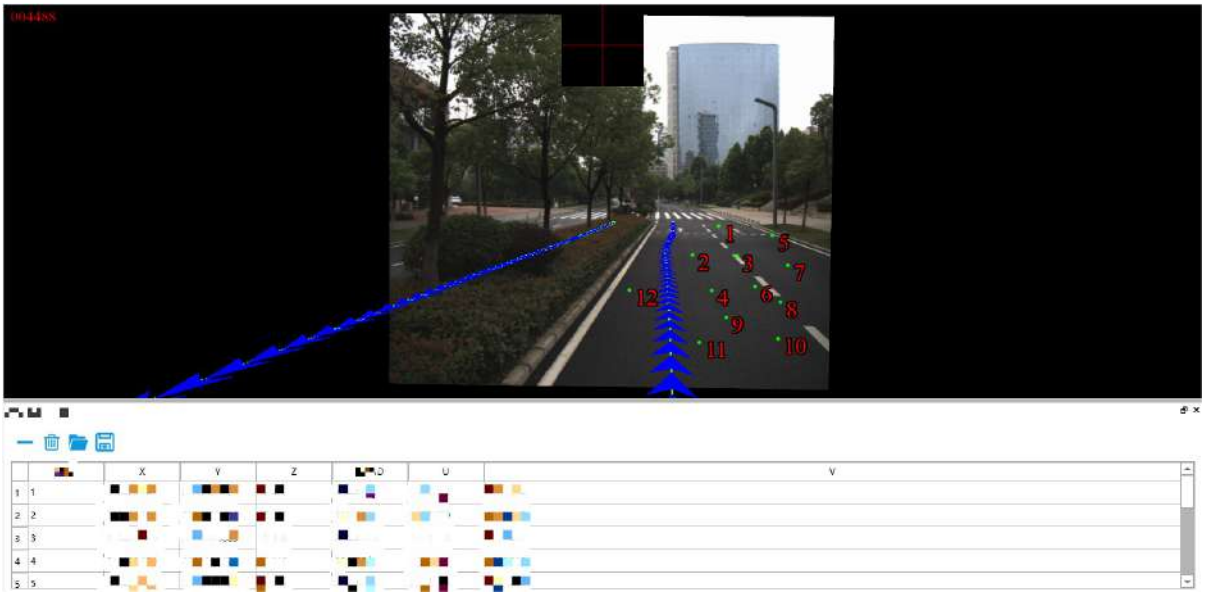
Multi Pick Point (Depth Interpolation)

Function Description： This tool uses the mouse to click interactively to query the attribute information of multiple points in the image, and supports the export of the selected points set in .txt format. The attributes that can be queried for the data include the serial number, position, image ID, and pixel position of the point.

Steps

1.Click the **Pick Multi-Point (image)** button in **Planar Measurement**, and use the left mouse button to click the single points in the scene in turn. The selected points are marked in the scene in the form of point labels, and at the same time the pop-up interface list displays the attribute information of the point set (as shown in the figure).

The point cloud attributes displayed in the list include point number, location, image ID, and pixel location.

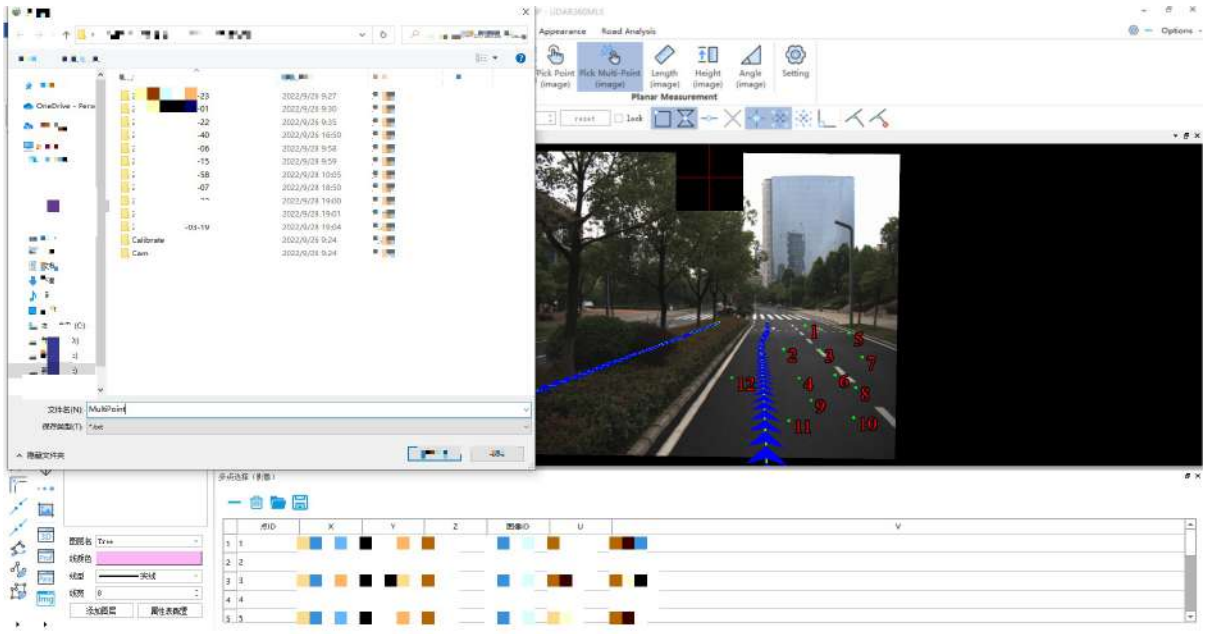


Parameter settings

- **X**: The X coordinate value of the point cloud data.
- **Y**: The Y coordinate value of point cloud data.
- **Z**: The Z coordinate value of the point cloud data.
- **Image ID**: The ID number of the image.
- **U**: The abscissa of the pixel.
- **V**: The vertical coordinate of the pixel.

2.Click any row in the selection list and click the button to delete the point.

3.The selected point set is exported in *.txt* format. Click the button to save the coordinates and other attribute information as a *.txt* file.



4.If a point has been created by user, click  to open the *.txt* file saved before.

5.At the same time, if the user does not need the multiple points currently selected, or if the user has saved multiple points and needs to continue to click multiple points, he can directly click the  button to clear the interface.

> Please note: This function can only be used in the PlanarImage view window.

Length Measurement (Depth Interpolation)

Function Description： This tool is applicable to point cloud data to calculate the distance between two consecutive points.

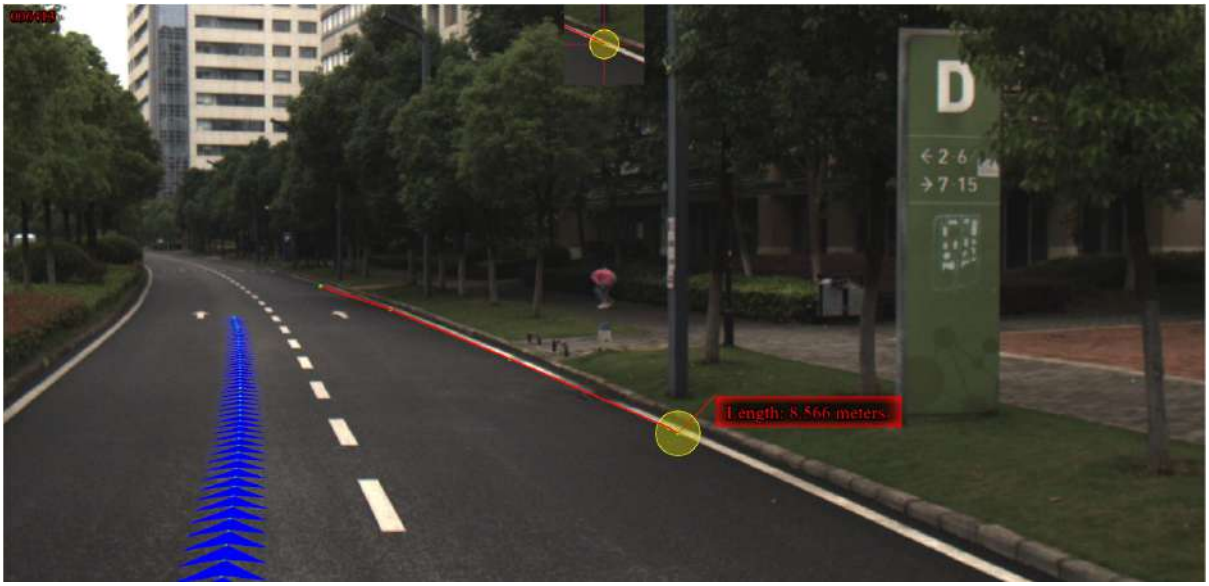
Steps

1.Click **Length(image)**  button via **Planar Measurement** mode.

2.Left click in the panoramic display window (*PlanarImage*) to select the starting point of length measurement.

3.Click the left mouse button to select the next measurement point.

4.Repeat the previous step until all measurement points are selected, and then double-click the left mouse button to complete the measurement, and the interface pop-up box will display the measurement length information.



5.Click the right mouse button before completing the measurement to cancel the last measurement point.

Height Measurement (Depth Interpolation)

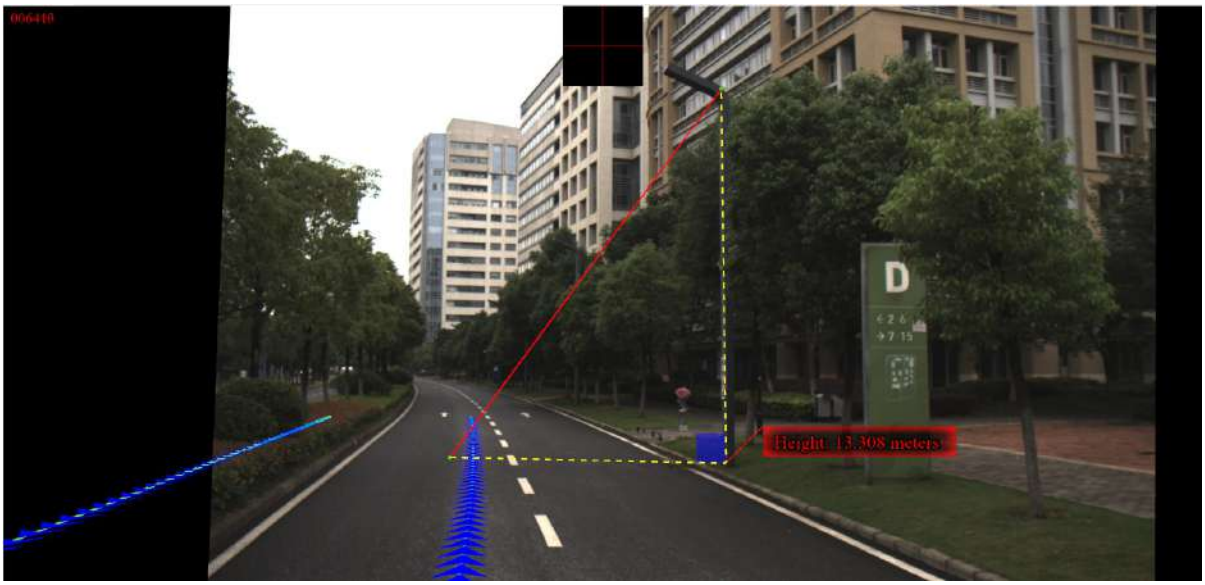
Function Description： This tool is applicable to point cloud data which calculates the relative height difference between two points in PlanarImage window.

Steps

1.Click **Height(image)** button via **Planar Measurement** mode.

2.Select the reference point of height measurement by left-clicking in PlanarImage window.

3.Double-click the left mouse button to select the end point of the height measurement to complete the measurement. The interface pop-up box will display the height information.



4.Right-click to go back to the previous point during the measurement.

Angle Measurement

Function Description： This tool is applicable to point cloud data which calculates the angle of pitch between two points in PlanarImage view.

Steps

1.Click **Angle(image)**  button via **Planar Measurement** mode.

2.Select the reference point of angle measurement by left-clicking in PlanarImage view.

3.Select the measurement point by double-clicking. The pitch angle between the reference point and the measurement point will be rendered in the scene and the measurement result will be displayed in a label as follows.

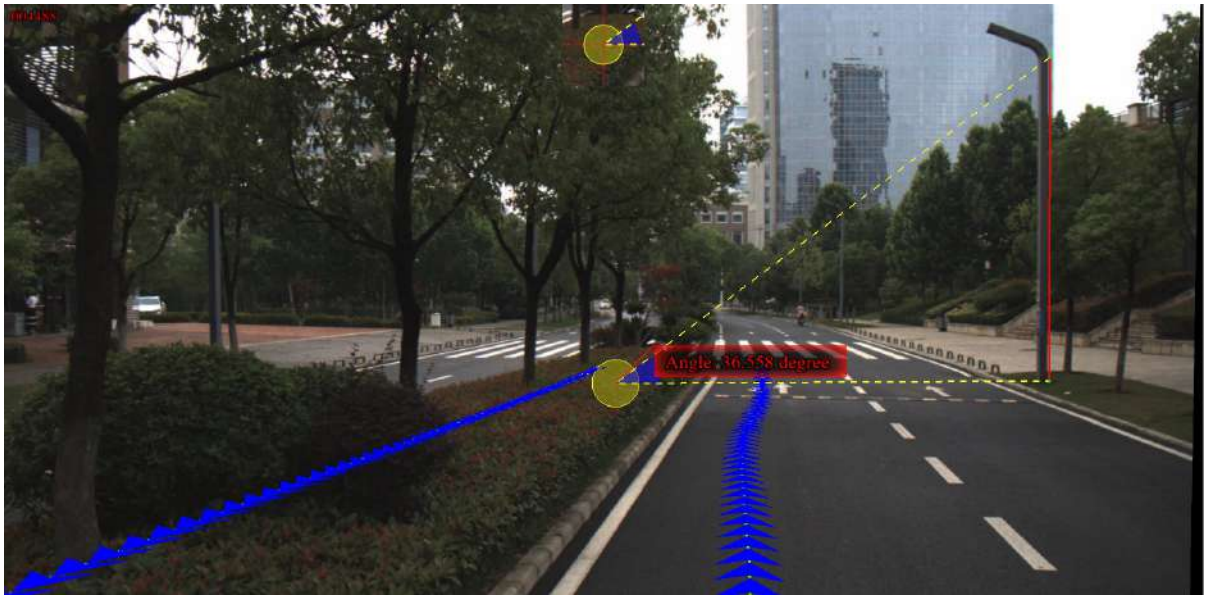


Image Undistorting

Function Description: The original image is distorted, remove the distortion of the original image.
Illustrate

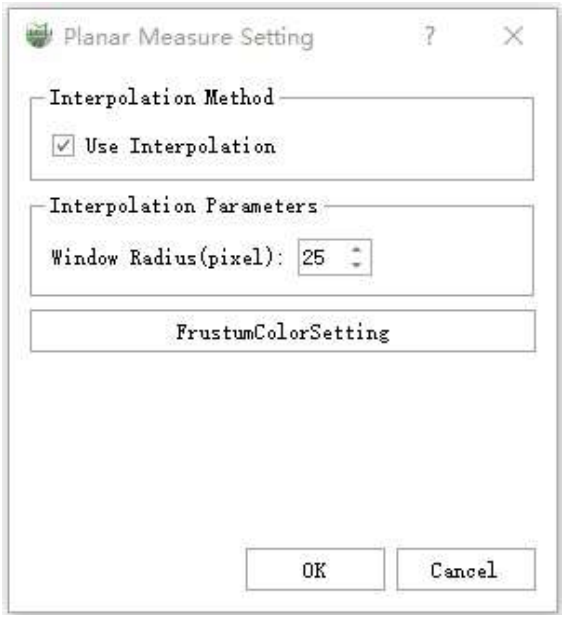
1.Image distortion is due to the lens manufacturing precision, where the deviation of the assembly process will introduce distortion, resulting in distortion of the original image. The distortion of the lens is divided into two categories: radial distortion and tangential distortion.

2.Run the image de-distortion function to restore the image to before distortion.

Dialog Setting

Function Description：Dialog Setting is used to set parameters for running the function of panorama measurements.

Click the **setting**  button.



Parameters Introduction

- **Positioning method**: This function is set by default. It is available to choose whether select the complex interpolation algorithm or not when using the function of Panorama Measurements based on Panorama Measurements based on Point Cloud Depth Interpolation. If unselect this function, nearest neighbor algorithm is used to calculate the point location information.
- **Interpolation Parameter**: It is the windows radius to be used when running interpolation algorithm. For example,when the windows radius is N, all the depth value around the measured points in the square area whose length value is 2N + 1 (the unit is pixel) will be read. If the point density is not high and measurements can not perform with default windows radius, users can increase the value of windows radius.
- **FrustumColorSettings**: Visual cone color settings.

Preprocessing

After completing the high-precision point cloud solution, the trajectory and point cloud of the result data can be further optimized through methods such as correction, splicing, and quality inspection.

Trajectory

Segmentation

Calibration

Boresight

GCP Adjustment

Strip Adjustment

Trajectory

- Trajectory Segmentation
- Segment Table
- Trajectory Graph
- Trajectory Quality Detection
- Trajectory Jump Repair

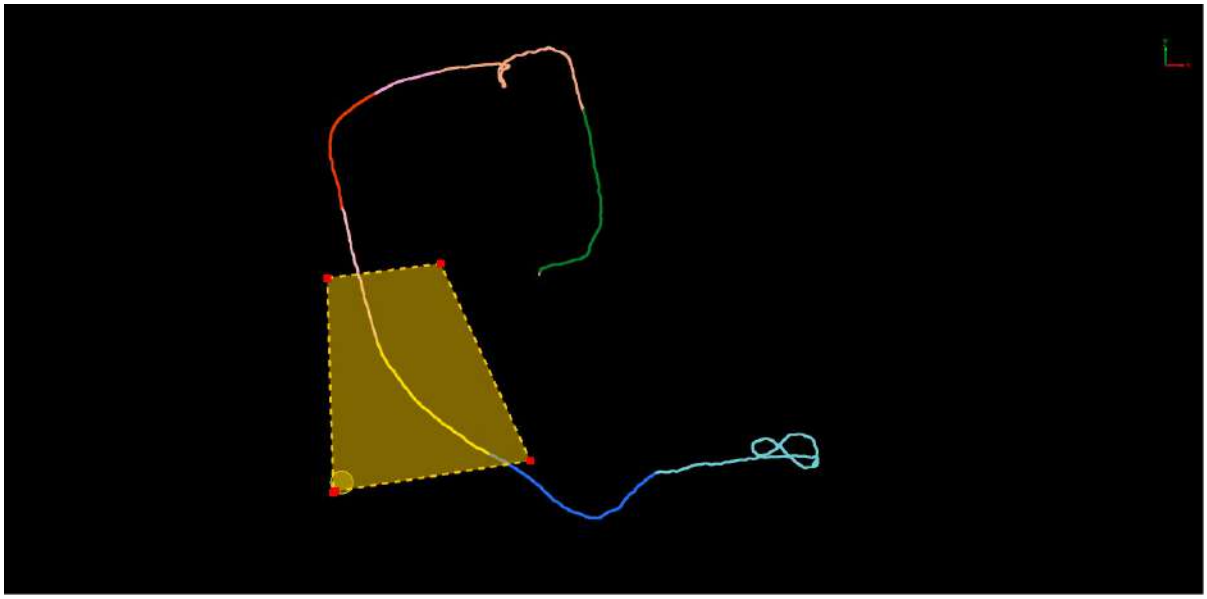
Trajectory Segmentation

Split by Polygon

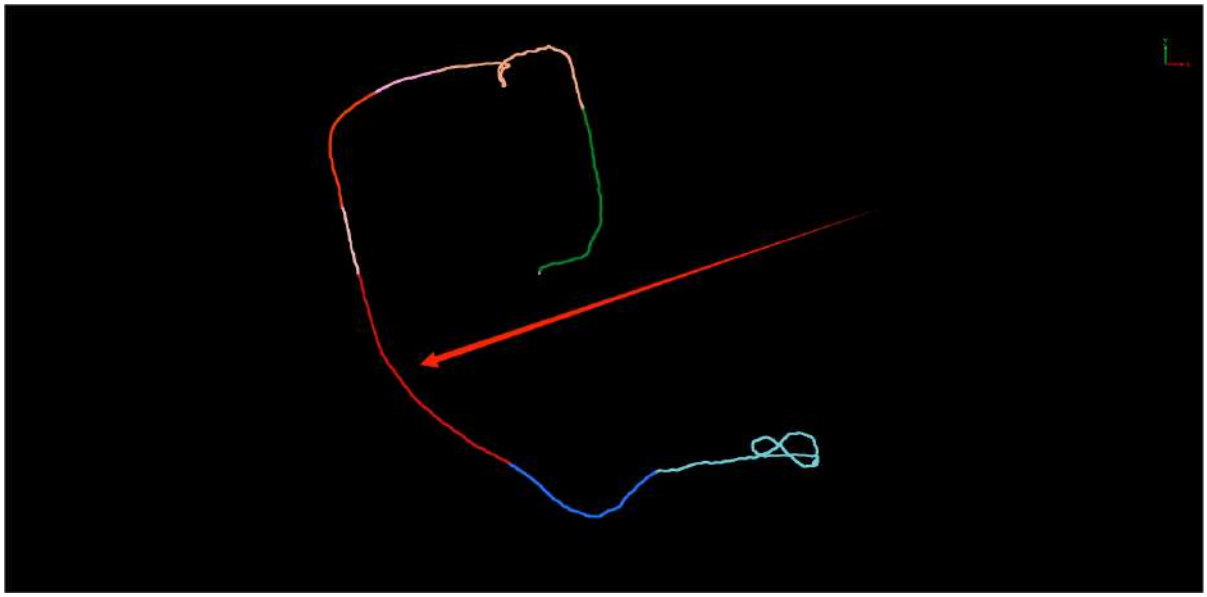
Steps

1.Click **Draw Polygon**  button.

2.Click the mouse around the area of interest in the 3D display window to select multiple points and draw polygons.



3.Click **Split Trajectory**  button to divide the track segments in the polygon area.

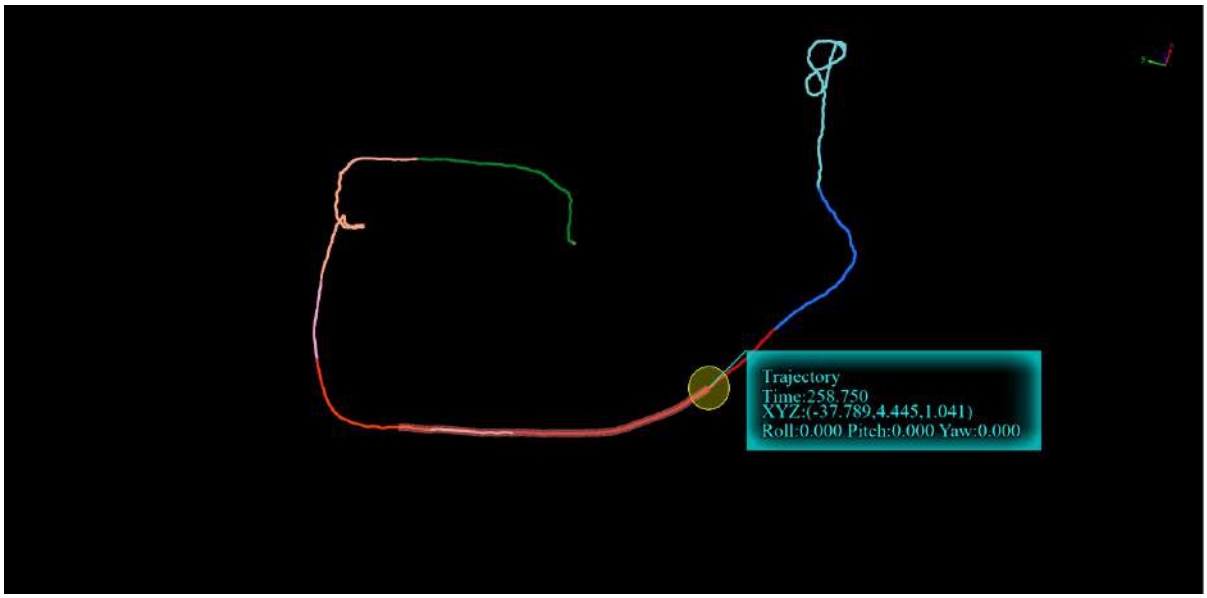


Select on Trajectory

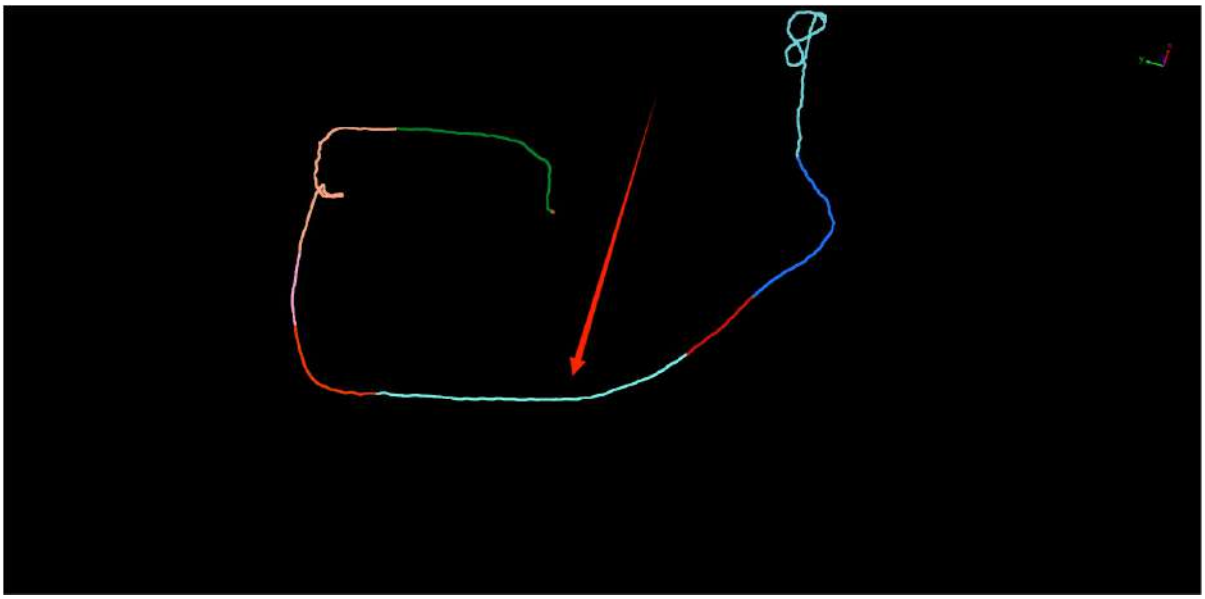
Steps

1.Click **Select on Trajectory** button.

2.Click on the resulting track in the 3D display window to select the starting point of the segment.



3.Select the end point of the segment along the track, and the track segment between the starting point and the end point will be divided.



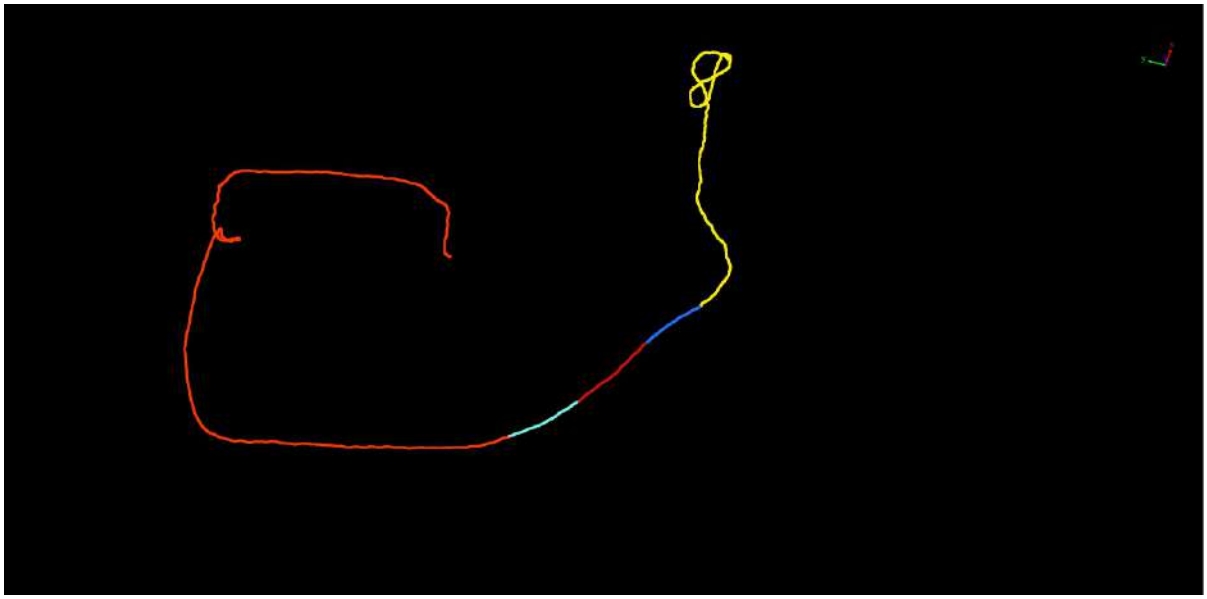
Invert Select on Trajectory

Steps

1.Click **Invert Select on Trajectory** button.

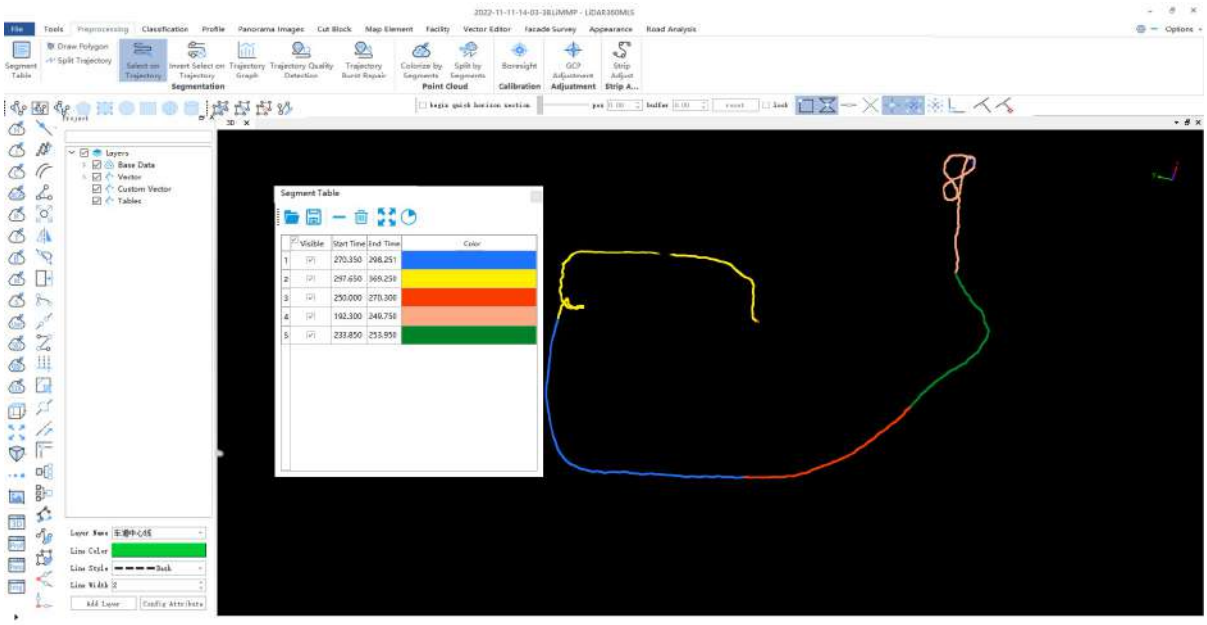
2.Click on the resulting track in the 3D display window to select the starting point of the segment.

3.Selecting the end point of the segment along the track, and the segment in the opposite direction of the track between the starting point and the end point will be divided.



Segment Table

Click **Segment Table** button, the segmentation result table will pop up in the lower right corner of the software interface, the table records the detailed information of the track segments divided using the above two methods (including the visibility of the track segments, the start and end times, and the color). Click the corresponding buttons on the toolbar at the top of the table to realize functions such as opening, saving, deleting, emptying, and hiding un-segmented areas.



Save File

Click the **Save File** button, and a pop-up window saves the track segment to the specified location on the hard disk, and the saves format is *.xml*.

Open File

Click the **Open File** button, and a pop-up window reads the saved track segment from the specified location on the hard disk, and the reading format is *.xml*.

Delete Segment

Click the **Delete Segment** button, to delete the selected track segment in the table.

Delete All Segments

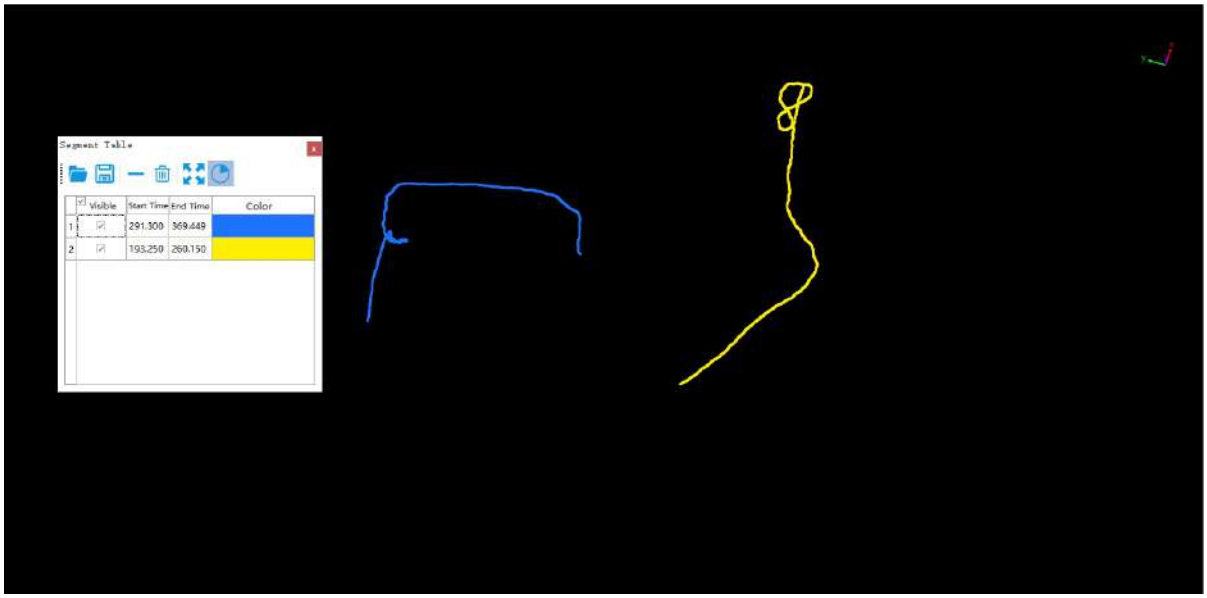
Click the **Delete All Segments** button, to delete all track segments in the table.

Segment Full Extent

First select the row where the track segment is located from the table, and then click the **Segment Full Extent** button, the 3D display window will focus on the currently selected track segment.

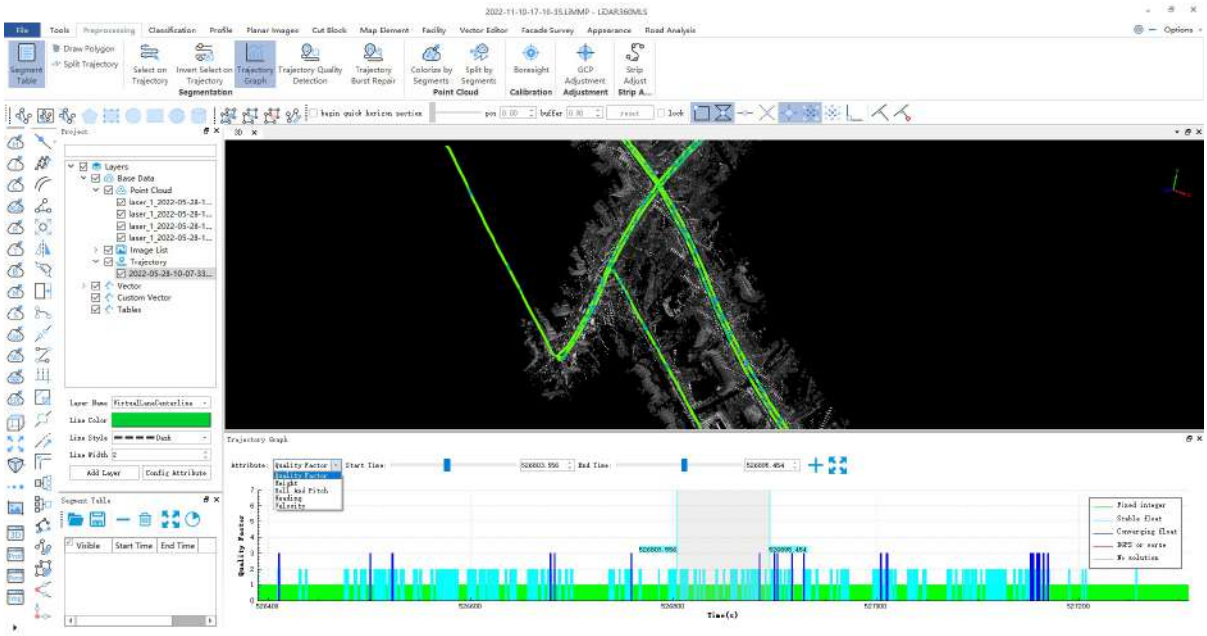
Hide Remaining Part

Click the **Hide Remaining Part** button, the 3D display window will hide the undivided area, that is, only the track segments in the list and their associated point clouds will be displayed.



Trajectory Graph

1.Click **Trajectory Graph**  button, a chart will pop up at the bottom of the user interface to display the track map, the track map shows the track segmentation information at each time. Through the track map menu bar, you can adjust the display properties, add track segments, and change the track display range.



Adjust Display Properties

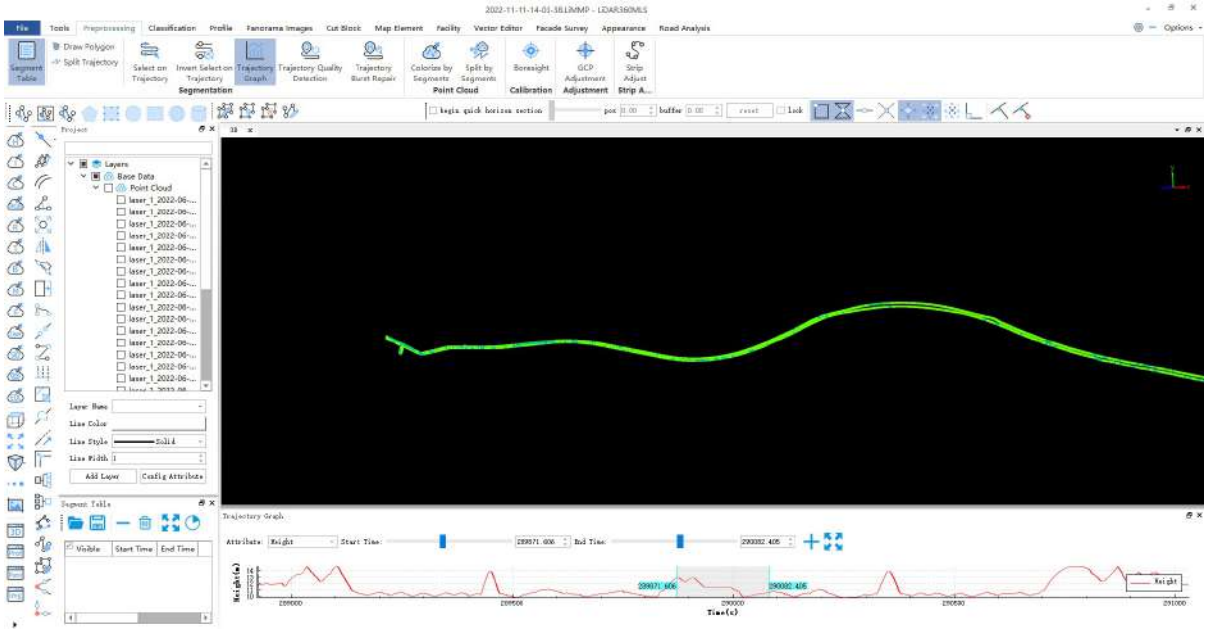
Click the selection box after the **Attribute** tab to adjust the information displayed in the segmented status chart. You can choose to display by Height, display by Quality Factor, display by Roll and Pitch, display by Heading, and display by Velocity.

Add Segment

First click the scroll bar after the **Start Time** to select the start time of the track segment, then click the scroll bar after the **End Time** to select the end time of the track segment, and finally click *Add Segment*  button completes the addition of the track segment.

Select the start and end time：

Add track segment：



Change the Display Range of the Trajectory Graph

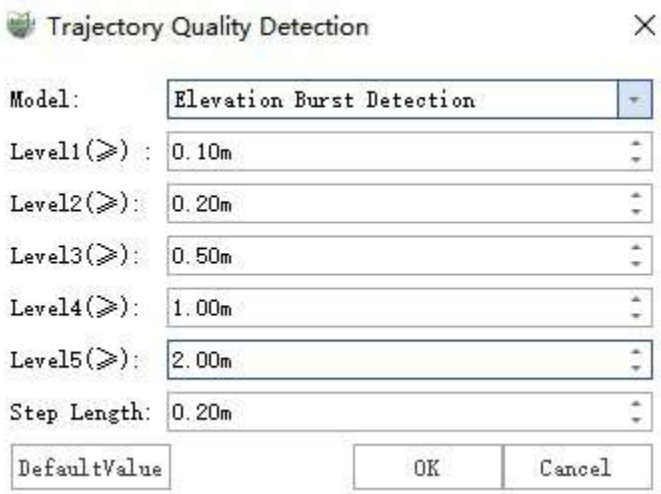
The trajectory graph displays the track segmentation information at all times by default. Use the mouse wheel to slide up and down on the track map to zoom in and out of the display range of the track map, click **Full Extent** button to restore the default display.

Trajectory Quality Detection

Function Description: Trajectory quality detection is divided into elevation transition detection and re-entry area relative elevation difference detection, which will be described in detail below.

Steps

1.Click **Trajectory Quality Detection** button, the segmentation result table will pop up in the lower right corner of the software interface, the table records the detailed information of the track segments divided using the above two methods (including the visibility of the track segments, the start and end times, and the color).Click the corresponding buttons on the toolbar at the top of the table to realize functions such as opening, saving, deleting, emptying, and hiding un-segmented areas.



2.**Elevation Transition Detection**: Used to detect elevation mutation points in the trajectory and display them in a hierarchical manner.

- **Level parameters**: Set five levels, and the mutation segments of different levels are displayed in different colors.
- **Step Length**: The planar distance detected between two points, where the smaller the distance, the more sensitive it is to the mutation value.

3.**Relative Height Difference Detection**: Used for two round-trip trajectories in the re-entry area to detect points where the relative elevation difference is greater than the threshold value.

- **Level parameters**: Set five levels, and the mutation segments of different levels are displayed in different colors.

Trajectory Jump Repair

Function Description: Used to repair the track segment with elevation transition in the track and needs to be detected for elevation transition first, which will be described in detail below.

1.Click **Trajectory Jump Repair** button, a pop-up box will appear, as shown in the figure：

2.**Level Threshold**: Set the transition level that needs to be repaired, and transition trajectory points greater than this threshold will be repaired.

3.**Recalculate Cloud**: If checked, the point cloud corresponding to the transition point will be re-solved after the trajectory is repaired, otherwise only the trajectory will be repaired.

Point Cloud

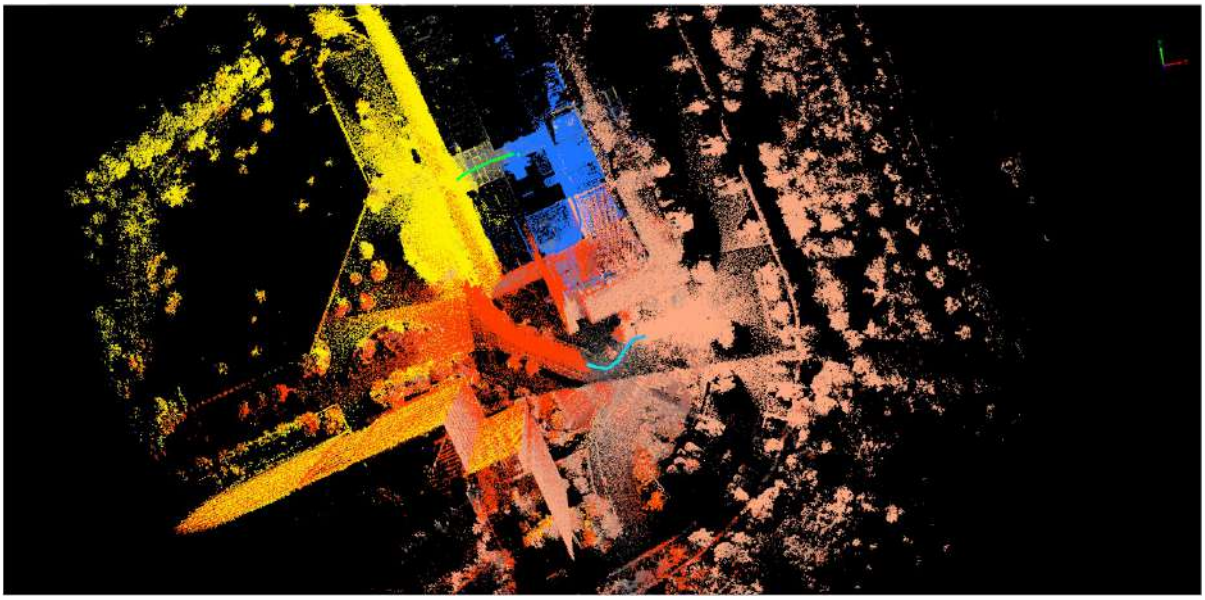
- Colorize by Segments
- Split by Segments

Colorize by Segments

Function Description: Based on the segmented track fragment, the corresponding point cloud is displayed.

Steps

1.Click **Colorize by Segments**  button, the point clouds in the display window will be colored according to the segment of the track to which they belong, that is, the point clouds are given a specific color according to the segment to which they belong, and the point clouds without segments are displayed in gray.

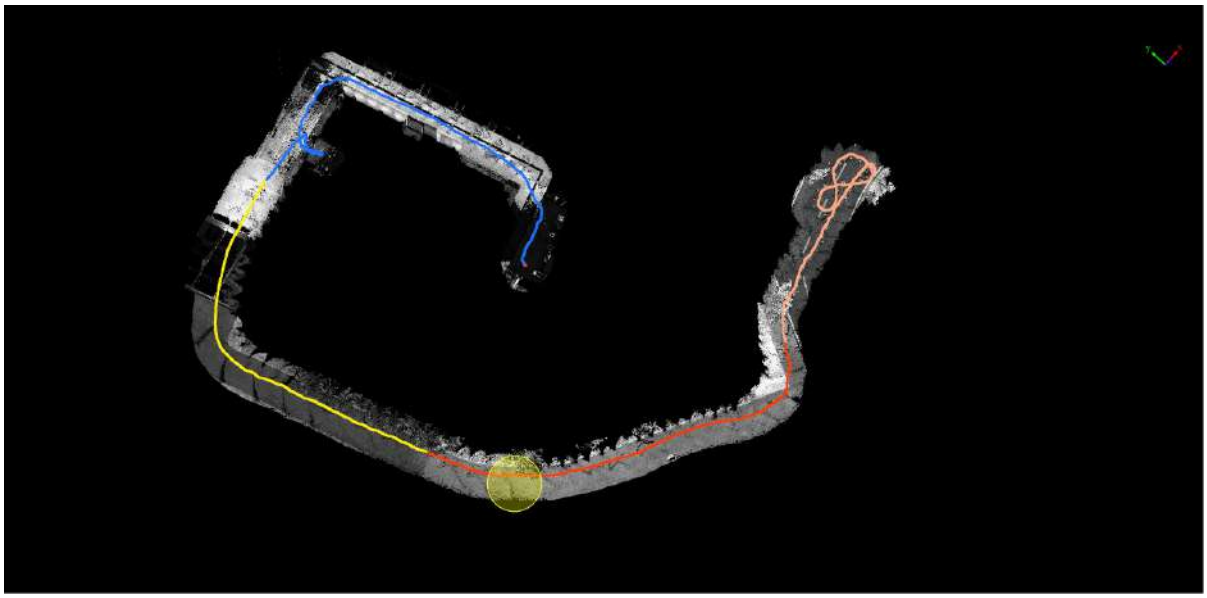


Split by Segments

Function Description: Based on the split track fragment, wherein the point cloud is re-divided.

Steps

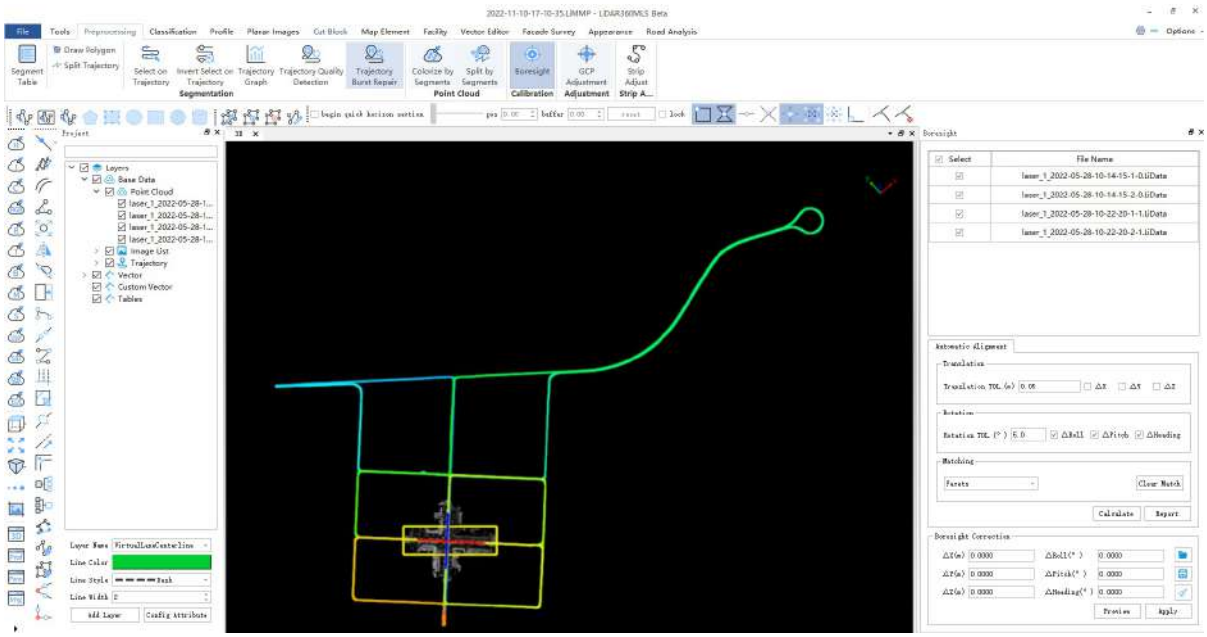
1.Click **Split by Segments**  button, click the **OK** button in the pop-up window, and the point cloud will be divided into multiple pieces according to the track segment to which it belongs. By checking **Cut by Trajectories'Buffer** and editing the **Buffer** value, you can limit the range of segmentation.



Segmented by track fragment

Boresight Calibration

1.Click **Boresight** button, the placement interface will pop up on the right side of the user interface. This function is mainly used to correct the placement error of the laser, and the point cloud data will be transformed according to the corrected value. To obtain high-quality point cloud data, the selection of laser placement error correction values is particularly important. This software provides two methods: automatic placement error correction and manual placement error correction.

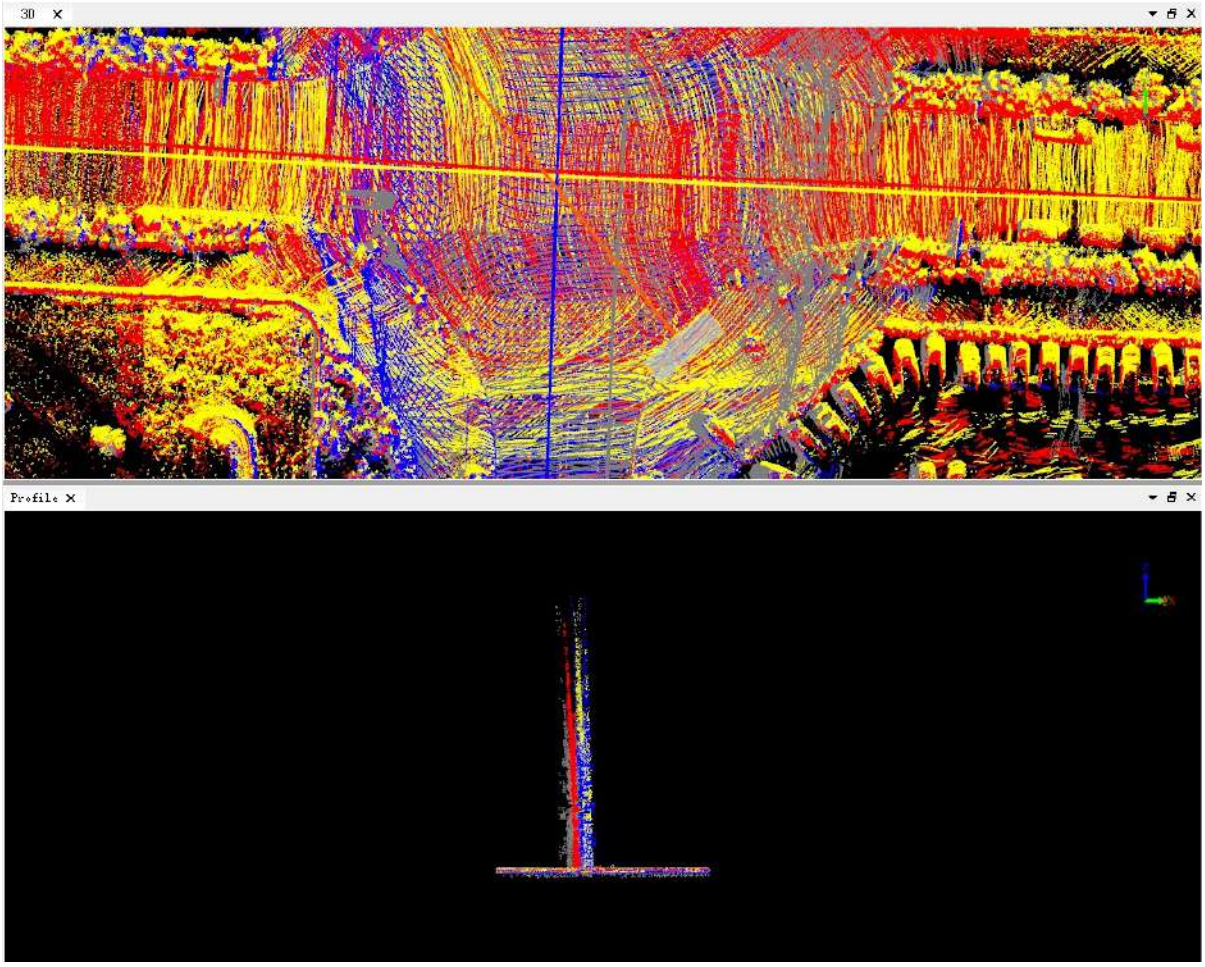


2.Automatic placement error correction：

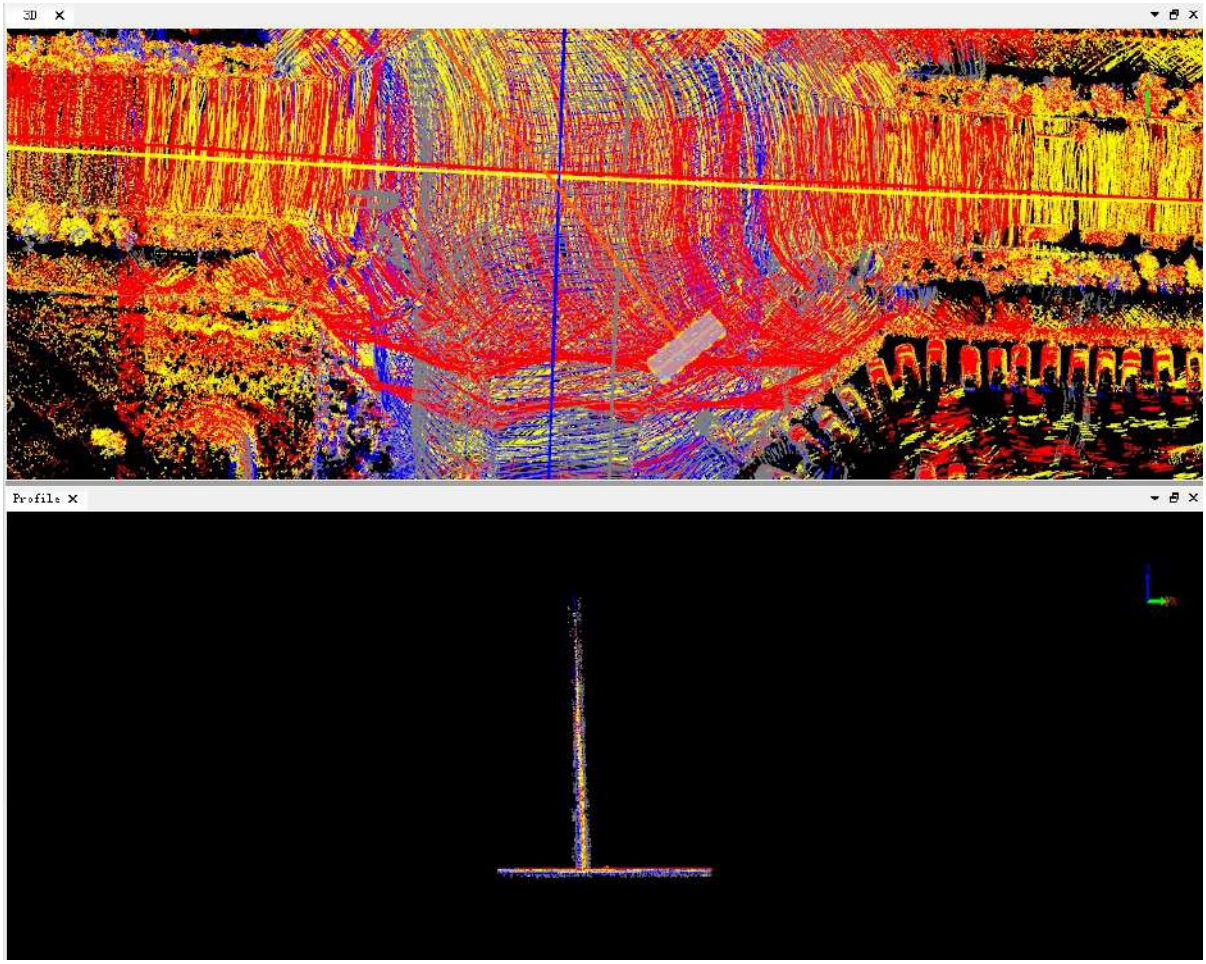
Check the final placement error that needs to be calculated in the placement interface, enter the translation tolerance and rotation tolerance, and Boresight After clicking calculate, the software will automatically calculate the placement error and display the results in the placement error correction parameter box. After the placement error is calculated in the automatic placement error correction mode, the software will count the alignment quality and generate an HTML report. Click on the report to open the generated HTML report. For the relevant principles of automatic placement error correction, please check **Automatic Calculation of Placement Error**.

3.Manual Calculation of Placement Error：

Users can also enter the placement error correction parameters by themselves. For manual calculation of placement error, please refer to **Manual Calculation of Placement Error**. Click the "Preview" button, the error correction value will temporarily act on the selected point cloud, but the point cloud file on the disk has not been modified. If the preview effect is better, you can click the “Apply” button to fix the error to the point cloud file that is actually applied to the disk. Whether it is an automatic method or a manual method, in the process of correction, you can use the profile tool to view the correction results, as shown in the figure.

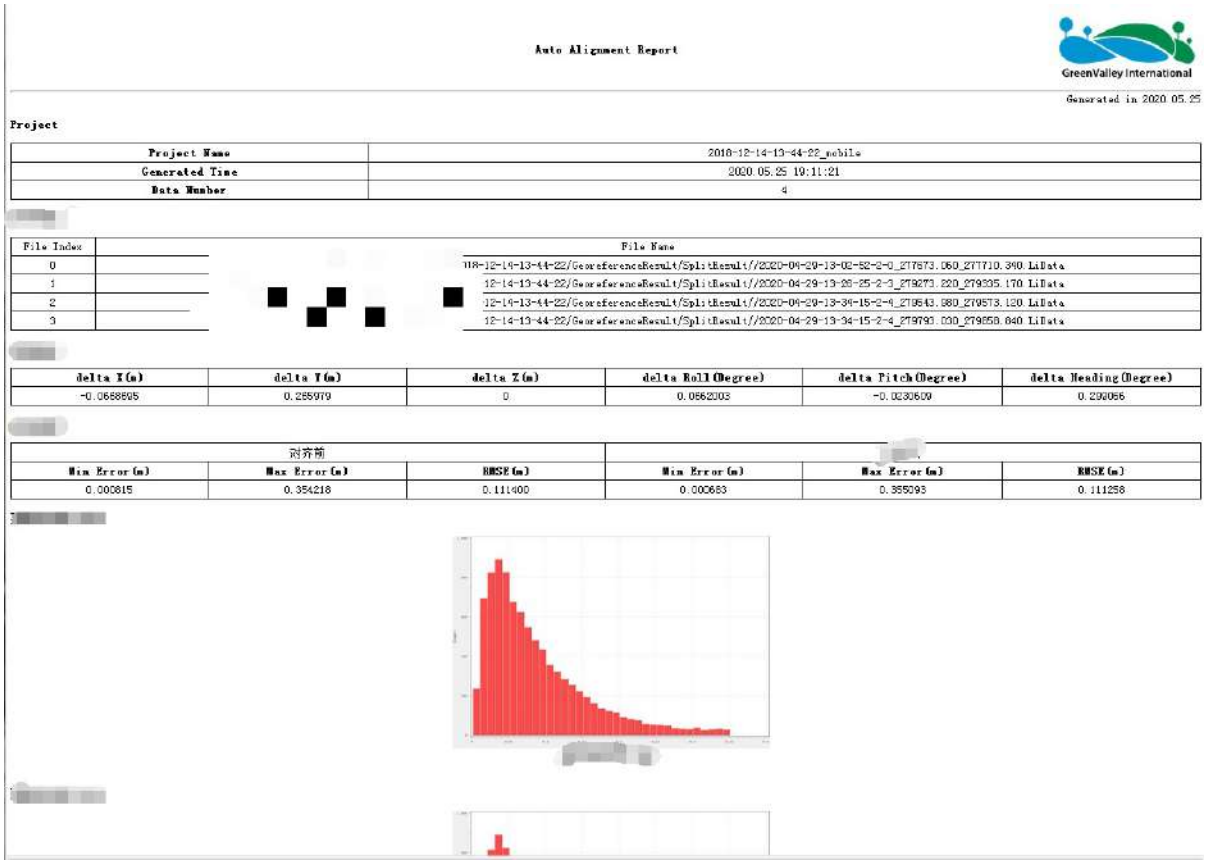


Before correction



After correction

4.Click the **Report** button to view the placement error correction quality report, as shown in the figure.



Auto Alignment Report

GreenValley International

Generated in 2020.05.25

Project

Project Name	2018-12-14-13-44-22_mobile
Generated Time	2020.05.25 19:11:21
Data Number	4

File Index	File Name
0	018-12-14-13-44-22/GeoreferenceResult/SplitResult//2020-04-29-13-02-52-2-0_277673.060_277710.340.LiData
1	12-14-13-44-22/GeoreferenceResult/SplitResult//2020-04-29-13-25-2-3_279273.220_279305.170.LiData
2	12-14-13-44-22/GeoreferenceResult/SplitResult//2020-04-29-13-34-15-2-4_279543.080_279573.120.LiData
3	12-14-13-44-22/GeoreferenceResult/SplitResult//2020-04-29-13-34-15-2-4_279793.030_279858.840.LiData

delta X(m)	delta Y(m)	delta Z(m)	delta Roll(Degree)	delta Pitch(Degree)	delta Heading(Degree)
-0.0688605	0.255979	0	0.0662003	-0.0230609	0.290066

对齐前					
Min Error(m)	Max Error(m)	RMSE(m)	Min Error(m)	Max Error(m)	RMSE(m)
0.000615	0.354218	0.111400	0.000683	0.355093	0.111258

5.Click the **Clear Match** button to delete the matching information, and calculate it again after changing the relevant parameters.

You can load the error correction file in turn through the three buttons on the right, save the current parameters as an error correction file, or clear the currently set correction value.

Parameter setting

- **Placement Error Correction**
  - **ΔX/ΔY/ΔZ**: Set the offset error correction parameter.
  - **ΔRoll/ΔPitch/ΔHeading**: Set the attitude error correction parameters (roll, pitch, yaw).
- **Automatic Alignment**: The program automatically matches the relevant points of the segmented point cloud, and calculates the optimal placement error correction value through the adjustment.
  - **Translation Tolerance**: The maximum translation adjustment range corresponds to ΔX, ΔY, and ΔZ, and the default setting of the program is 0.05 meters.
  - **Rotation Tolerance**: The maximum rotation adjustment range corresponds to ΔRoll, ΔPitch, and ΔHeading. The default setting of the program is 5°.
  - **Optional Items**: The user can freely choose whether 3 translation quantities and 3 rotation quantities participate in the adjustment and calculate the correction value. It is not recommended to correct ΔZ when processing aviation down-view data. All rotation quantities are checked by default, while all translation quantities are not checked. Better results can be obtained by using the default values.
  - **Matching**: Users can choose two matching methods: facet piece matching and sampling point matching.
    - **Facets**: Uniformly extract high-order facets combinations in the overlapping area of the data to be matched (relatively robust on data with this feature).
    - **Sampled Points**: The sampling point is extracted by analyzing the significance of the data to be matched, and then matched based on the sampling point and its normal vector.

Placement Error Calculation

The placement error between the laser scanning reference coordinate system and the positioning inertial navigation platform reference coordinate system is the largest source of systematic error in airborne lidar. The influence of these placement errors on the coordinates of the ground laser foot point also depends on the altitude of the flight and the size of the scanning angle.

The software provides two ways to eliminate placement errors：

1.Manual measurement and inspection.

2.Automatic adjustment correction.

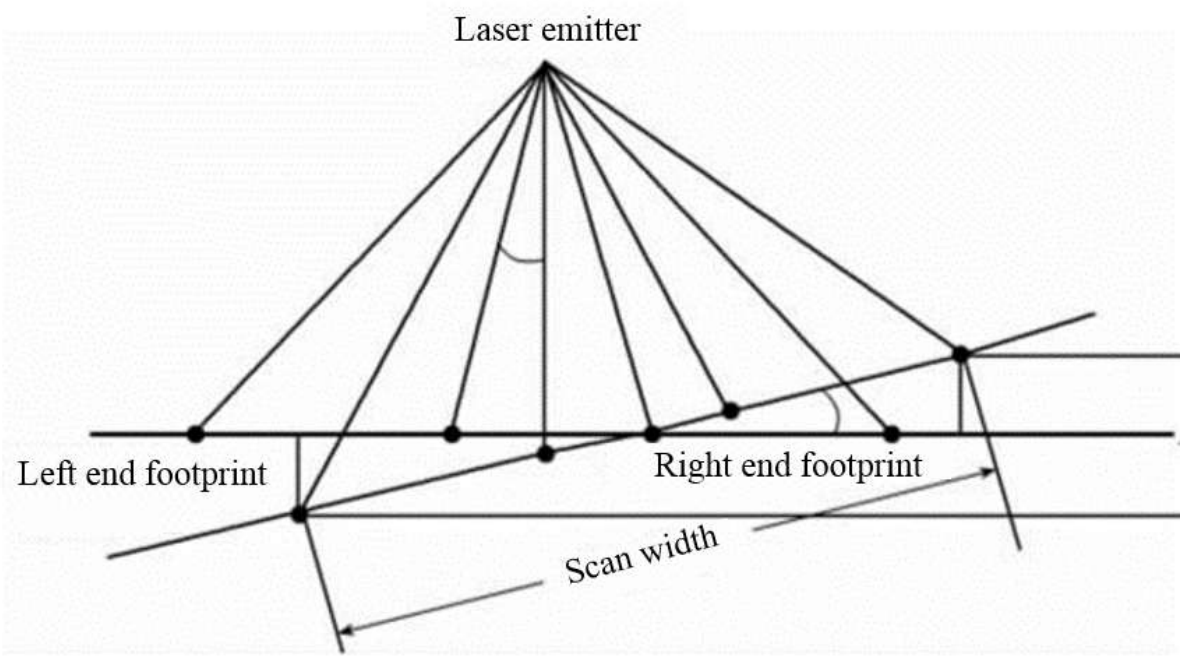
Their principles are described in detail below.

Manual Measurement and Inspection

First, you need to prepare the inspection data, usually flying in four directions perpendicular to each other, front, rear, left and right. After the flight, regular objects on the ground (such as playgrounds, regular houses, etc.) are measured. Based on the overlapping laser foot point data, step-by-step geometry can be used (Zhang et al., 2010) to recover the correction value of the placement angle error (that is, the amount of rotation). However, the placement offset error (that is, the translation amount) has a small impact, and manual measurement and inspection will not correct it.

Estimate the Roll Angle Error (ΔRoll)

The placement angle error of the side roll will cause the plane scanning line to tilt (as shown in the figure below), and the plane position of the scanned object will be offset along the scanning direction (perpendicular to the flight direction).



In the data of the two air belts flying back and forth at the same altitude

- Open the profile perpendicular to the direction of flight to measure the height difference of the approximate

object of the same name.

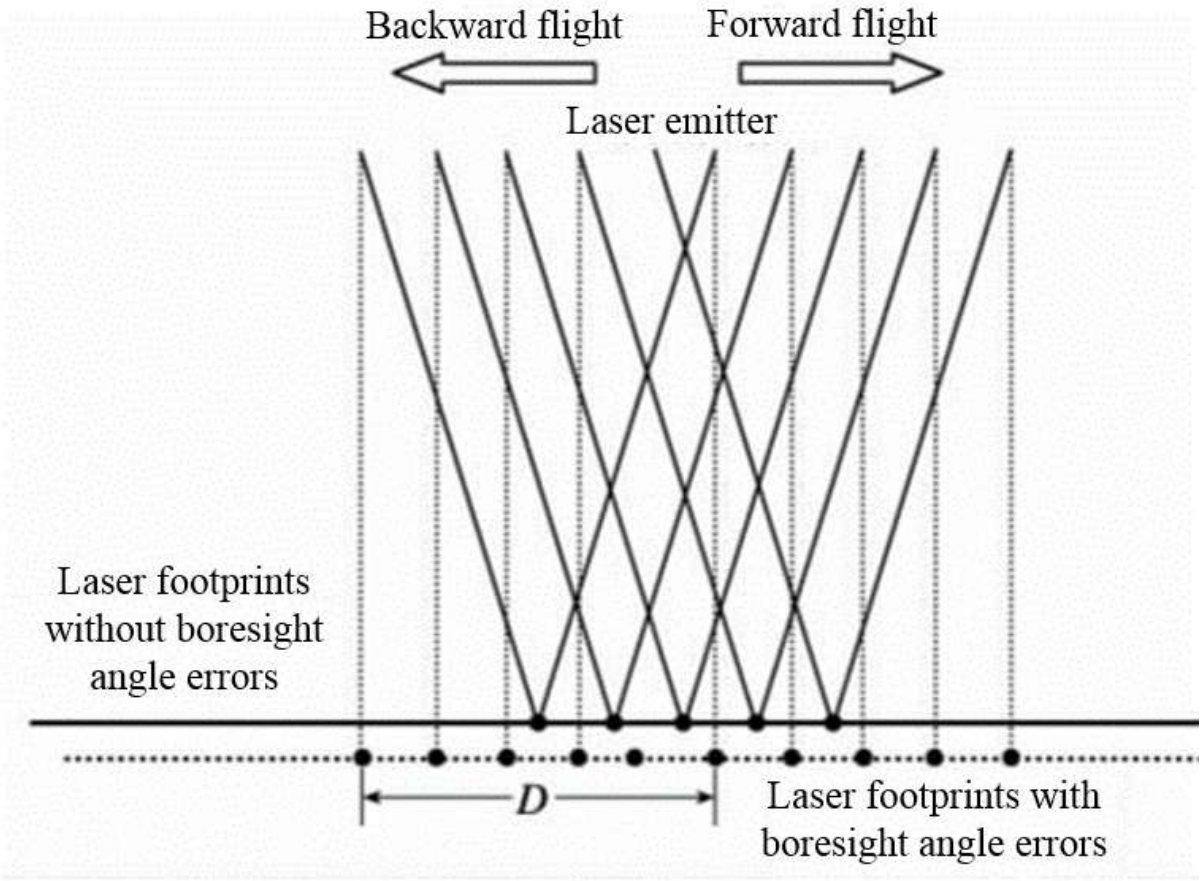
- Measure the horizontal distance r between the approximate object of the same name and the center line of the two air belts in the 2D view.

The estimation formula for the roll angle error is as follows：

$$\Delta Roll \approx arctan\left(\frac{\Delta h}{2r}\right)$$

Estimate Pitch Angle Error (ΔPitch)

In the online scanning mode, the placement error of the pitch direction mainly deviates the true position of the scanned object in a direction perpendicular to the scanning line. The following is a schematic diagram of the influence of pitch direction placement error on the scanning laser foot point.



In the data of the two air belts of the round-trip flight：

- Open the profile parallel to the flight direction to measure the distance difference D between the center position of the same object along the flight direction.

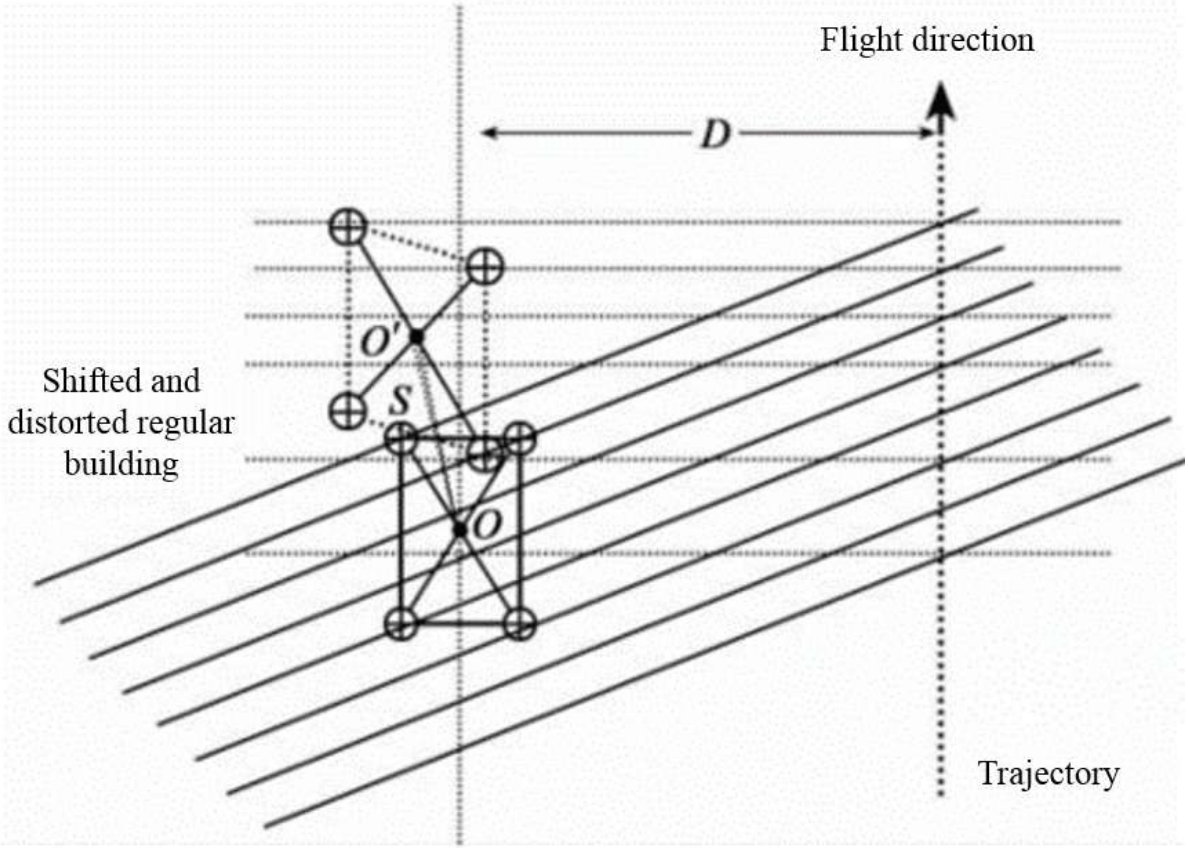
- Calculate the average flight altitude H based on the trajectory (try to keep the round-trip flight altitude as consistent as possible).

The estimation formula for the pitch angle error is as follows：

$$\Delta Pitch \approx arctan\left(\frac{D}{2H}\right)$$

Estimate Heading Angle Error (ΔHeading)

The heading angle placement error will change the center position of the scanned object and deform the object at the same time, as shown in the figure below.



Open a 2D view in the data of the two air belts of forward and backward flight

- Measure the distance S between the average center position of the laser foot of the object twice.
- Measure the distance D between the two air belts.

The estimation formula for heading angle error is as follows：

$$\Delta Heading \approx arctan\left(\frac{S}{D}\right)$$

Automatic Adjustment Correction

Manual measurement estimation requires relevant professional knowledge and proficient operation of the software, while automatic calculation can greatly reduce the workload of the operators. In data with obvious characteristics, automatic calculation can completely replace manual calculation and achieve the same or even higher accuracy.

The automatic algorithm can not only correct the placement angle error (that is, the amount of rotation), but also correct the placement offset error (that is, the amount of translation). You can freely choose the values that need to be corrected, and it is recommended to correct only the placement angle errors because they have the greatest impact. The principle of the algorithm is as follows：

1.Extract feature points and normal vectors in adjacent air belts, refer to the algorithm (Glira et al., 2015).

2.Match the feature points extracted from the adjacent air band to obtain the relevant point pairs.

3.Establish a placement error correction model and calculate the distance of the relevant point pair along the normal vector.

4.The least squares method is used to minimize the correlation distance and obtain the optimal solution of the corrected value at the same time.

@inproceedings{
author={Zhang Xiaohong, Forsberg Rene},
title={Retrieval of Airborne Lidar Misalignments Based on the Stepwise Geometric Method},
booktitle={Survey Review 42(316):176-192 April 2010},
year={2010}, }

@inproceedings{
author={Philipp Glira, Norbert Pfeifer, Christan Briese and Camillo Ressl},
title={A Correspondence Framework for ALS Strip Adjustments based on Variants of the ICP Algorithm},
booktitle={PFG Photogrammetrie, Fernerkundung, Geoinformation Jahrgang 2015 Heft 4},
year={2015},
}

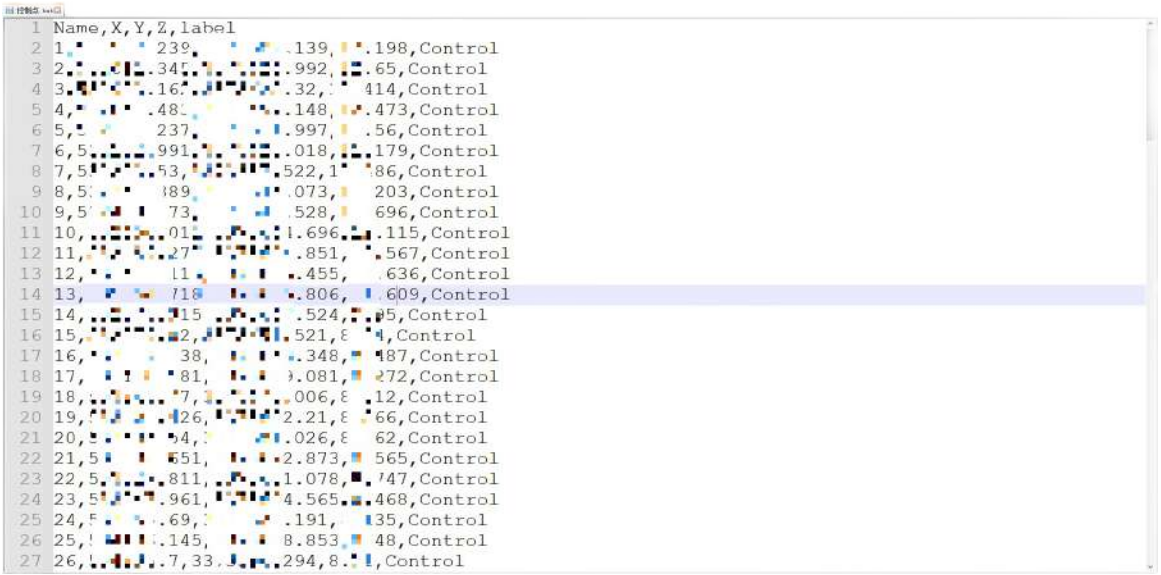
GCP Adjustment

Function Description: The control point correction function is used to perform partial or single-pass correction on the calculated data of the vehicle-mounted mobile measurement device.

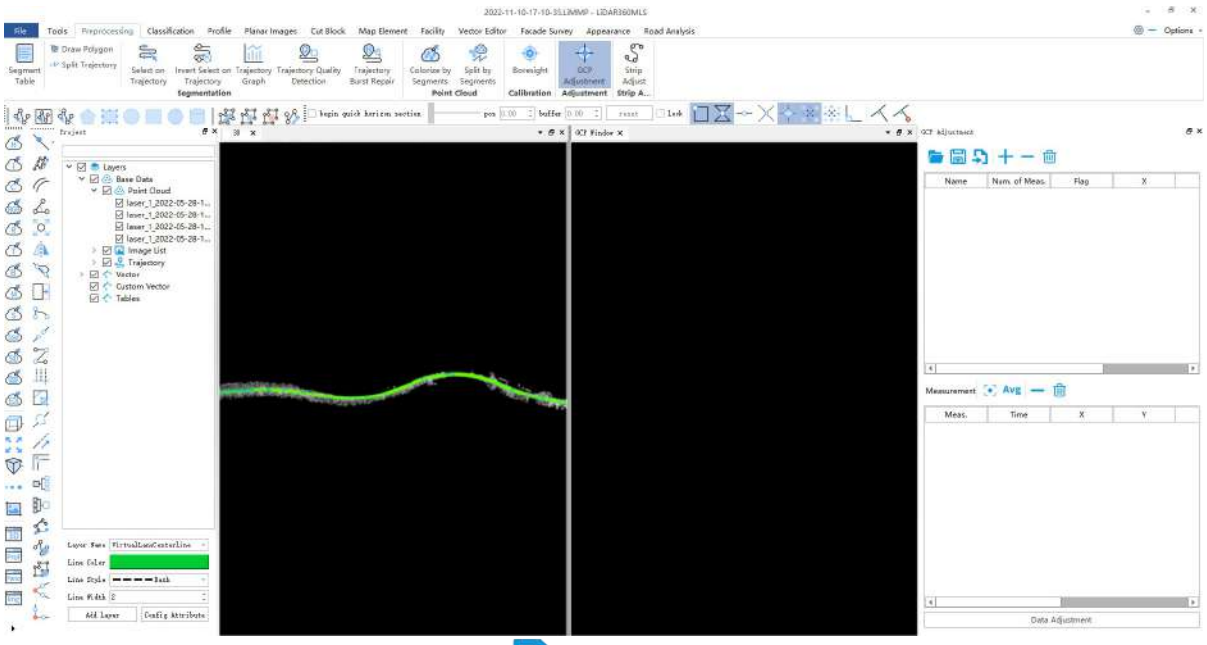
Steps

1.Data preparation: Data preparation includes the following two aspects

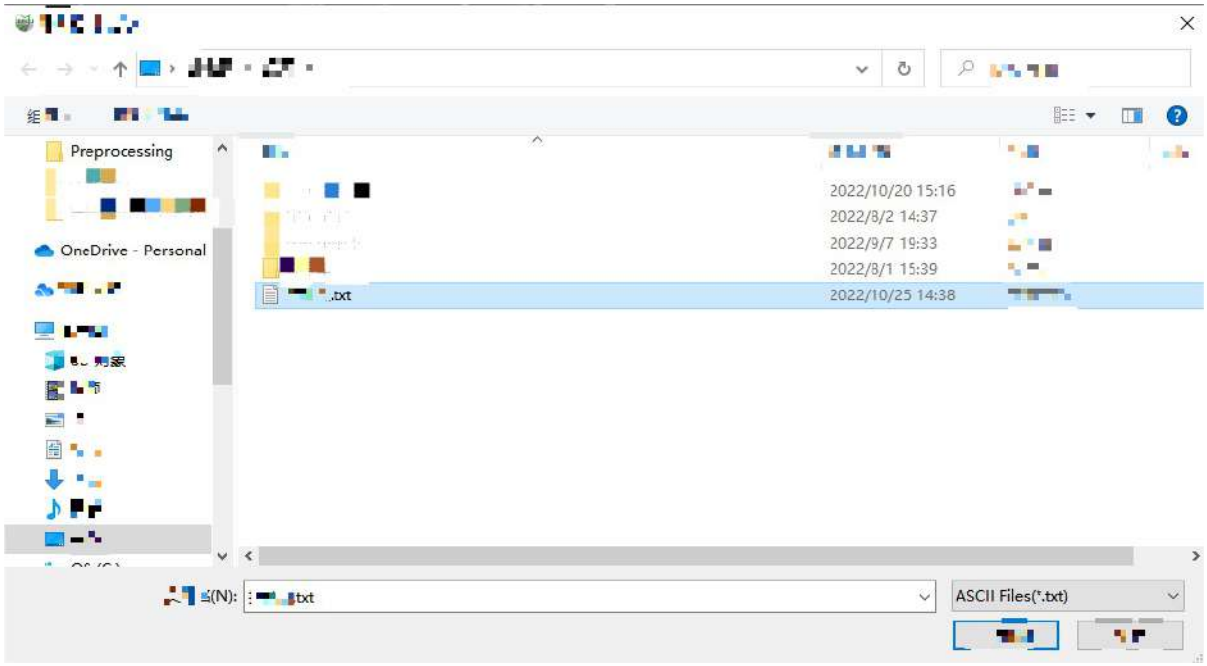
- 1.1. Source data: It contains the original point cloud to be corrected and the corresponding trajectory file. If there is an image file during the data calculation, the image data is also required. Among them, LiDAR360 MLS software can be used to directly open the .ligeo project file solved by Ligeo, or you can also refer to New Project to configure point cloud, trajectory, Image, to create a new .LiMMP project.
- 1.2. Control point file: The control point file needs to be organized into a .txt, which includes five columns of information: point name, X, Y, Z, label. Among them, the first four columns can generally be obtained when external dots are made. The label needs to use Control/Check to distinguish whether it is a control point or a check point.



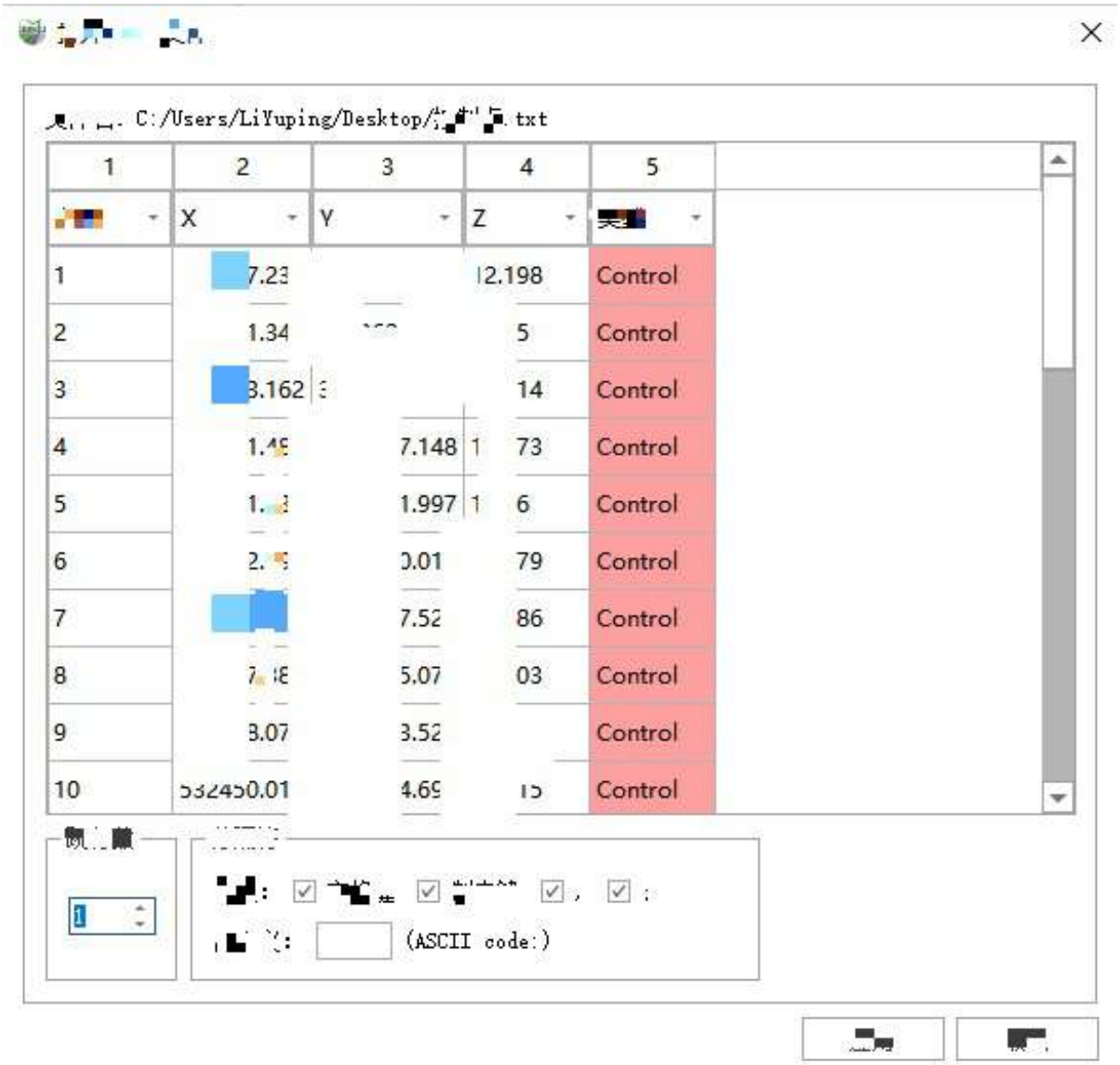
2.Click **GCP Adjustment** button to activate this function, the control point correction dialog box will appear on the right side of the main page, and the GCP window will appear at the same time.



3.Import external control point file: Click **Import** button to import the control point TXT file prepared in step.



Select the file to import, and you will be prompted to select the corresponding column name. You can select and match the column according to the actual situation of the control point file.



4.If the control point file is not prepared in the early stage, but the control point information is known, you can add different control points by **clicking on the ground control point**. Click the + button in the upper half of the Control Point Alignment window, the Click Ground Control Point dialog box will appear:

By entering the control point information in the text boxes corresponding to XYZ, and clicking the Add button, you can add a new control point. If there is no RTK control point collected in the field, but the collected data has poor accuracy on a certain channel, you can click on the corresponding point cloud of the flight belt in the 3D window to collect the feature point coordinates of the point cloud in the reverse direction as the reference control point.

5.After the control point file is imported normally, you can see that the software interface displays the following contents:

(1) 3D window: The data from the point cloud file where the first control point is located will be displayed by default. In the upper part of the control point correction window, click on any control point to control the display the section of the point cloud where the current control point is located in the 3D window.

(2) The GCP window displays the currently selected control point and the point cloud data of the current air strip within a certain slice range. In the upper part of the control point correction window, click on any control point to control the data within the single air strip slice range in the current window.

(3) Control point guide dialog:

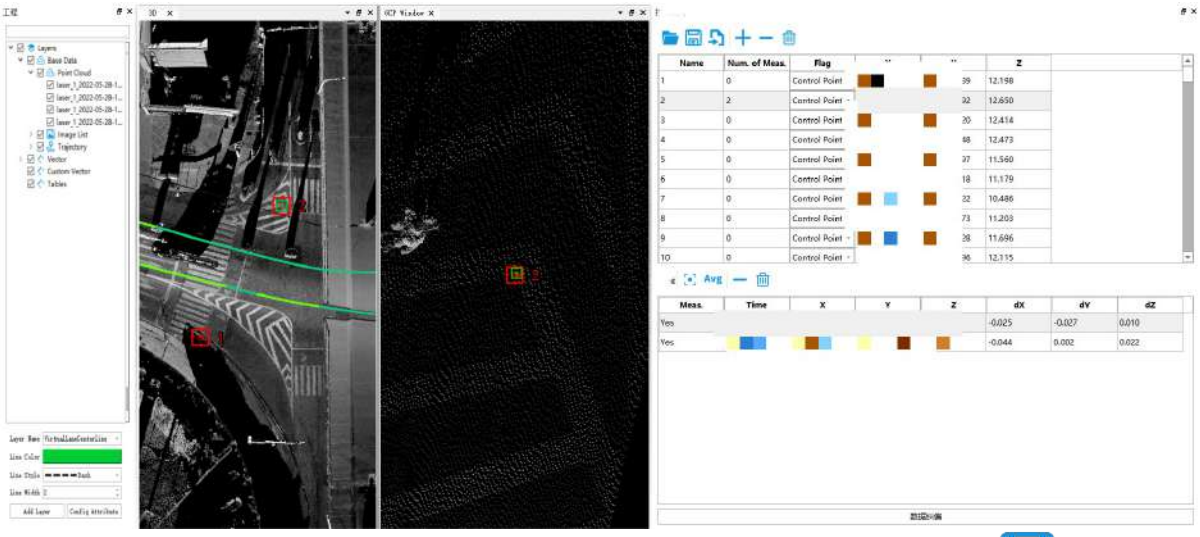
(3.1) The upper part shows the imported point-by-point information:

- Name: The imported control point name.
- Num. of Meas.: The current control point that can control several air strips, that is, the current control point in the lower half has been punctured correspondingly on several air strips.
- Flag: The type of control point, Control Point indicates that the current point is the control point during the deviation correction process; Check Point indicates that the current point is the check point during the deviation correction process.
- X: East coordinate of control point coordinates.
- Y: North coordinate of the control point file.
- Z: The elevation value of the control point.

(3.2) The lower part displays the information about each control corresponding point to be punctured, click any record, and then use the mouse to click the point of the point cloud corresponding to the current control point in the GCP window with the left mouse button to puncture:

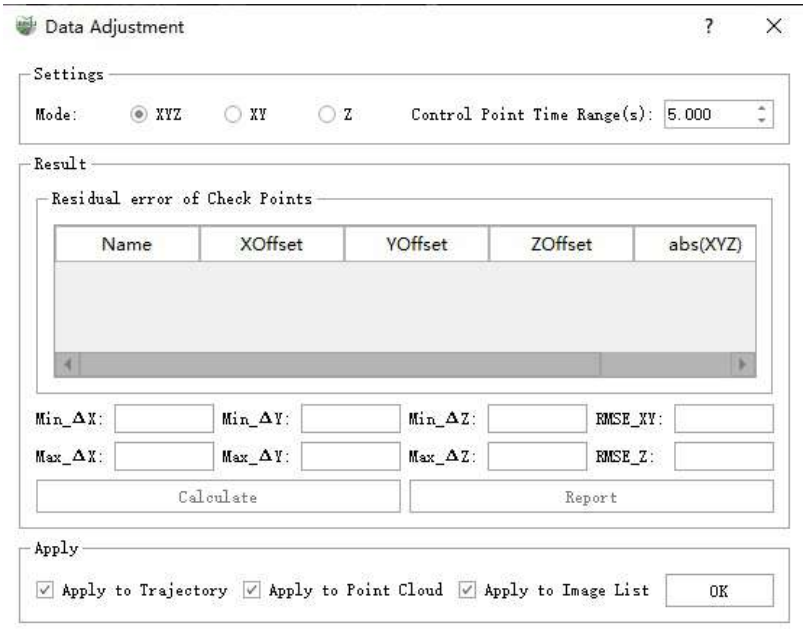
- Meas.: No means that the flight zone is not constrained by the corresponding control point, Yes means the flight zone is constrained by the current control point.
- Time: The collection time of the current puncture point position.

- X: The east coordinate of the point cloud representing the current puncture point position.
- Y: The north coordinate of the point cloud representing the current puncture point position.
- Z: The point cloud elevation value representing the current puncture point position.
- dx: The offset between the point cloud and the east coordinate of the current control point.
- dY: The offset of the point cloud from the north coordinate of the current control point.
- dZ: The coordinate offset between the point cloud and the current control point elevation.



6.When all the punctures corresponding to all the control points are completed, click **Save File** button to export the GCP control point information, the exported xml file can not only continue to be loaded later and the only current function continues to be modified and used, and this file is also used as the control point file information in the splicing process in the StripAdjust function.

7.Click **Data Adjustment**, and the **Data Adjustment** dialog box will pop up. In this dialog box, set the correction mode, perform the correction calculation, and view the checkpoint residuals. Click the **Calculate** button to calculate the control point correction parameters according to the puncture point information.

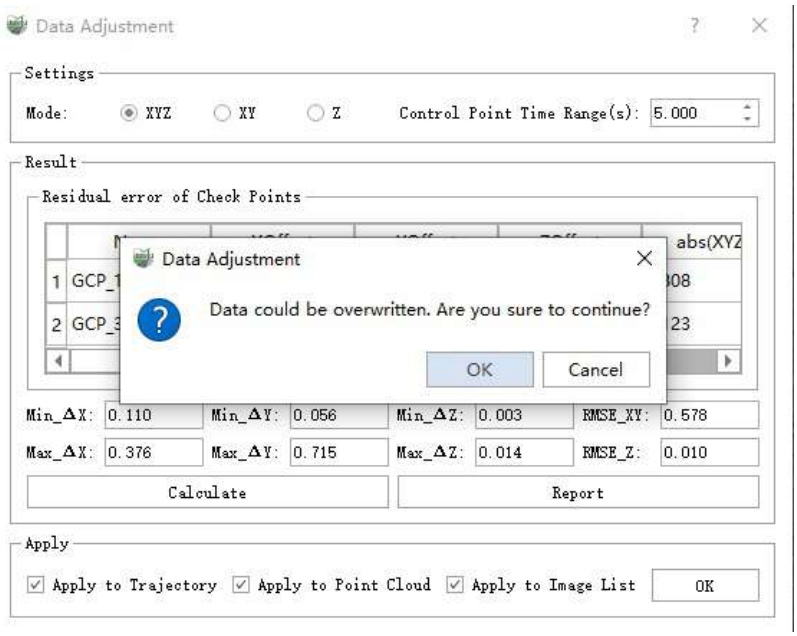


Parameter Description:

- Setting Mode: The mode used to set the skew correction.

    - XYZ: Represents correcting coordinates in three directions of XYZ at the same time.
    - XY: Represents only correct coordinates in both XY directions.
    - Z: Means to correct only the value in the Z direction.
    - Control time range (seconds): Represents the sampling interval by time.
- Result: To calculate the correction parameters of the control point, and check the correction accuracy of the set check point.
- Apply: After the correction parameters are confirmed to be correct, you can click the OK button to apply the results to the selected data type. It is recommended to select all of them here. At the same time, the image data must be set when creating a new project. Otherwise, it will not modify the image information, which may cause the point cloud and the image to be misaligned after the offset correction.

> Note: This step will rewrite the original data, please make sure that there is no problem in the parameter calculation before applying it.



Strip Adjustment

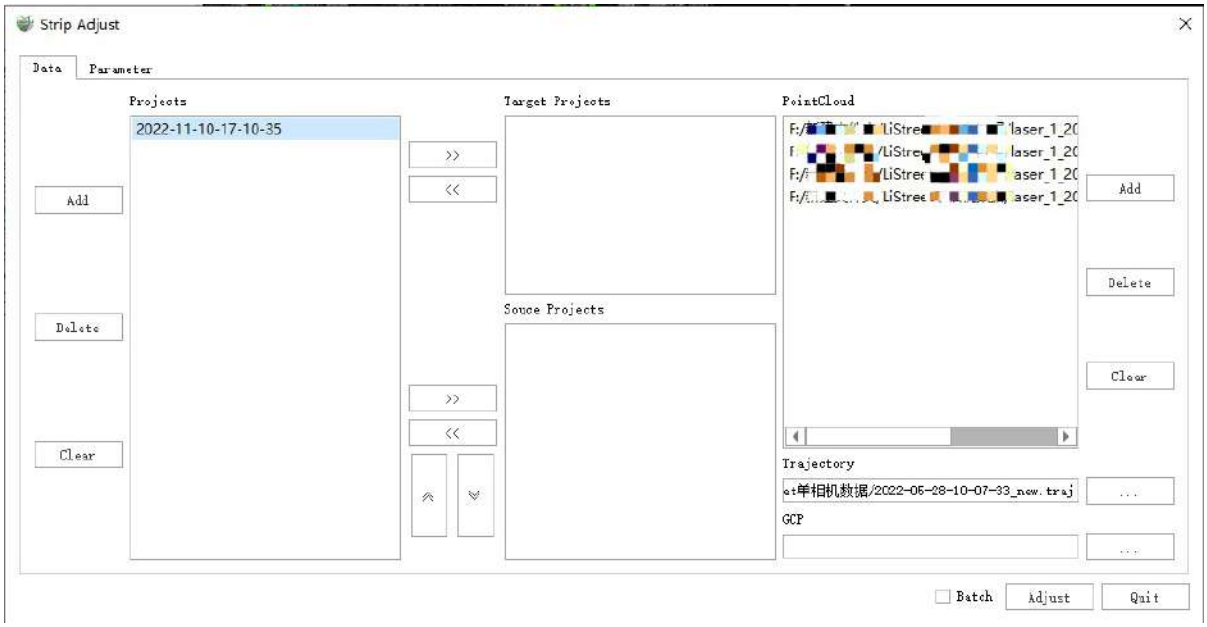
Function Description: The strip adjustment supports the adjustment of the point cloud revisited area within the vehicle data of a single project; supports the adjustment calculation of multiple groups of projects, and aligns the registration project point cloud to the reference to the engineering point cloud to achieve the purpose of weakening the layering error of the revisited area; and support the control points to participate in the adjustment. The parameter setting and usage steps will be described in detail below.

Parameter Description

1.Click **Strip Adjustment**  After pressing the button, a pop-up box pops up:

- **Project Data**: Supports setting reference projects and registration projects participating in the adjustment, and supports setting point clouds, trajectories, and control points for each project.
- **Adjustment Parameters**: Supports setting the parameters of trajectory segmentation, segmentation registration, and SPLIEN adjustment steps.

2.**Project Data**



- **Add Project**: Add project files (.LiStreet, .LiMMP) from outside.
- **Delete Project**: Delete the added project.
- **Clear Project**: Clear all projects.
- **Add target project**: Select one in the project bar and click Add to target project.
- **Remove Target Project**: Remove the added project from the target project to the project list.
- **Add Matching Project**: Select one in the project bar and click Add to Matching Project.
- **Remove Matching Projects**: Remove a project from the matching project column to the project list.
- **Move Matching Project Up**: Move the selected registration project down one place to adjust the order of the registration projects. It is sorted by the acquisition time or the location of the acquisition area, and the adjustment calculation will be performed in the order of the projects.
- **Move Matching Projects Down**: Move the selected registration project up one place to adjust the registration project sequence. It is sorted by the acquisition time or the location of the acquisition area, and the adjustment calculation will be performed according to the project order.

- **Add Point Cloud**: Select any target/matching project and add point cloud file.
- **Delete Point Cloud**: Select any target/matching project, and delete a certain segment/several segments of the point cloud file.
- **Clear Point Cloud**: Select any target/matching project, clear all point cloud files in the project and set again.
- **Select Track**: Select any target/matching project and reset the track file.
- **Select Control Point**: Select any target/matching project, import the GCP control point file corresponding to the current project, and the file will be saved and generated by saving the puncture points through the [Control Point Correction] (GCPAdjustment.md) function.

3.**Adjustment Parameters**

- **Trajectory Split**:
  - Min Length of Segment: The minimum length of the track segment result. It is recommended to set the default value.
  - Max Length of Segment: The maximum length of the track segment result. It is recommended to set the default value.
  - It can be adjusted according to the size of the collection area, the open area can be appropriately enlarged, and the shaded area should be reduced. Segmented point cloud is based on trajectory segmentation and provides matching point cloud for segment matching.
- **Registration**:
  - Max Correspond Distance: Matched pairs whose distance is greater than this value do not participate in the adjustment. It is recommended to set the default value.
  - Subsample Space: point cloud thinning distance. It is recommended to set the default value.
  - Min Distance from Ground to Trajectory: Ground points with a distance from the track height less than this value will be filtered. It is recommended to set the default value.
  - Overlap Threshold: Matching pairs with overlap less than the threshold will be filtered. It is recommended to set the default value.
  - Thread Number: Match the number of threads, which is related to the computer configuration. It is recommended to follow the default. Segment matching performs relative splicing based on the revisited endpoint cloud, which provides matching constraint information for the subsequent SPLINE overall adjustment optimization, which directly affects the final splicing effect.
- **SpLine Optimization**:

- Time Interval: SPLINE node time interval, the smaller the interval, the finer the pose correction and the greater the calculation amount. It is recommended to set the default value.
- Max Correction of Position: The maximum correction amount of SPLINE node position. It is recommended to set the default value.
- Max Correction of Posture: The SPLINE overall adjustment takes the trajectory pose corresponding to the SPLINE node as the basic adjustment unit, and adjusts the trajectory pose correction based on the SPLINE assumption to perform adjustment optimization to achieve the corrected trajectory pose splicing point cloud and guarantee the corrected trajectory pose smooth and specific effects. The maximum amount of correction for the attitude of SPLINE nodes. It is recommended to set the default value.

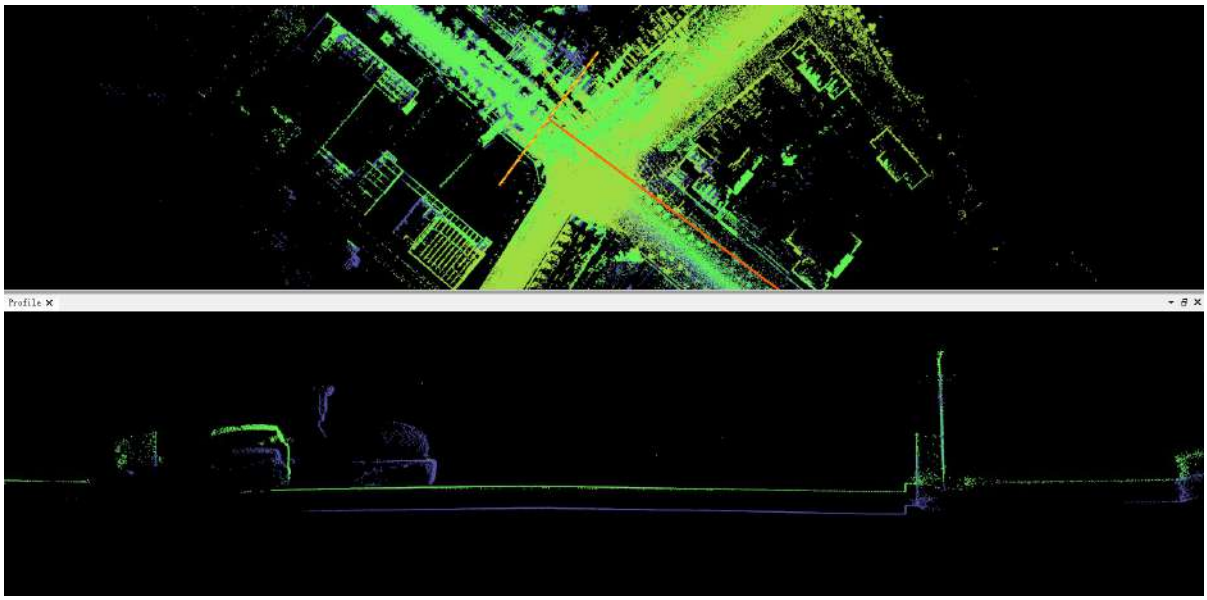
Single Project Strip Adjust

1.Through New Project or directly Open ligeo Project method, open the vehicle engineering data to be optimized, and it is recommended to configure the image information here.

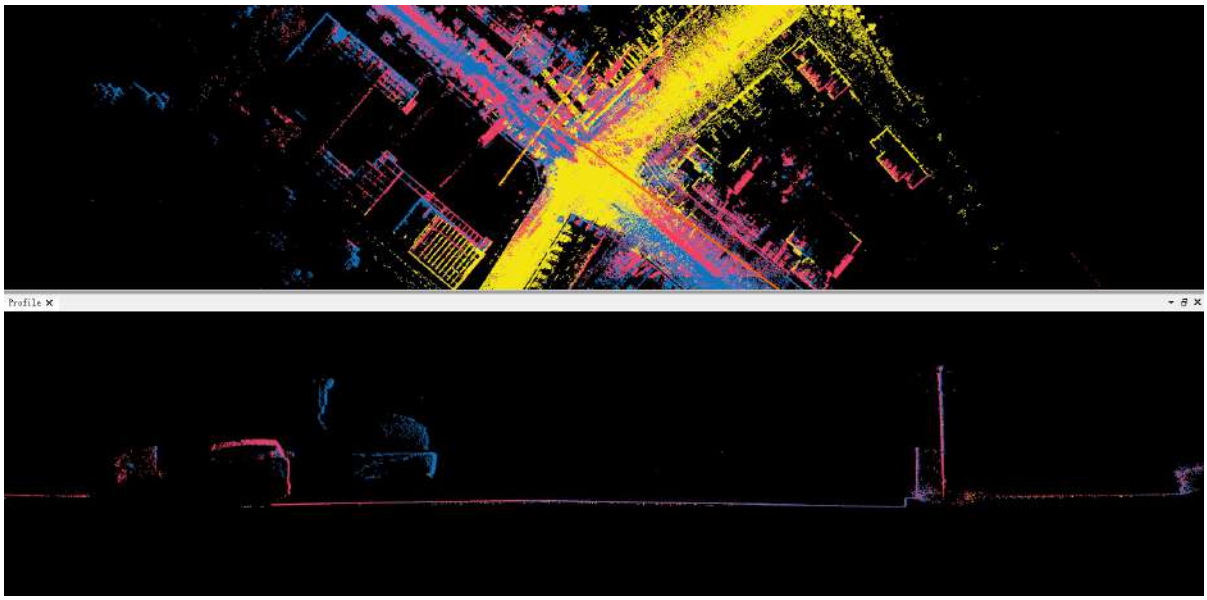
2.Click the  function button, in the setting of the air belt splicing project, move the current project to the matching project. If internal splicing of control point parameters is required, you need to pass Control Point Correction in advance to puncture the control point and save it as an .xml file, and then load the modified file into the control point file setting of the current project of the air belt splicing.

3.Click the Splicing button to activate the splicing function. The air belt splicing will not recreate the original data. Under the folder where the current project is located, a new project and folder with "current project name-adjust" will be generated. The spliced point cloud, trajectory and other data are stored in "\adjust\optimize_result" path.

4.If there are multiple sets of single projects that need to be spliced, you only need to add all single projects to the project list by adding projects, and then move them into matching projects, and check the batch check box processing at the top and bottom. The internal adjustment operation is performed on each group of projects in sequence. The adjusted results of each group of projects are in the adjust folder under the folder where the respective projects are located.



Before the single-engineering air belt splicing



After the single-engineering air belt splicing

Multi-project Strip Adjust

1.Assuming that there are four groups of ABCD projects, the AB, BC and CD projects have a certain range of public areas respectively. Due to the unobstructed ground and other reasons, the point clouds of the two projects with the same name do not completely overlap, and there are certain layers, and A project has participated in single project splicing through control points, ensuring absolute accuracy. At this time, multi-project splicing is required to deal with problems such as layering between the four groups of projects.

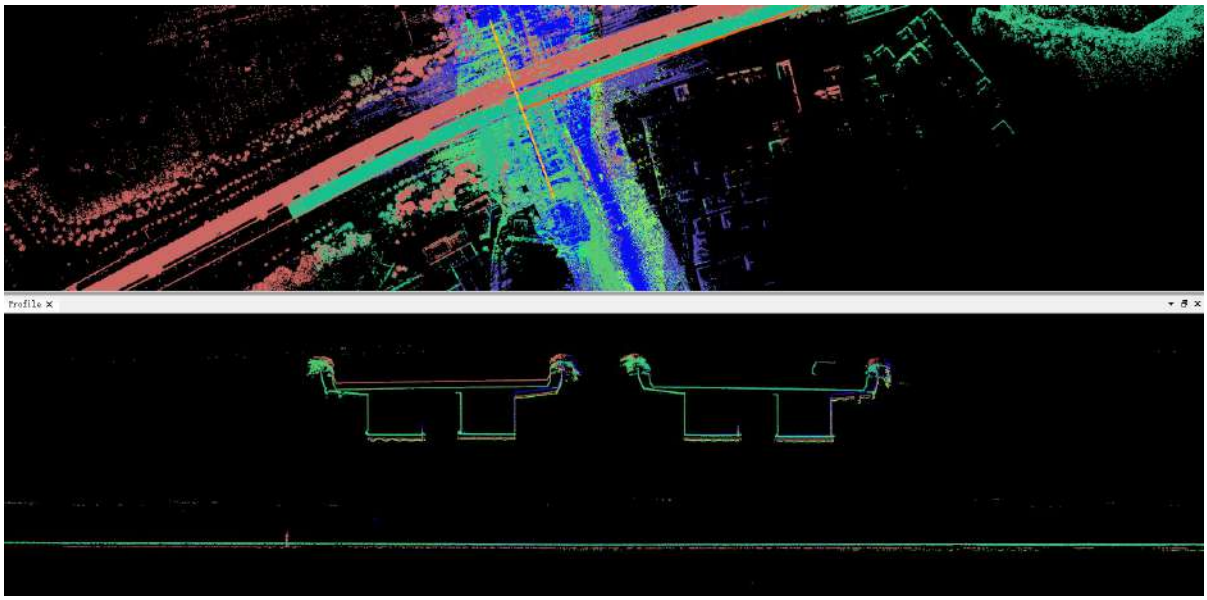
2.Open any group of projects, assuming project A is opened here, click **Strip Adjustment** button, on the left side of the project list, click the "Add" button to add the BCD project to the project list.

3.According to step 1, it is known that project A can be used as a target project, and BCD needs to be used as a matching project, then in the project list, select project A, and move project A into the project A through the **Add Target Project** button on the left side of the target project list. Then in the project list, select the BCD project respectively, and move the BCD project into the matching project list by pressing the **Add Matching Project** button on the left side of the matching project. And according to step 1, the splicing sequence is project A←project B←project C←project D. Therefore, you can adjust the order of BCD projects by matching the up and down buttons on the left side of the project.

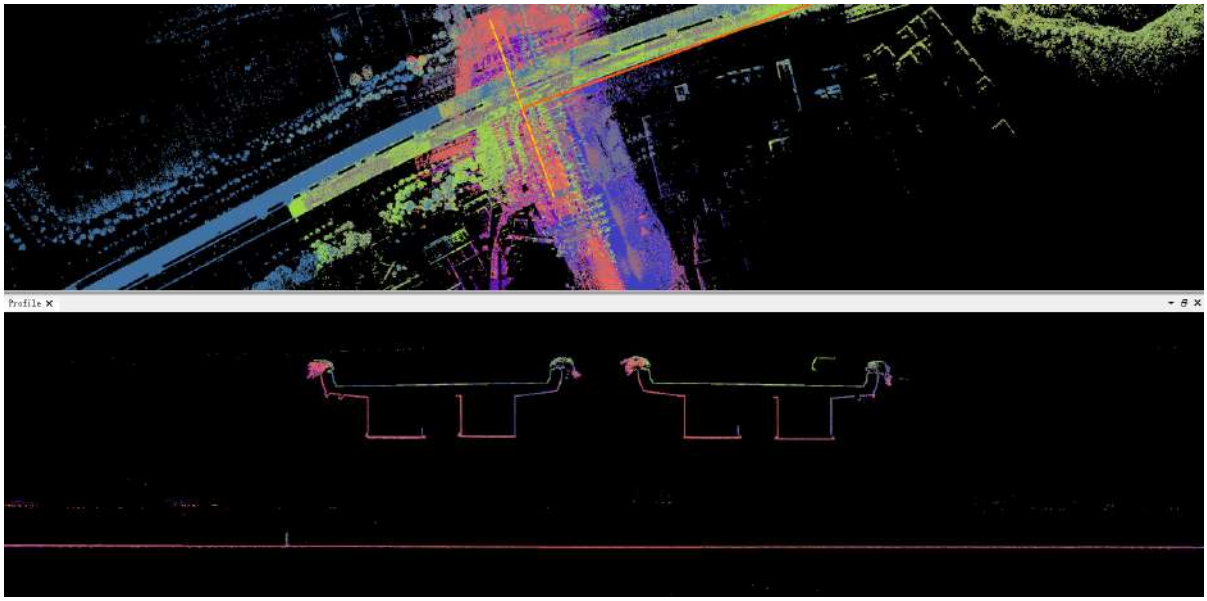
4.Click on each project to view and modify the point cloud, trajectory and control point data of the current project on the right side of the air belt splicing interface. If the control point data is set in a single project, the control points will participate in the adjustment during the process of two-by-two project adjustment, ensuring that the spliced data can not only eliminate the relative stratification, but also ensure the absolute accuracy.

5.After the above steps are confirmed to be correct, click the splicing button directly to perform multi-project splicing of four groups of projects.

6.The results of BCD project splicing are all in the suffix adjust folder under the same name directory of the respective project files.



Before multi-engineering air belt splicing



After the multi-engineering air belt splicing

Cut Block

The functions included under the Data Fragmentation menu bar are:

- Subdivision
- Cut Block
- Block Display

Subdivision

Point cloud framing includes framing according to point cloud, framing according to trajectory, and framing area block display, among which, operations such as map node editing can be performed according to the framing frame.

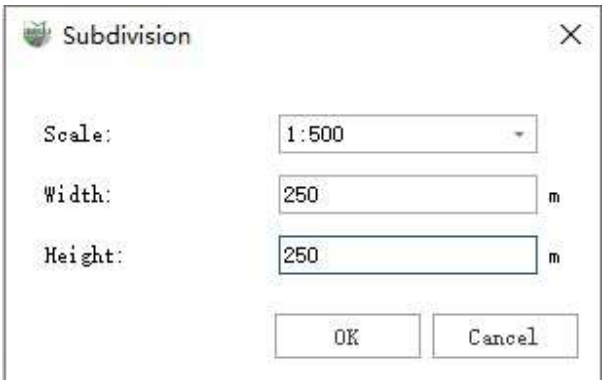
- Point Cloud Subdivision
- Point Cloud Cut Block
- Create Along Trajectory
- Create Along Center Line

Point Cloud Subdivision

Function Description: LiDAR360MLS uses scale-based point cloud segmentation, and can perform point cloud data subdivision without loading the trajectory file, and automatically segment the point cloud according to the set scale, to provide users with a fast and efficient means of data collaborative processing.

Steps

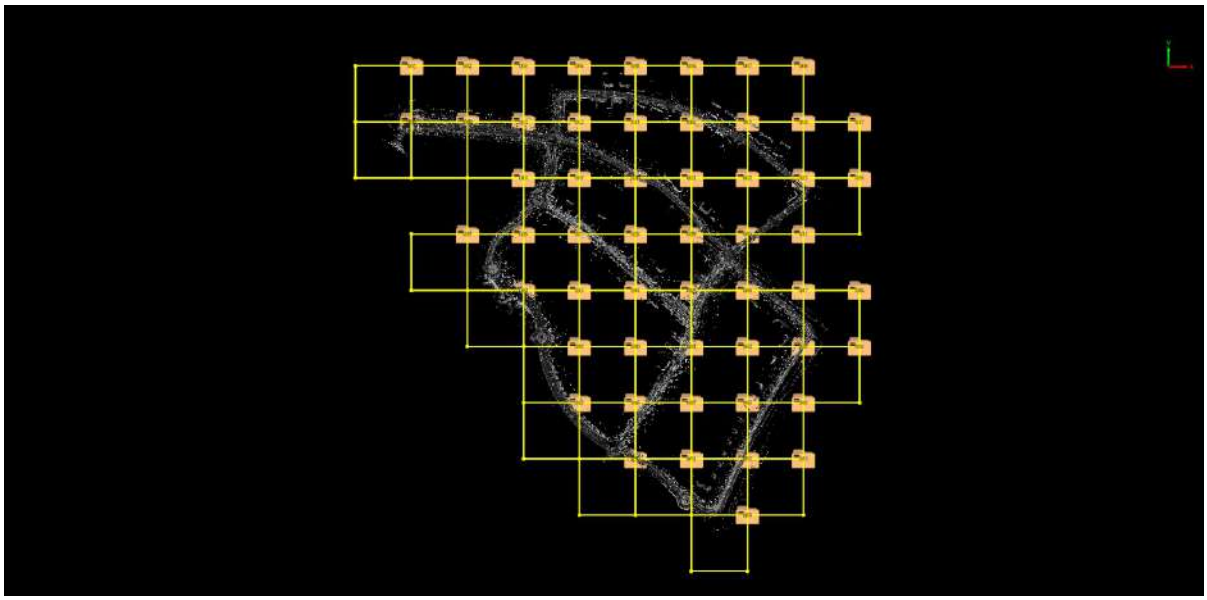
1.Click **Point Cloud Subdivision** button and the following interface will appear.



Parameter Description

- **Scale**: The precision can be selected as 1:500, 1:1000, 1:2000, 1:5000.
- **Width**: Controls the width of the frame.
- **Height**: Controls the height of the frame.

2.After clicking the OK button, the result of the framing is shown in the following figure:

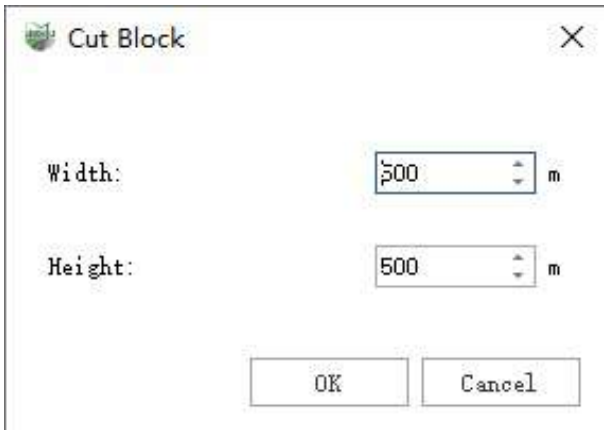


Point Cloud Cut Block

Function Description: This function supports point cloud cutting into blocks mode with user-defined rectangle length and width.

Steps

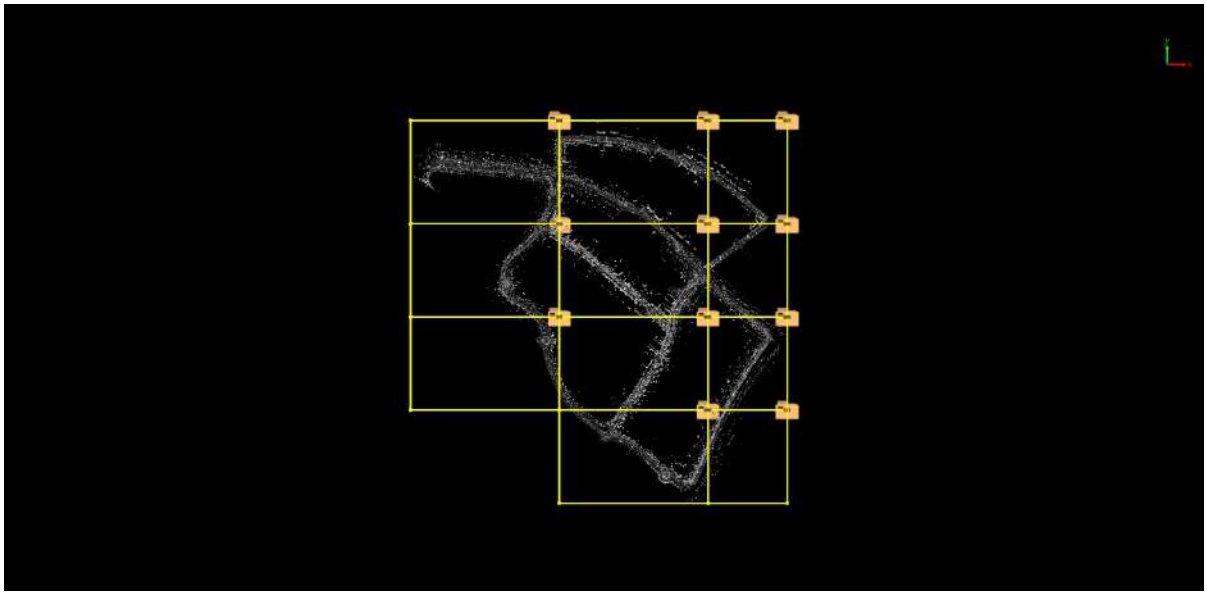
1.Click **Point Cloud Cut Block**, the following interface pops up.



Parameter Description

- **Width**: Controls the width of the frame.
- **Height**: Controls the height of the frame.

2.After clicking the OK button, the result of the framing is shown in the following figure:



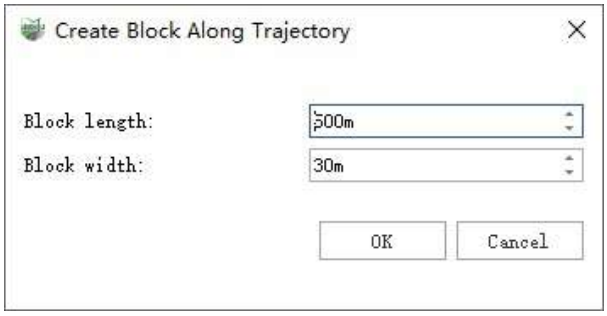
Create Along Trajectory

Function Description: This function only performs framing according to the trajectory file, and will not segment the point cloud.

> Note: You need to add track files before framing, otherwise a pop-up warning will appear.

Steps

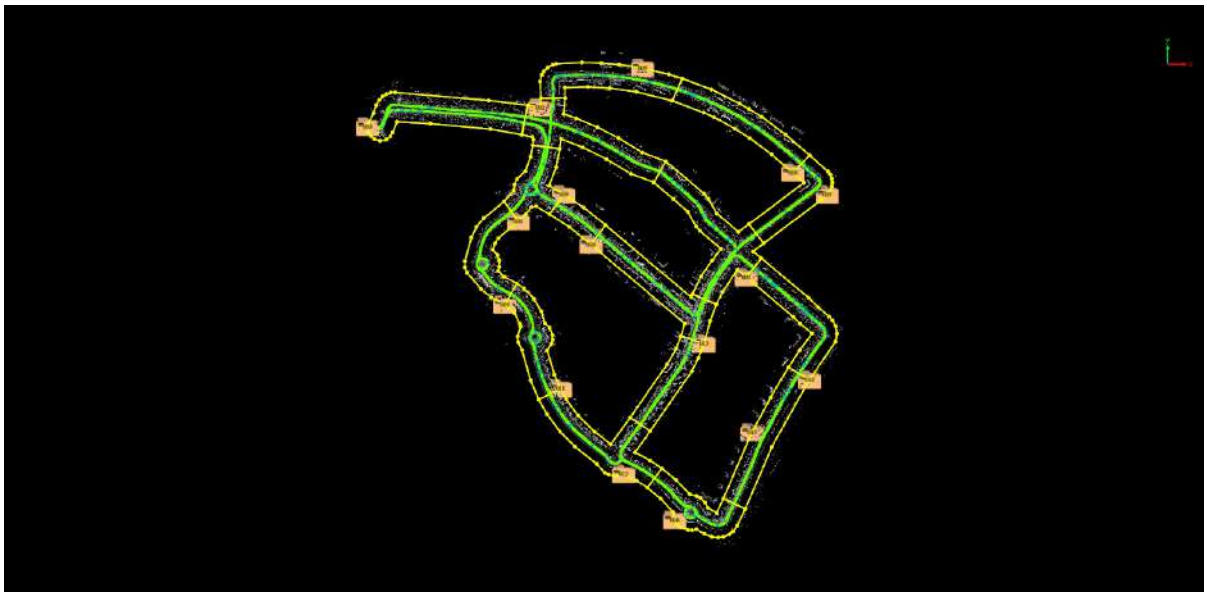
1.Click **Create Along Trajectory**  , the following interface pops up.



Parameter Description

- **Block length**: The length of the frame, unit: meters.
- **Block width**: The width of the frame, unit: meters.

2.After clicking the OK button, the result of segmenting according to the trajectory is shown in the following figure:



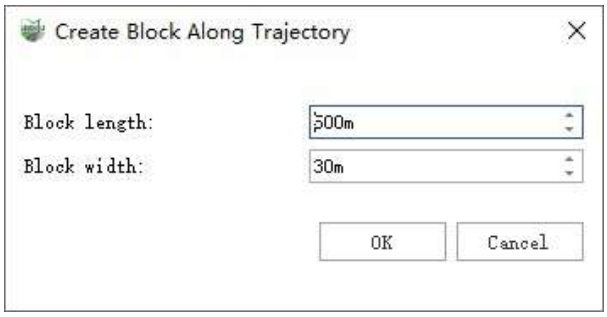
Create Along Center Line

Function Description: This function only performs framing according to the vector line, and does not segment the point cloud. Note: Before framing, you need to draw a vector line and click on this line, otherwise a pop-up warning will appear.

Note: Before framing, you need to draw a vector line and click on this line, otherwise a pop-up warning will appear.

Steps

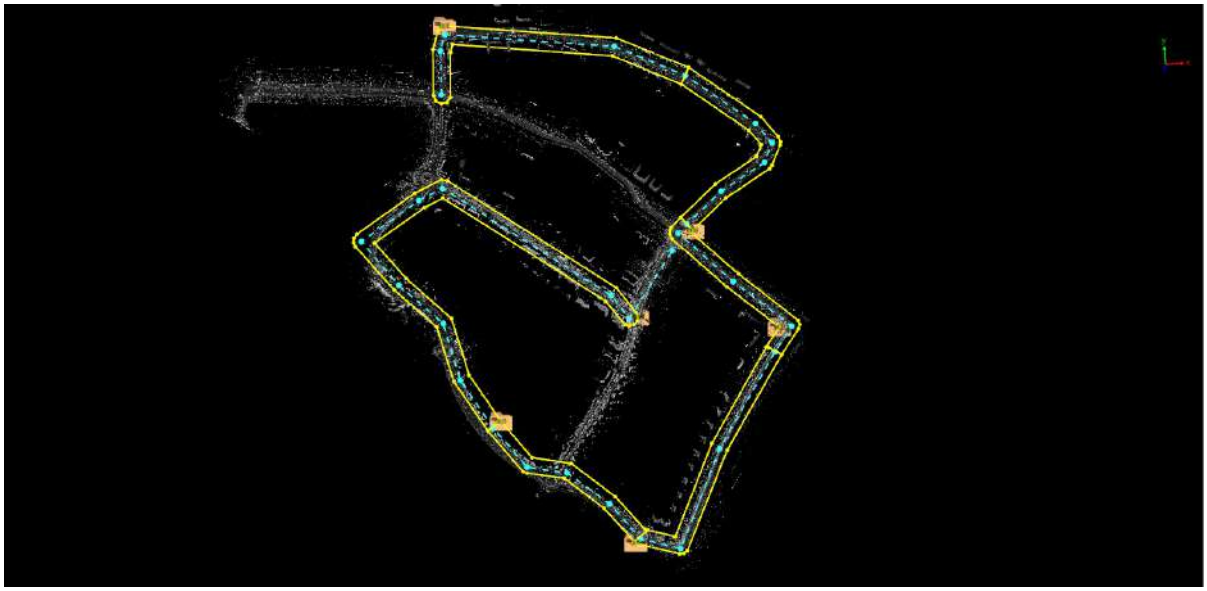
1.Click **Create Along Center Line**  , the following interface will appear.



Parameter Description

- **Block length**: The length of the frame, unit: meters.
- **Block width**: The width of the frame, unit: meters.

2.After clicking the OK button, the result of segmenting according to the trajectory is shown in the following figure:



Cut Block

- Cut Block Edit
- Split Point Cloud

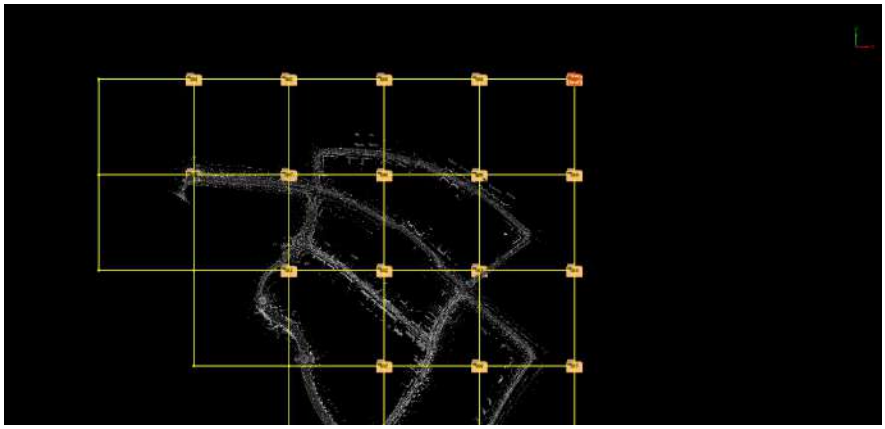
Cut Block Edit

Function Description: This function is used to edit the area block after framing based on the track.

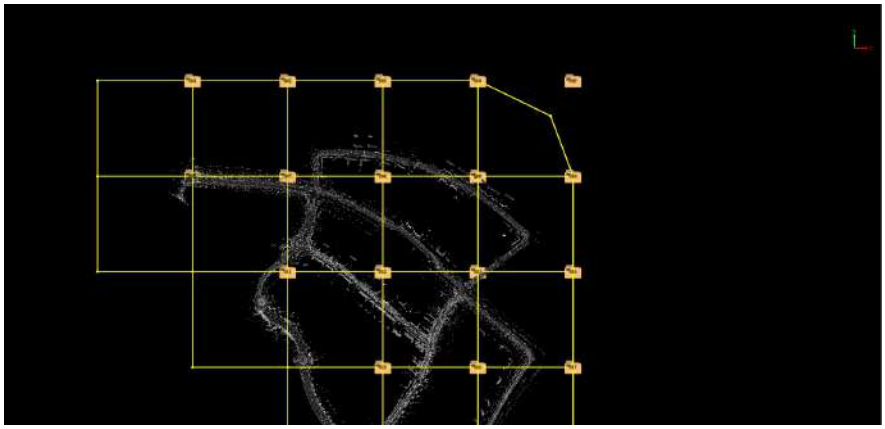
Steps

- **Edit Block Node** : Click the  button, move the mouse to the node of the frame, when the color of the mouse turns red, it means that the current node is captured. Press and hold the left mouse button and drag the current node to a new position, as shown in the figure below.

  Before editing:

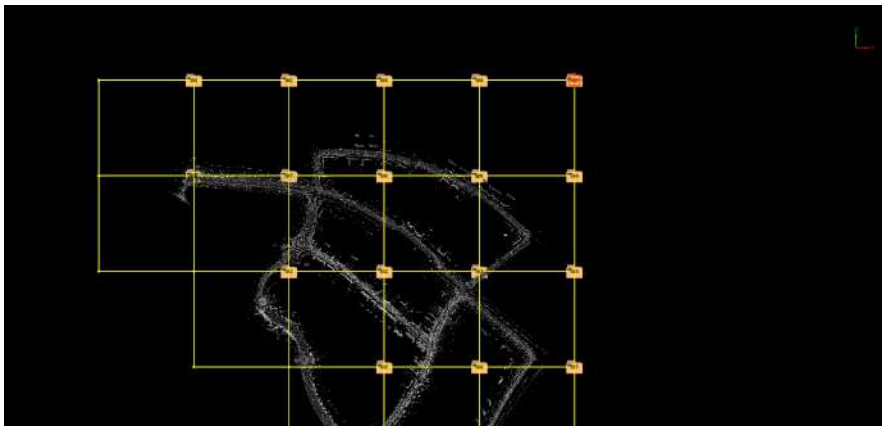
  

  After editing:

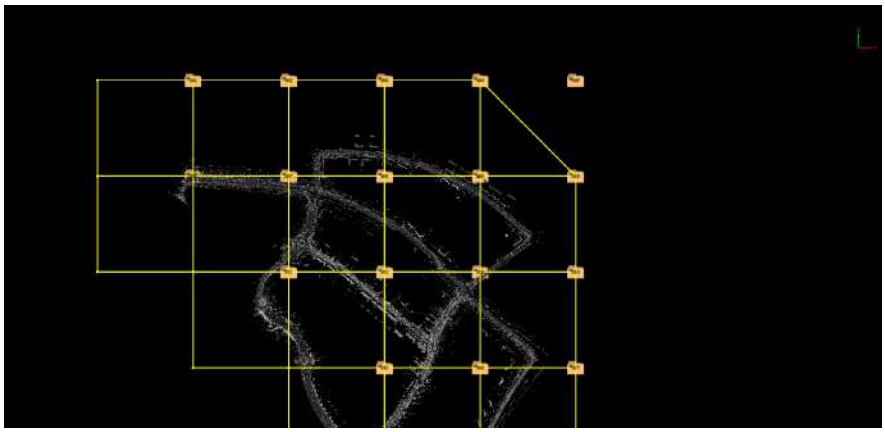
  

- **Remove Block Node** : Click  button, move the mouse to the node of the frame. When the color of the mouse changes to red, it means that the current node is captured, and the left mouse button can be held down. As shown below,

  Before editing:

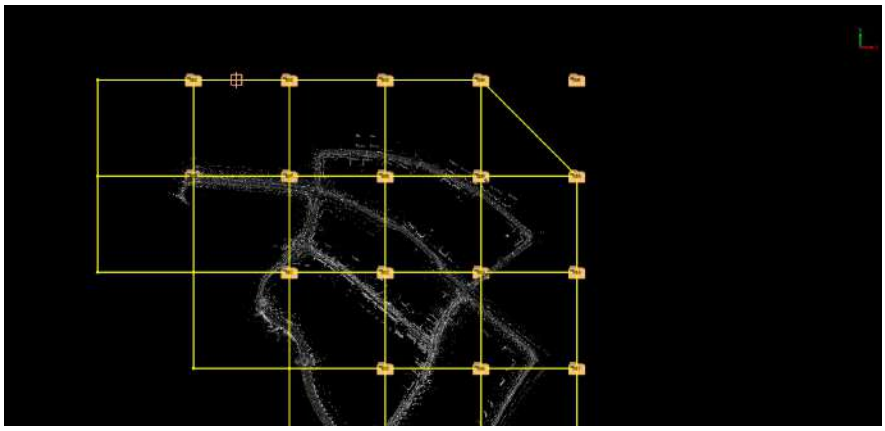


After removing the node:

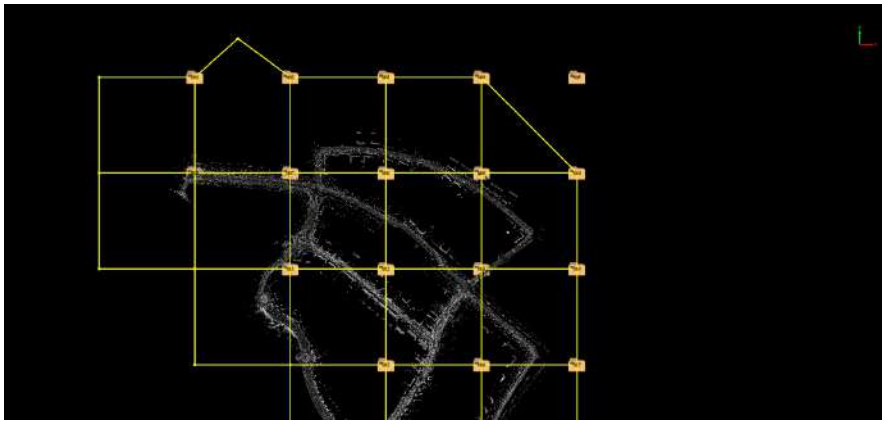


- **Insert Block Node** : Click the  button Then, move the mouse to the position where the node needs to be inserted in the frame, and click the left button of the mouse to insert a node at this position. As shown below,

  Before inserting a node:

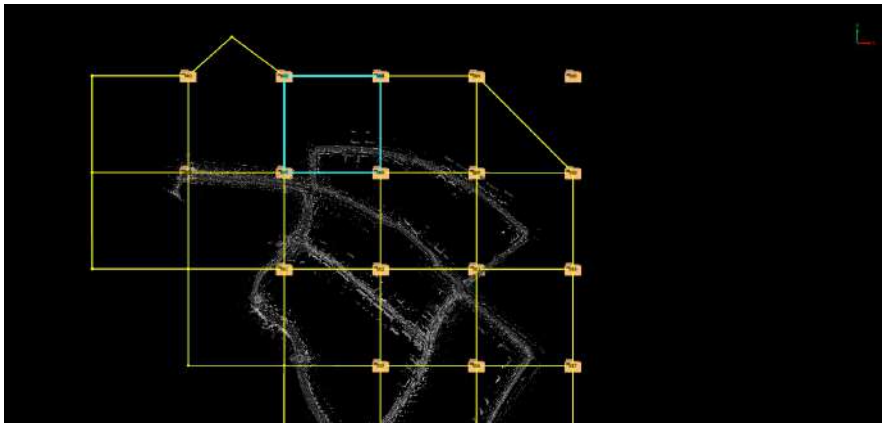


After inserting the node and moving it:

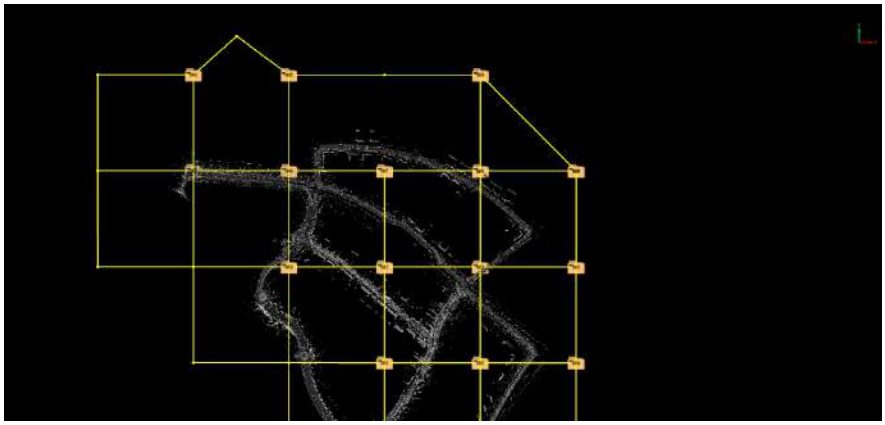


- **Merge Blocks**: Click  button, move the mouse to the label of the frame. When the color of the mouse turns red, it means that the current area block is captured. Press and hold the left mouse button to select the current area block, and then move the mouse to the label of another adjacent area block. Click the left mouse button to merge the two selected area blocks. As shown below,

  Before the region blocks are merged:



  After the region blocks are merged:

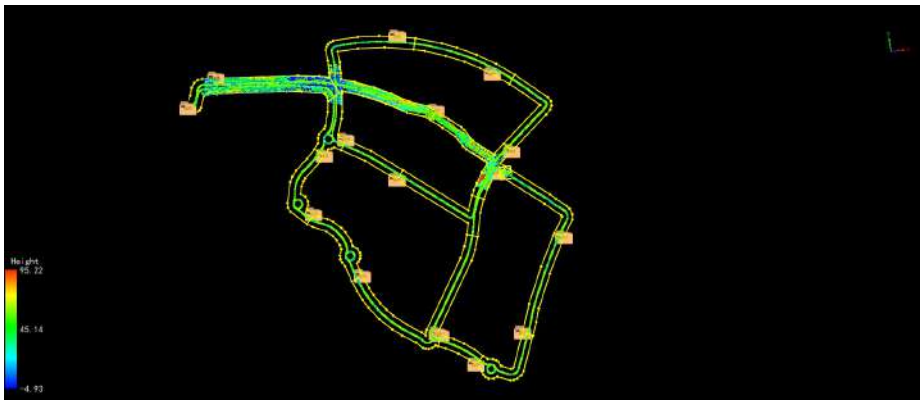


Split Point Cloud

Function Description: This function is used to segment the point cloud according to the framing frame after the trajectory framing is used.

Steps

1.Click **Split Point Cloud** , which will directly segment the point cloud.



2.If the point cloud has been segmented, a dialog box will appear:

3.Click the **Yes** button to re-segment the point cloud, and click **No** to cancel the point cloud segmentation.

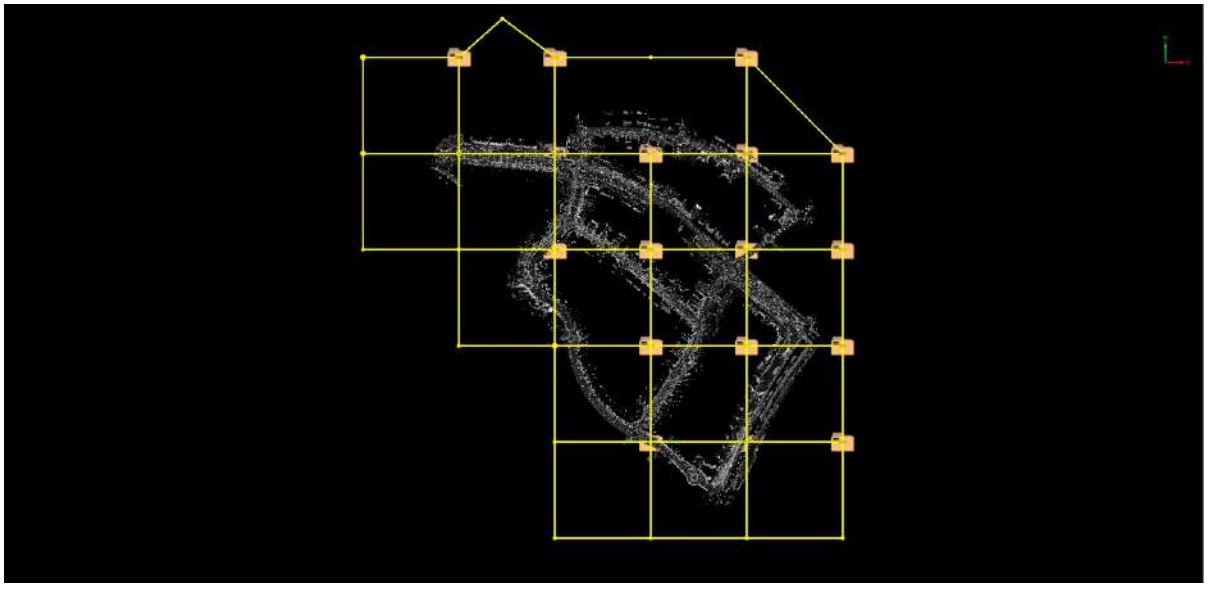
Cut Block Display

Function Description: This function selects, highlights, jumps, shows and hides each area block after framing, which is convenient for users to adjust and browse the area blocks after framing.

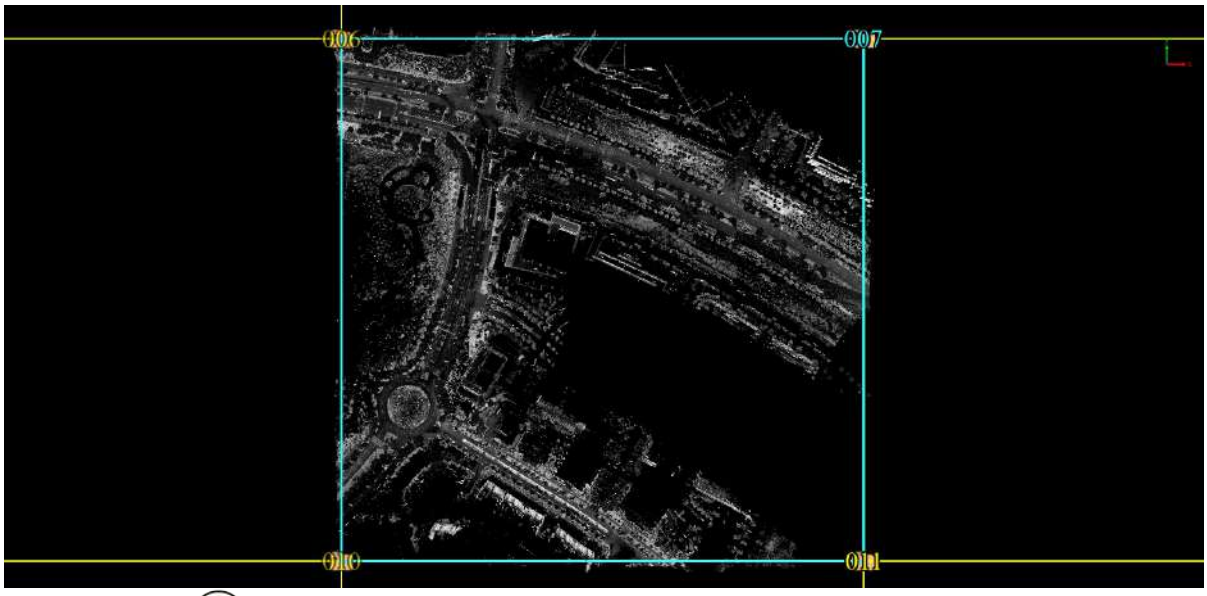
Steps

1.After the point cloud segmentation is completed, in the area block display, click **Select Frame** , and then click the label of the block area with the mouse, the software will automatically hide the point cloud of all block areas except the currently selected frame, and center the selected area block to enlarge it.

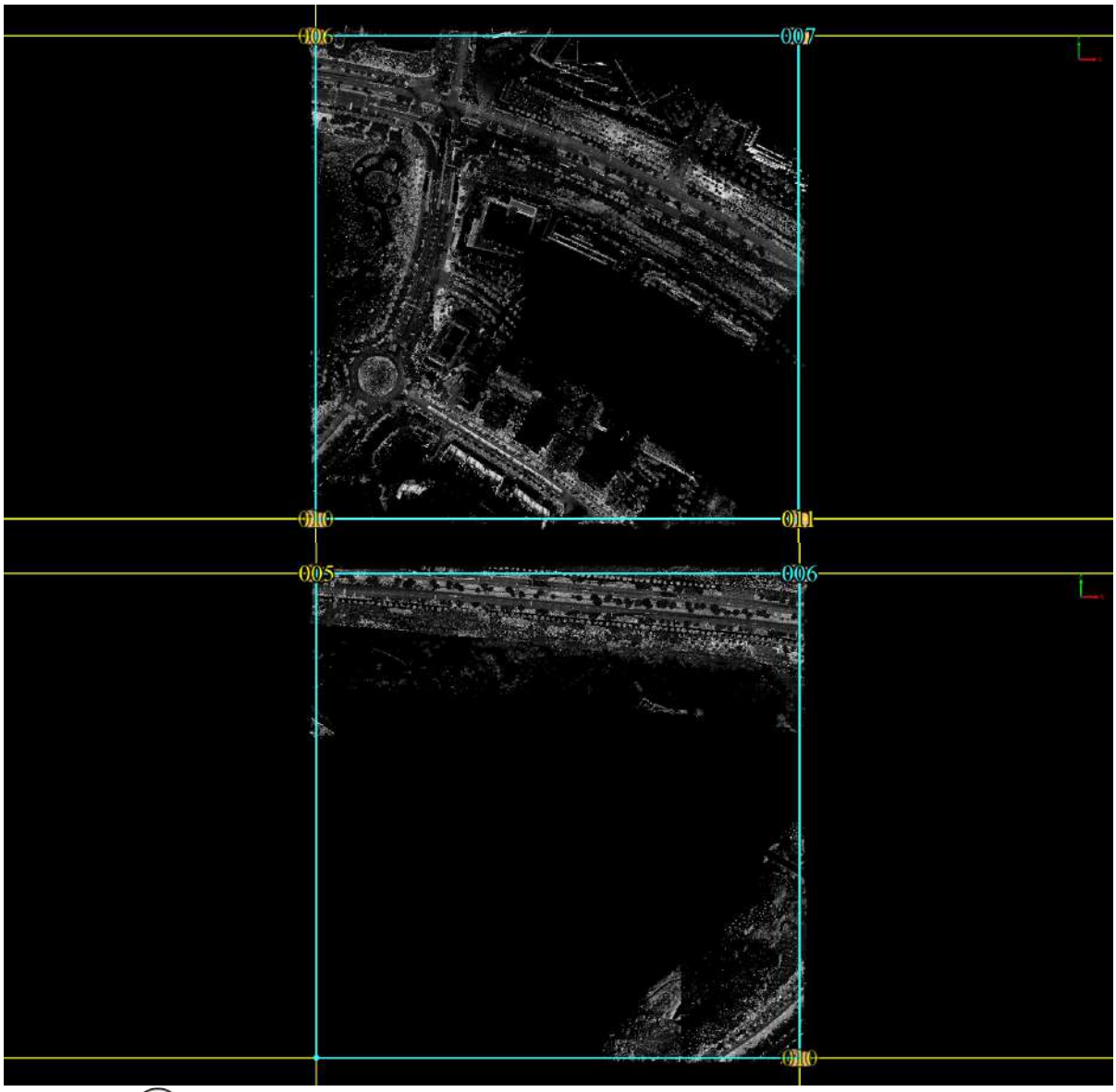
Before frame selection:



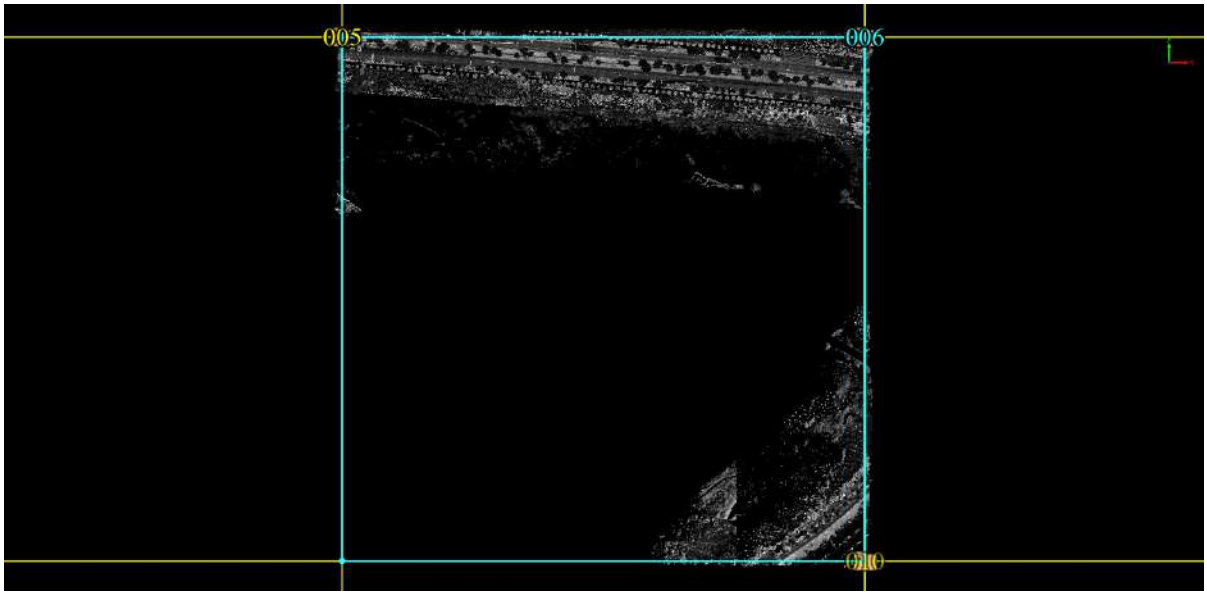
After frame selection:

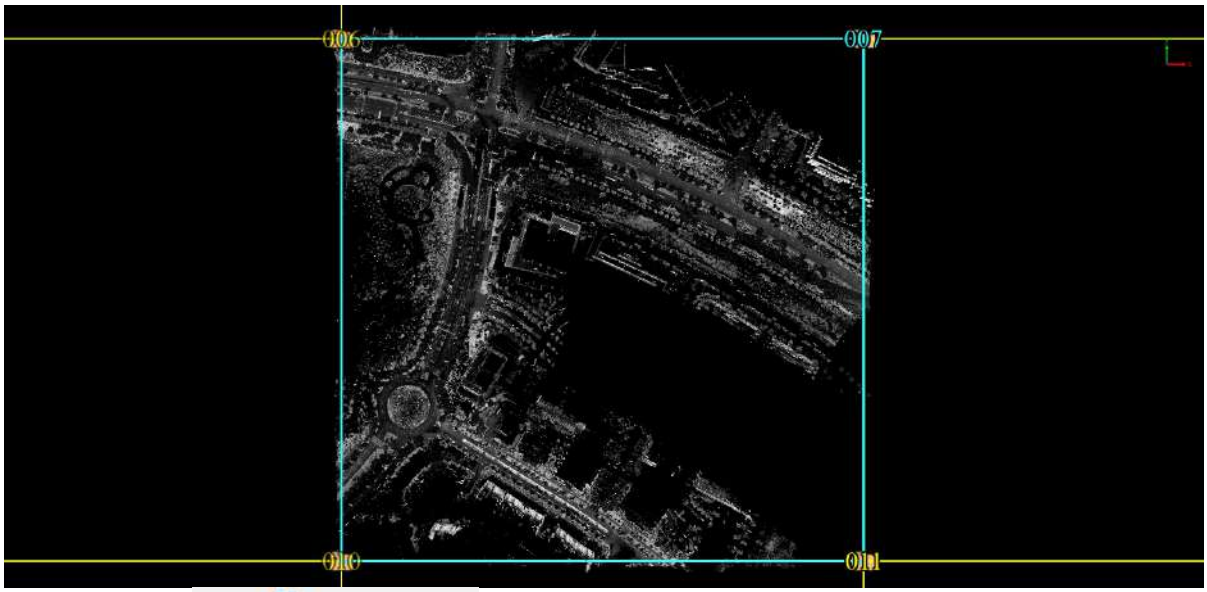


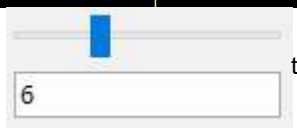
2.Click **Previous** to jump to the previous area block of the current area block number.



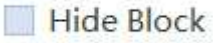
3.Click **Next**  to jump Go to the next area block of the current area block number.





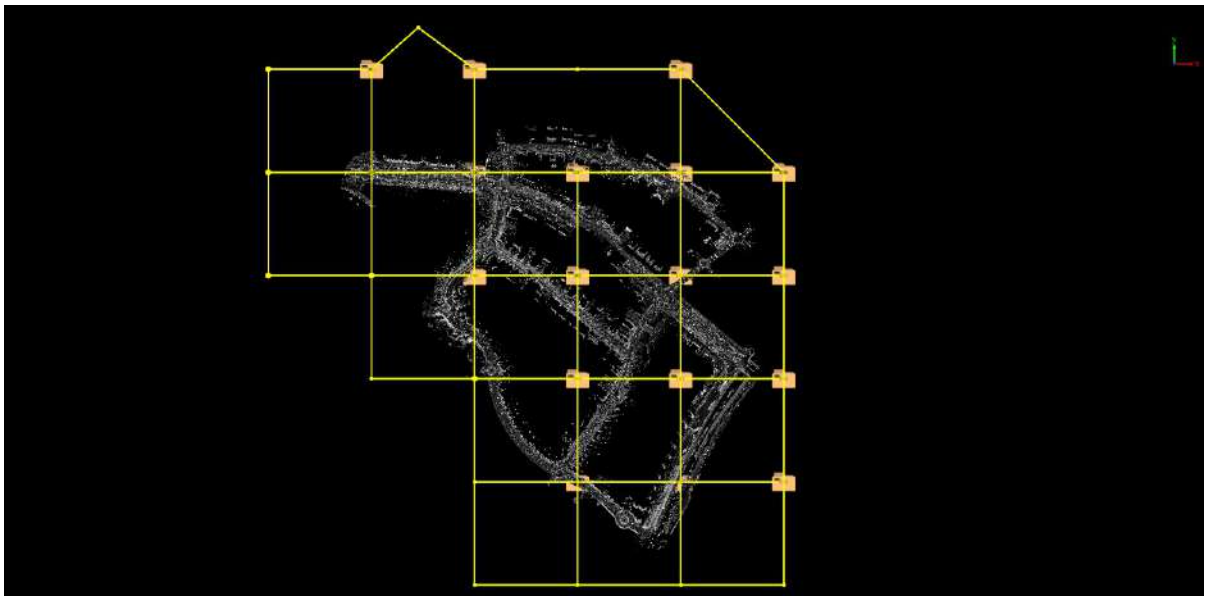
4.Drag the **slider** 

to switch different area blocks to select and highlight;

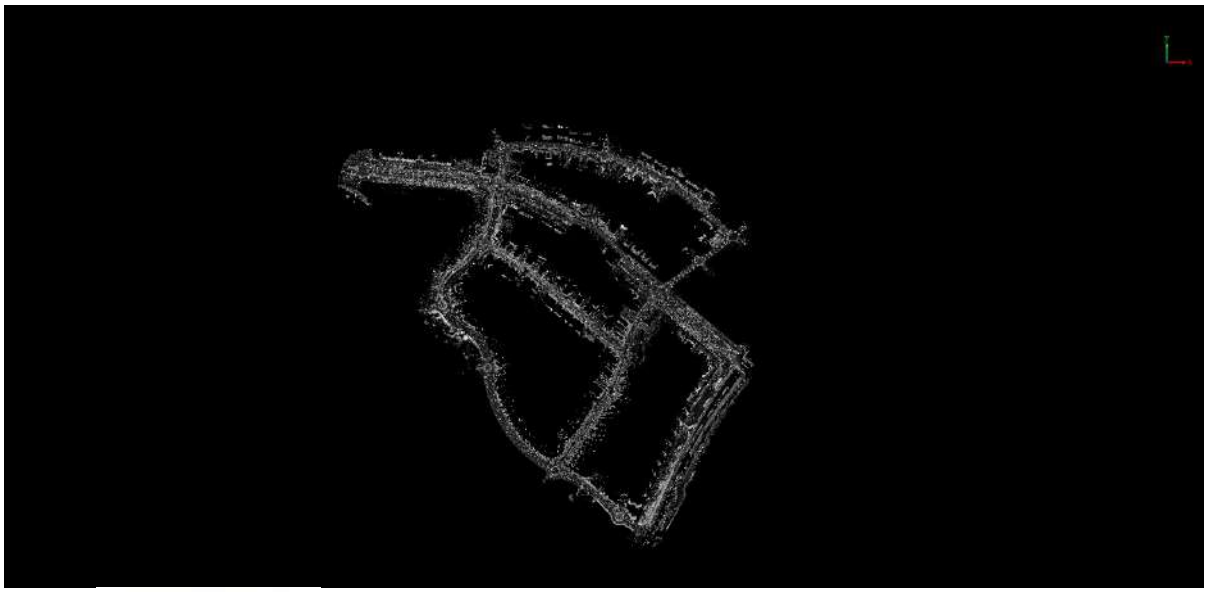
5.Select 

in the menu bar to display or hide area blocks and labels;

Display area block:

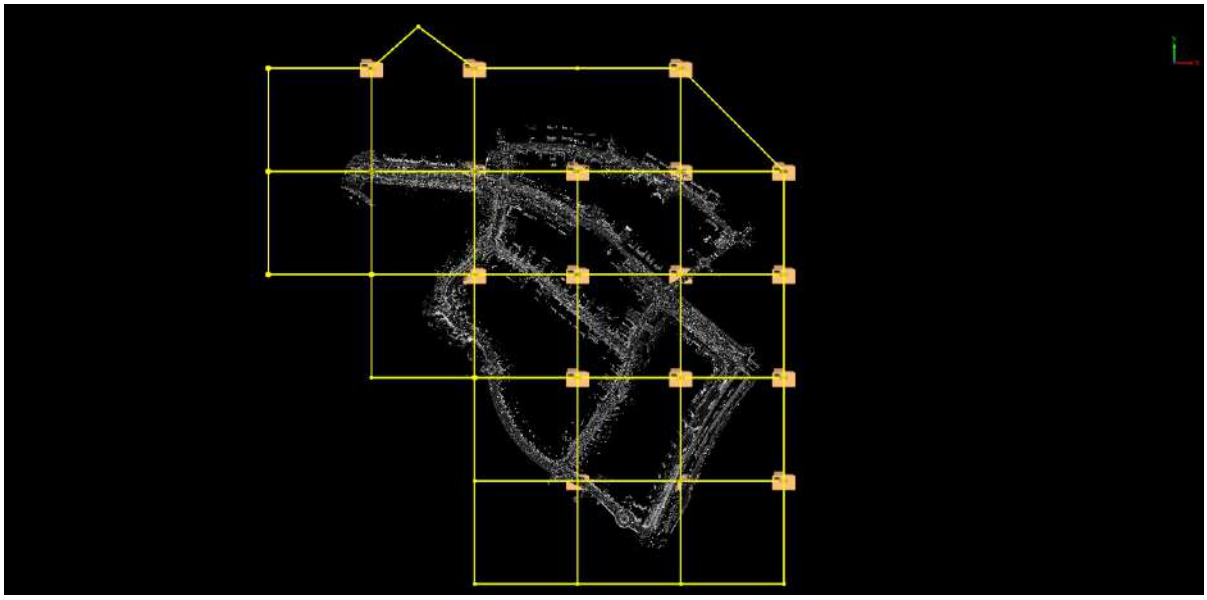


Hidden area block:

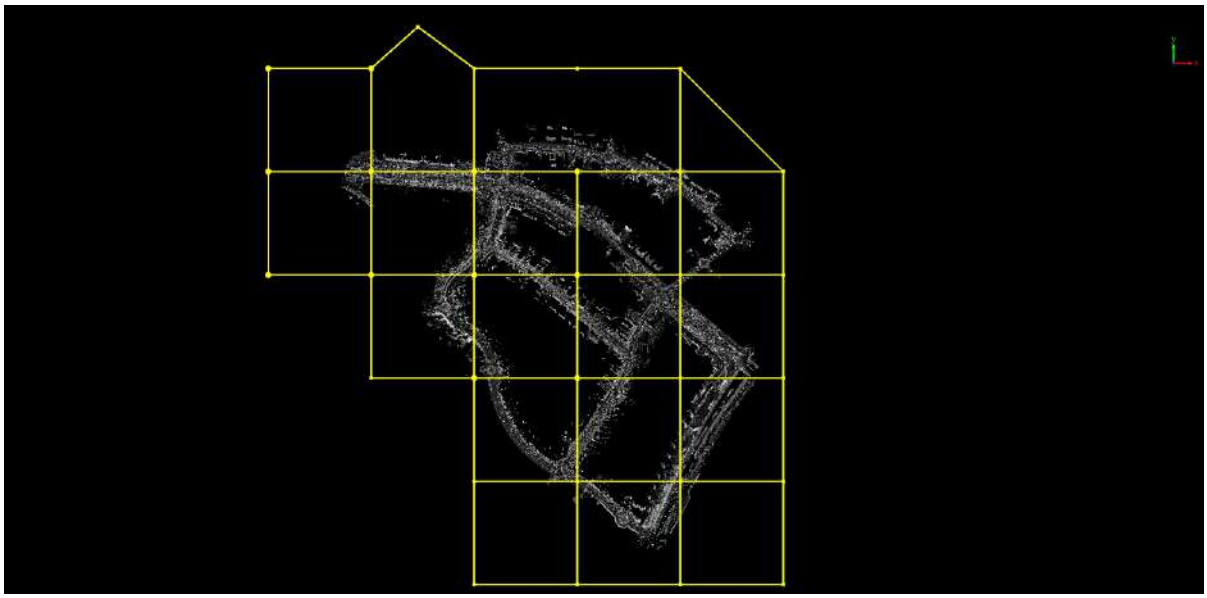


6.Select ☐ Hide Label in the menu bar to display or hide the corresponding label of the area.

Show labels:



Hide labels:



Map Elements

The map element module mainly uses manual drawing or semi-automatic detection to carry out rapid extraction of road surface elements. It is primarily used to generate road markings such as lane lines, arrows, and stop lines on the **road surface**, as well as the vectorized production of **road attachments** such as poles, power lines and other facilities.

Mainly includes：

- Road Shape
- Road Surface

Road Shape

The road shape module is mainly used to generate vectorized results related to **road shape**.

The **vectorized result generation methods** can be divided into **semi-automatic** and **manual**.

Manual Function

- Add Virtual Lane Centerline
- Add Lane SolidLine
- Add Lane DashLine
- Add Stopline
- Add Road Side Line

Semi-Automatic Functions

- Solid Lane
- Dash Lane
- Detect Road
- Generate Virtual Lane Centerline

Add Virtual Lane Centerline

Function Description： The Add Virtual Lane Centerline is a virtual line located at the center of the two lane lines. The software uses a dotted line to express the lane center line. This tool can be used to manually select the nodes of the line to generate the center line of the lane.

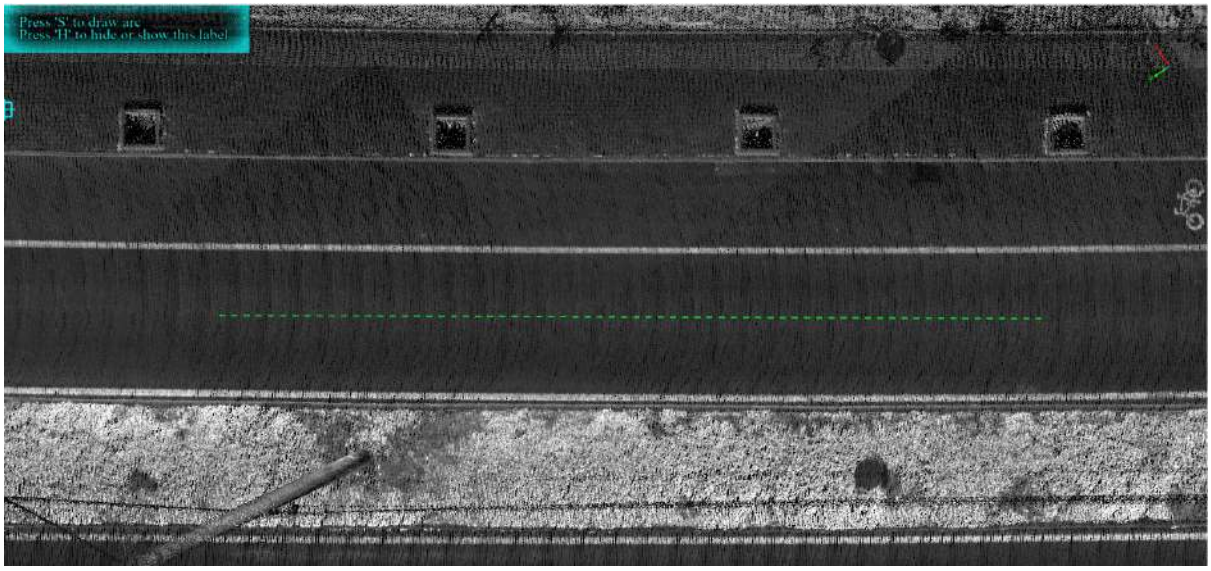
Steps

1.Click the **Add Virtual Lane Centerline** button in the Road Shape panel.

2.Left click to select the starting point of the line in the point cloud.

3.Left click the selected line node until the end point, and quickly double-click the left mouse button to determine a lane center line.

> - When the point cloud is outside the current window, the user can drag and drop the point cloud by pressing and holding down the right mouse button.
> - **Hold down the left mouse button** to adjust the viewing angle.



Drawing lane center line

Note:

> - In the process of clicking on the node, you can use the mouse **wheel and right button** to respectively zoom and move the point cloud.
> - This function is the same Add Lane SolidLine function with the right mouse button and shortcut key.

Add Lane SolidLine

Function Description： This tool is used to manually select the nodes of the line, multi-point or double-point to generate the Lane Solidline.

Steps

1.In the Road Shape panel, click the drop-down mark on the right side of the lane line, and select the **Add Lane SolidLine** button.

2.Left click to select the starting point of the lane line.

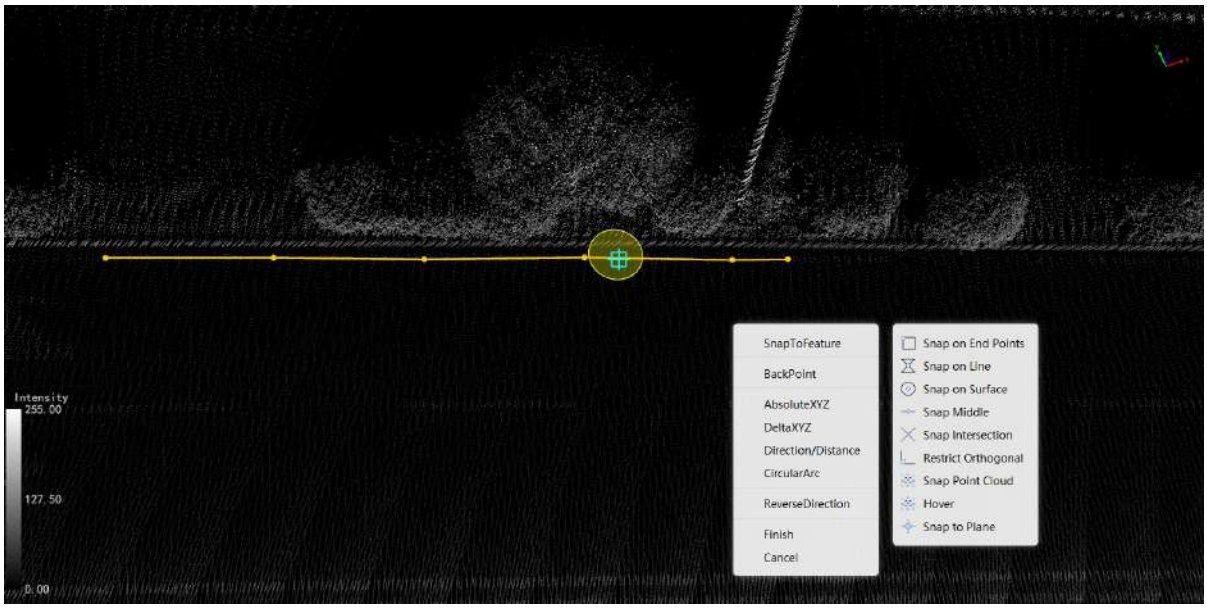
3.Move the mouse, and **left click** in turn to select the node of the Lane Solidline.

> - When the point cloud is outside the current window, the user can drag the point cloud by **pressing and holding down the right mouse button**.
> - **Press and hold the left mouse button** to adjust the viewing angle.

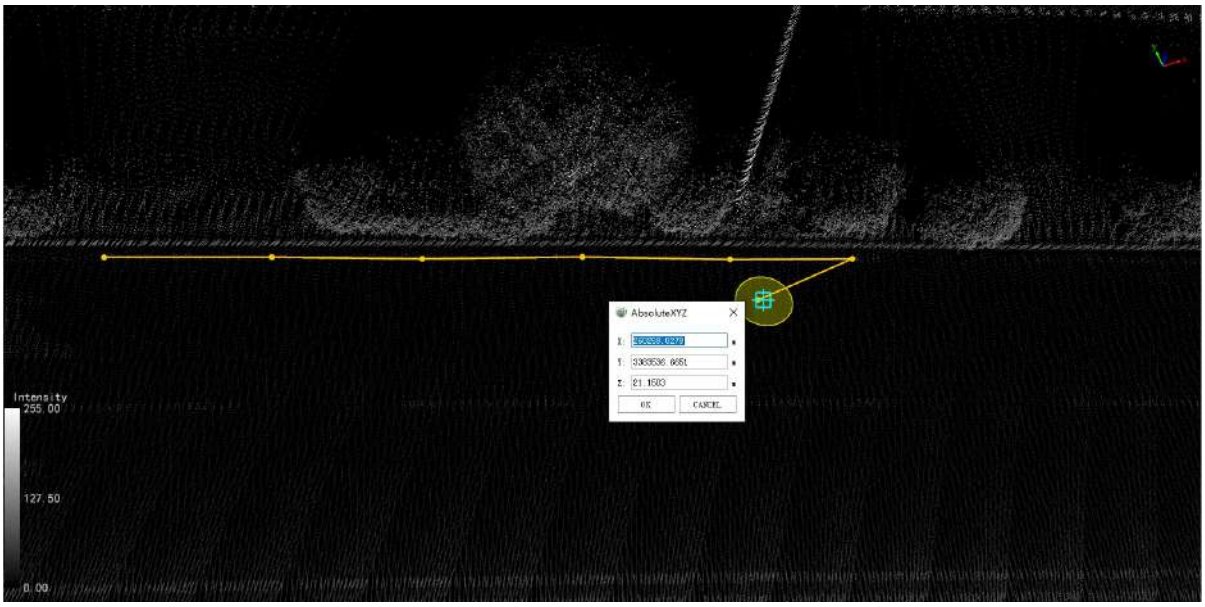
4.Switch between different drawing modes through shortcut keys during drawing.

Function	Shortcut key	Description
Line, Polygon	S/s	Short press the “s” key to switch the drawing mode from drawing a straight line to drawing an arc. The arc adopts a three-point mode. The first point is the last point before short pressing the “s” key, the second point is the end point of the arc, and the third point controls the radian
Line, Polygon	B/b	Short press “b” to back up the drawn node

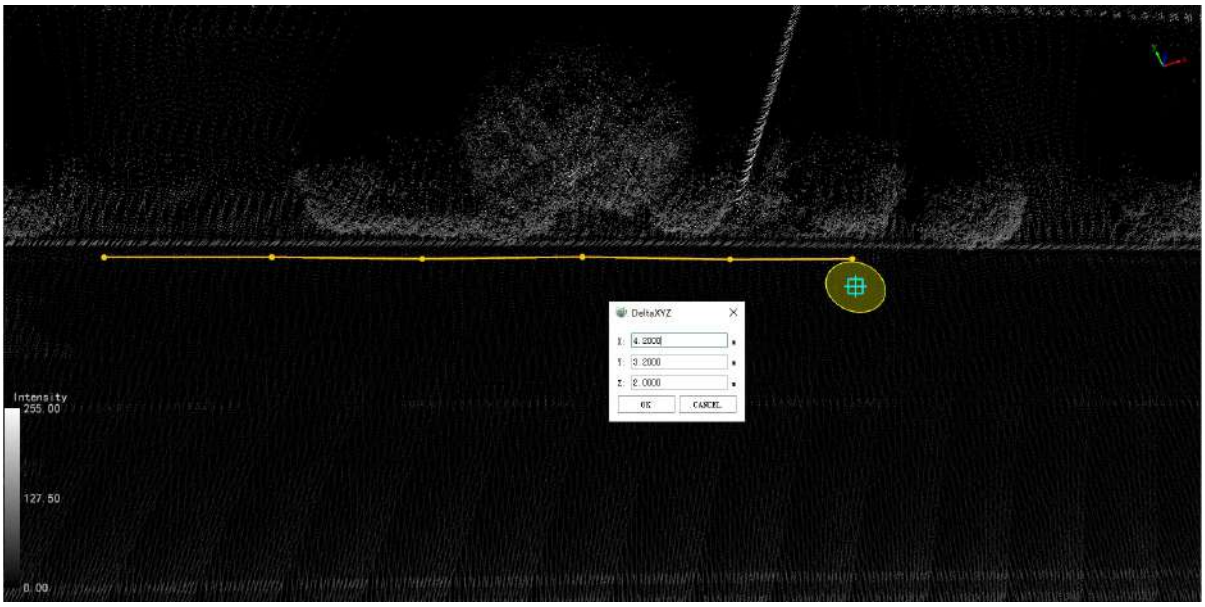
5.The right mouse button contains the following operations：



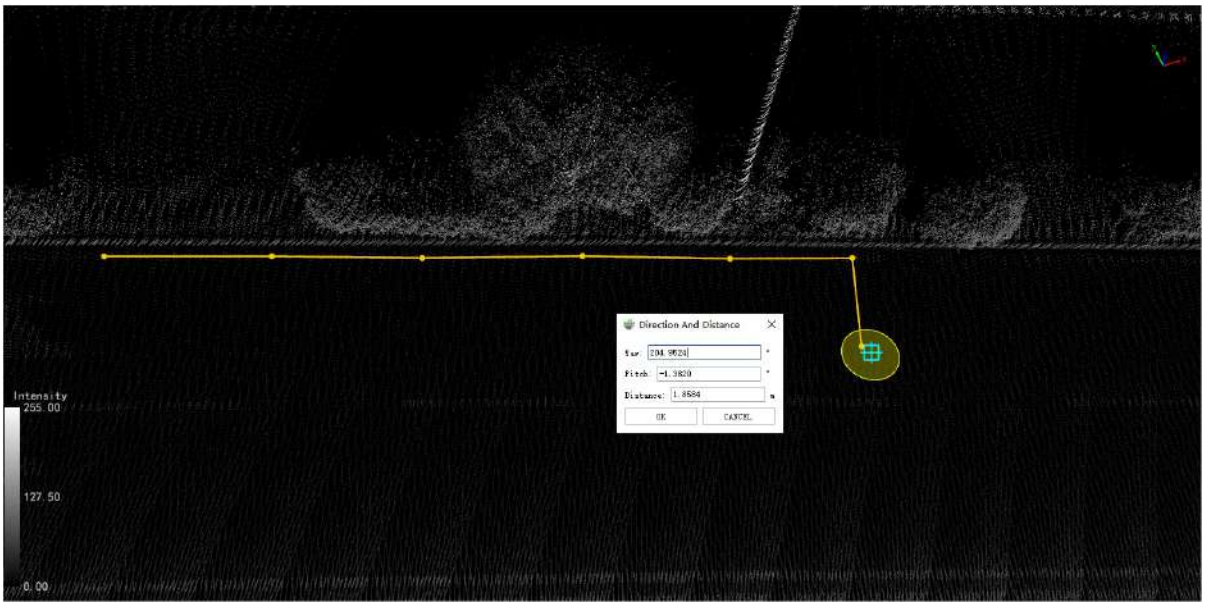
- SnapToFeature: Various snap modes can be switched at any time in the SnapToFeature menu.
- BackPoint: Click the BackPoint to back out nodes in order. (The first node cannot be rolled back.)
- AbsoluteXYZ: With the AbsoluteXYZ function, the user can specify the AbsoluteXYZ coordinates during the drawing process. The coordinate value entered in the pop-up box is the position drawn by the node.



- DeltaXYZ: With the DeltaXYZ function, the user can specify the coordinate position of the next node relative to the previous node in the drawing process. The coordinate value entered in the pop-up box is the position relationship between the next node and the previous node.



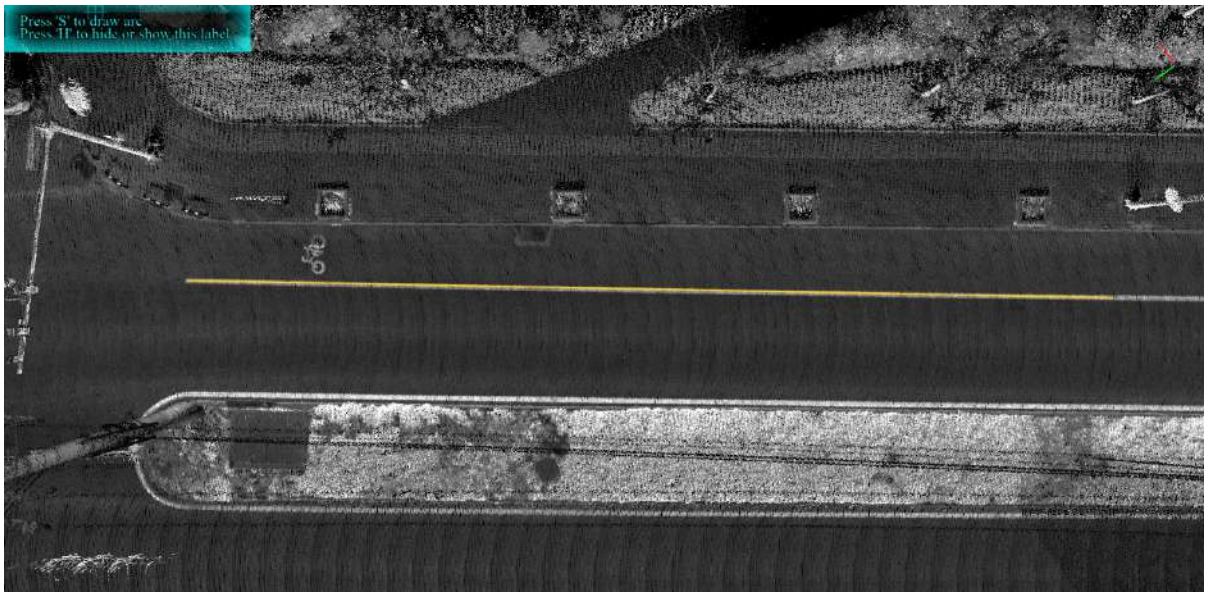
- Direction And Distance: With the Direction And Distance function, the user can specify the direction and distance of the next node relative to the previous node in the drawing process. The deflection angle is the included angle with the Y axis of the coordinate system, the pitch angle is the included angle with the XY plane of the coordinate system, and the distance is the distance between the next point and the next point in the above angle direction. In the section window, the deflection angle does not need to be modified.



- Circular Arc: With the circular arc function, an arc with specified direction, size and position can be generated during drawing. As shown in the figure below, ArcDirection refers to the angle of the arc relative to the Y axis of the coordinate system. ArcPitch refers to the direction relative to the XY plane of the coordinate axis. ArcRoll refers to the rotation angle around the XY plane of the coordinate axis; Radius refers to the length of the two ends of the generated arc. ArcLength refers to the side length of the generated arc. Side selection is controlled to the left or right of the current plane position. (Note: the section window is not applicable to arc calculation, and the generated arc will not be displayed completely in the section window)

- ReverseDirection: During line drawing, the reverse function can reverse the direction of the drawn vector line, and continue to draw based on the new direction.
- Finish: In the drawing process, the user can double-click the last node with the left mouse button to finish the drawing, or the user can click the “Finish” button with the right mouse button to finish the drawing of the current object.
- Cancel: In the drawing process, the user can click “Cancel” with the right mouse button to exit the current drawing.

6.**Double click the left** button quickly to determine the end point of the line and generate a lane line.



Lane SolidLine

> Note: In the process of clicking the node, the user can use the mouse **wheel, right button** to respectively zoom and move the point cloud.

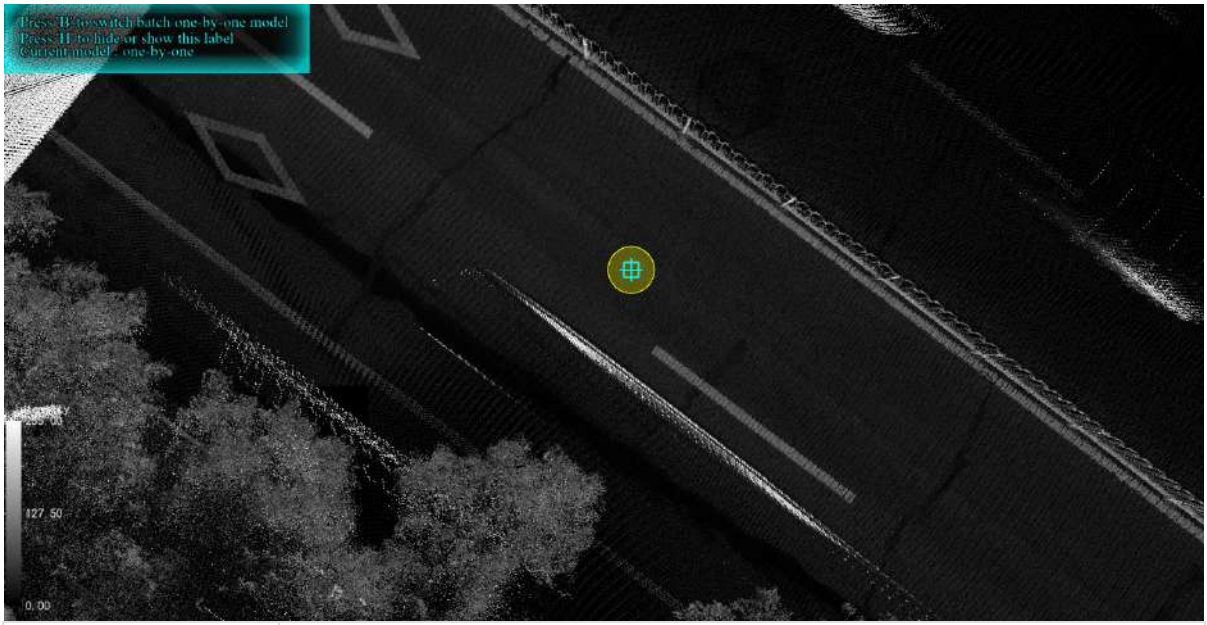
Add Lane DashLine

Function Description： This tool is used to manually select the nodes of the line, and the three key points can quickly generate the dashed line of the lane.

Steps

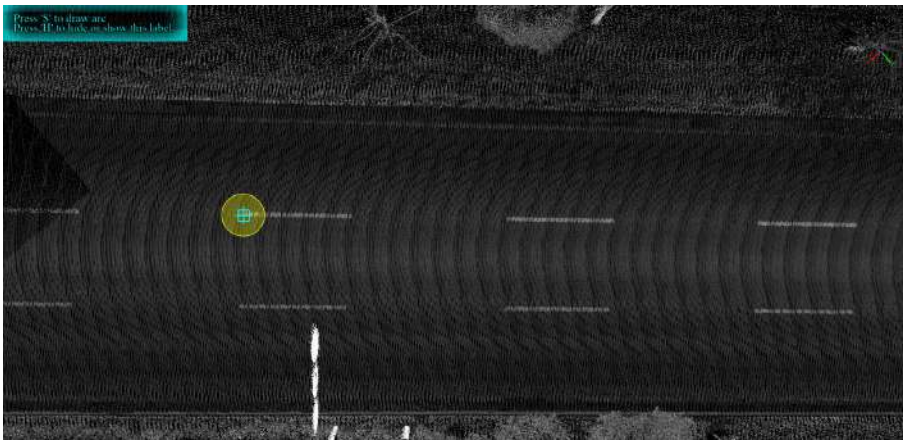
1.In the Road Shape panel, click the drop-down mark on the right side of the lane line, and select the **Add Lane DashLine** button.

2.3D window tips：



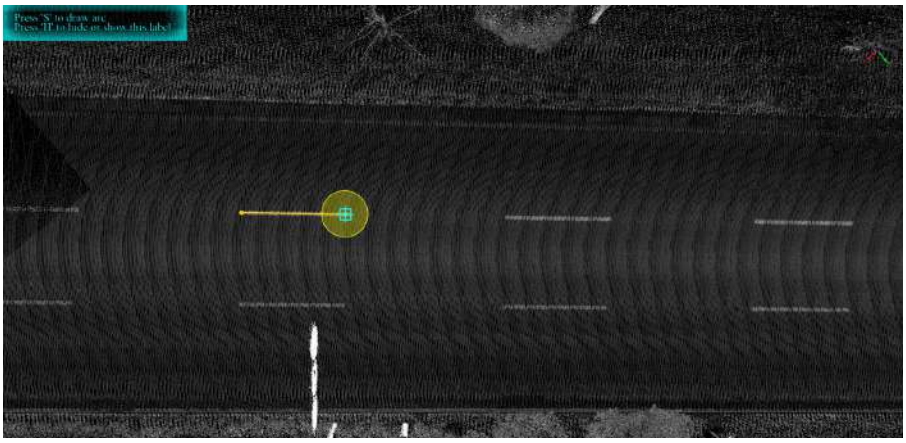
Shortcut name	Function
B	Toggles drawing mode (batch drawing/one by one)
H	Turn off/on the top left corner prompt Right mouse button
Current model	Current model
Right mouse button	Retracts a dashed line and clears the preview vector if in batch drawing mode

3.The initial drawing method is "one by one". Click the left mouse button to select the start point as **the first key point, determine the starting point of the dashed line**.



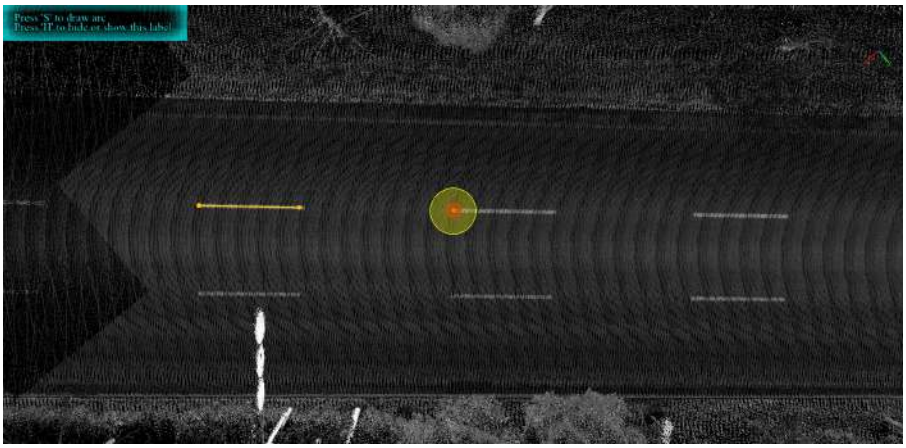
The first key point

4.Move the mouse, **left click the second key point**, the details are shown in the figure, **used to determine the length of a single dashed line**.



The second key point

5.Move the mouse, **left click the third key point**, the details are shown in the figure, **used to determine the length between the dashed segments**.

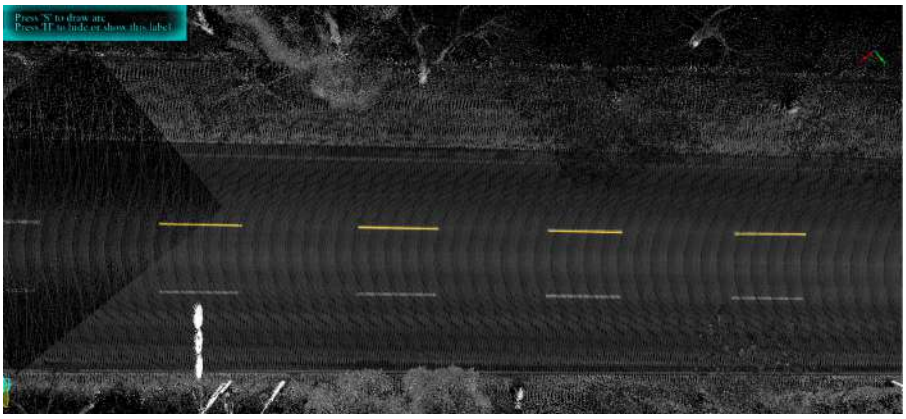


The third key point

6.**Move the mouse to the position to end**, it will automatically generate a dashed line segment based on the previously determined **single dashed line segment length and the length between the dashed line segments** two lengths, **double-click the left mouse button** to finish.

> - When the point cloud is outside the current window, the user can drag the point cloud by **pressing**

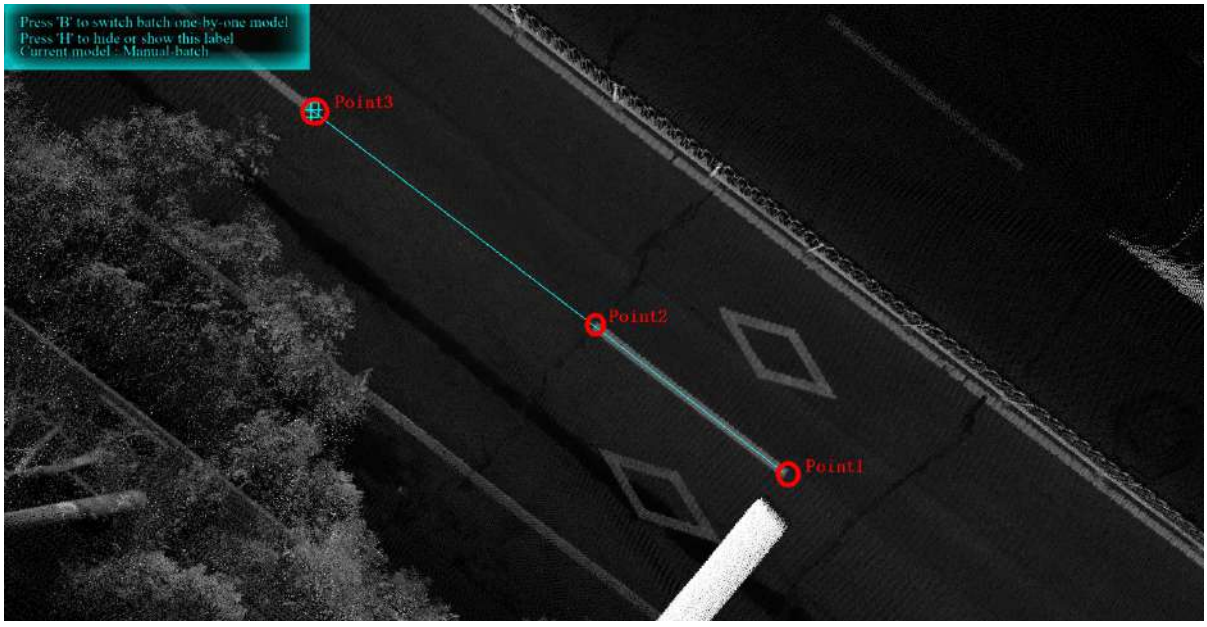
> **and holding down the right mouse button**.
- **Press and hold the left mouse button** to adjust the viewing angle.



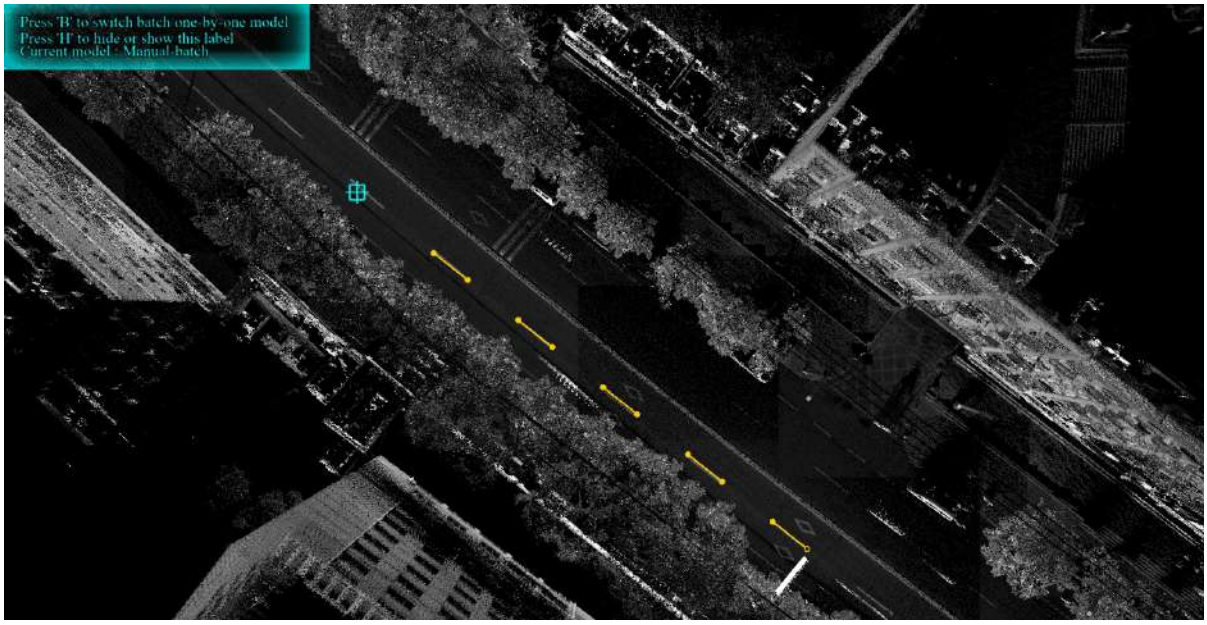
Drawing lane dashed line

7.The shortcut key **B** can be used to switch to "Batch" mode during the default "Section-by-section" mode or when the function is first activated. In this mode, you need to click on three key points in sequence, the two endpoints of a lane dashed line and the endpoint of another lane dashed line adjacent to the second key point, to determine the length and spacing of the lane dashed line, then simply drag the mouse to pull out the live vector along the extension of the lane dashed line and double click the mouse to finish drawing when the live vector matches the lane dashed line point cloud.

> Note: This mode applies to the drawing of lane dashes when the road is in a straight shape.



Key points for drawing lane dashed lines (batch mode)



Real-time vector for drawing lane dashed lines (batch mode)

> Note:
>
> - In the process of clicking on a node, the user can use the mouse **wheel and right click** to zoom and move the point cloud separately.
> - Right mouse button operation and shortcut keys for this function are the same as for the Lane Solidline function.

Add Stopline

Function Description： The lane stop line is located where the lane line enters the intersection. This tool is used to manually click the nodes of the line to generate the lane stop line.

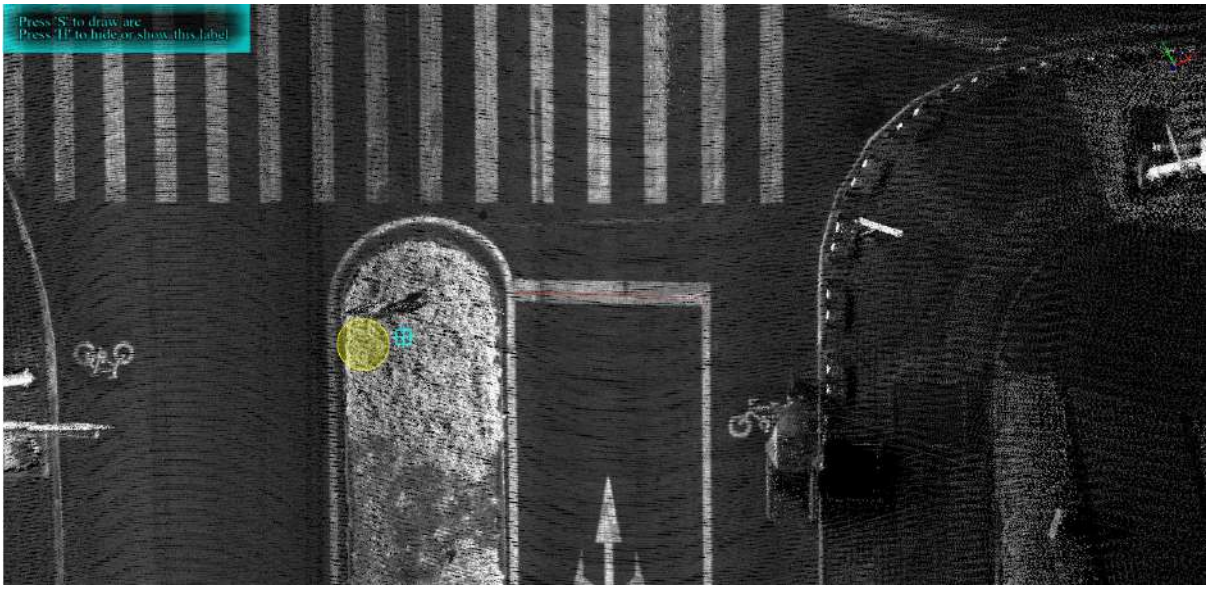
Steps

1.Click the **Add Stopline**  button in the Road Shape panel.

2.Mouse **left button click** to select the starting point of the line in the point cloud.

3.Move the mouse, **click the left mouse button** to select the line node, until the end point, quickly double-click the left mouse button to confirm a lane stop line.

> - When the point cloud is outside the current window, the user can drag the point cloud by **pressing and holding down the right mouse button**.



Drawing stop line

> Note:
>
> - In the process of clicking on a node, the user can use the mouse **wheel and right button** to zoom and move the point cloud separately.
> - The right mouse button operation and shortcut button of this function are the same as the Lane Solidline function.

Add Road Side Line

Function Description： The edge of the road is located at the physical boundary of the road surface. This function is used to manually click the nodes of the line to generate road edges.

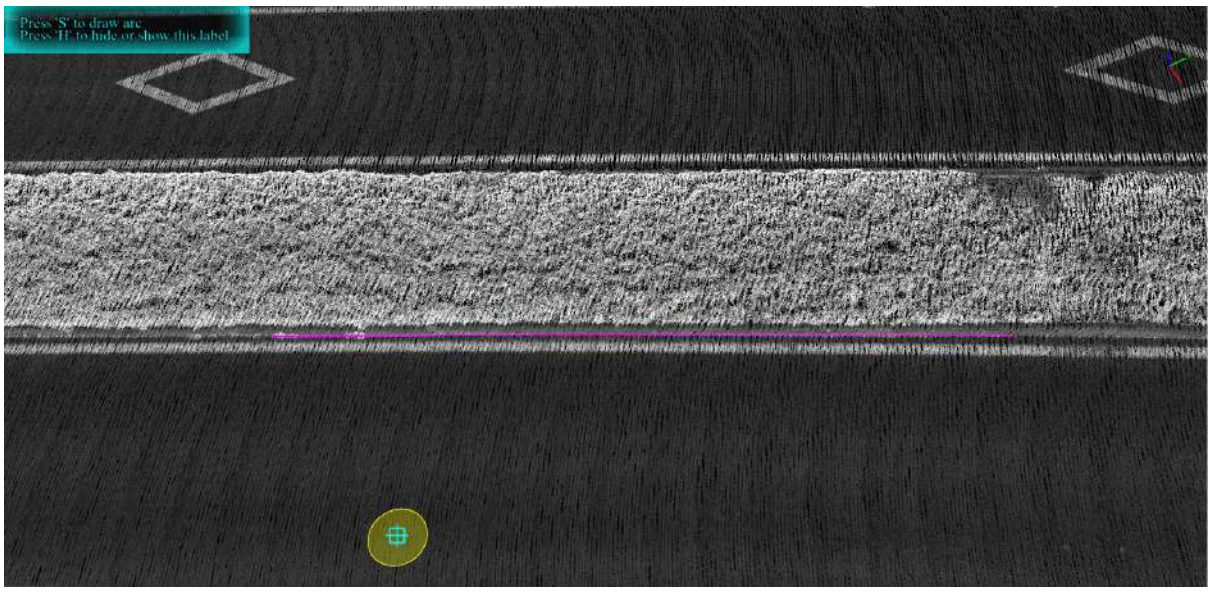
Steps

1.Click the **Add Road Side Line** button in the Road Shape panel.

2.**Left-click** to select the starting point of the line.

3.Move the mouse, **Left-click** the selected line node until the end point, and quickly double click the left mouse button to confirm a road side line.

> - When the point cloud is outside the current window, the user can drag and drop the point cloud by **pressing and holding the right mouse button**.
> - **Press and hold the left mouse button** to adjust the viewing angle.



Drawing road side line

> Note:
>
> - In the process of clicking on the node, the user can use the mouse **wheel and right button** to respectively zoom and move the point cloud.
> - The right mouse button operation and shortcut button of this function are the same as the Lane Solidline function.

Generate Virtual Lane Centerline

Function Description： This tool can be used to automatically generate the centerline of the lane by clicking the two lane lines with the left mouse button on the premise that there are two lane lines.

Steps

1.Click the **Generate Virtual Lane Centerline**  button in the Road Shape panel.

2.**Click with the left mouse button** and click **adjacent** the left lane line and the right lane line in turn to automatically generate the lane center line.

> - When the point cloud is outside the current window, the user can drag the point cloud by **pressing and holding the right mouse button**.
> - **Press and hold the left mouse button** to adjust the viewing angle.

Before generating

After generating

> Note: In the process of clicking, the user can use the mouse **wheel, right button, and left button** to respectively zoom and move the point cloud, adjust the angle of view.

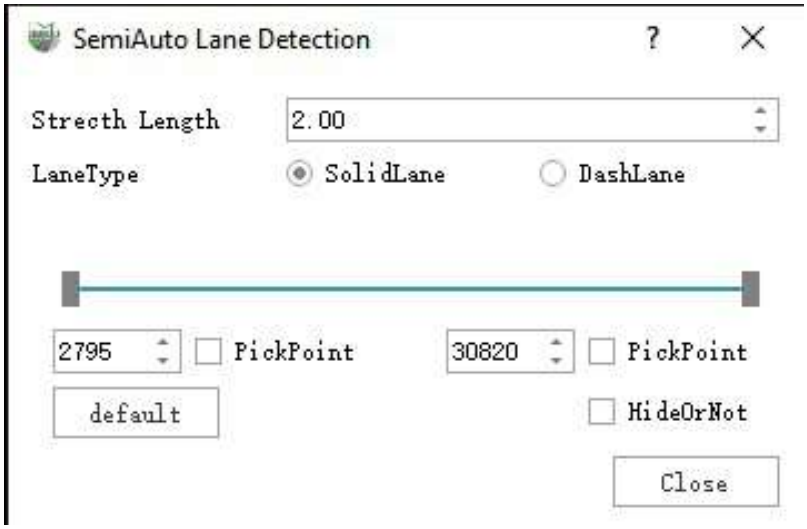
Detect Lane

Function Description: This tool enables the detection of lanes using points. Left click to select two points to obtain the starting point and direction point. Along the selected direction, automatic roadside detection will be carried out from the starting point.

> Note: This function is only effective for the area where lane lines and road surface have high recognition in point cloud intensity.

1.Click **Detect Lane** button in Road Shape panel.

2.The line **Detection Settings dialog box** should pop up:



Solid Lane dialog

Parameter Description:

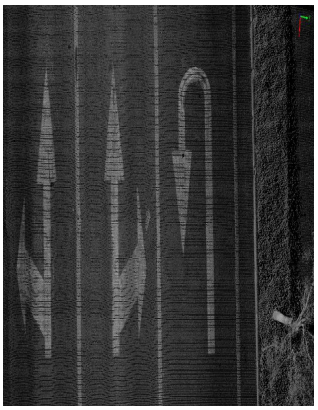
- **Maximum Angle**：The default is 20, which means the deviation angle between the current segment and the previous segment. If this angle is exceeded in a local area, the growth will stop. **Maximum Angle is suggested below 30 degrees**.
- **Maximum Diff Intensity**：The default value is 0.5, which indicates the tolerance value of the intensity difference between the front and rear segments of the solid lane. If the intensity difference is greater than the set value, the growth will stop. **The default value is recommended**.
- **Stretch Length**：The default value is 2 meters. If there is a certain defect in the middle of the line and it can be extended without stopping within the set distance, **the default value is recommended and can be adjusted according to the situation**.
- **LaneType**: The default type is SolidLane, if not, the user can **selects SolidLane**.
- **Intensity Adjustment**: For stretching the intensity by manually selecting the maximum and minimum values where the strength is not easily distinguished.
- **PickPoint**：Sets the minimum and maximum intensity and can be determined by clicking the point cloud with the left mouse button after checking the selection box. The left is the minimum intensity and the right is the maximum intensity.

> The scroll bar, value box and point selection box are linked together, and the intensity can be adjusted in any operation mode. Meanwhile, the point cloud display will also be linked in real time (intensity stretching of the whole point cloud).

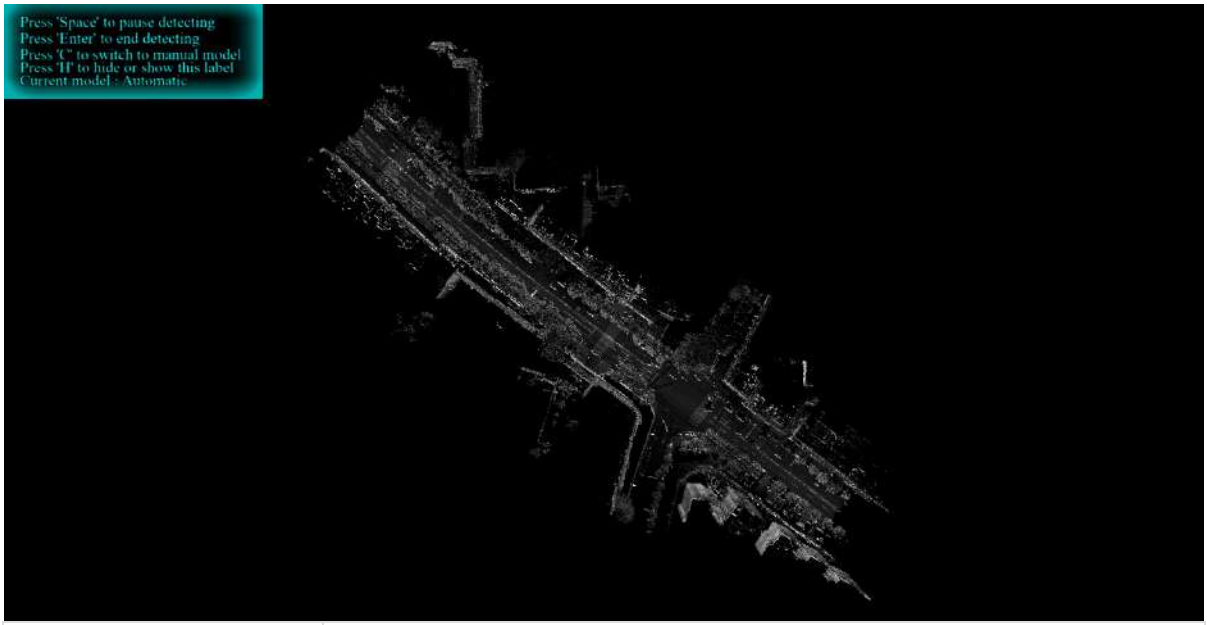
- **Default**：Restores the original maximum and minimum intensity, and the point cloud display will also be linked in real time.
- **HideOrNot**：After adjusting the intensity through the scroll bar, value box, and check box, the point cloud display will be updated accordingly.

> Unchecked: For intensity stretching of the whole point cloud, points below the minimum value are displayed as black, and points above the maximum value are displayed as white.
>
> Checked: The intensity of the whole point cloud is stretched, and the points below the minimum value and above the maximum value are not displayed.

Intensity adjustment effect

		
Raw point cloud screenshot	The minimum value is adjusted to 14, the maximum value is adjusted to 18, and the box is not selected to hide	The minimum value is adjusted to 14, the maximum value is adjusted to 18, and the box selection is hidden

3.3D window tips:



Shortcut name	Function
Space	Interrupt growth
Enter	Confirm growth result
C	Toggles between automatic/manual drawing mode
B	Toggles between batch or one-by-one plotting in manual mode
H	switch off/on the top left corner prompt
Current model	Current model
Right mouse button	Rewind node (only works after interruption, not during growth)

The lane line detection function offers two modes, a semi-automatic mode, where automatic detection is carried out in the "Start + Direction" mode, and a manual mode, which follows the manual drawing mode of the "Draw lane lines" function. Both modes can be switched between during the drawing process by means of C.

4.The initial drawing mode is "semi-automatic". **Click left mouse** on a point on the lane line point cloud as the **starting point**, and then **left mouse click** to select another point on the lane line as the **growth direction**.

5.Automatic lane detection begins from **the starting point along the selected direction**.

> During the growth process, the center of the screen is automatically adjusted to the current growing point; at the same time, the point cloud can be viewed by **angle adjusting, dragging, and zooming** through **left mouse button, right mouse button, and scroll wheel**.

6.Use shortcut keys for **real-time shape adjustment** :

- he program is automatically **interrupted** due to the reasons such as drastic changes in intensity, low curb height, actual termination, actual interruption, etc.; if the automatic detection does not grow correctly, users can use the **Space** to manually **interrupt**.
- After **interruption**, users can **right click** to go back to adjust the shape.
- After **interruption**, users 3 and 4 can be repeated, followed by automatic growing.
- If the point cloud is missing in a large area or the intensity contrast is very low due to occlusion etc. and the program cannot detect it automatically, you can switch to manual mode with **C** and then draw by manually clicking on the key points, when the manual drawing is finished, you can double click to end the current mode or switch to automatic mode with **C**.

7.When the growth is completed and the shape is acceptable, press the Enter key (Enter, confirm key) to complete the growth of the solid line of the current lane.

Solid Line

- **Click the starting point and direction point**

Click the starting point and direction point

- **Growth results**


Solid line growth results

Dashed Lane Detection

Function Description: This tool enables the detection of dashed lanes using points. Use the left mouse button to select two points to obtain the starting point and direction point along the clicked direction, and start automatic lane line detection from the starting point.

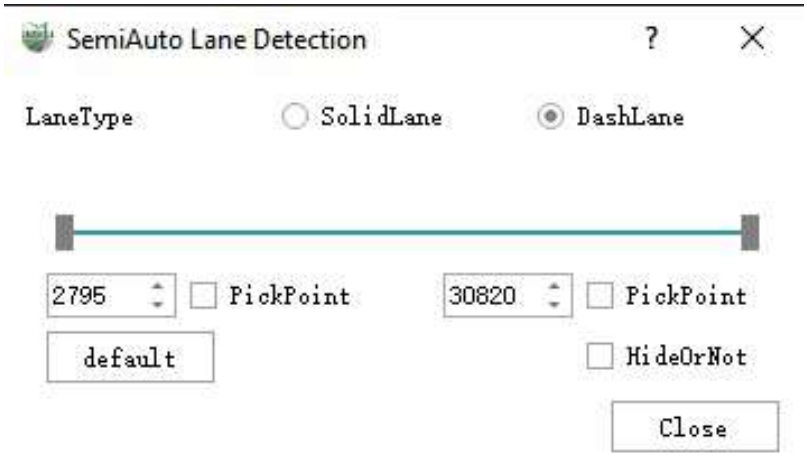
> Note: This function is only effective for areas where the lane line and road surface have high recognition in point cloud intensity.

Steps

1.Click **Detect Lane**  button in the Road Shape panel.

2.The **Detection Settings dialog box** will appear.

Dashed Lane Dialog



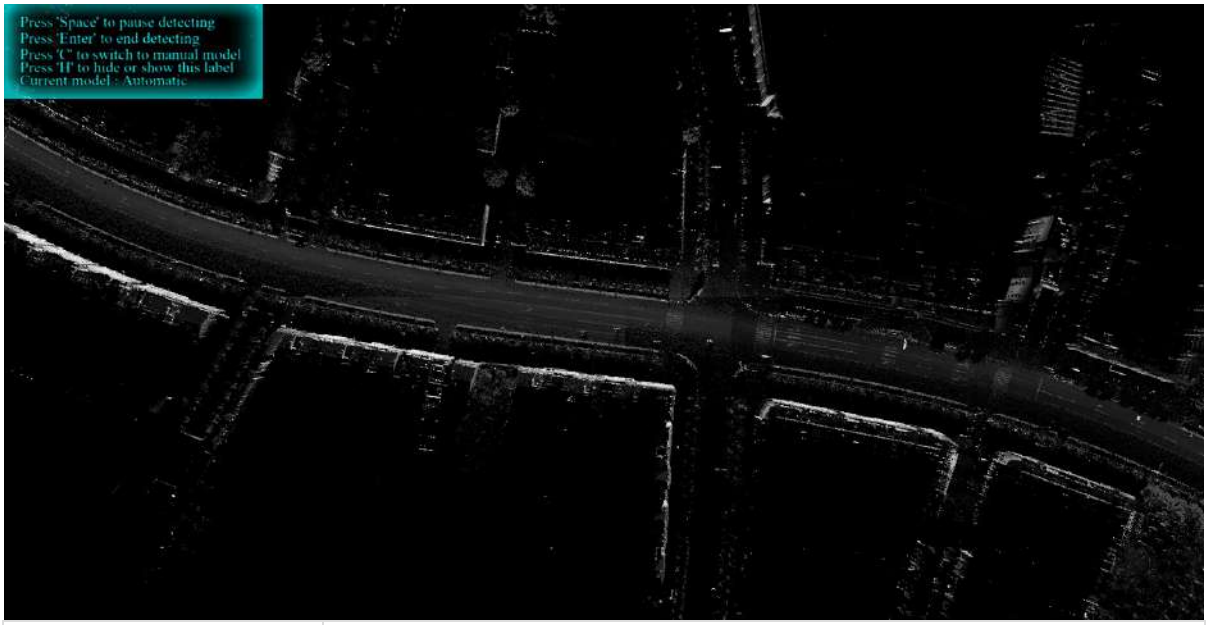
Dashed Lane Dialog

Parameter Description:

- **LaneType**：Line type. Select dashed line.
- **Intensity Adjustment**：

> Intensity adjustment, please refer to see Lane Solid Line intensity adjustment module

3.3D window tips:



Shortcut name	Function
Space	Interrupt growth
Enter	Confirm growth result
C	Toggles between automatic/manual drawing mode
B	Toggles between batch or one-by-one plotting in manual mode
H	switch off/on the top left corner prompt
Current model	Current model
Right mouse button	Rewind node (only works after interruption, not during growth)

The lane dash detection function offers two modes, a semi-automatic mode, which automatically detects the lane by means of "Start + Length + Spacing", and a manual mode, which follows the manual drawing mode of the "Draw lane dash" function. Both modes can be switched between during the drawing process by means of C.

4.The initial drawing mode is "semi-automatic". **Click Left on the mouse** on a point on the lane line point cloud as the **start point**, and then **left mouse click again** to select another point on the lane line as the **growth direction**.

5.Automatic lane detection begins from **Click the starting point, direction point**.

> During the growth process, the center of the screen is automatically adjusted to the current growing point; at the same time, the point cloud can be viewed by **angle adjusting, dragging, and zooming** through **left mouse button, right mouse button, and scroll wheel**.

6.Use shortcut keys for **real-time shape adjustment** :

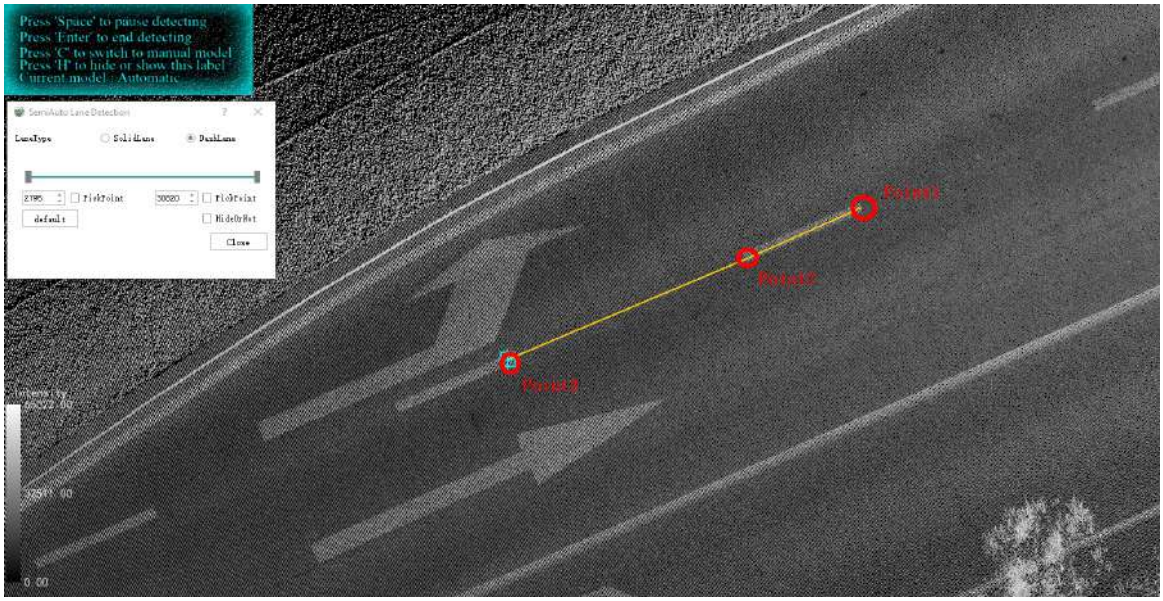
- The program is automatically **interrupted** due to drastic changes in intensity, actual termination, actual interruption, etc.; Users can use the **space** to manually **interrupt** if the automatic growth is not correct growth.
- After **interruption**, Users can use **right mouse button** to go back (go back a dashed line) to adjust the shape.
- After **interruption**, Users can repeat operations 3 and 4, followed by automatic growth.
- If the point cloud is missing in a large area or the intensity contrast is very low due to occlusion etc. and the

program cannot detect it automatically, you can switch to manual mode with **C** and then draw it manually by batch drawing or section by section (you can switch with the shortcut key **B**, for details refer to Lane Dash Lines Drawing). When finished, you can double-click to end the current mode or use **C** to switch to automatic mode.

7.When the growth is completed and the shape is acceptable, press the **Enter** (Enter, confirm key) to complete the growth of the dotted line of the current lane, otherwise, the user can continue to **right-click** to go back and adjust the threshold again for growth.

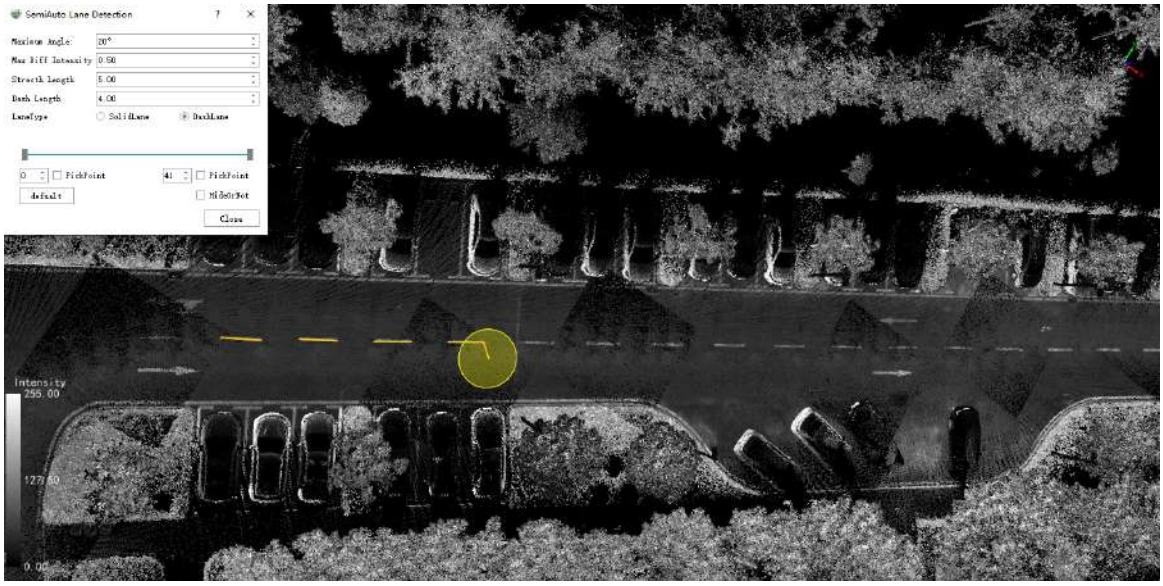
Rendering

- **Click the starting point and direction point**



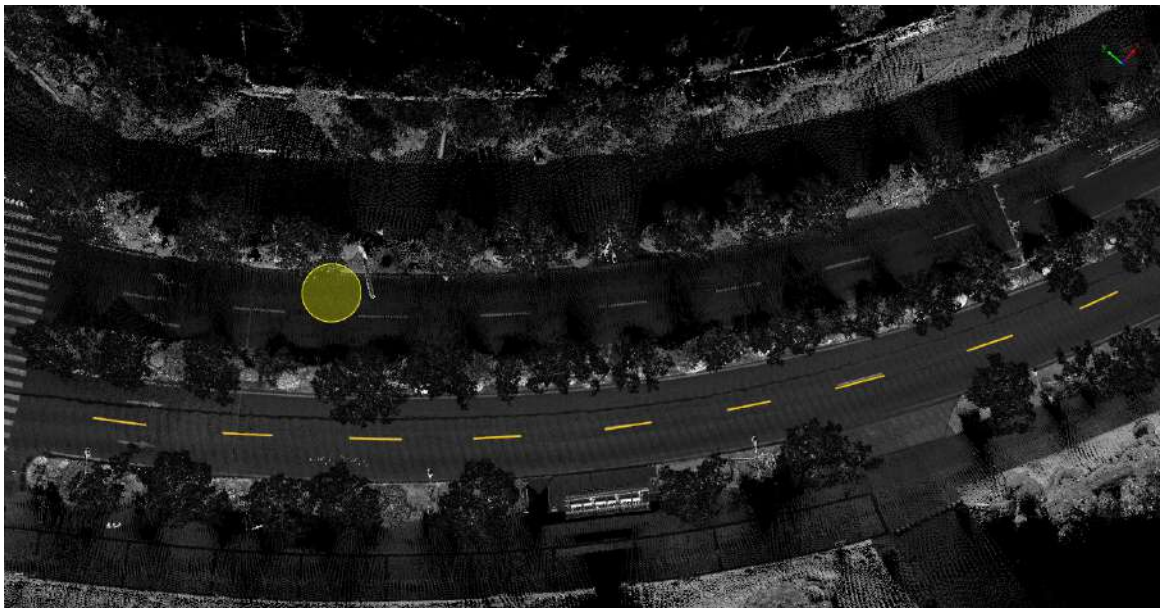
Click the starting point and direction point/p>

- **Automatic or manual interruption during growth**



Dashed line growth results

- **Growth results**



Dashed line growth results

Detect Road

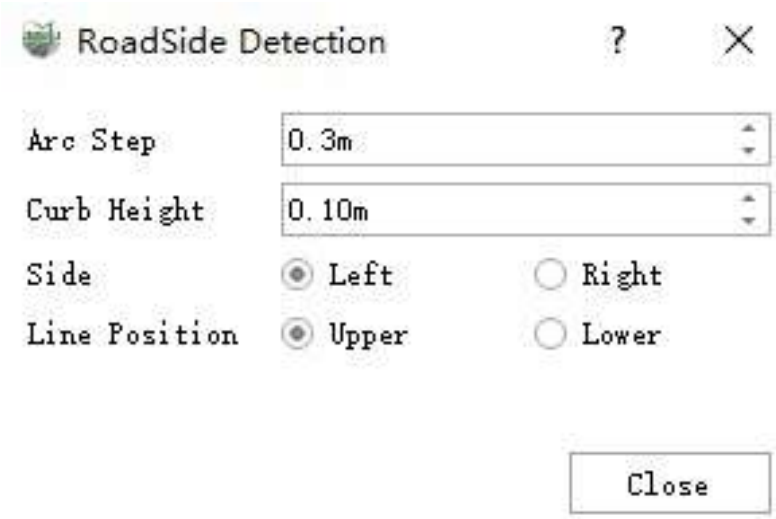
Function Description： This tool enables the detection of roadsides using points. Left click to select two points to obtain the starting point and direction point. Along the selected direction, automatic roadside detection will be carried out from the starting point.

> Note: This function is only effective for areas with clear curbs and the curb height is above 10 cm.

Steps

1.Click the **Detect Road**  button in the Road Shape panel.

2.A **Detection Settings dialog box** will appear:

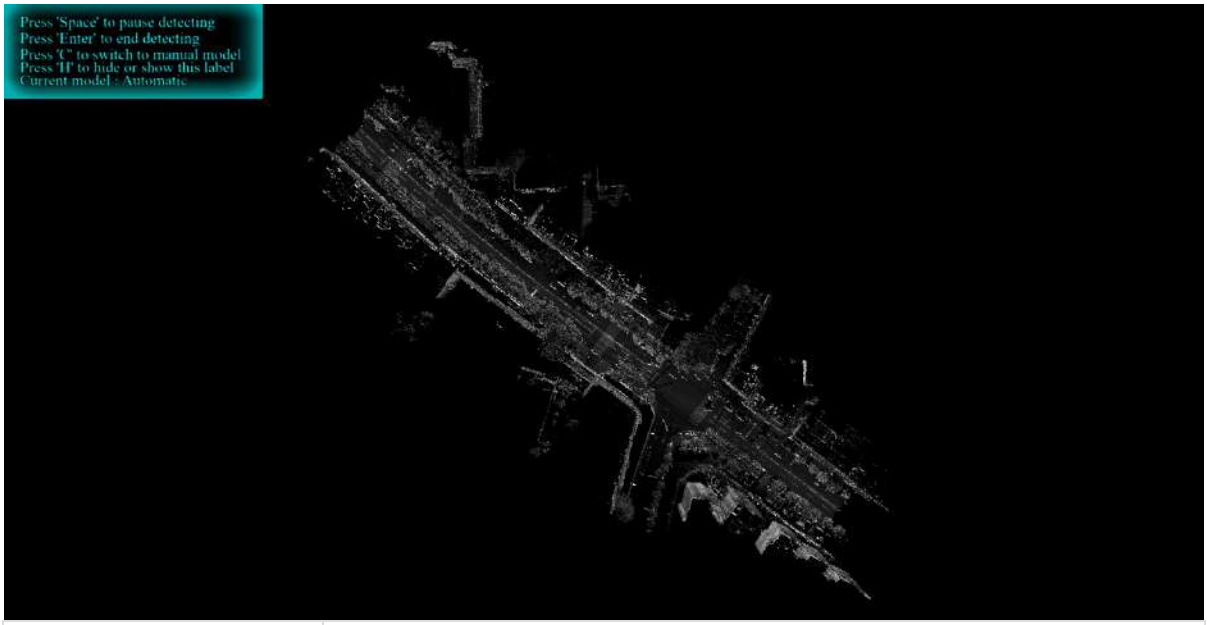


Pop up Detection Settings dialog box

Parameter Description:

- **Arc Step**: The default is 0.3 meters, and the curb may have a larger arc. If the step length is set too large, the arc part will not fit the actual point cloud enough. It is recommended to be less than 0.3 meters, 0.1 meters above.
- **Curb Height**: The default is 0.1. If the peripheral curb is lower than the set value, it cannot be detected. The default value is recommended.
- **Side**: The default is the left side. It indicates which side of the straight line formed by the starting point and direction points of the point selection is located in the actual path point cloud, which is selected according to the actual situation.
- **Line Position**: The default is upper, which means that the final generated line is located on the upper or lower edge of the curb. The **upper line** is calculated by adding the height of the curb to the point Z value of the **lower line**. Therefore, the position of the upper line can be adjusted by **adjusting the height of the curb**.

3.3D window tips:



Shortcut name	Function
Space	Interrupt growth
Enter	Confirm growth result
C	Toggles between automatic/manual drawing mode
B	Toggles between batch or one-by-one plotting in manual mode
H	switch off/on the top left corner prompt
Current model	Current model
Right mouse button	Rewind node (only works after interruption, not during growth)

Two modes are available in the curb detection function, a semi-automatic mode, where automatic detection is carried out by means of the "start point + direction" mode, and a manual mode, which follows the manual drawing mode of the "draw curb line" function. Both modes can be switched between during the drawing process by means of C.

4.The initial drawing mode is "semi-automatic". Left click to select a point on the road surface point cloud as the starting point, and then left click again to select another point on the road surface as the direction point.

> Note: The clicked starting point and direction point must be clicked on the surrounding road surface. The distance between the clicked point and the actual curb point cloud is recommended to be more than 0.1 meters and within 0.5 meters.

5.Along the selected direction, automatic roadside detection starts from the starting point.

> During the growth process, the center of the screen is automatically adjusted to the point currently growing; at the same time, the angle of the point cloud can be adjusted, dragged, and zoomed to view the point cloud through the left, right, and scroll wheels of the mouse.

6.Use shortcut keys for **real-time road shape adjustment**:

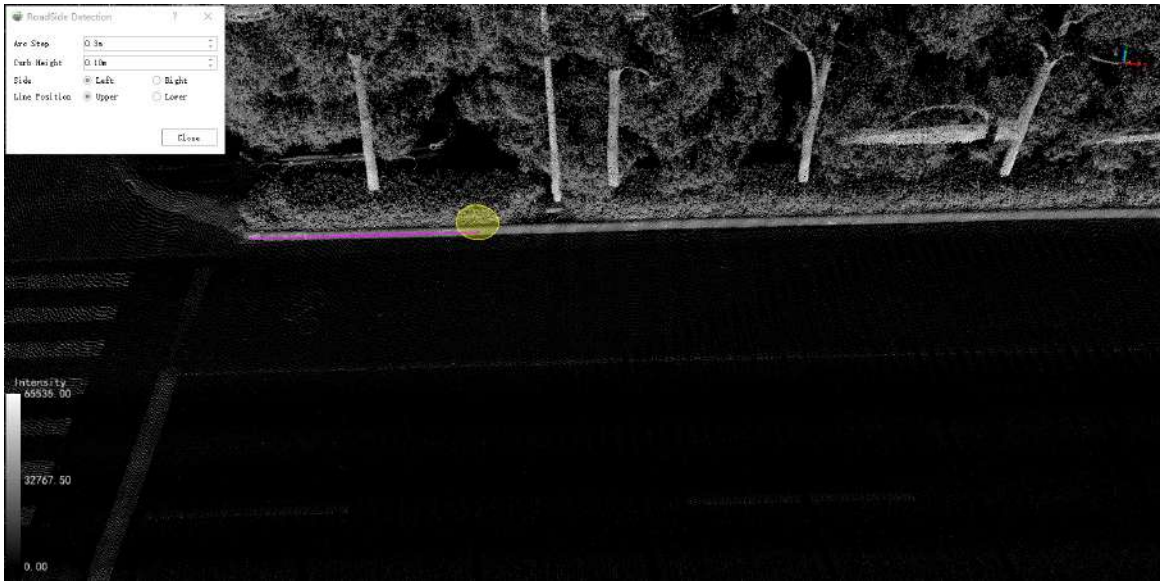
- The program is automatically **interrupted** due to the reasons such as low curb height, actual termination, actual interruption, etc.; if the automatic growth does not grow correctly, the user can use the **Space** to manually **interrupt**.

- After **interruption**, the user can **right click** to go back to adjust the shape.
- After **interruption**, operations 3 and 4 can be repeated, followed by automatic growth.
- If the point cloud is missing in a large area due to occlusion, etc., and the program cannot detect it automatically, you can switch to manual mode with **C** and then draw by manually clicking on the key points, and when the manual drawing is finished, you can double-click to end the current mode or switch to automatic mode with **C**.

7.When the growth is completed and the shape is acceptable, press the **Enter**(Enter, confirm key) to complete the growth of the solid line of the current lane.

Straight Road Curb

- **Click the starting point and direction point**

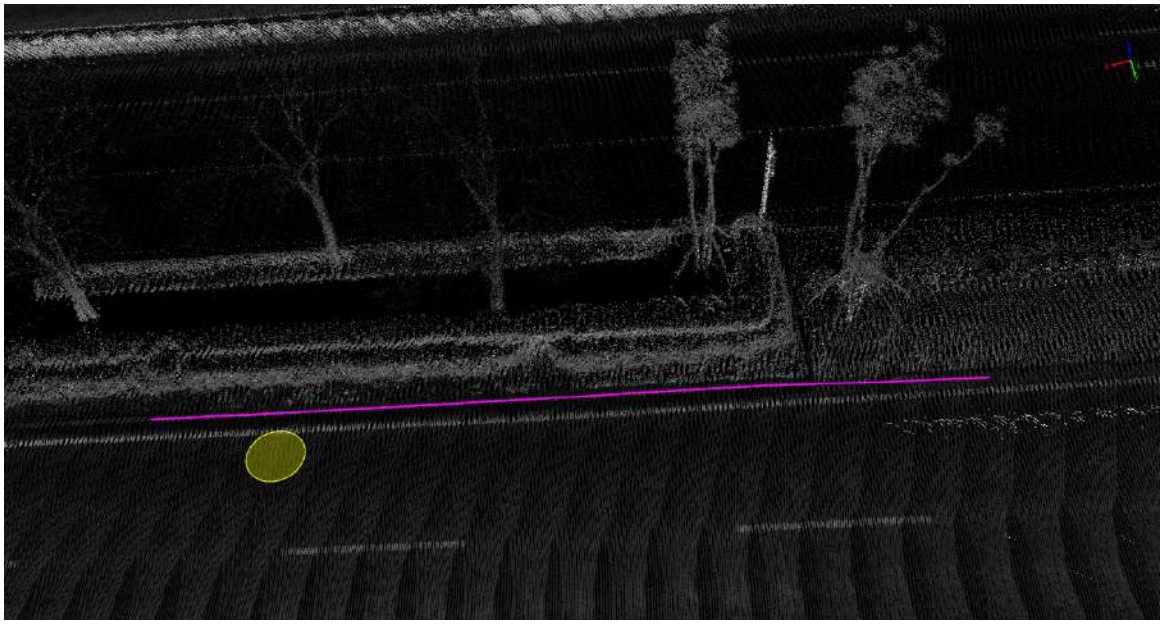


Click the starting point and direction point

- **Interrupt** and click again

Interrupt

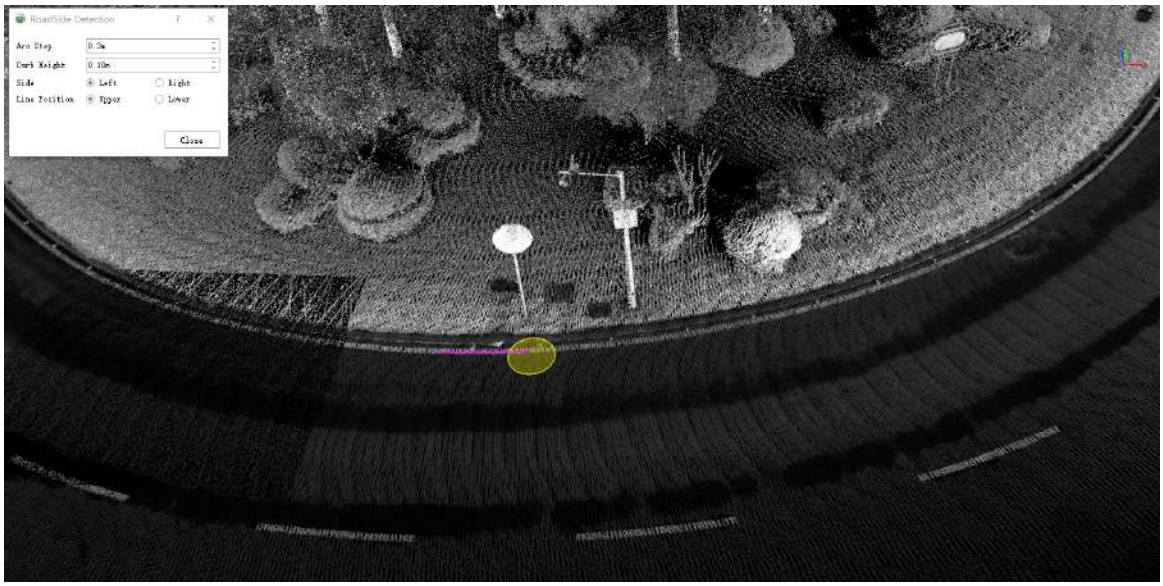
- **Growth result**



Curb growth result

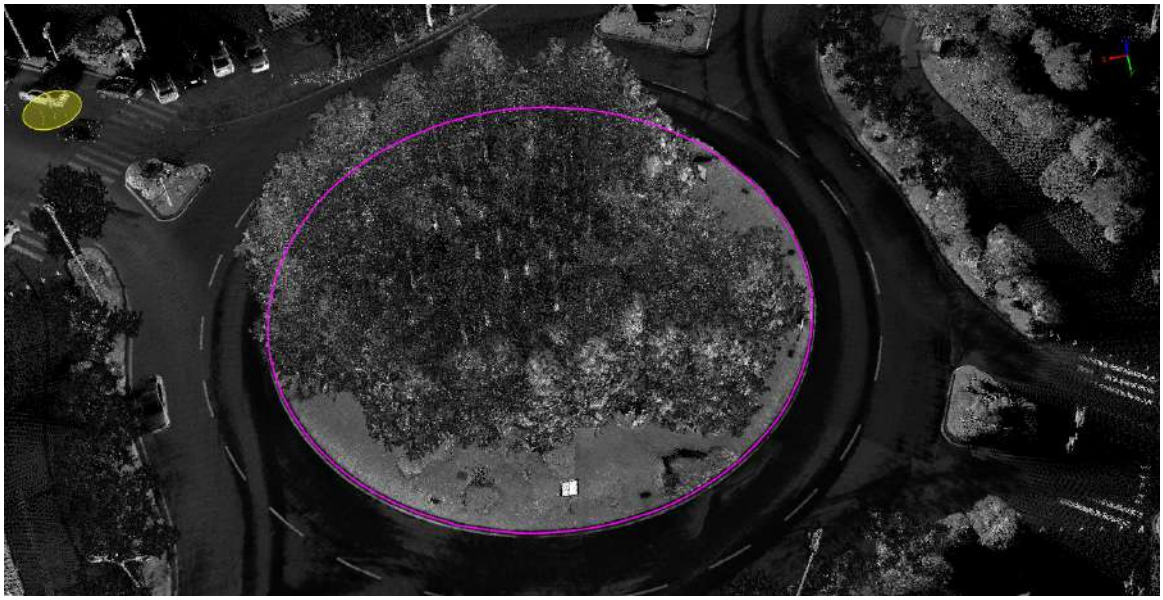
Closed Round Curb

- **Click the starting point and direction point**



Click the starting point and direction point

- **Growth result**



Round curb growth result

Road Surface

The road surface module is mainly used for the **vectorized manual** production of various ground signs such as various ground arrows, crosswalks, deceleration signs, and no-stop lines.

Includes:

- Guide arrow
  - Add Straight
  - Add Left
  - Add Right
  - Add Left/Right
  - Add Straight/Left
  - Add Straight/Right
  - Add Straight/Left/Right
  - Add Turn Round
  - Add Straight/Turn Round
  - Add Left/Turn Round
  - Add Left Merge
  - Add Right Merge
  - Add Right/Turn Round
  - Add Straight/Right/Turn Round
  - Add Left/Right/Turn Round
- Horizontal marking
  - Add CrossWalk
  - Add Longitudinal Speed Reduction
- Pavement graphic markings
  - Add CrossWalk Warning
  - Add Speed Reduction
  - Add No-Parking
  - Add BicycleLane
  - Add No U-Turn
  - Add Font
- Template Matching
  - TemplateMatchingSidebarParametersDetail
  - TemplateMatchingCustomTables
  - TemplateMatchingAutoMatch
  - TemplateMatchingManual
  - TemplateMatchingDatabaseFile
  - New Template
- Ground Marking Batch Process

Add Straight

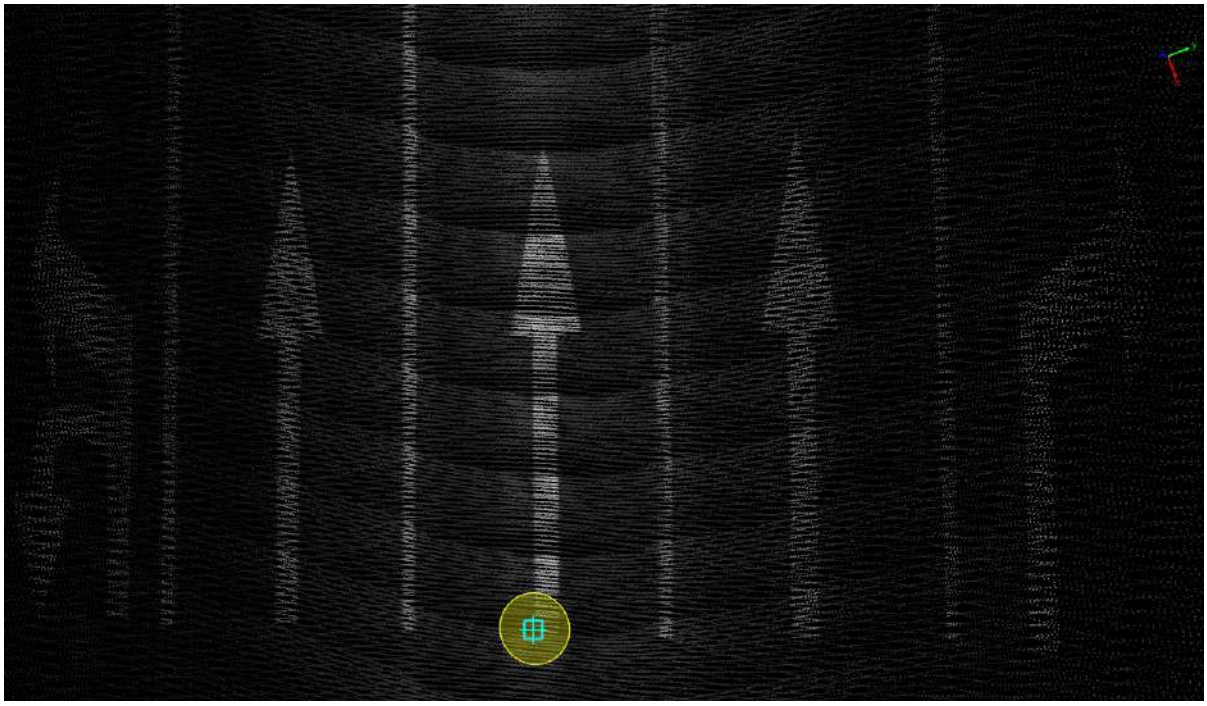
Function Description： The straight arrow means that the current lane can only go straight. This tool is used to manually select the point cloud to vectorize the straight arrow.

Steps

1.In the Road Surface panel, click the **Add Straight**  button.

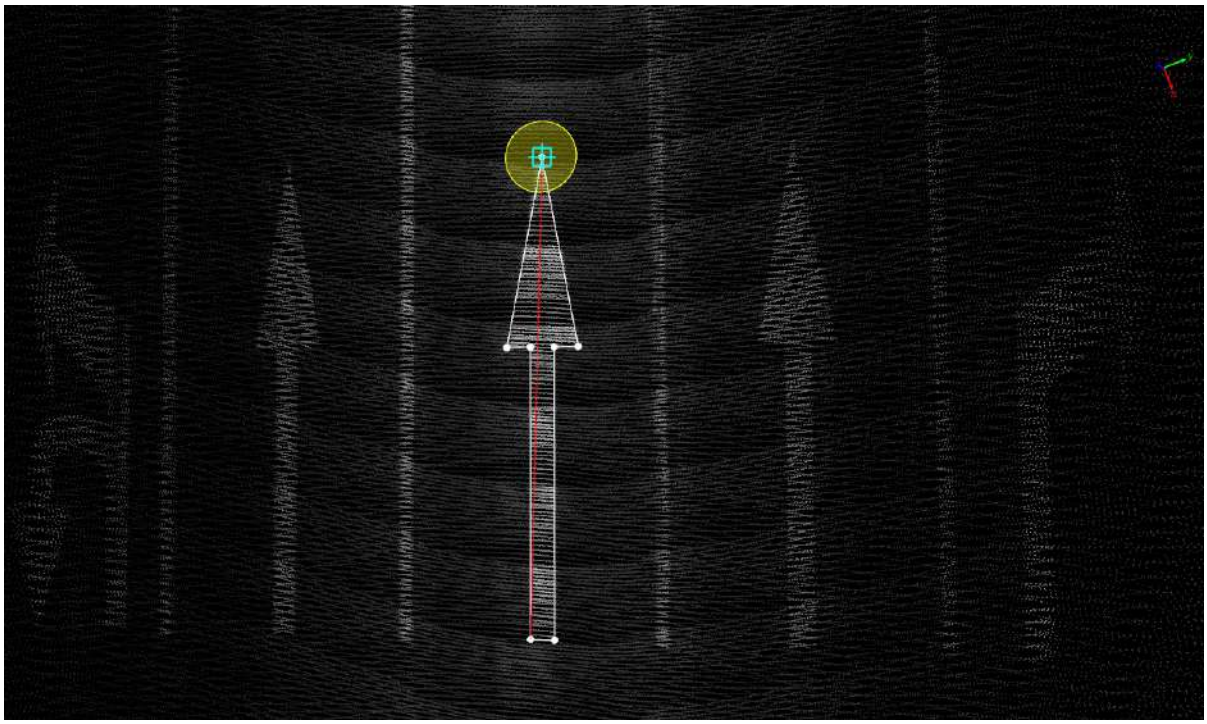
> - The key points in the arrow icons need to be aligned with the actual point cloud.
> - The red point is the first key point and the green point is the second key point.

2.**Left mouse click** to select the lower center of the straight line arrow point cloud with the following details and the **first key point** in the icon.



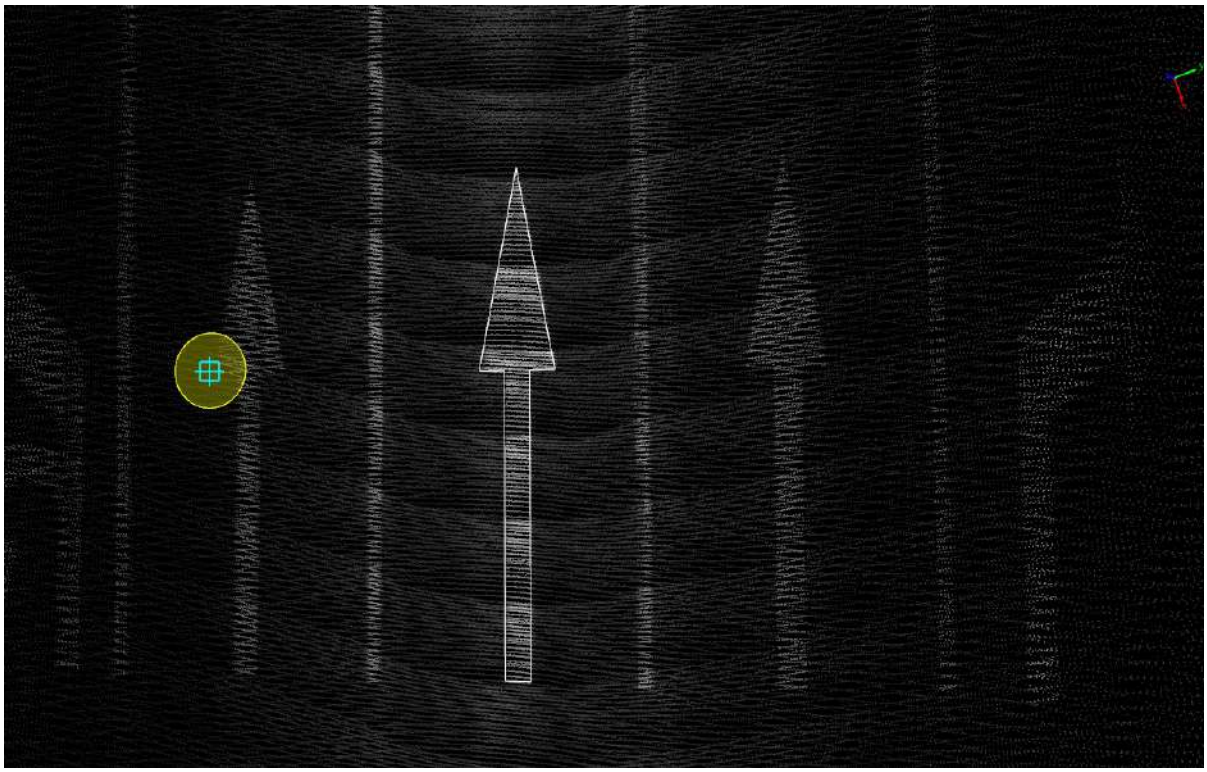
First key point

3.Move the mouse to see **template** and **actual point cloud** fit, **left mouse click to select the second key point** to complete the straight arrow production, details are as follows, icon in **second key point**.



Second key point

Results Show



Straight arrow drawing

Add Left

Function Description： The left turn arrow indicates that the current lane can only be turned left, this tool is used to manually tap the point cloud for vectorization of the left turn arrow.

Steps

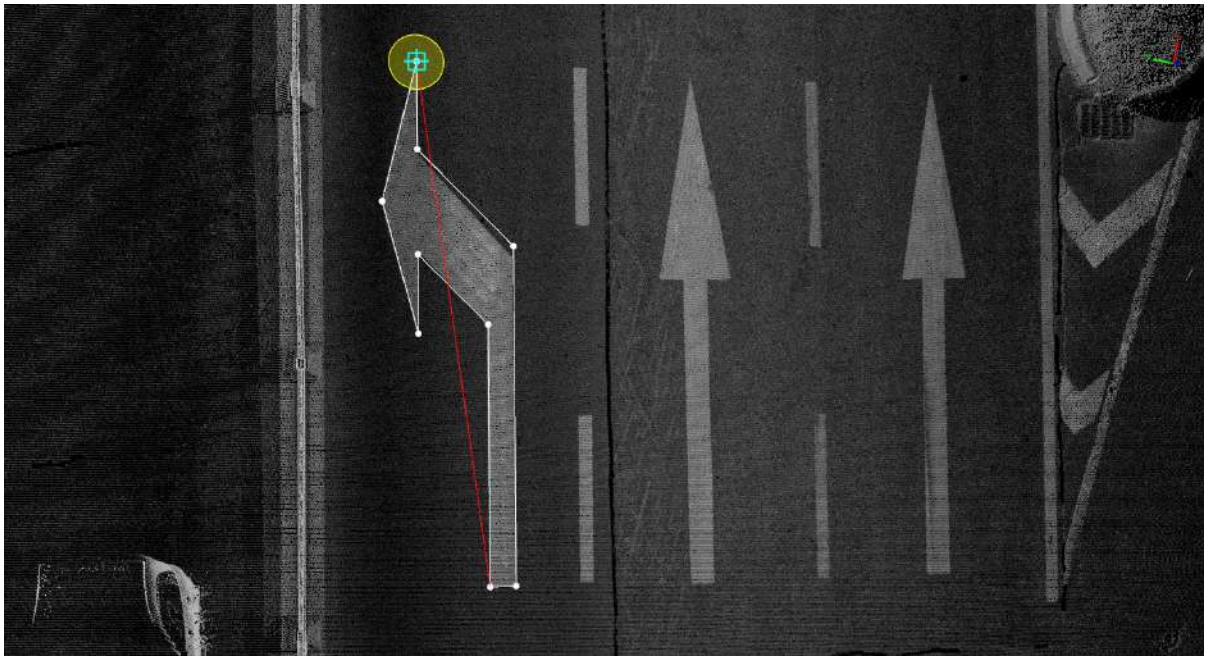
1.Click the **Add Left**  button in the Road Surface panel.

> - The key points in the arrow icons need to be aligned with the actual point cloud point cloud.
> - The red point is the first key point and the green point is the second key point.

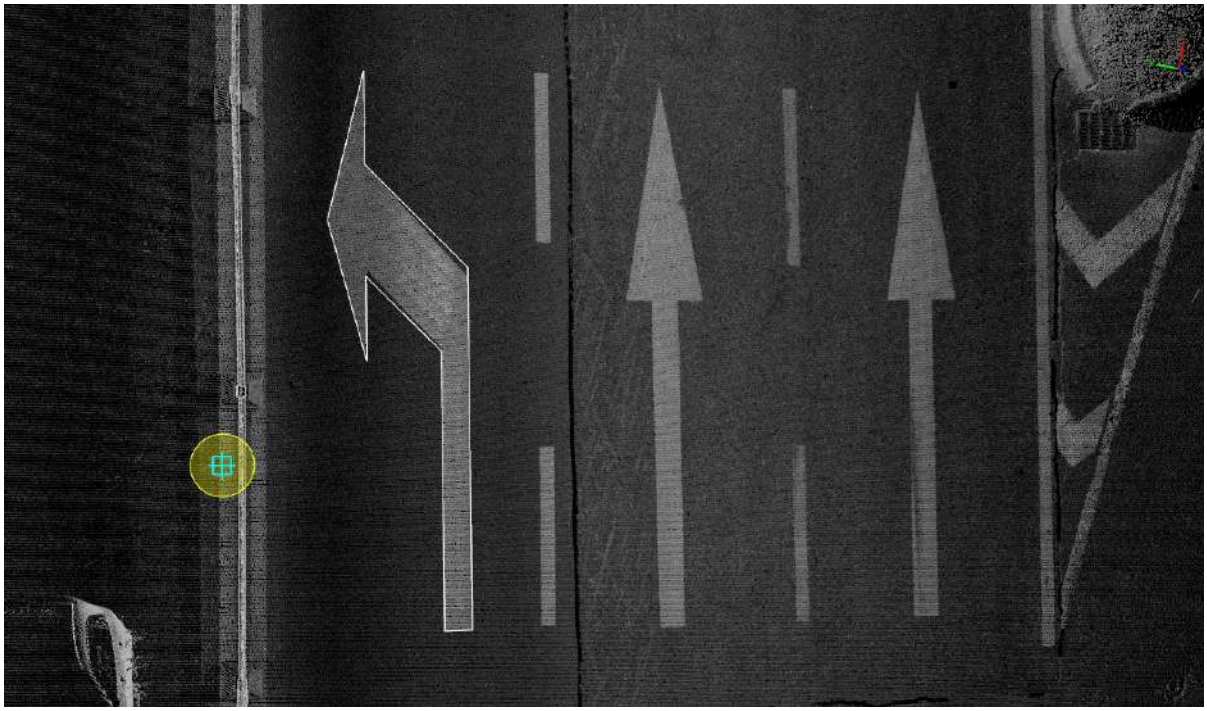
2.**Left mouse click to select the key point** of the left arrow point cloud, the details are as follows, **the first key point** in the icon.

First key point

3.Move the mouse to check the fit between the **template** and the **actual point cloud**, left mouse click **the second key point** to complete the left arrow production,, the details are as follows, **the second key point** in the icon.



Second key point



Turn left arrow drawing

Add Right

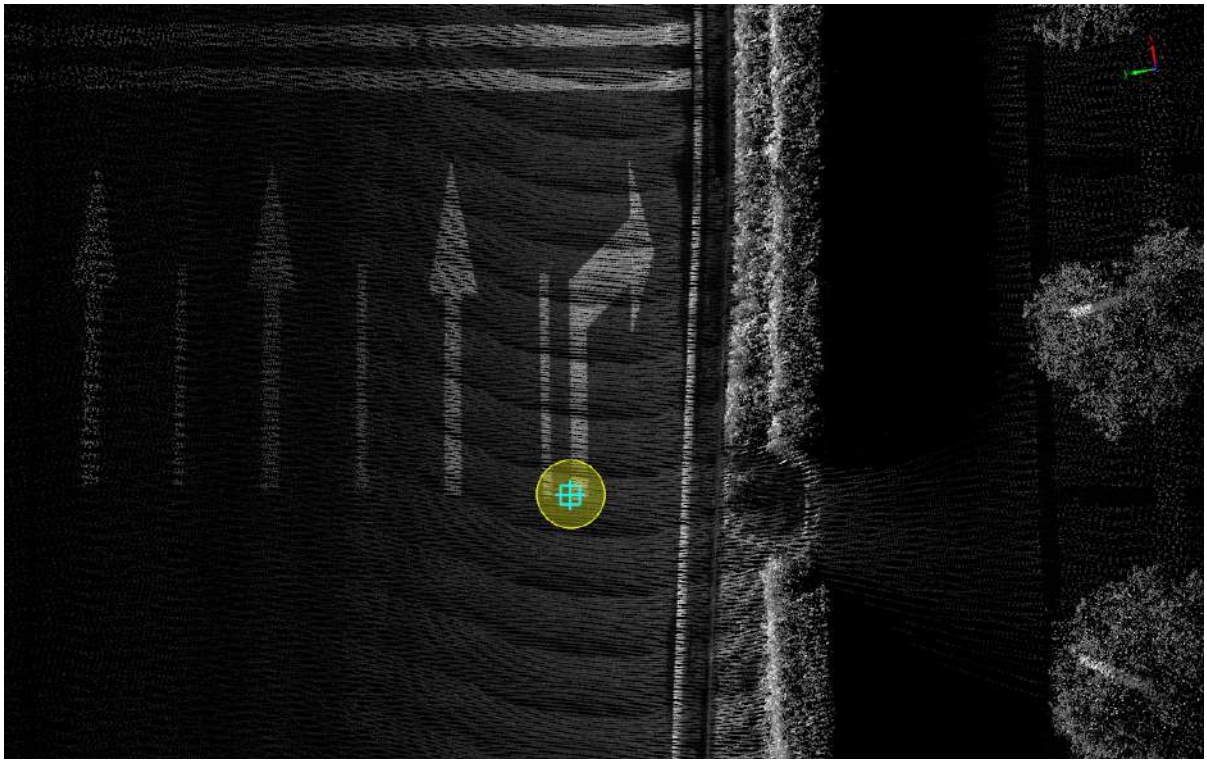
Function Description： The turn right arrow indicates that the current lane can only be turned right, this tool is used to manually tap the point cloud for vectorization of the right turn arrow.

Steps

1.Click the **Add Right**  button in the Road Surface panel.

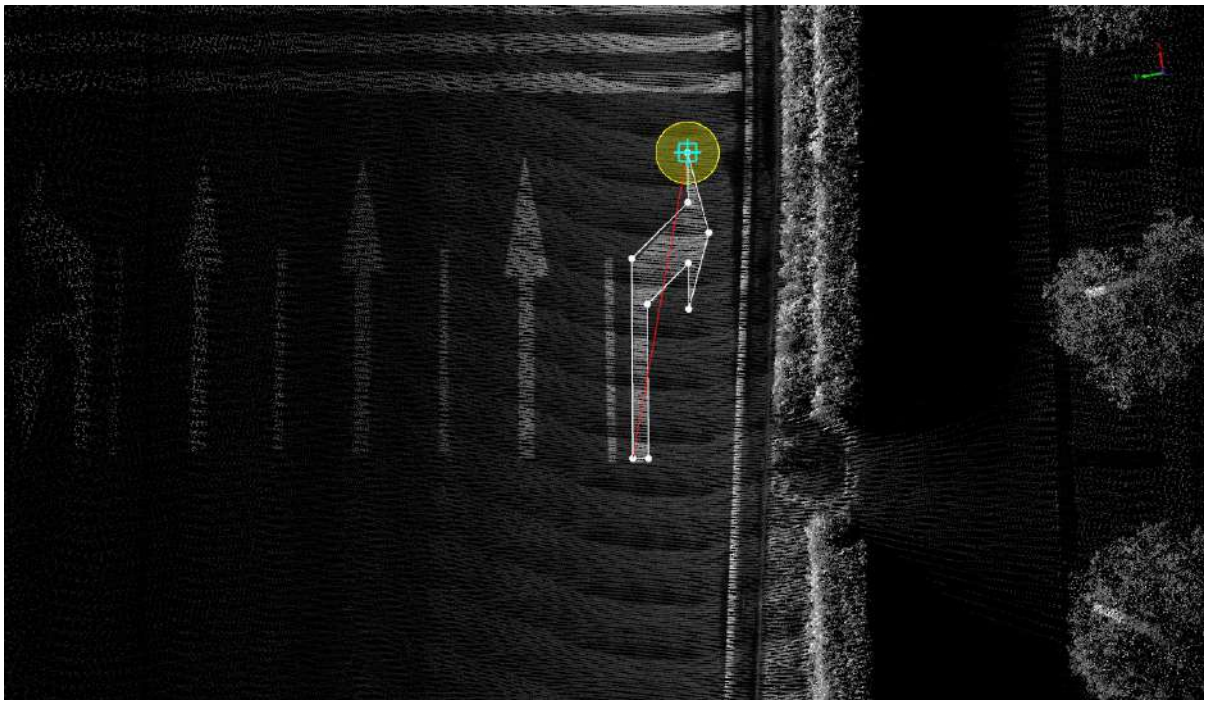
> - The key points in the arrow icons need to be aligned with the actual point cloud.
> - The red point is the first key point and the green point is the second key point.

2.**Left mouse click** to select the right turn arrow point cloud key point with the following details and the **first key point** in the icon.

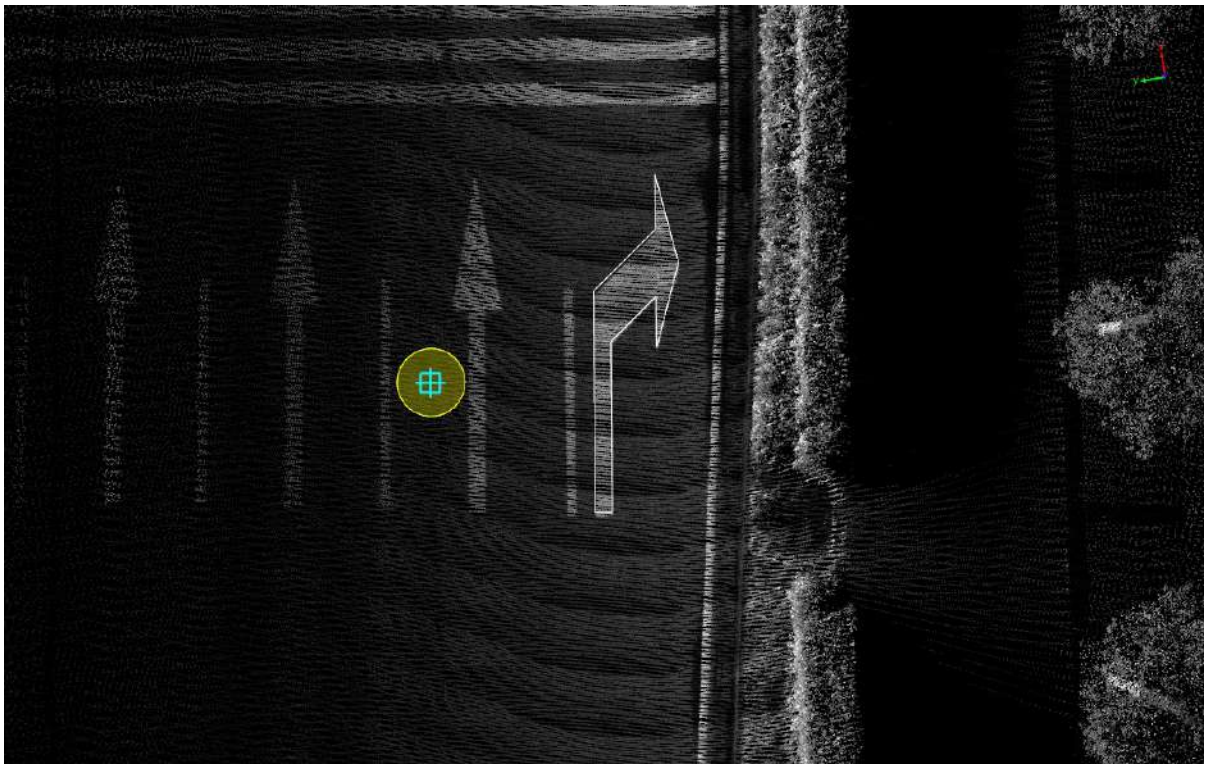


First key point

3.Move the mouse to see how the **template** fits into the **actual point cloud**, **left mouse button click** to select the **second key point** to complete the turn right arrow production, details are as follows, icon in **second key point**.



Second key point



Turn right arrow drawing

Add Left/Right

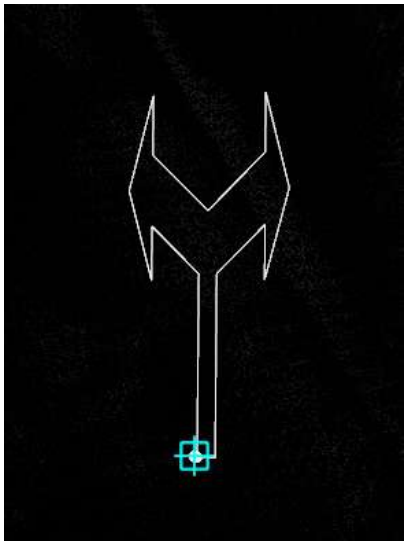
Function Description： The left/right arrow indicates that the current lane can only turn left or right. This tool is used to manually click on the point cloud to vectorize the left/right arrow.

Steps

1.Click the **Add Left/Right**  button in the Road Surface panel.

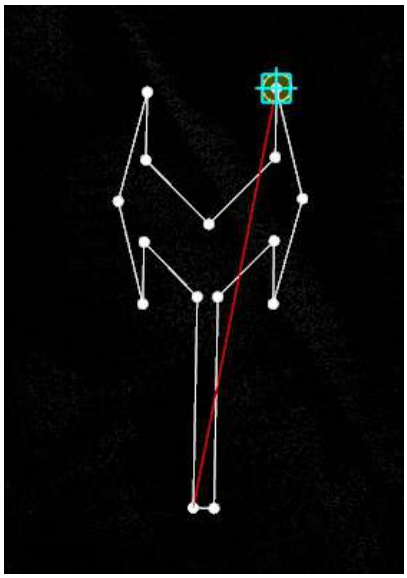
> - The key points in the arrow icons need to be aligned with the actual point cloud.
> - The red point is the first key point and the green point is the second key point.

2.**Left mouse click to select the key point** of the left/right arrow point cloud, the details are as follows, **the first key point** in the icon.

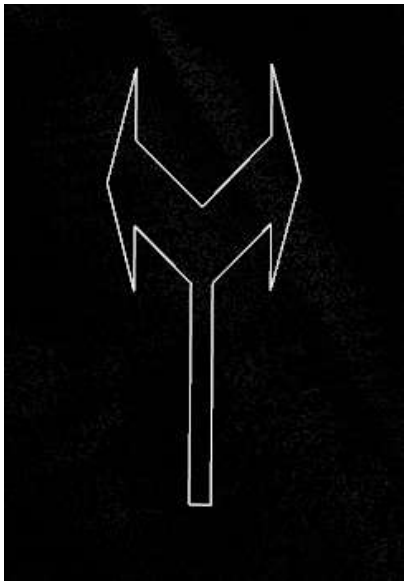


First key point

3.Move the mouse to check the fit between the **template** and the **actual point cloud**, left mouse click **the second key point** to complete the left/right arrow production, the details are as follows, the second key point in the icon.



Second key point



Turn Left/turn right arrow drawing

Add Straight/Left

Function Description： The straight/turn left arrow indicates that the current lane can turn left or go straight. This tool is used to manually click on the point cloud to vectorize the straight/left arrow.

Steps

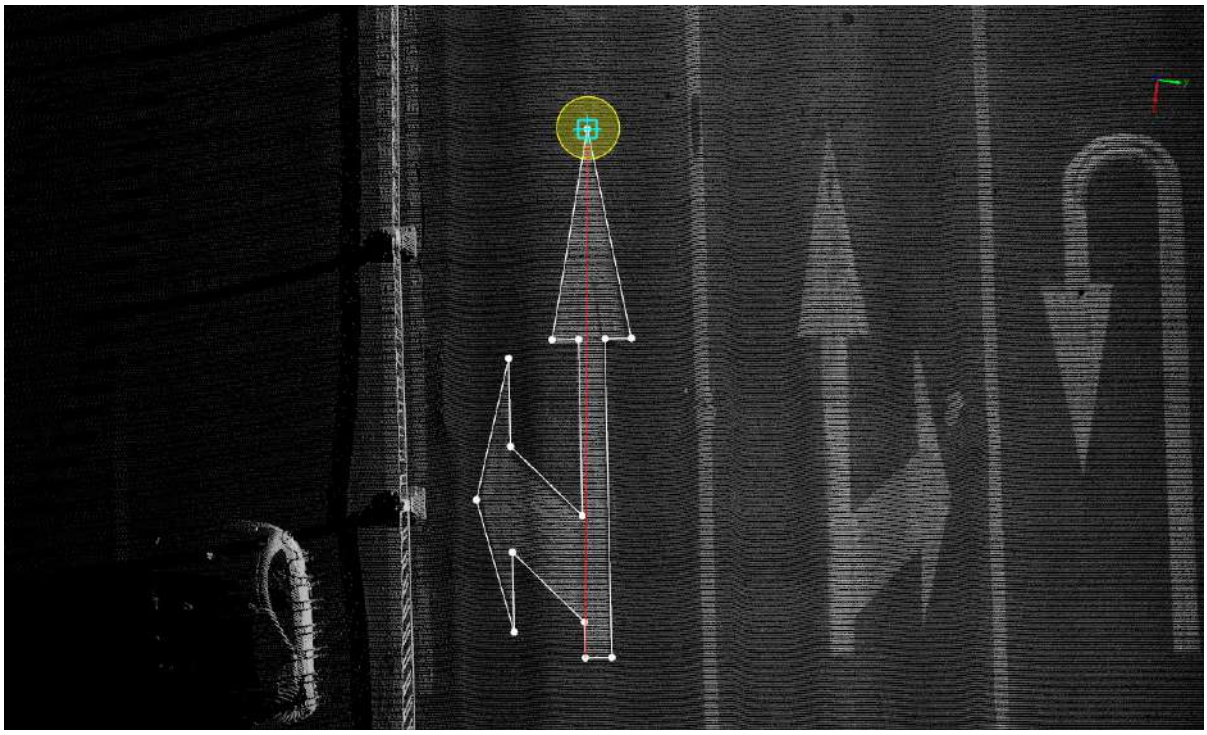
1.Click the **Add Straight/Left**  button in the Road Surface panel.

> - The key points in the arrow icons need to be aligned with the actual point cloud.
> - The red point is the first key point and the green point is the second key point.

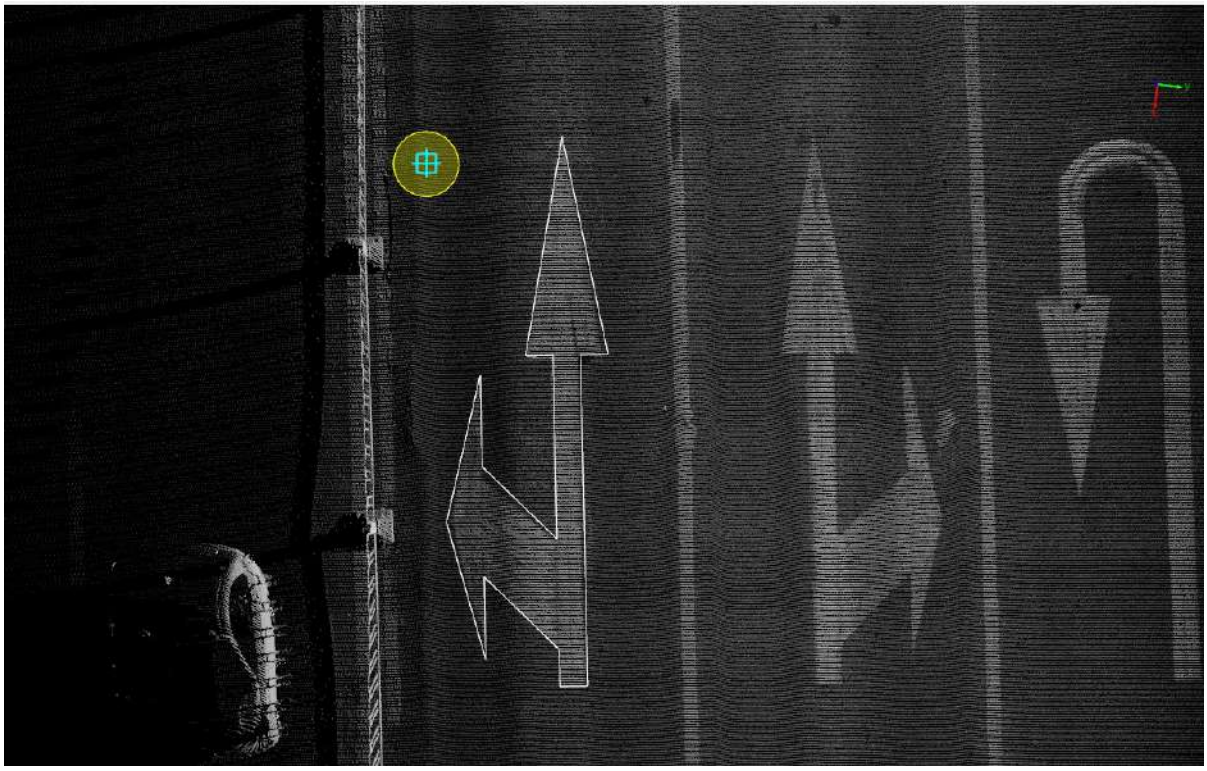
2.**Left mouse click** to select the straight/turn left arrow point cloud key point, the details are as follows, the icon in the **first key point**.

First key point

3.Move the mouse to view **template** and **actual point cloud** fit, **left mouse click** to select **second key point** to complete the straight/turn left arrow production, details are as follows, icon in **second key point**.



Second key point



Straight/turn left arrow drawing

Add Straight/Right

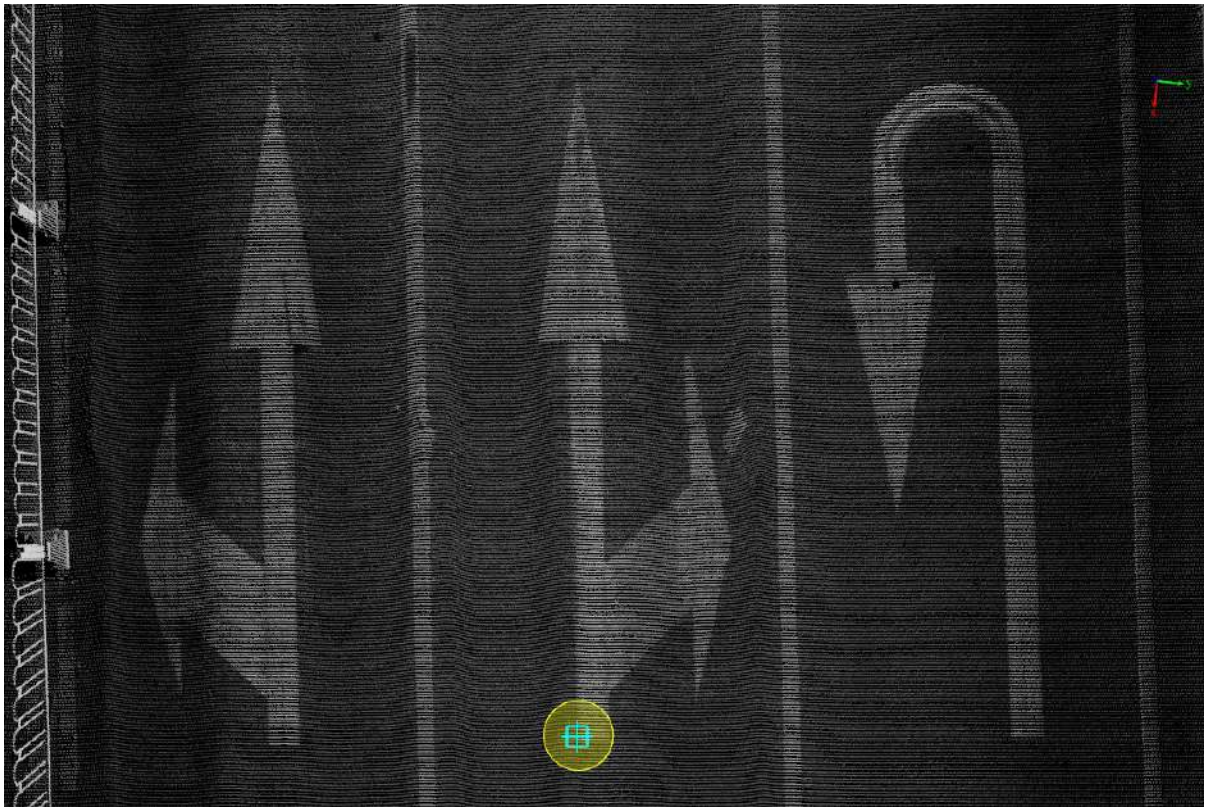
Function Description： The straight/turn right arrow indicates that the current lane can turn right or go straight. This tool is used to manually click on the point cloud to vectorize the straight/right arrow.

Steps

1.Click the **Add Straight/Right**  button in the Road Surface panel.

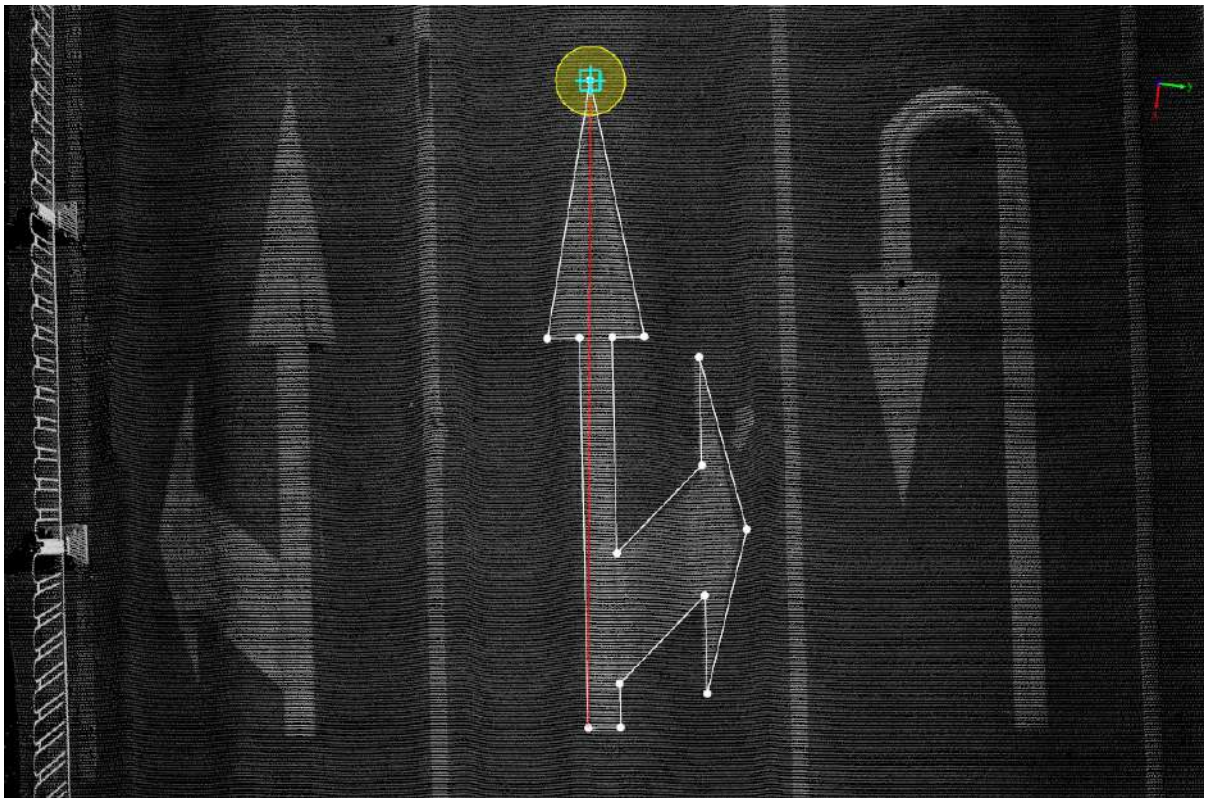
> - The key points in the arrow icons need to be aligned with the actual point cloud.
> - The red point is the first key point and the green point is the second key point.

2.**Left mouse click** to select the straight/turn right arrow point cloud key point, the details are as follows, the icon in the **first key point**.

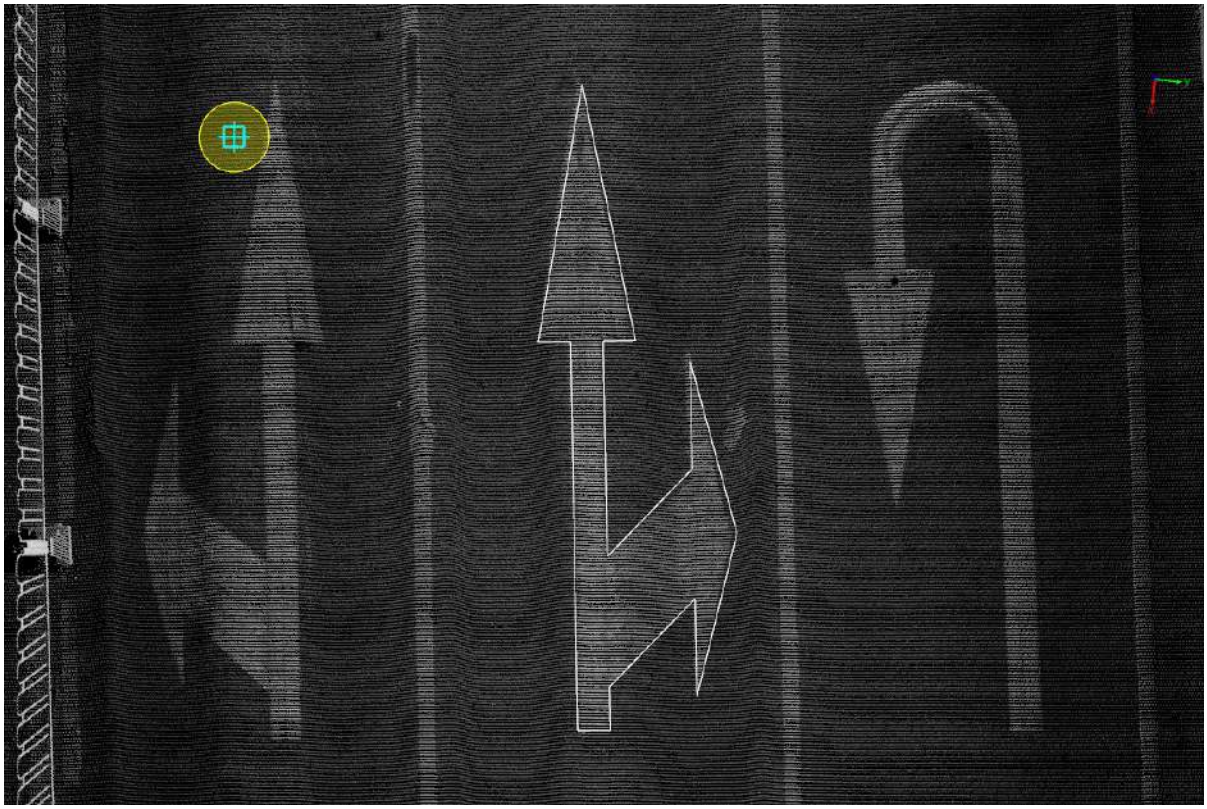


First key point

3.Move the mouse to see **template** and **actual point cloud** fit, **left mouse click to second key point** to complete the straight/turn right arrow production, details are as follows, icon in **second key point**.



Second key point



Straight/turn right arrow drawing

Add Straight/Left/Right

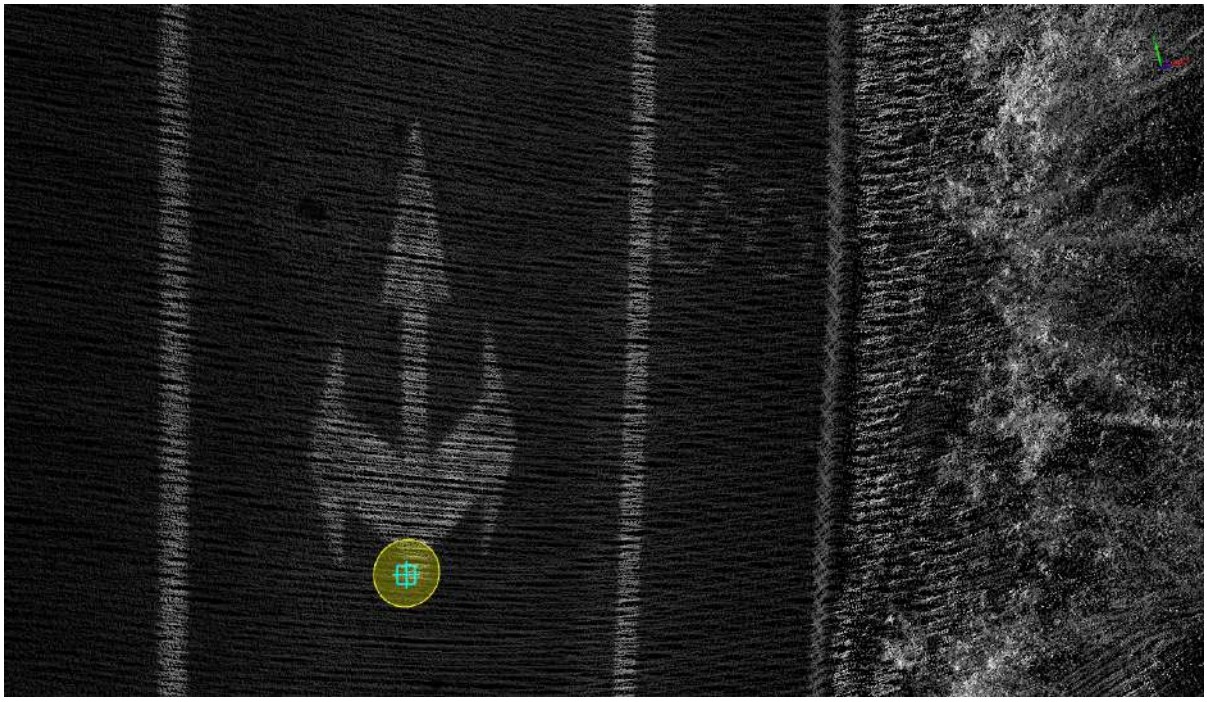
Function Description： The straight/turn left/turn right arrows indicate that the current lane can go straight, turn left, or turn right. This tool is used to manually click on the point cloud for vectorization of straight/turn left/turn right arrows.

Steps

1.Click the **Add Straight/Left/Right**  button in the Road Surface panel.

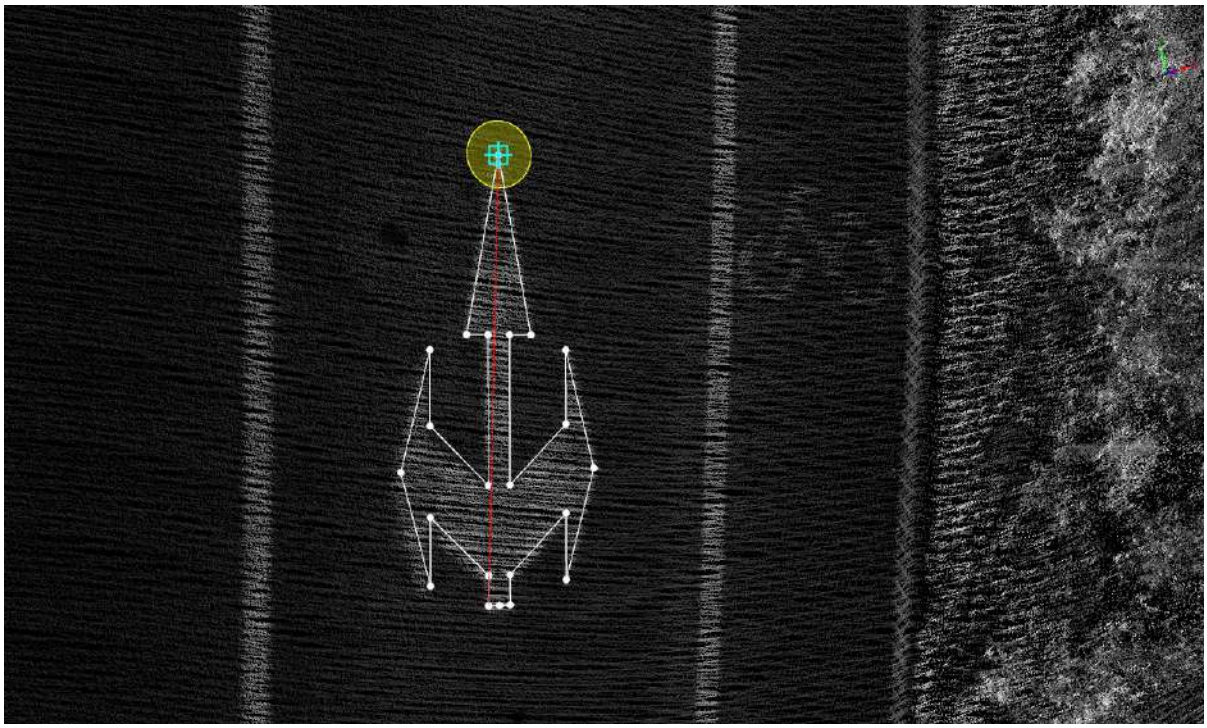
> - The key points in the arrow icons need to be aligned with the actual point cloud.
> - The red point is the first key point and the green point is the second key point.

2.**Left mouse click** to select the straight/turn left/turn right arrow point cloud key point, the details are as follows, the icon in the **first key point**.

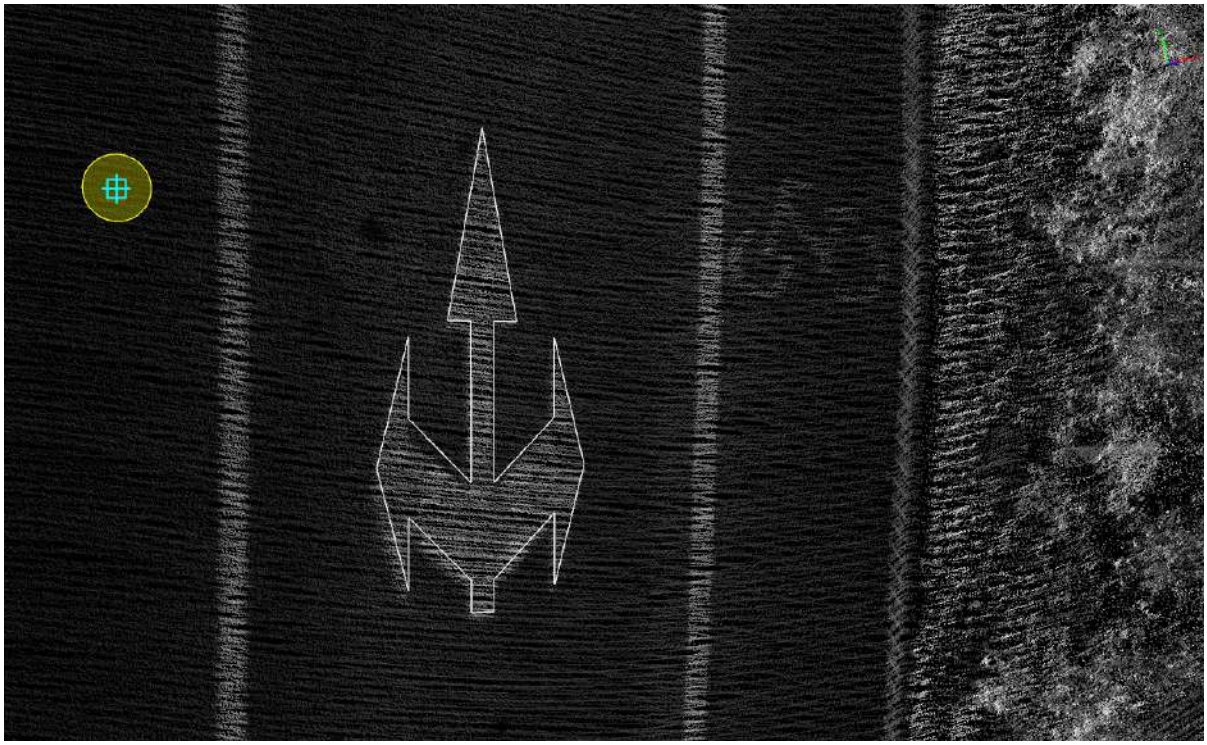


First key point

3.Move the mouse to see **template** and **actual point cloud** fit, **left mouse click to second key point** to complete the straight/turn left/turn right arrow production, details are as follows, icon in **second key point**.



Second key point



Straight/turn left/turn right arrow drawing

Add Turn Round

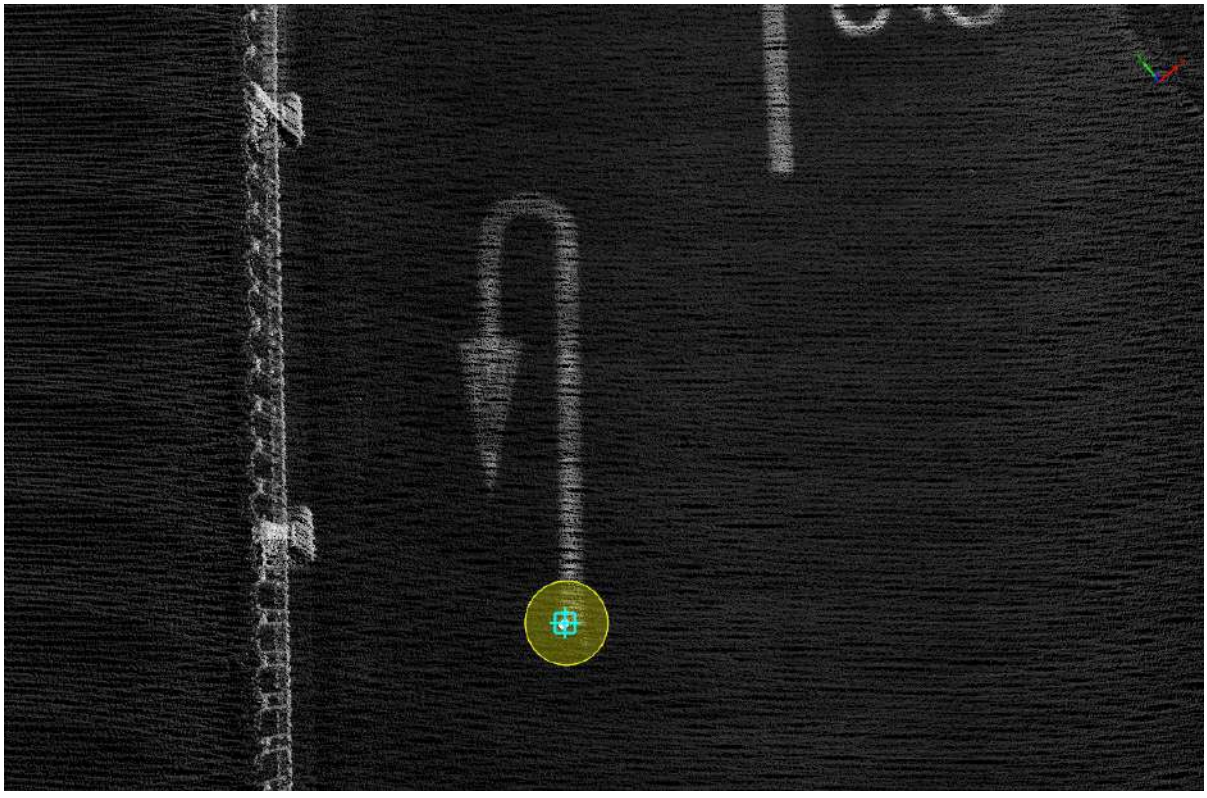
Function Description： U-turn arrow indicates that the current lane will be a U-turn. This tool is used to manually click on the point cloud to vectorize the U-turn arrow.

Steps

1.Click the **Add Turn Round**  button in the Road Surface panel.

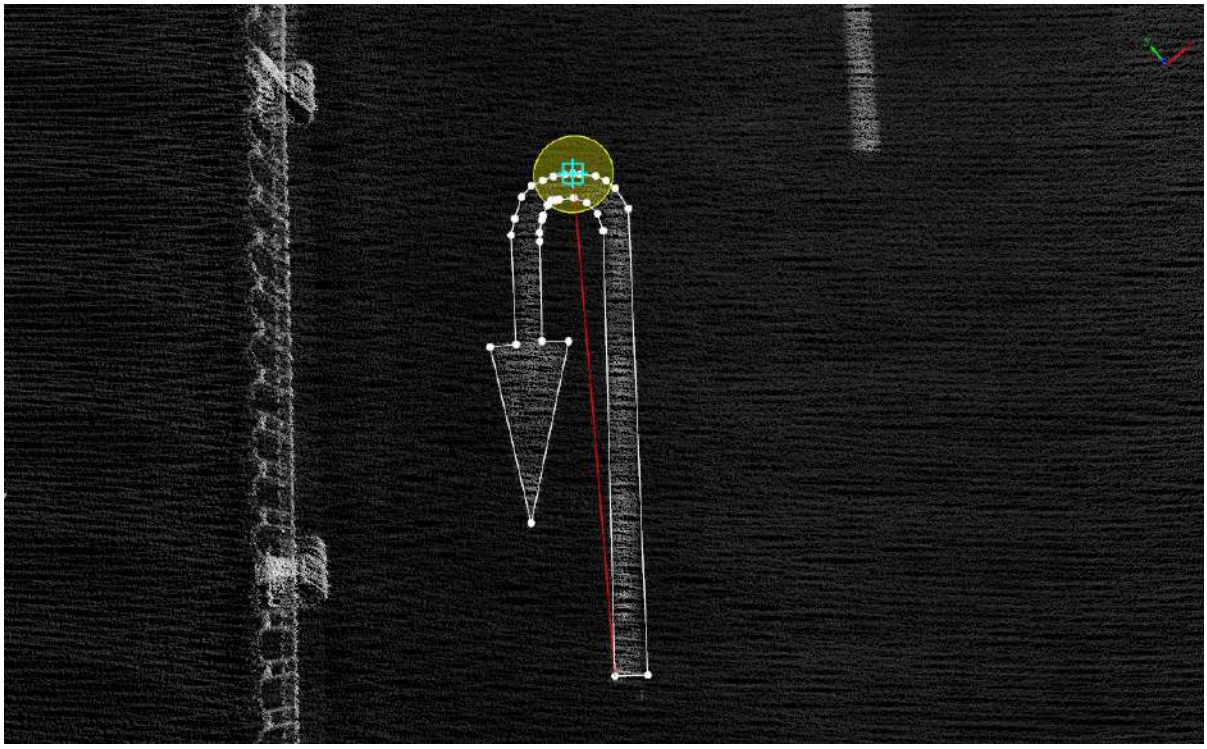
> - The key points in the arrow icons need to be aligned with the actual point cloud.
> - The red point is the first key point and the green point is the second key point.

2.**Left mouse click** to select the 2. U-turn arrow point cloud key point, the details are as follows, the icon in **the first key point**.

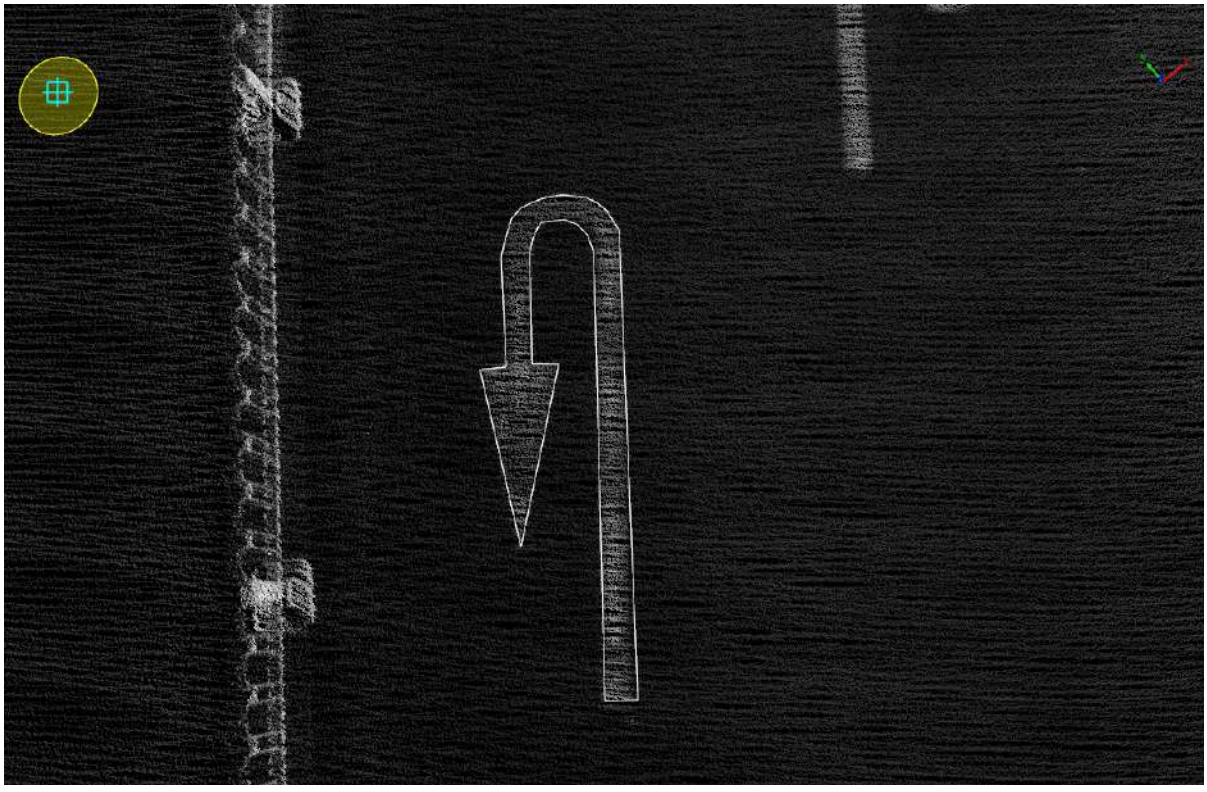


First key point

3.Move the mouse to see **template** and **actual point cloud** fit, **left mouse button click to select second key point** to complete the production of the U-turn arrow, the details are as follows, the icon in **second key point**.



Second key point



U-turn arrow drawing

Add Straight/Turn Round

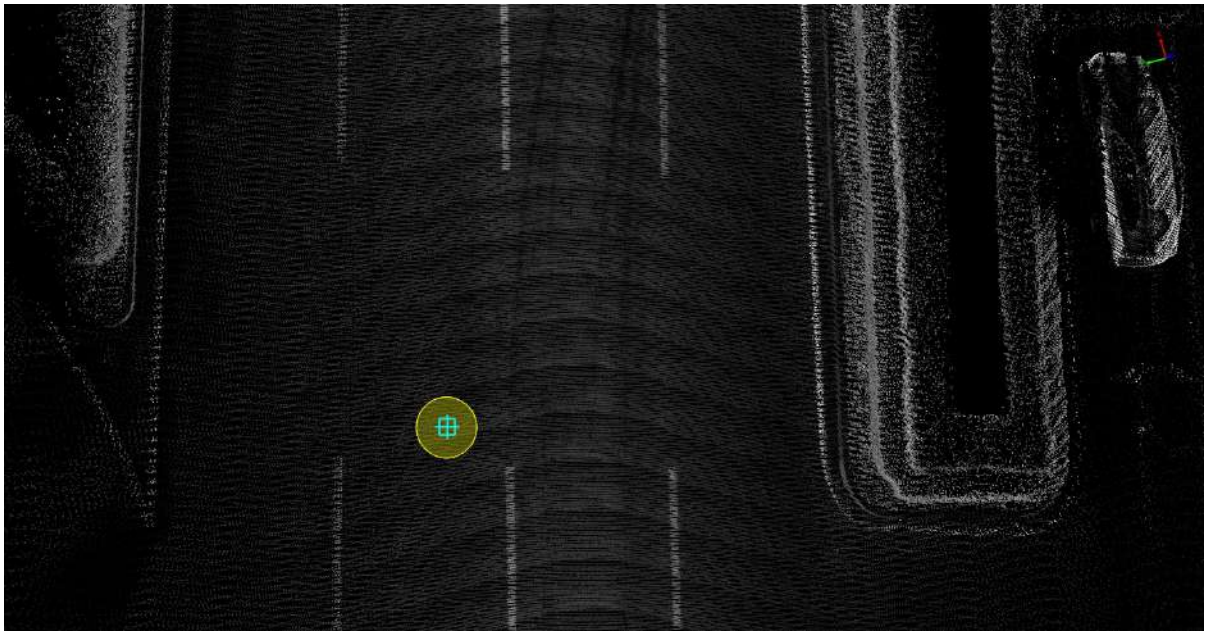
Function Description： Straight/U-turn arrow indicates that the current lane can be U-turn or go straight. This function is used to manually click on the point cloud to vectorize the straight/U-turn arrow.

Steps

1.Click the **Add Straight/Turn Round**  button in the Road Surface panel.

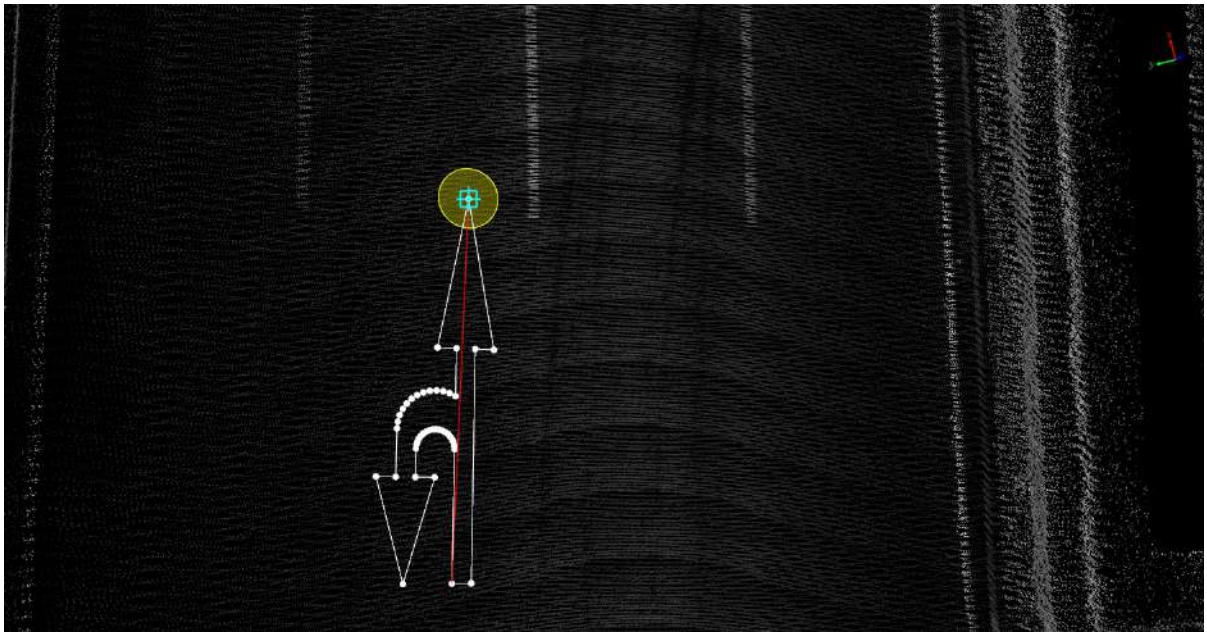
> - The key points in the arrow icons need to be aligned with the actual point cloud.
> - The red point is the first key point and the green point is the second key point.

2.**Left mouse button click** to select the straight/U-turn arrow point cloud key point, the details are as follows, the icon in the **first key point**.

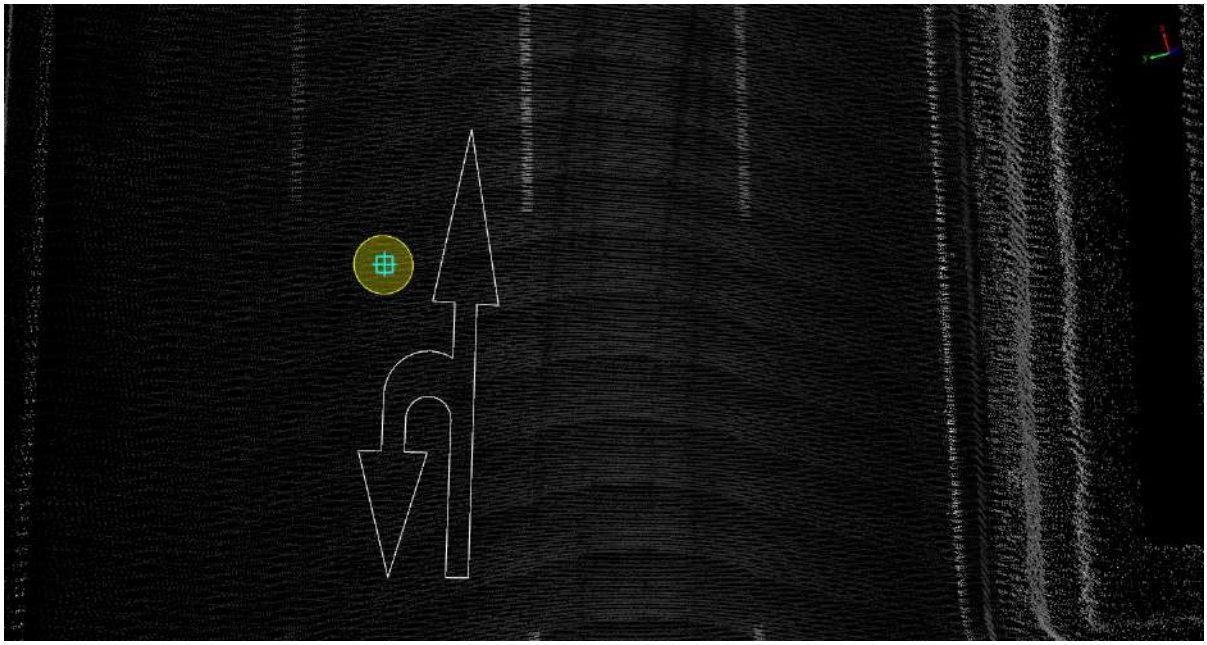


First key point

3.Move the mouse to view the **template** and **actual point cloud** fit, **left mouse click to second key point** to complete the production of straight/U-turn arrow, details are as follows, icon in **second key point**.



Second key point



Straight/U-Turn arrow drawing

Add Left/Turn Round

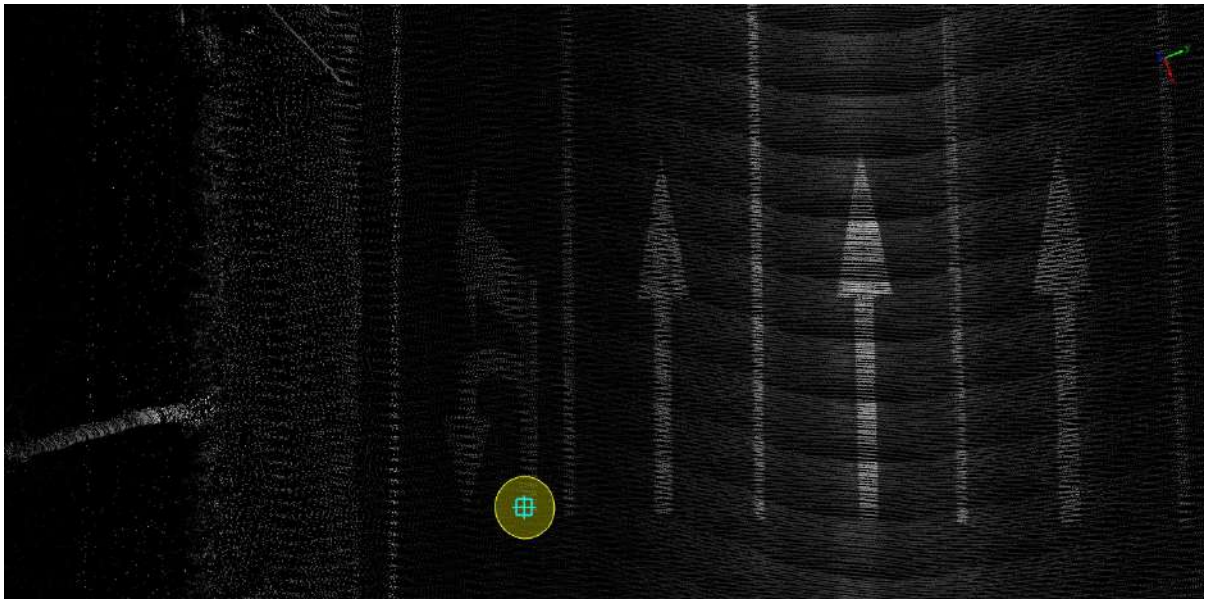
Function Description： Turn left/U-turn arrows indicate that the current lane can turn left or turn around, this tool is used to manually click on the point cloud for vectorization of turn left/U-turn arrows.

Steps

1.Click the **Add Left/Turn Round**  button in the Road Surface panel.

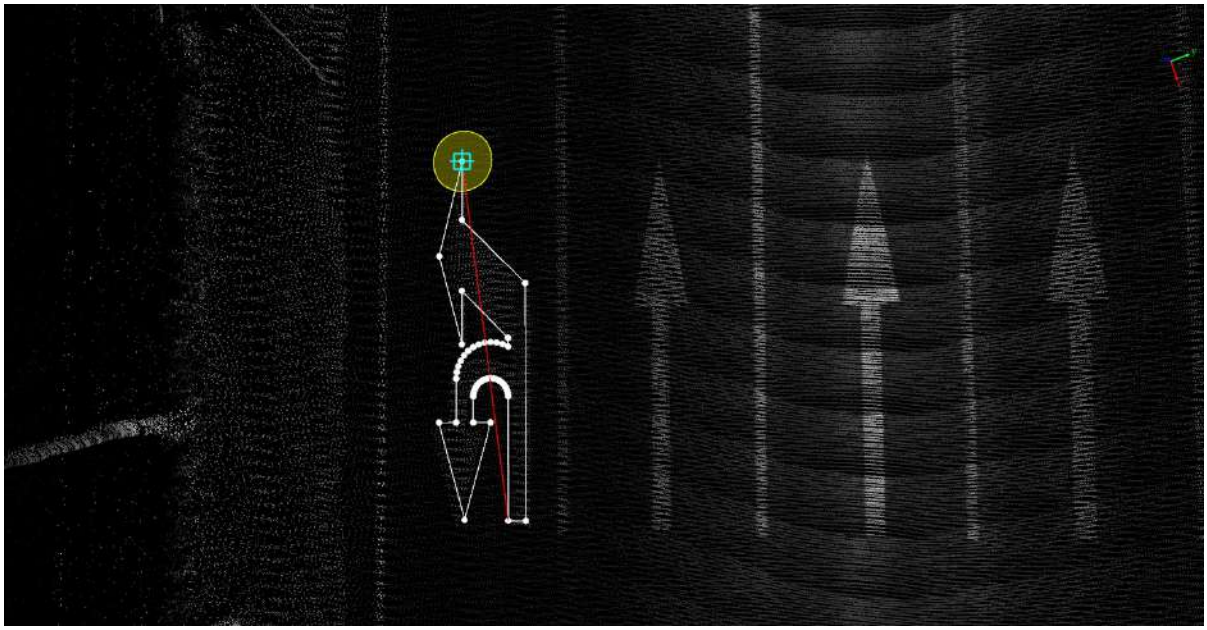
> - The key points in the arrow icons need to be aligned with the actual point cloud point cloud.
> - The red point is the first key point and the green point is the second key point.

2.**Left mouse click** to select the turn left/U-turn arrow point cloud key point, the details are as follows, the icon in **the first key point**.

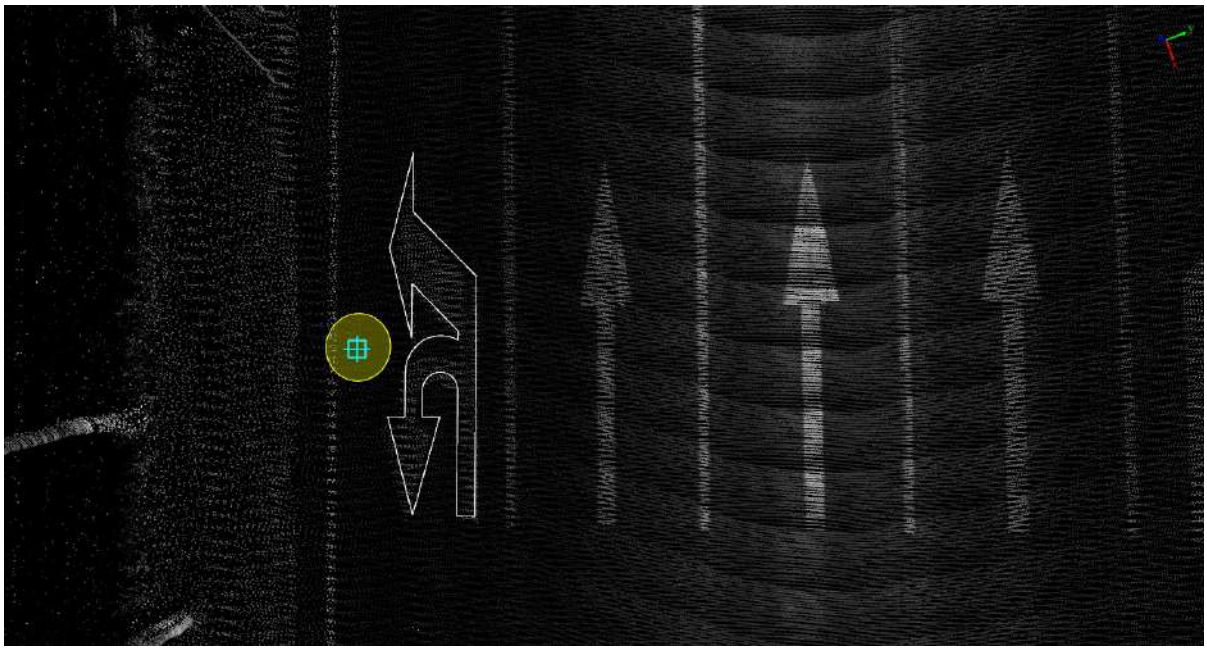


First key point

3.Move the mouse to view **template** and **actual point cloud** fit, **left mouse click to second key point** to complete the turn left/U-turn arrow production, details are as follows, icon in **second key point**.



Second key point



Turn left/U-turn arrow drawing

Add Left Merge

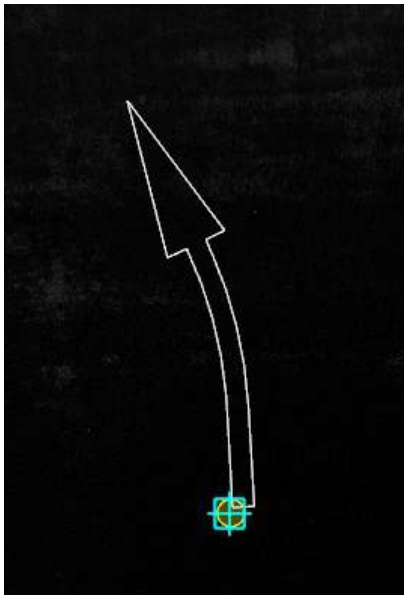
Function Description： The left merge arrow indicates that the current lane merges to the left. This tool is used to manually select the point cloud to vectorize the left merge arrow.

Steps

1.Click the **Add Left Merge**  button in the Road Surface panel.

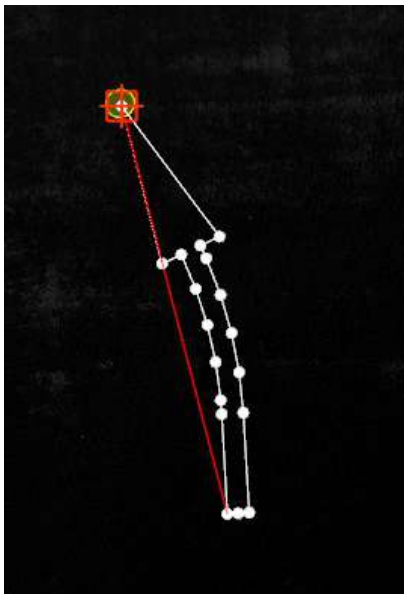
> - The key points in the arrow icons need to be aligned with the actual point cloud point cloud.
> - The red point is the first key point and the green point is the second key point.

2.**Left mouse click** to select the key point of the left merge arrow point cloud, the details are as follows, **the first key point** in the icon.

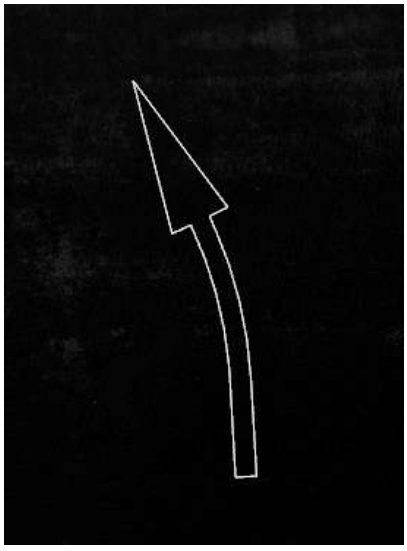


First key point

3.Move the mouse to check the fit of the **template** and the **actual point cloud**, left click **the second key point** to complete the left merge arrow. The details are as follows, **the second key point** in the icon.



Second key point



Left merge arrow drawing

Add Right Merge

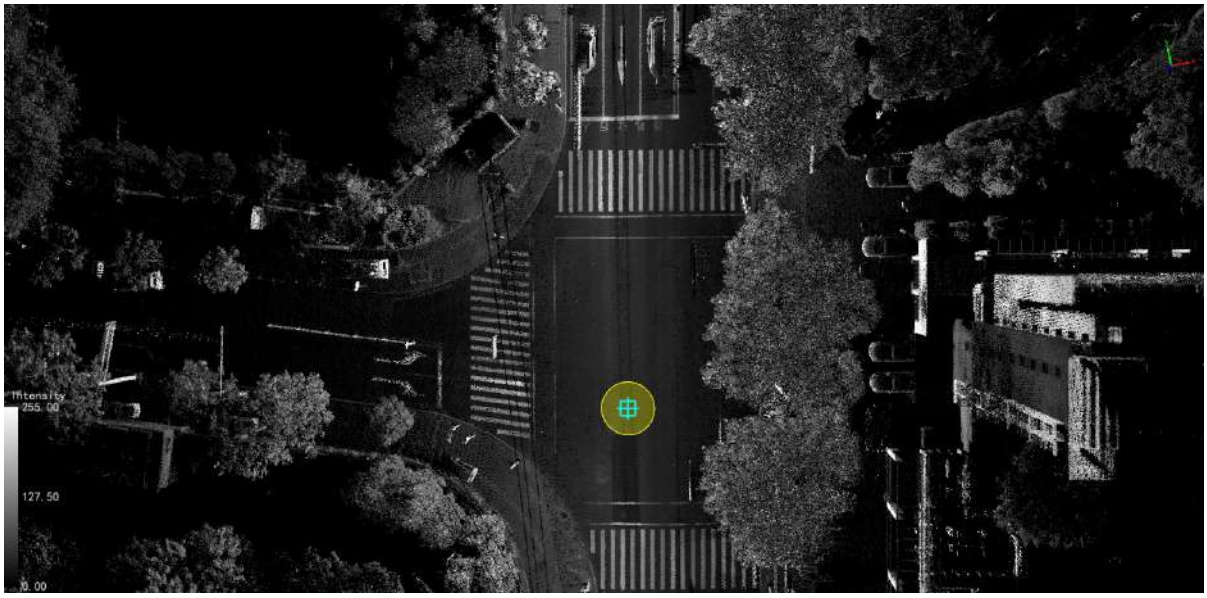
Function Description： The right merge arrow indicates that the current lane is merging to the right, this tool is used to manually click on the point cloud to vectorize the right merge arrow.

Steps

1.Click the **Add Right Merge**  button in the Road Surface panel.

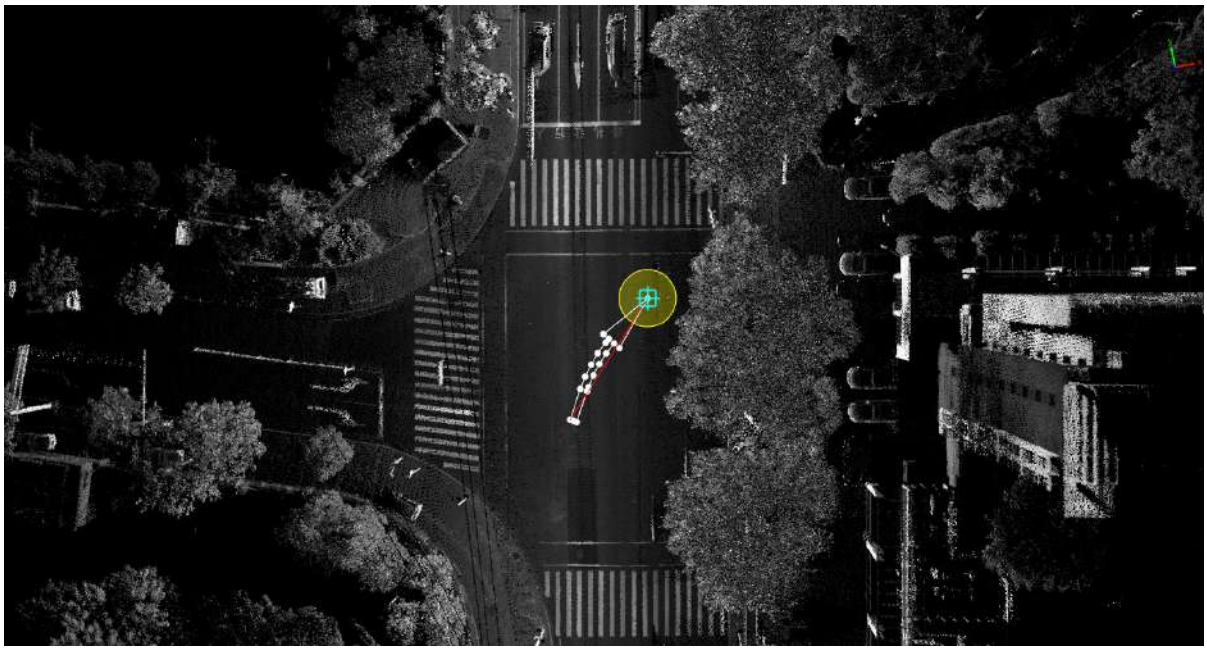
> - The key points in the arrow icons need to be aligned with the actual point cloud.
> - The red point is the first key point and the green point is the second key point.

2.**Left mouse click** to select right merge arrow point cloud key point, details below, icon in **first key point**.

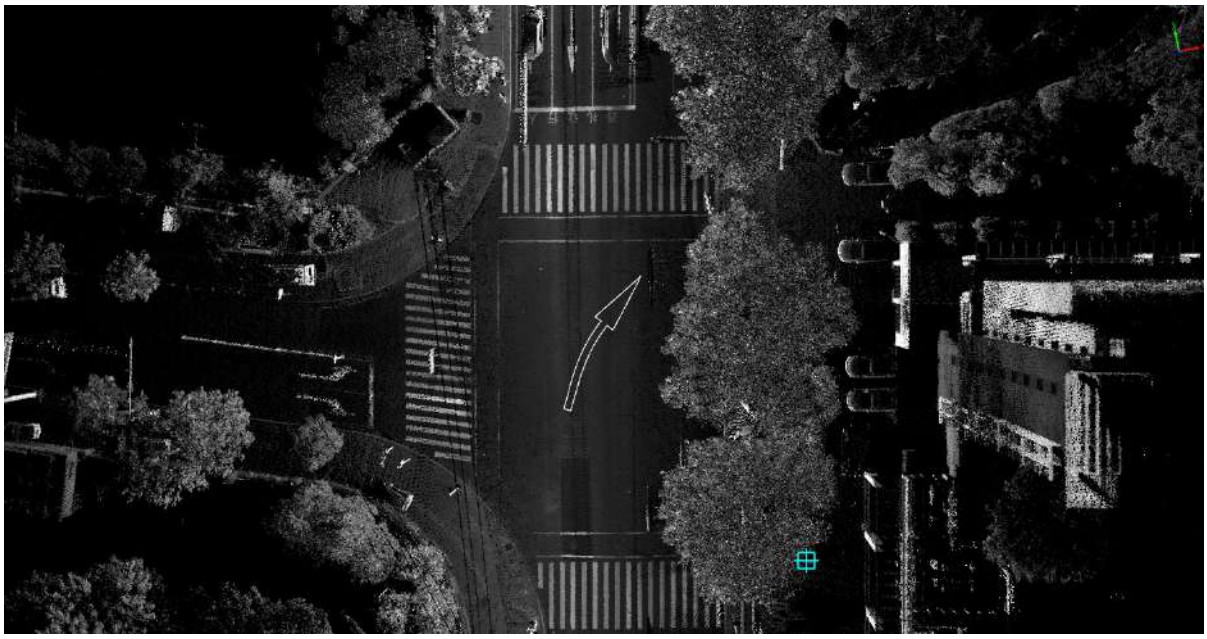


First key point

3.Move the mouse to see how the **template** fits into the **actual point cloud**, **left mouse click** to select the **second key point** to complete the creation of the merging arrow to the right, details are as follows, icon in **second key point**.



Second key point



Left merge arrow drawing

Add Right/Turn Round

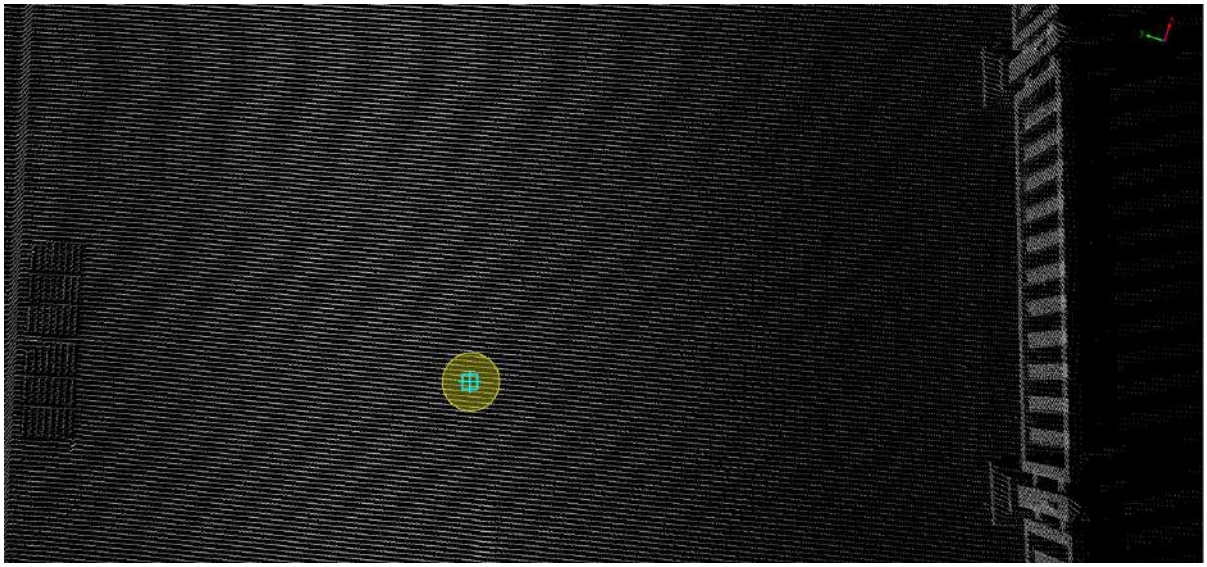
Function Description： The turn right/U-turn arrow indicates that the current lane allows a turn right or U-turn. This tool is used to manually click on the point cloud to vectorize the turn right /U-turn arrows.

Steps

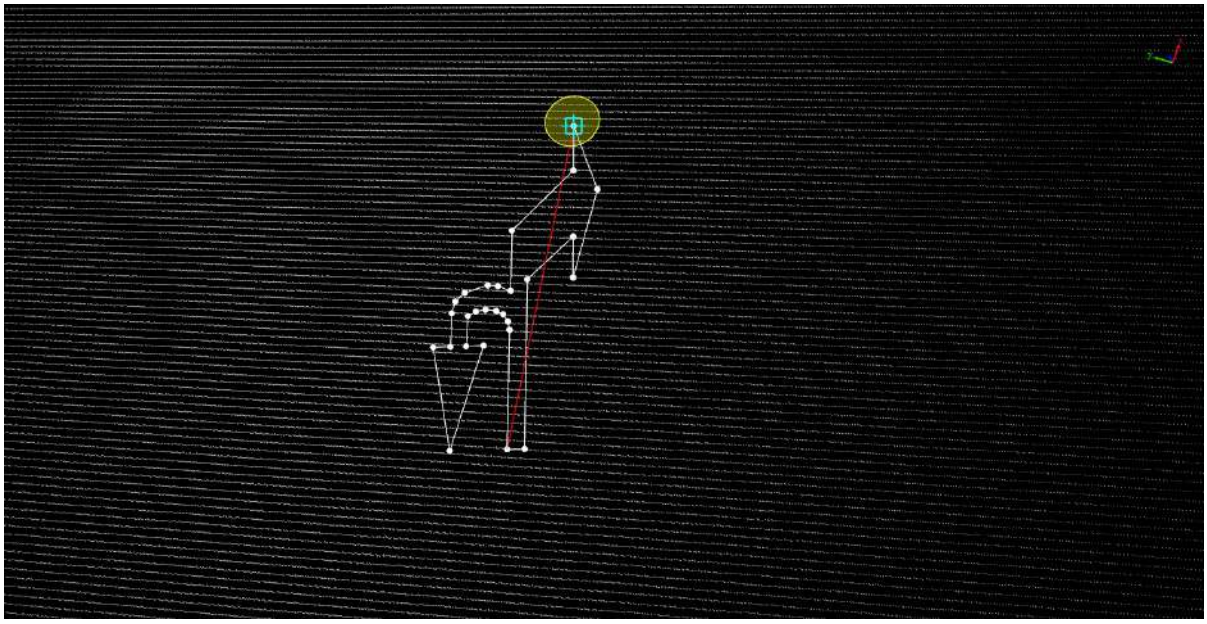
1.Click the **Add Right/Turn Round**  button in the Road Surface panel.

> - The key points in the arrow icons need to be aligned with the actual point cloud.
> - The red point is the first key point and the green point is the second key point.

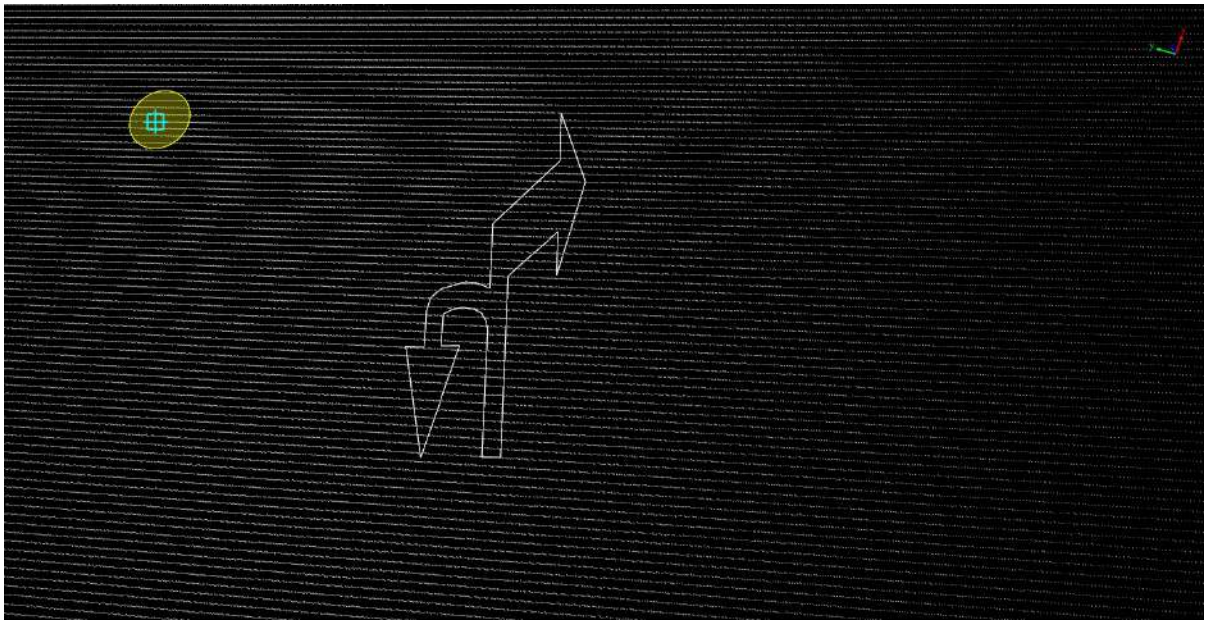
2.**Left mouse click** to select the point cloud key point, the details are as follows.



First key point



Second key point



Turn right /U-turn arrow drawing

Add Straight/Right/Turn Round

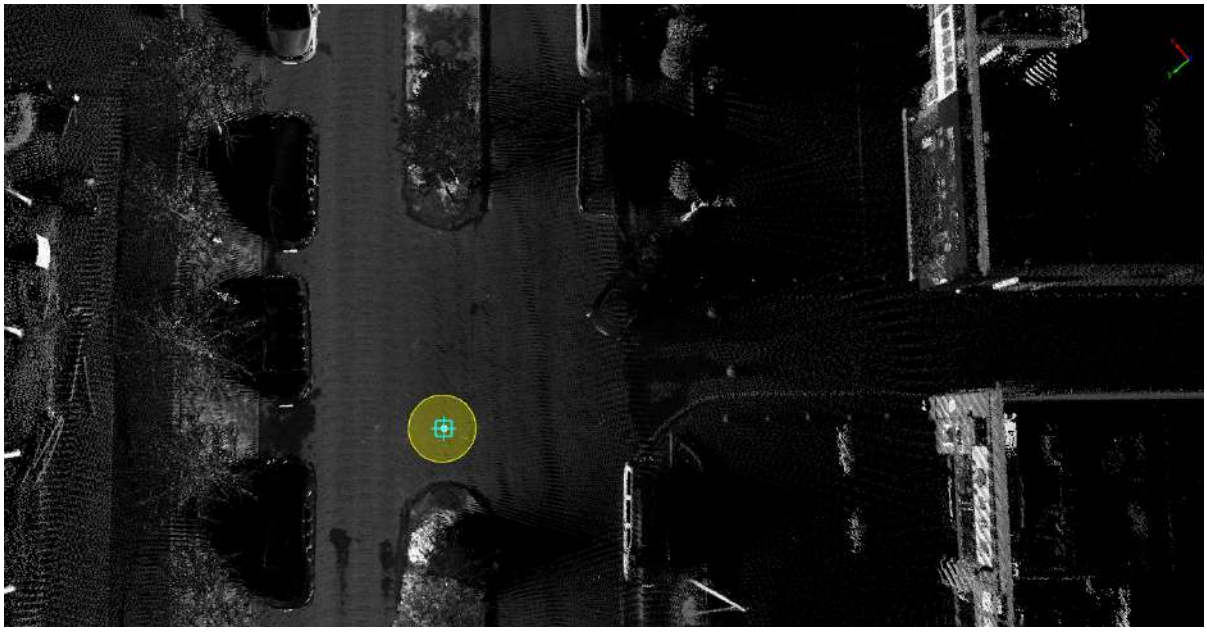
Function Description： Straight/turn right/U-turn arrows indicate that the current lane allows going straight, right-turn and U-turn. This tool is used to manually click on the point cloud to vectorize the straight/turn right/U-turn arrows.

Steps

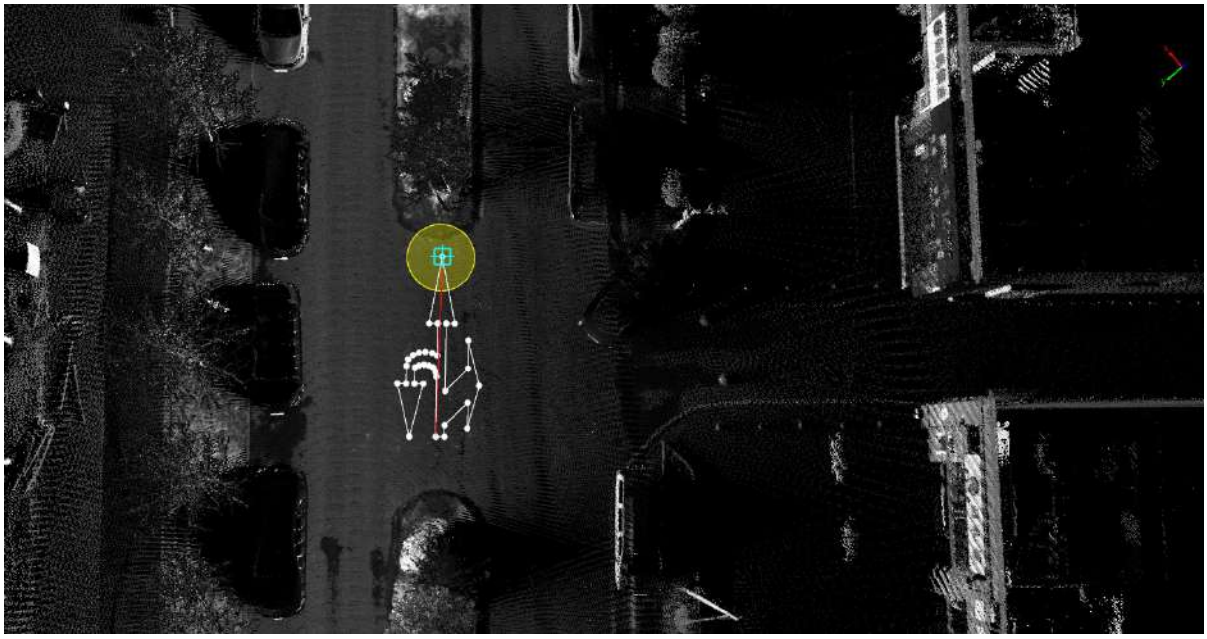
1.Click the **Add Straight/Right/Turn Round**  button in the Road Surface panel.

> - The key points in the arrow icons need to be aligned with the actual point cloud.
> - The red point is the first key point and the green point is the second key point.

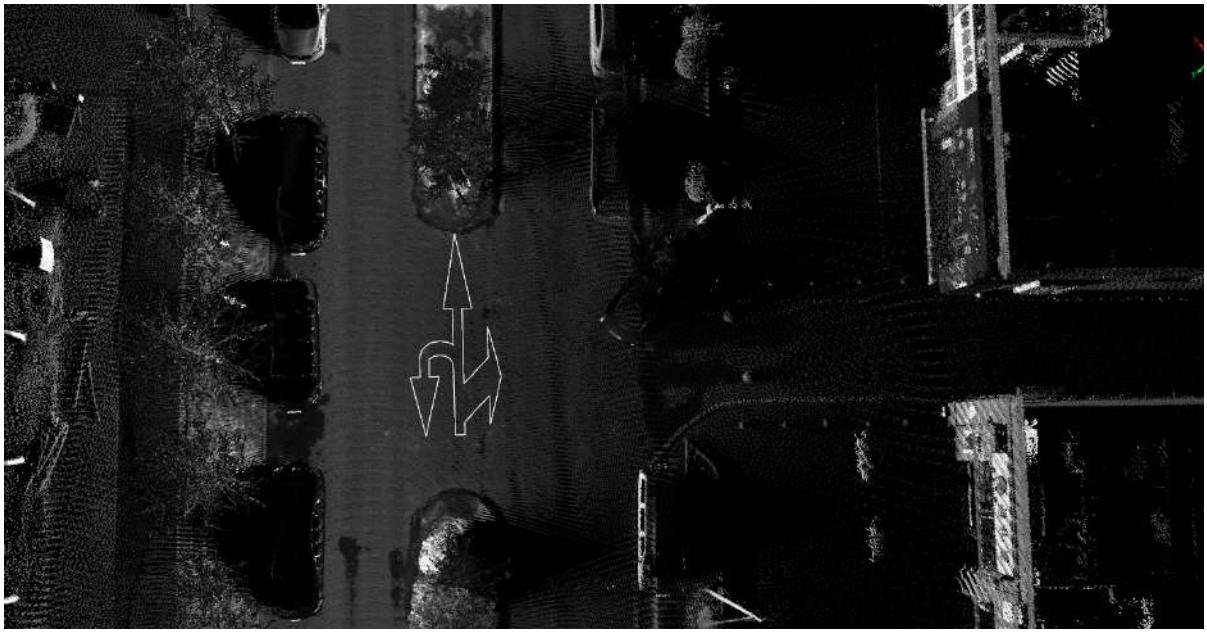
2.**Left mouse click** to select the point cloud key points, the details are as follows, the icon of **first key point**.



First key point



Second key point



Straight/turn right/U-turn arrow drawing

Add Left/Right/Turn Round

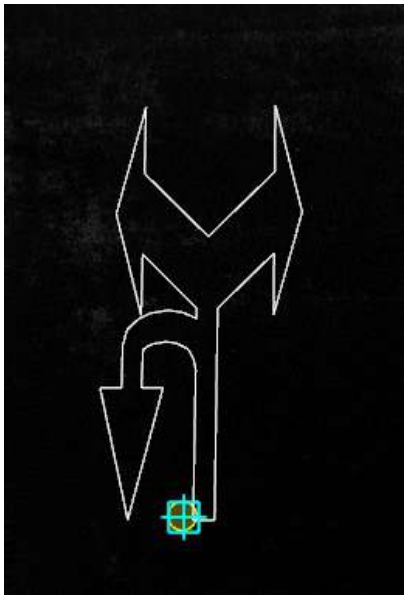
Function Description： Turn left/turn right/U-turn arrows indicate that the current lane can turn left, right, or turn around. This tool is used to manually click on the point cloud for vectorization of turn left/turn right/U-turn arrows.

Steps

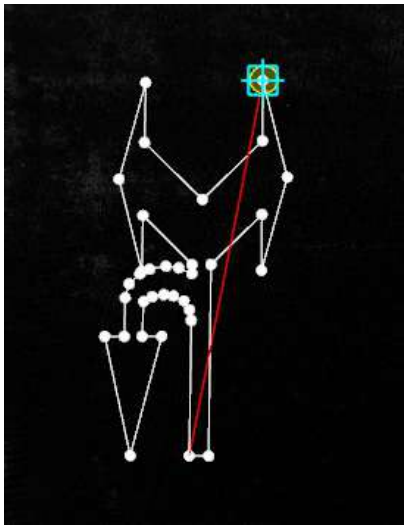
1.Click the **Add Left/Right/Turn Round**  button in the Road Surface panel.

> - The key points in the arrow icons need to be aligned with the actual point cloud point cloud.
> - The red point is the first key point and the green point is the second key point.

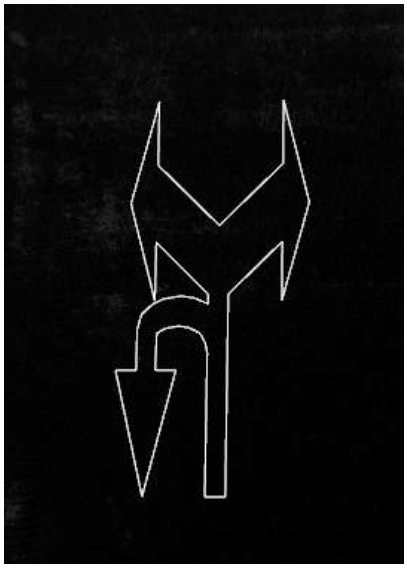
2.**Left mouse click** to select the point cloud key points, the details are as follows, the icon of **first key point**.



First key point



Second key point



Turn left/turn right/U-turn arrow drawing

Add CrossWalk

Function Description：This tool enables the outlining of crosswalks. Use three key points, manually click the point cloud, and make the vectorization of the crosswalk.

Steps

1.In the drop-down icon in the Road Surface panel, click the **Add CrossWalk**  button.

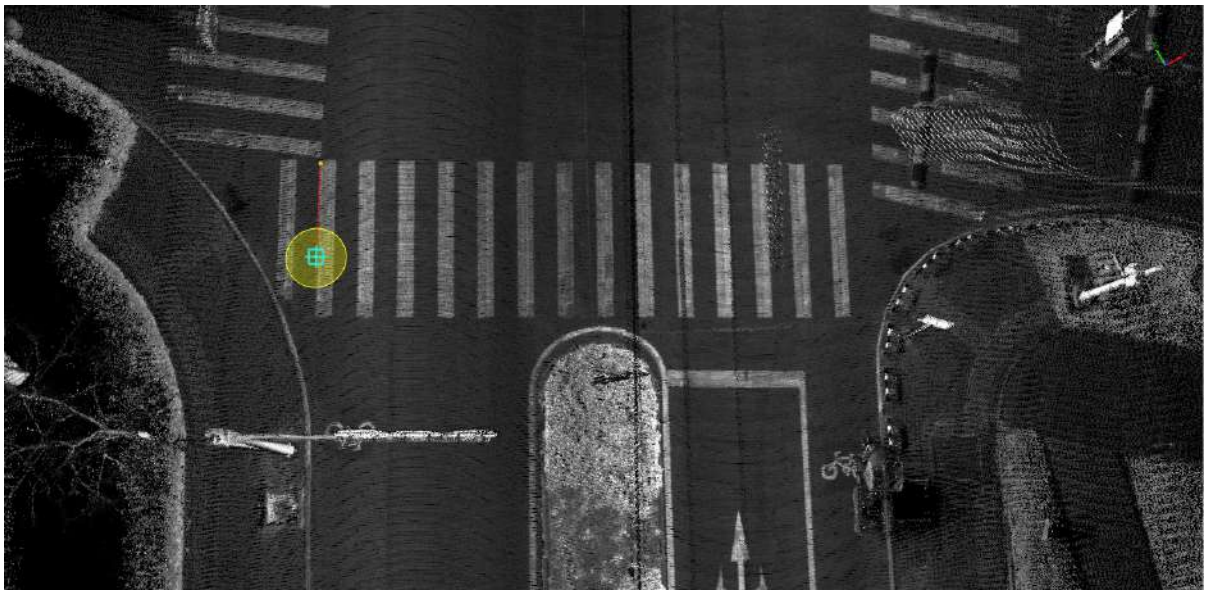
> - The **yellow point** in the figure below is the key point, and the key point needs to be aligned with the actual point cloud.
> - **Yellow point** position is the **position of the left click**.

2.3D window tips.：

Shortcut name	Role
C	Switch between standard/custom drawing modes
H	Turn off/on top left corner alert
Current model	Current model

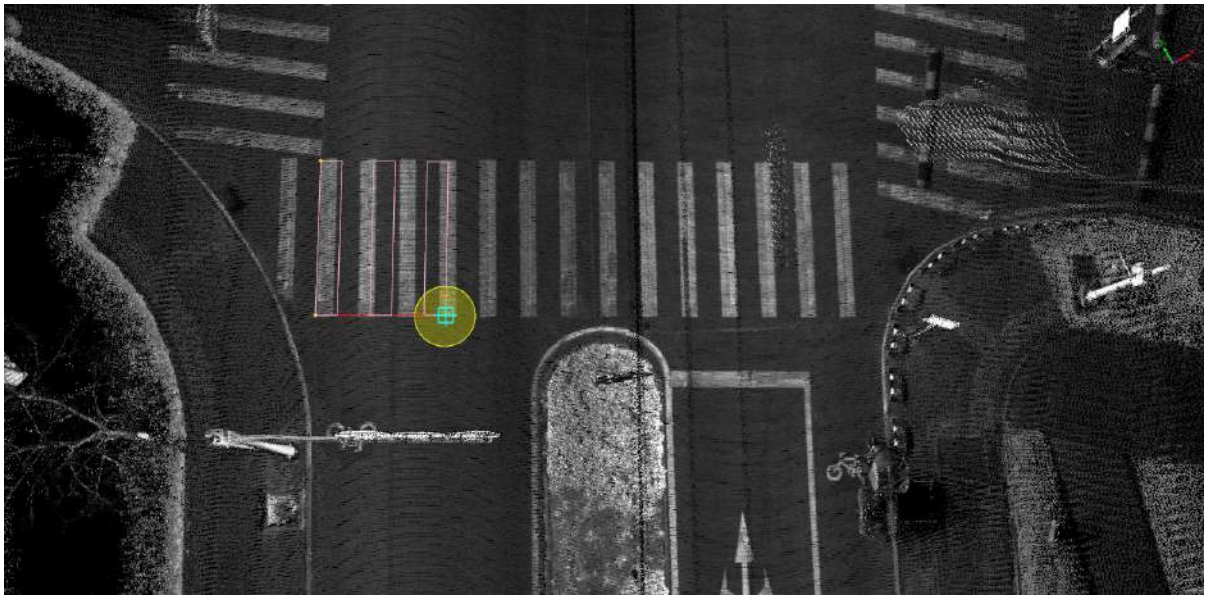
3.Standard model：

（1）**Left mouse click** to select the **first key point** of the crosswalk point cloud, the details are as follows, the **yellow point** position is the **first key point** in the icon.



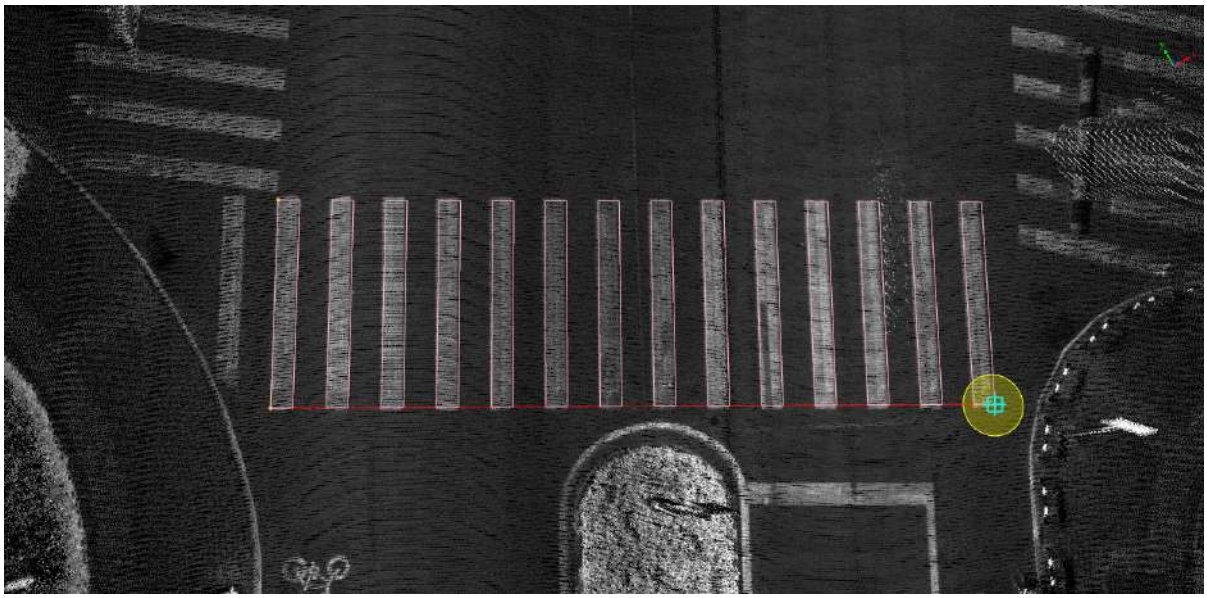
First key point

（2）Move the mouse to check the fit between the **template** and the **actual point cloud**, left click **the second key point**, and determine **crosswalk width**, The details are as follows, the **yellow point** is the **second key point in the icon**.

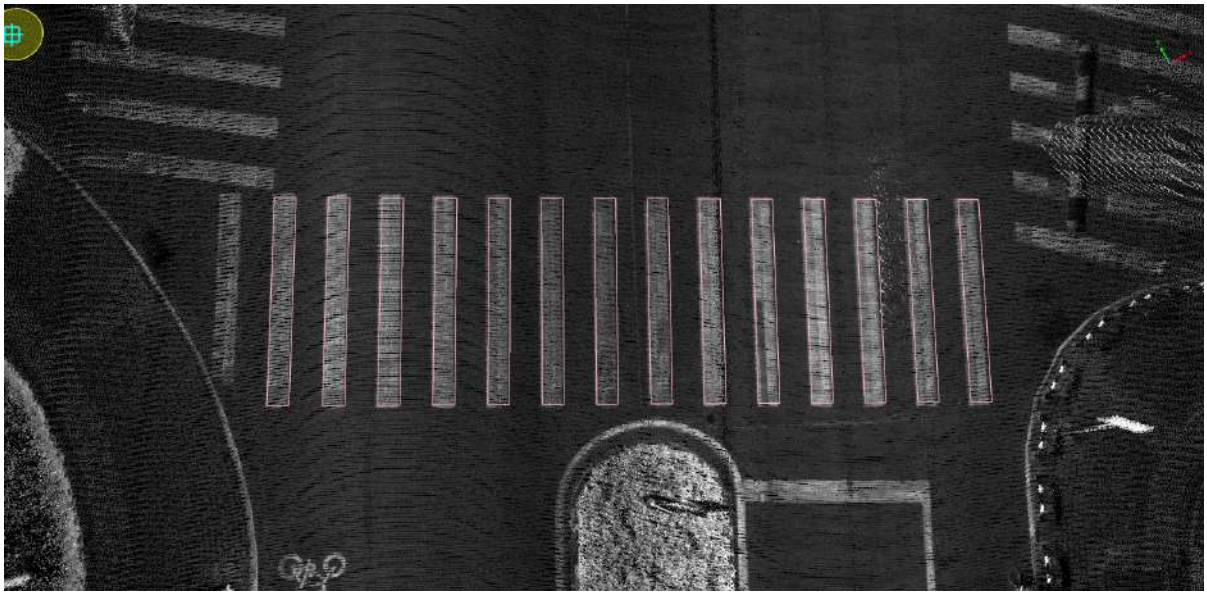


Second key point

（3）Move the mouse to check the fit of the **template** and the **actual point cloud**, Left click **the third key point**, and determine the **crosswalk length**, The details are as follows, the **yellow point** is the **third key point in the icon**.



Third key point

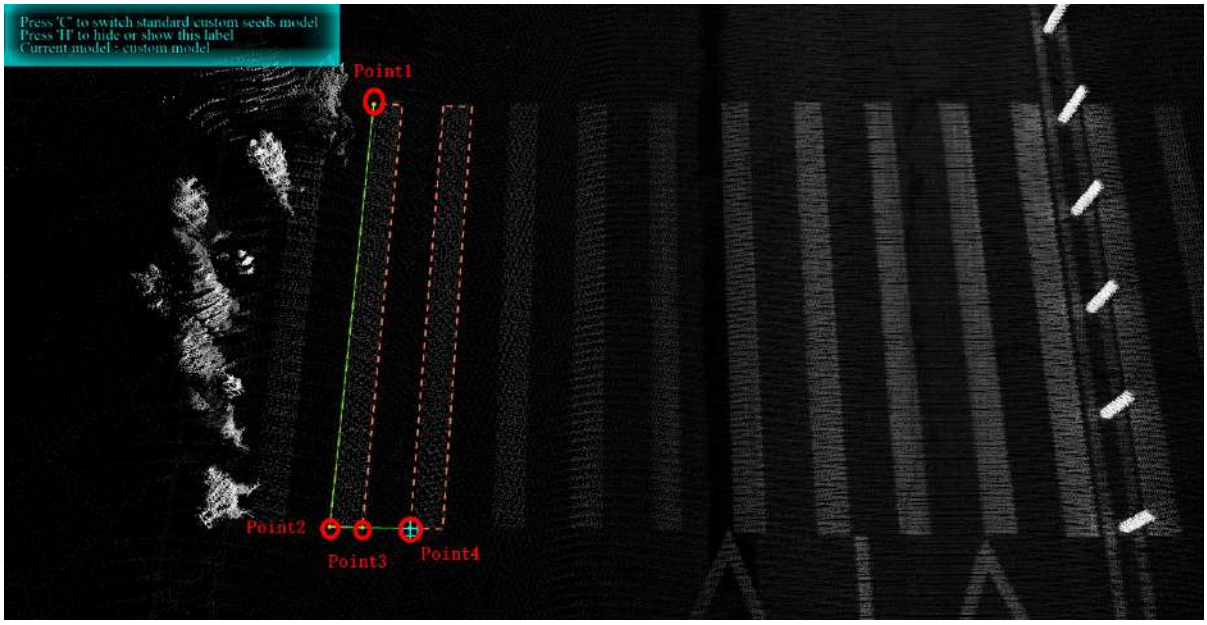


Crosswalk drawing

4.Customised models：

（1）Use the shortcut key **C** to switch the mode to custom model, the current model is displayed as custom model.

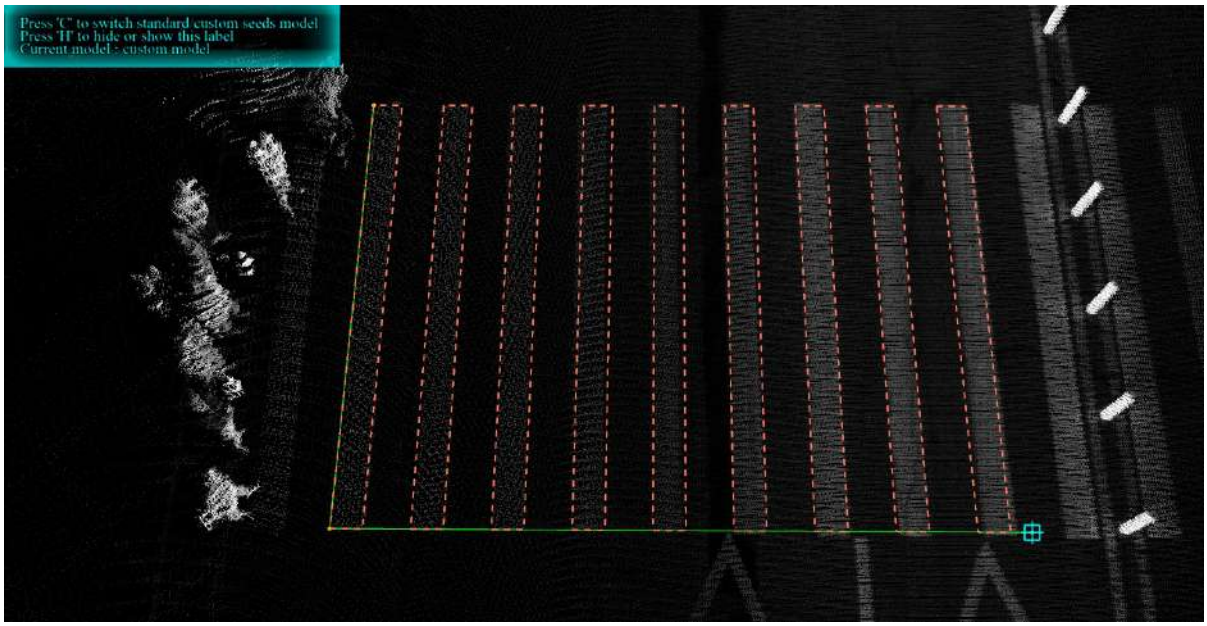
（2）Click with the left mouse button on each of the four key points shown below



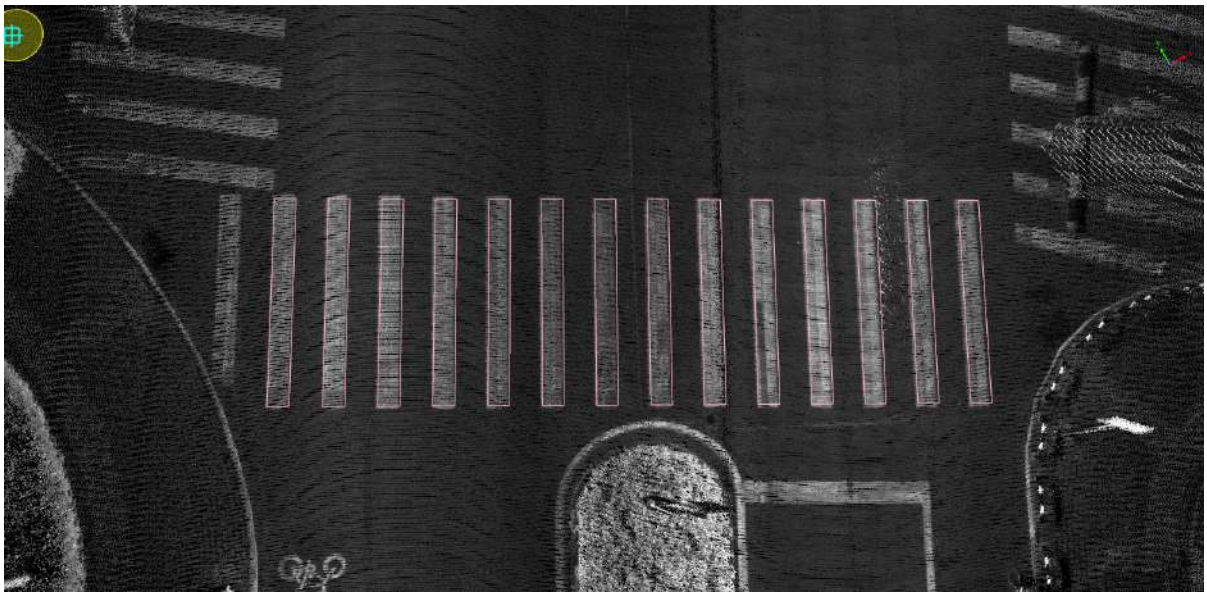
Custom Mode Key Points

> Notes：
>
> - For the first two points, click on the long side of the outside of the pedestrian crossing
> - for the third point, click on the intersection of the pedestrian crossing in the same line as the second point, the first three points define the shape of a pedestrian crossing
> - For the fourth point, click on the corner point where the adjacent pedestrian crossing is in line with points 2 and 3 in the direction parallel to the pedestrian crossing, thus determining the spacing of the pedestrian crossing

（3）Finally, drag the mouse to pull out the live vector and double-click to end the drawing when it fits with the pedestrian crossing point cloud



Custom mode real time vector



Pedestrian crossing drawing

Add Longitudinal Speed Reduction

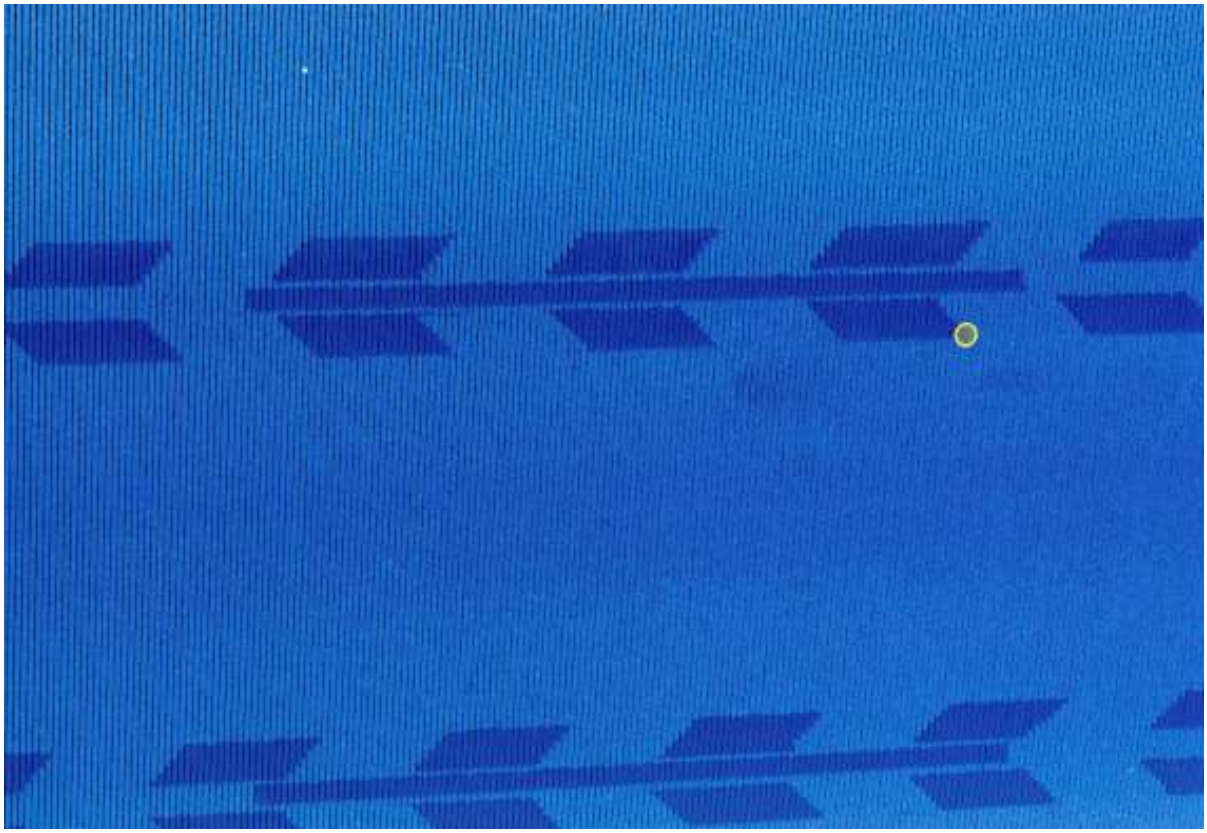
Function Description： This tool can be used to outline Vertical Deceleration Markings. For three key points, manual point selection point cloud for vectorization of vertical deceleration markings.

Steps

1.In the Road Surface panel, click the drop-down icon on the right, and select the **Add Longitudinal** button.

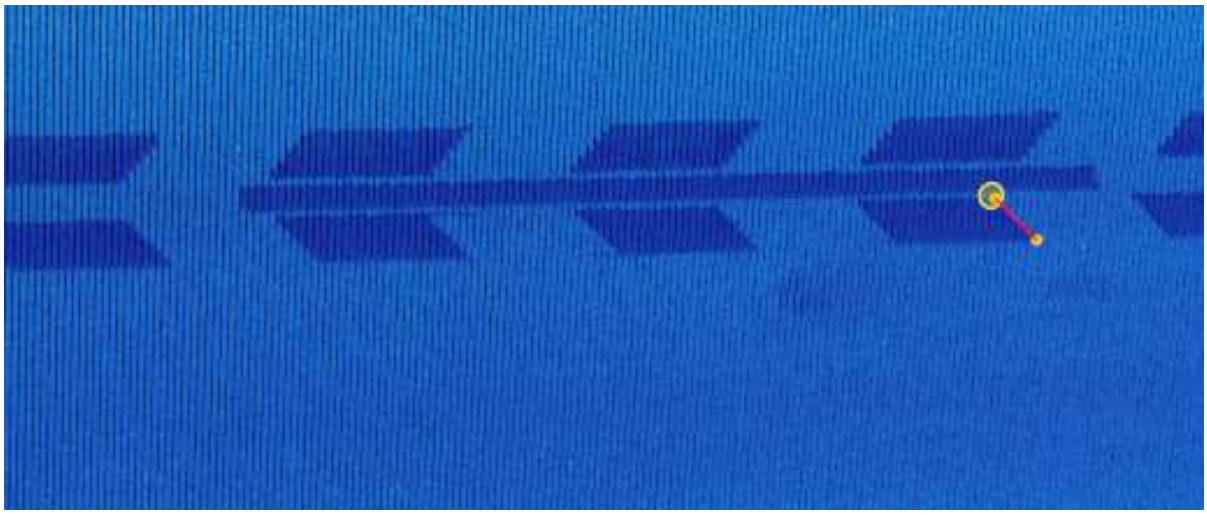
> - The **yellow point** in the figure below is the key point, which needs to be aligned with the actual point cloud.
> - **Yellow point** position is **left mouse click** position button click.

2.**Left mouse click** to select the **first key point** of the vertical deceleration markings point cloud, the details are as follows, **yellow point** position is the **first key point** in the icon.



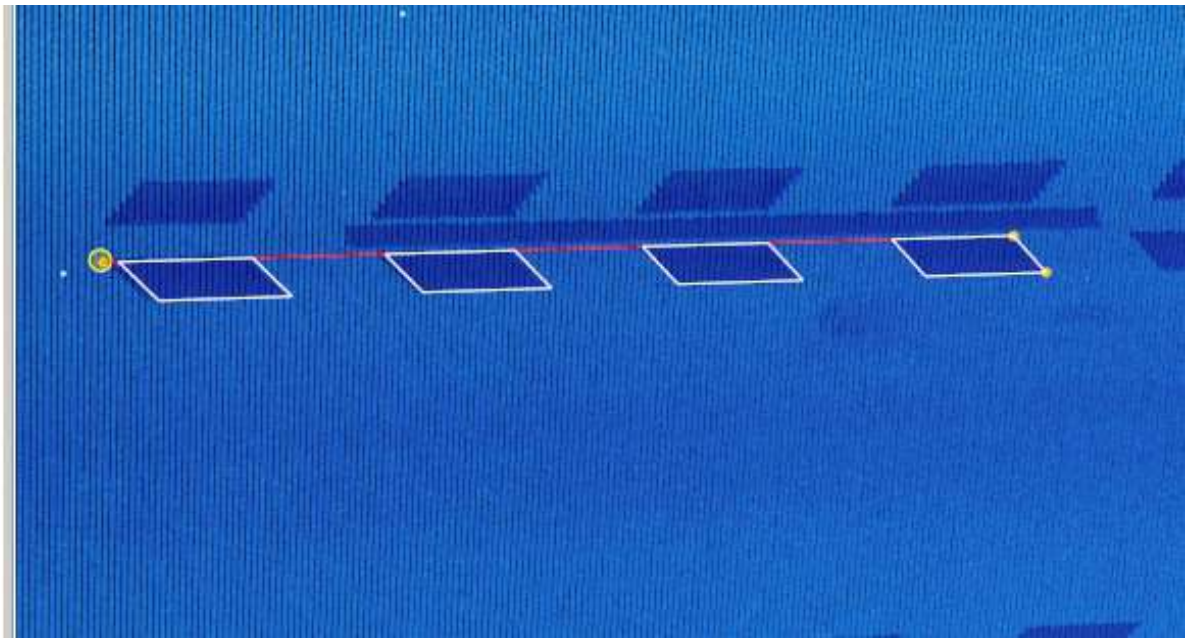
First key point

3.Move the mouse to view **template** and **actual point cloud** fit, **left mouse button click second key point**, determine **width**, details are as follows, **yellow point** is the icon in **second key point**.

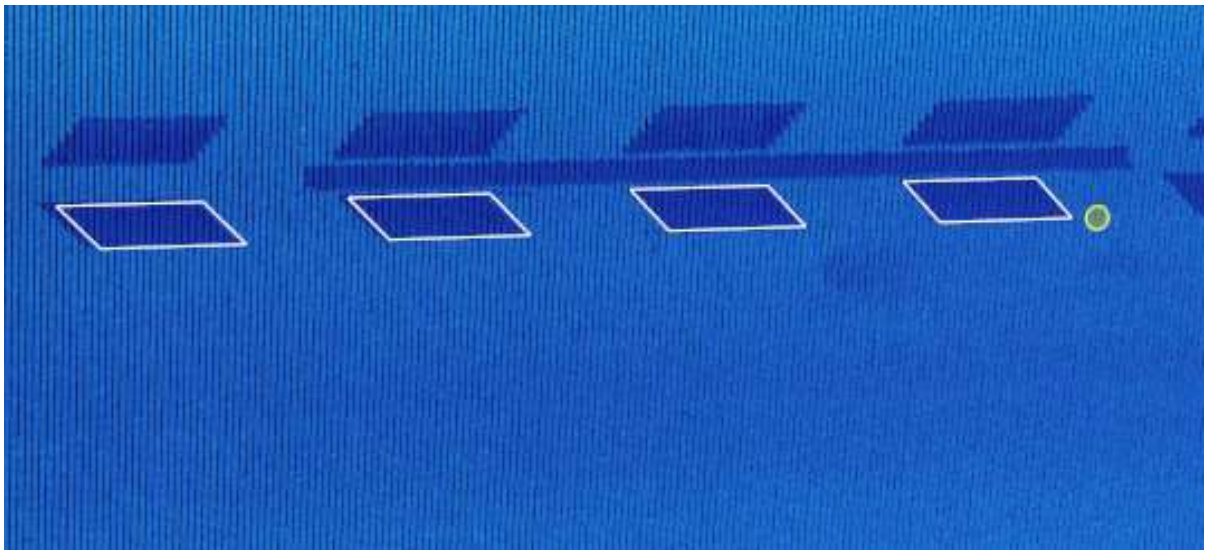


Second key point

4.Move the mouse to view **template** and **actual point cloud** fit, **left mouse button click third key point**, determine **length**, and the user can complete the production, the details are as follows, **yellow point** that is, the icon in **third key point**.



Third key point



Vertical deceleration markings drawing

Add Speed Reduction

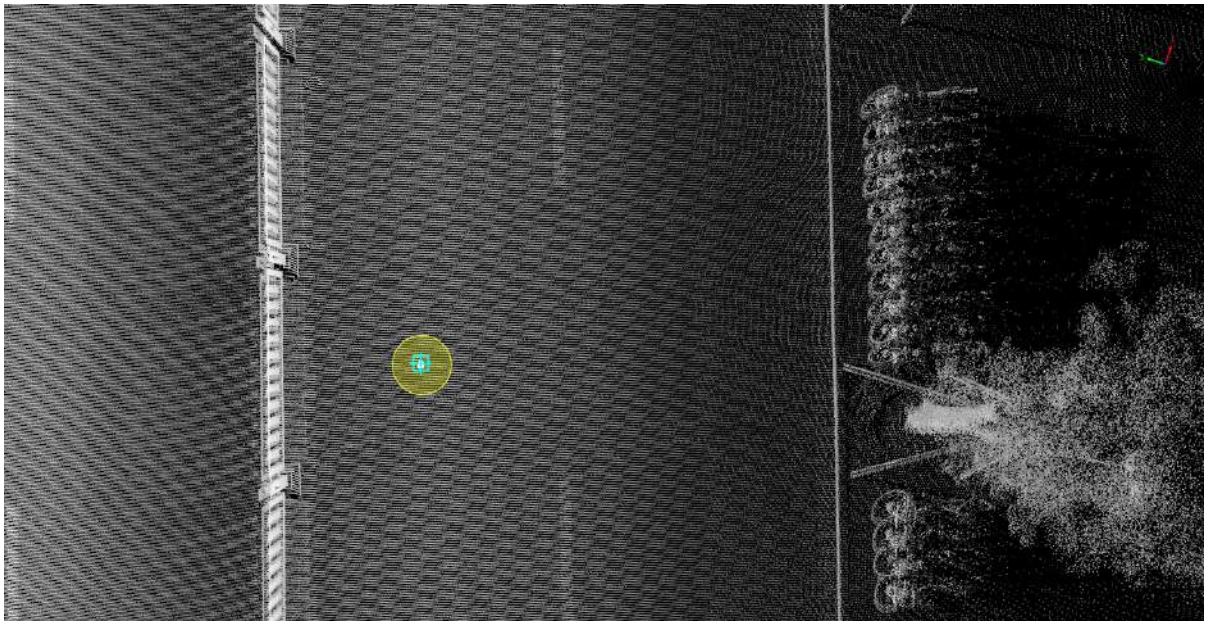
Function Description： This tool can be used to use two key points, manually click on the point cloud, and make the vectorized production of the reduce speed mark.

Steps

1.In the Road Surface panel, click the drop-down icon on the right, and select the **Add Speed Reduction**  button.

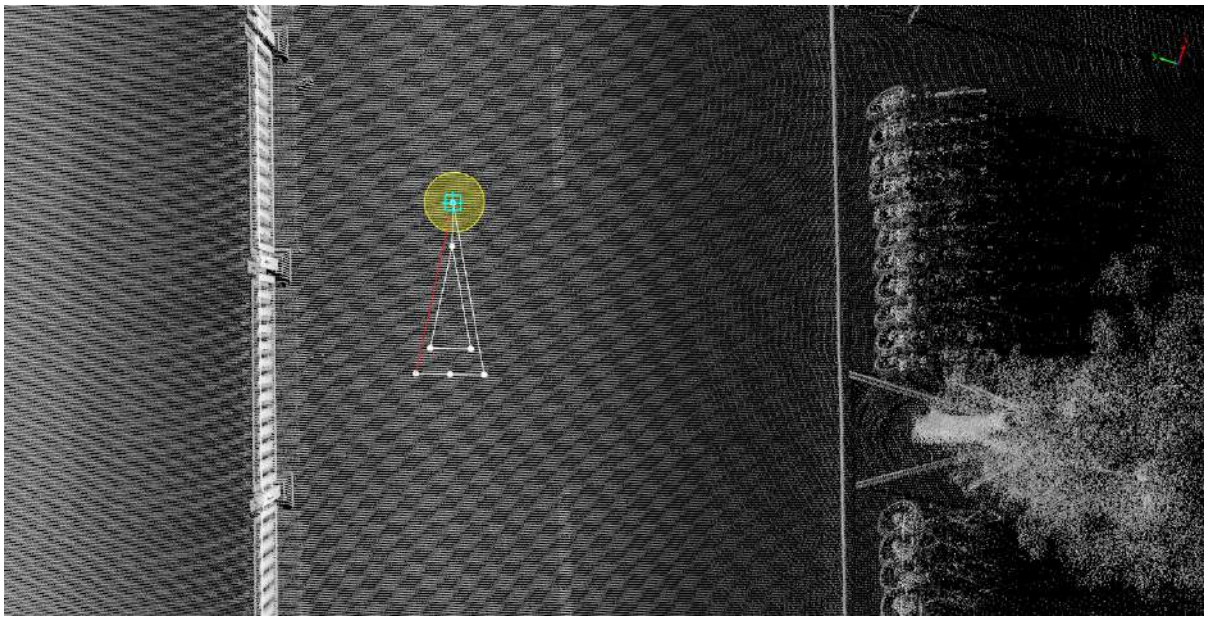
> - The key points in the arrow icons need to be aligned with the actual point cloud.
> - The red point is the first key point and the green point is the second key point.

2.**Left mouse click** to select the **first key point** of the reduce speed marker point cloud, the details are as follows, the icon in the **first key point**.

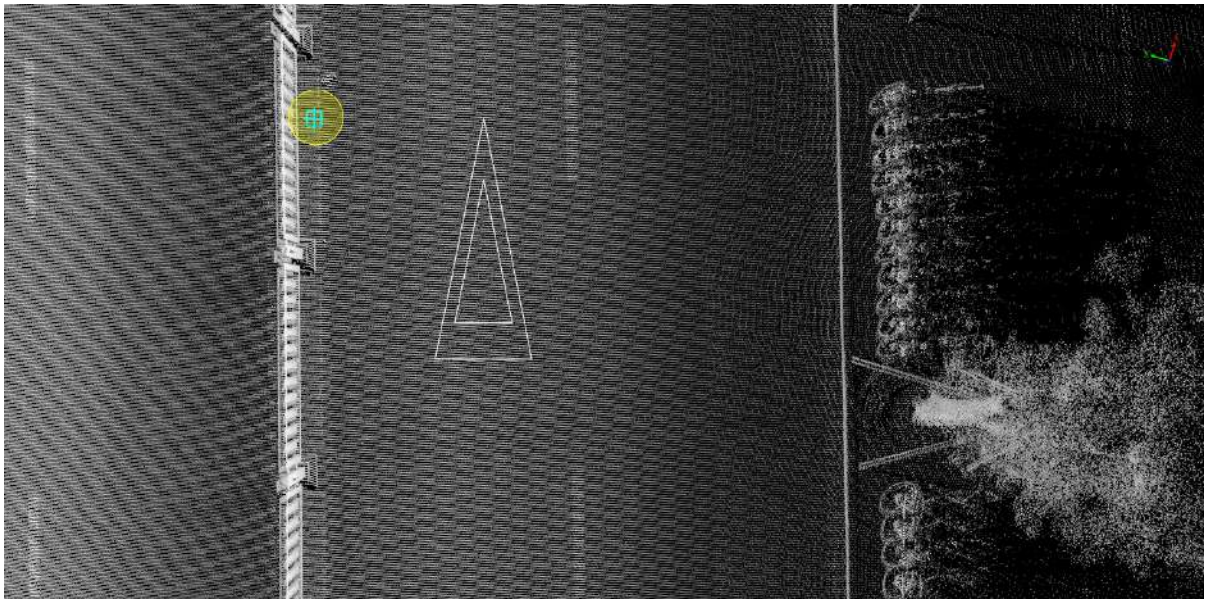


First key point

3.Move the mouse to view **template** and **actual point cloud** fit, **left mouse button click** click **second key point**, determine **reduce speed to let the line mark size**, and the user can complete the production, the details are as follows. The icon in **second key point**.



Second key point



Reduce speed mark drawing

Add CrossWalk Warning

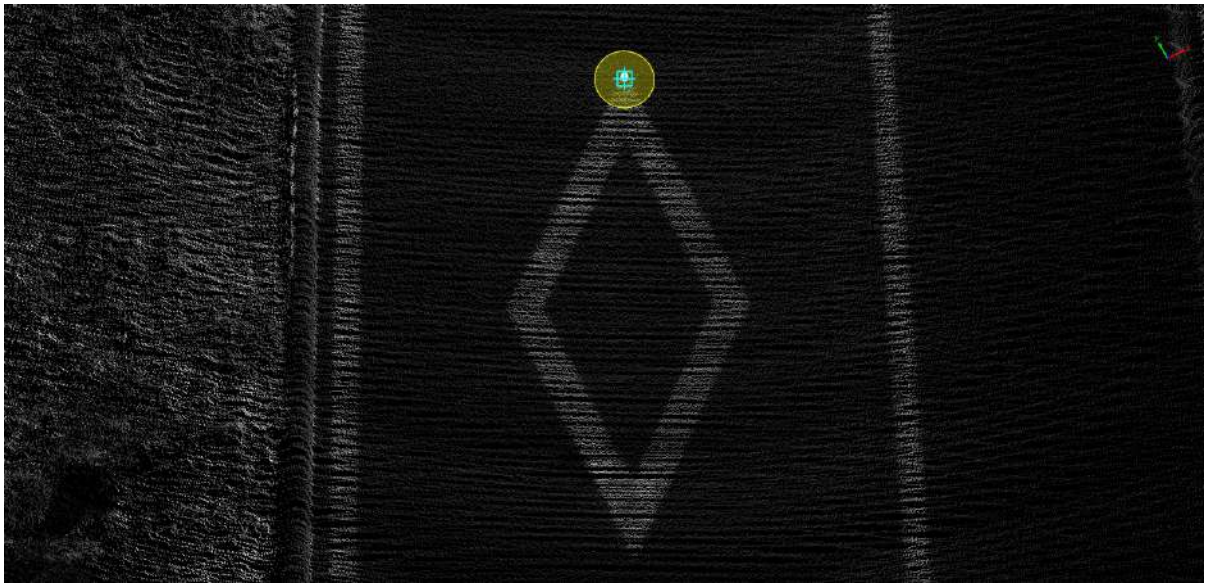
Function Description： The crosswalk warning sign is used to predict the crosswalk appearing ahead. This tool can be applied using two key points. Manually click on the point cloud to vectorize the crosswalk warning.

Steps

1.Click the drop-down arrow on the right in the Road Surface panel, and select the **Add CrossWalk Warning**  button.

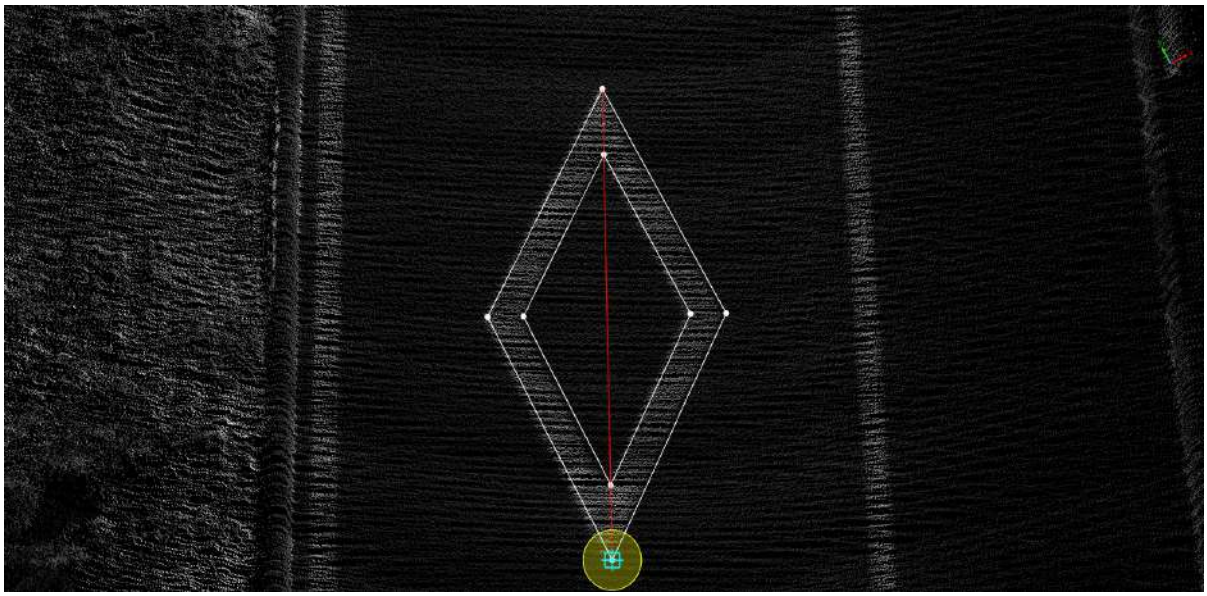
> - The key points in the arrow icons need to be aligned with the actual point cloud.
> - The red point is the first key point and the green point is the second key point.

2.**Left mouse click** to select the **first key point** of the crosswalk warning point cloud, the details are as follows, the **yellow point** position is the **first key point** in the icon. First key poi

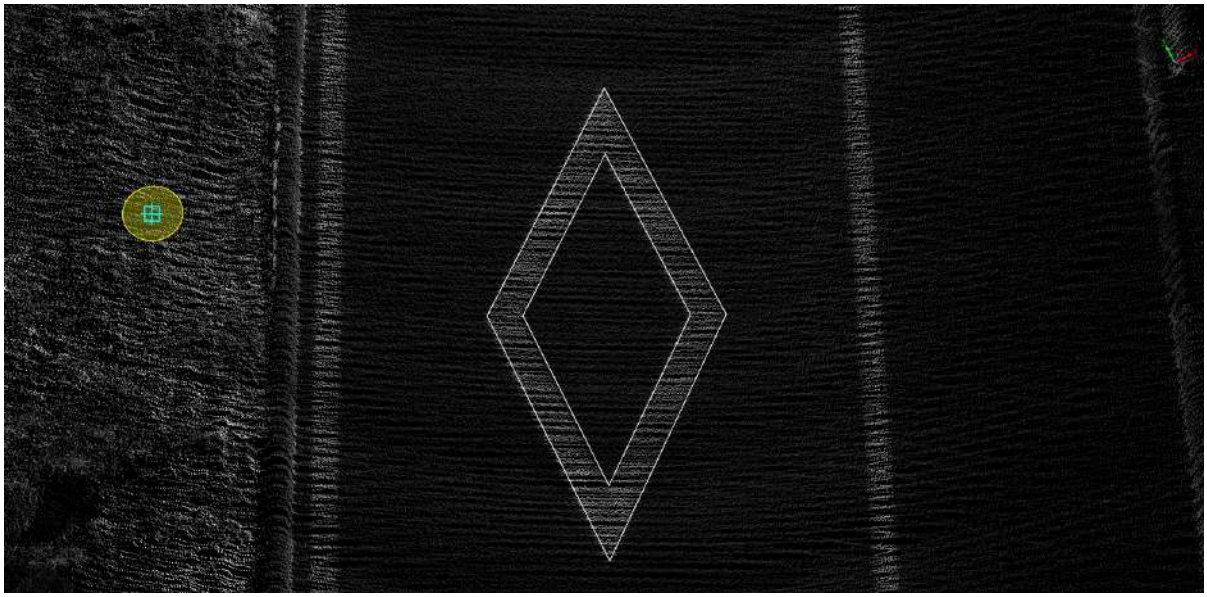


First key point

3.Move the mouse to check the fit between the **template** and the **actual point cloud**, Left click **the second key point** to determine the size of the **crosswalk warning** , and the user can complete the production, the details are as follows, the **yellow point** is the second key point in the icon.



Second key point



Crosswalk warning drawing

Add No U-Turn

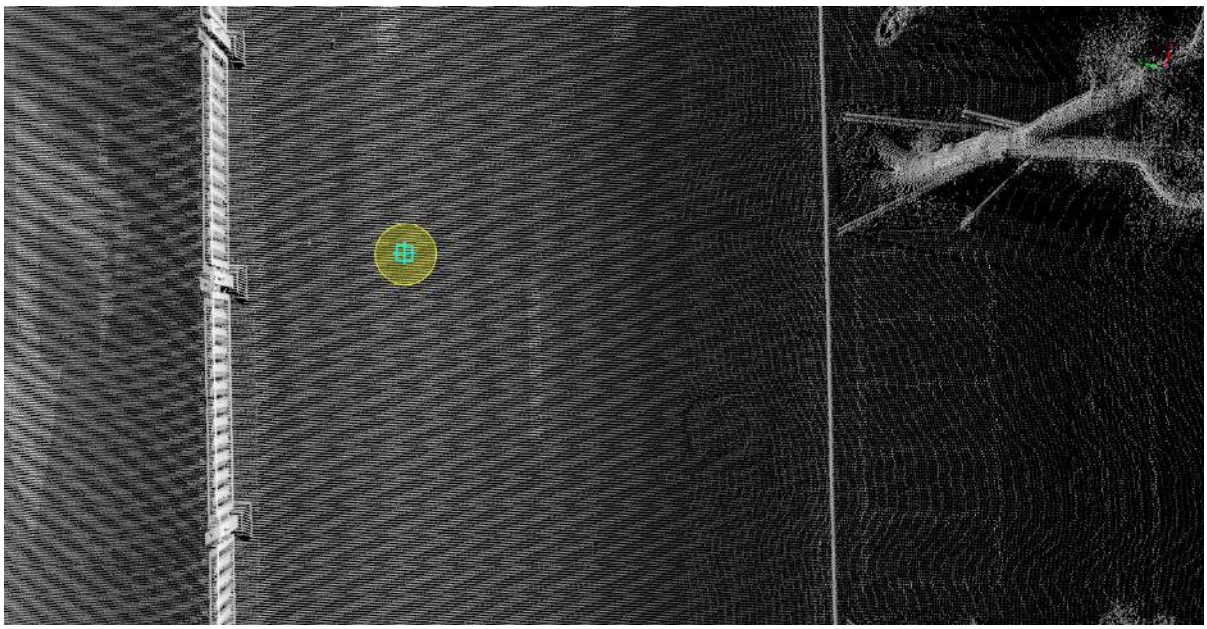
Function Description： The no U-turn marker indicates that the current lane does not allow U-Turns. This tool is used to manually click on the point cloud to create the vectorized production of the no U-turn marker.

Steps

1.Click the **Add No U-Turn**  button in the Road Surface panel.

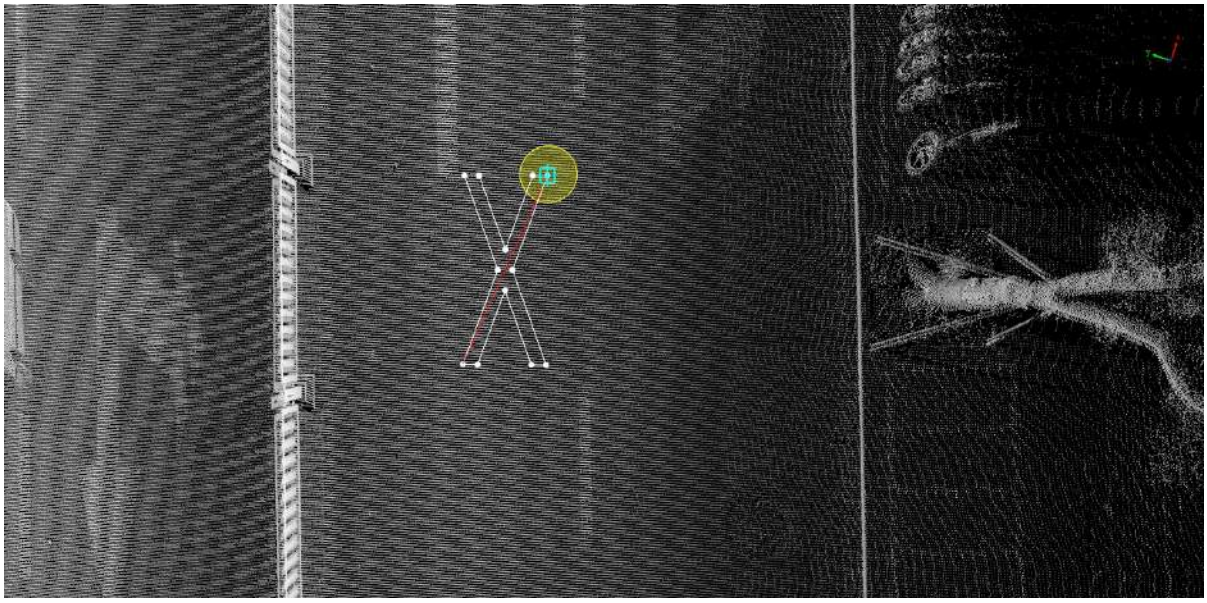
> - The **red points** in the arrow icons are key points, and the user needs to align the key points with the actual point cloud.
> - **Red point** position is **left mouse button click** position.

2.**Left mouse click** to select the key point of the No U-turn marker point cloud, details are as follows, **yellow point position is the first key point** in the icon.

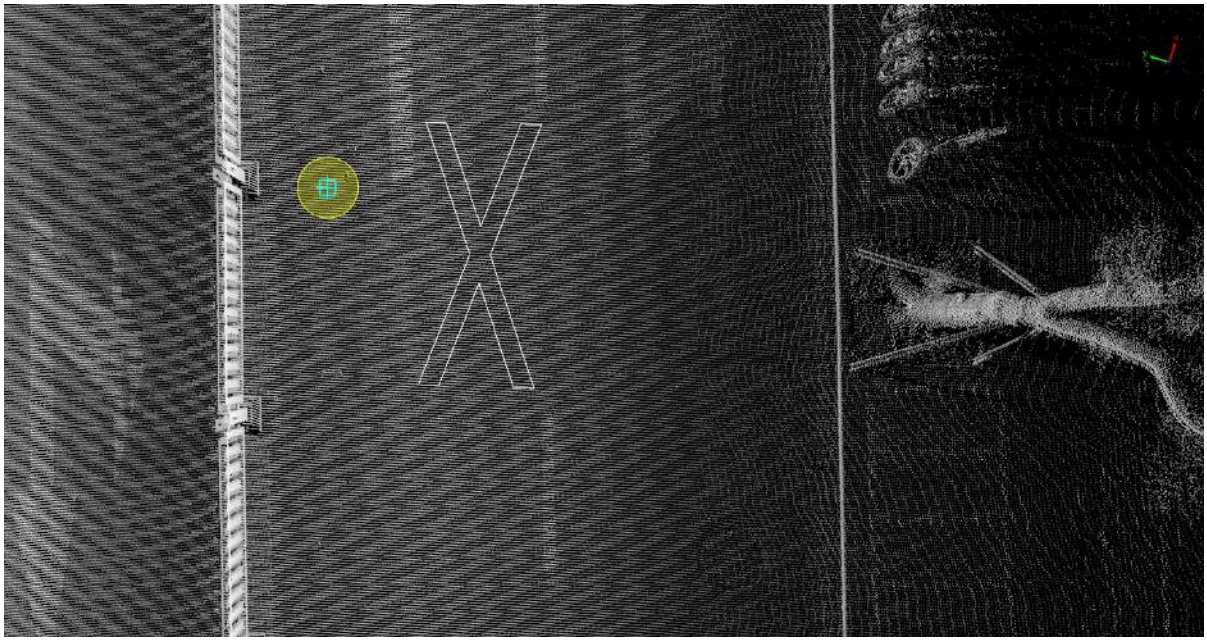


First key point

3.Move the mouse to see how the **template** fits into the **actual point cloud**, **left mouse click** to select the **second key point** to complete the production, details are as follows, **yellow point** is the **second key point** in the icon.



Second key point



No U-Turn marker drawing

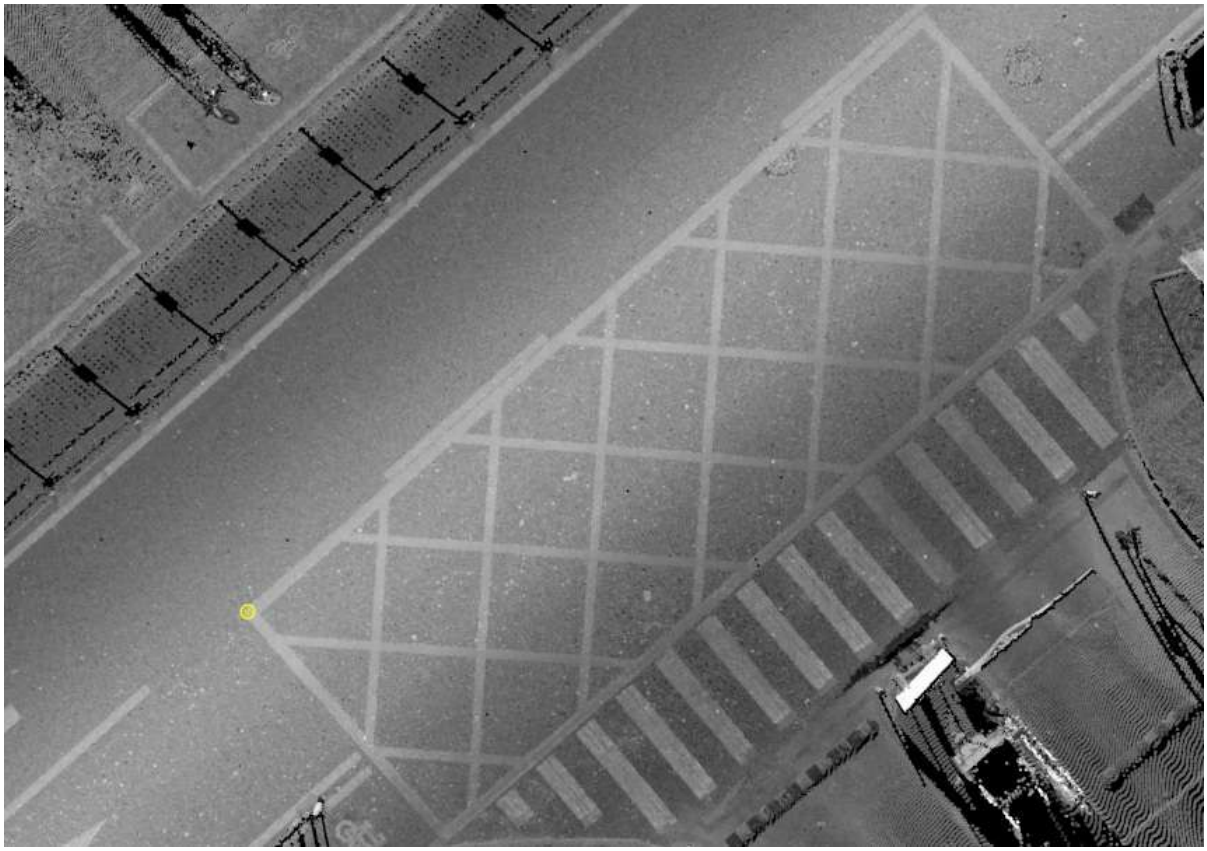
Add No-Parking

Function Description： This tool can be applied using three key points to determine the size of the outer rectangle, and internal auxiliary three key points to determine the interior. Manually click on the point cloud to vectorize the no-stop line.

Steps

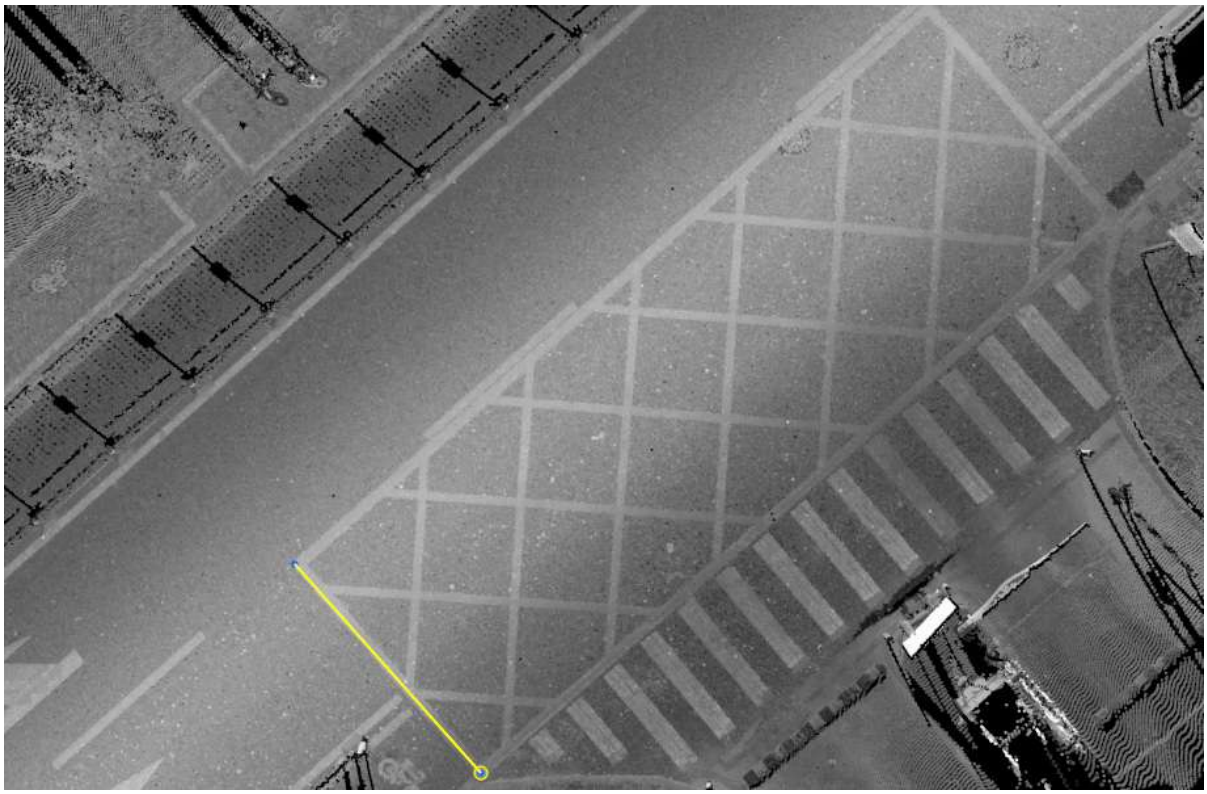
1.In the Road Surface panel, click the drop-down icon on the right, and select the **Add No-Parking**  button.

2.**Left mouse click** to select the **first key point** of the no-stop line point cloud, the details are as follows, **yellow point** position is the **first key point** of the icon.



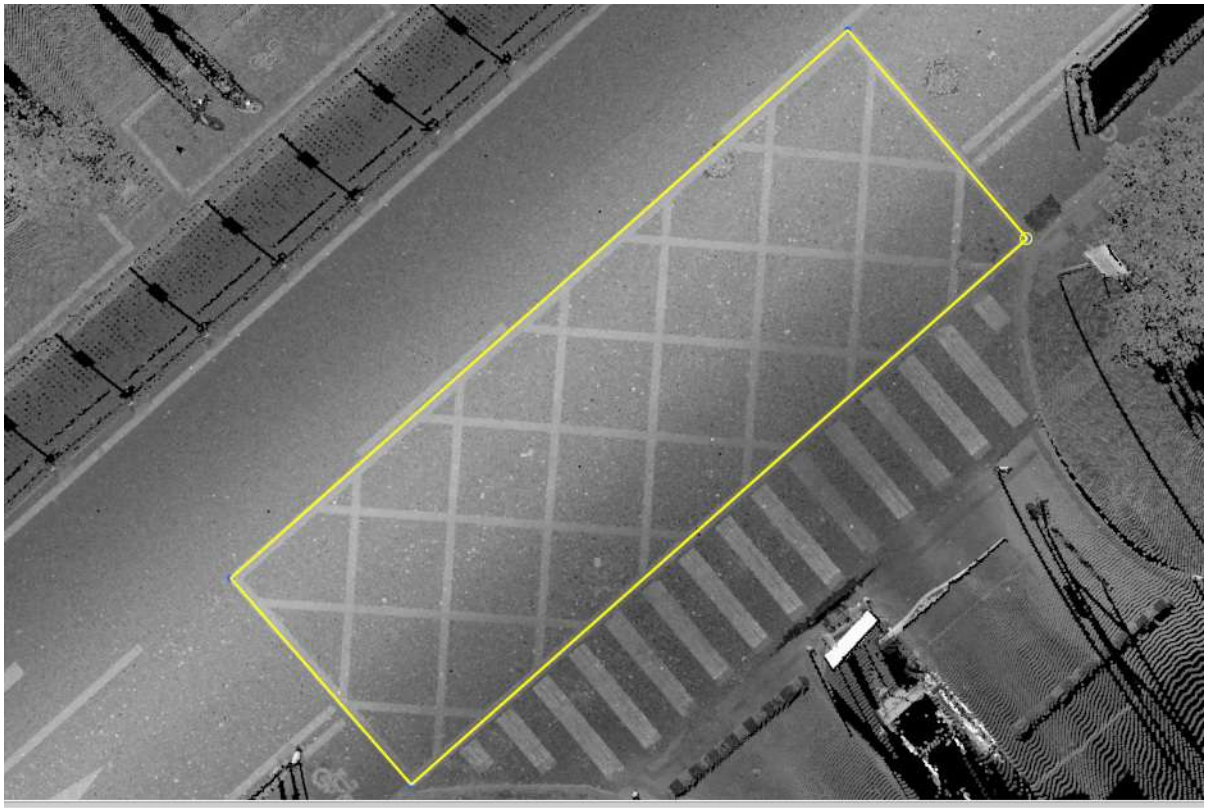
First key point

3.Move the mouse to view **template** and **actual point cloud** fit, **left mouse button click second key point**, determine **external rectangle width**, details are as follows, **yellow point** that is, the icon in **second key point**.



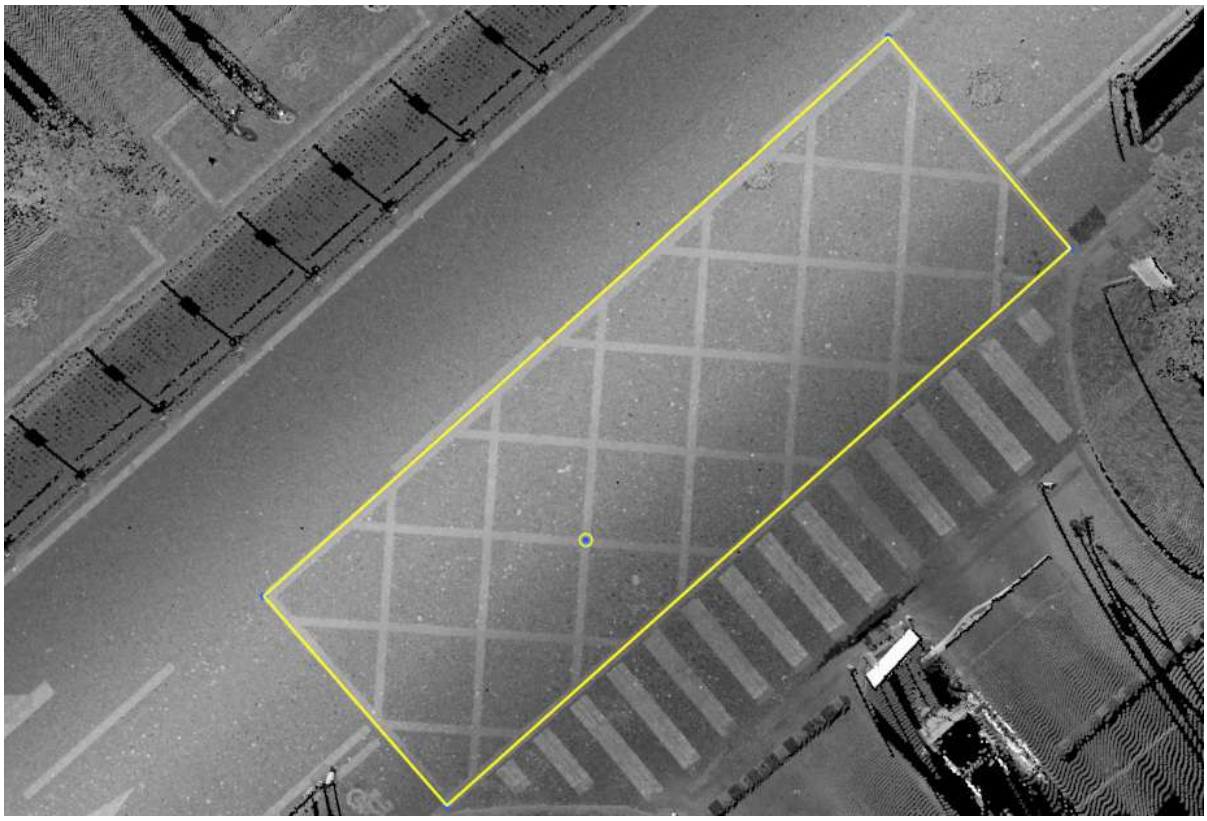
Second key point

4.Move the mouse to view the **template** and **actual point cloud** fit, **left mouse button click third key point** to determine the **external rectangle length**, details are as follows, **yellow point** that is, the icon in the **third key point**.



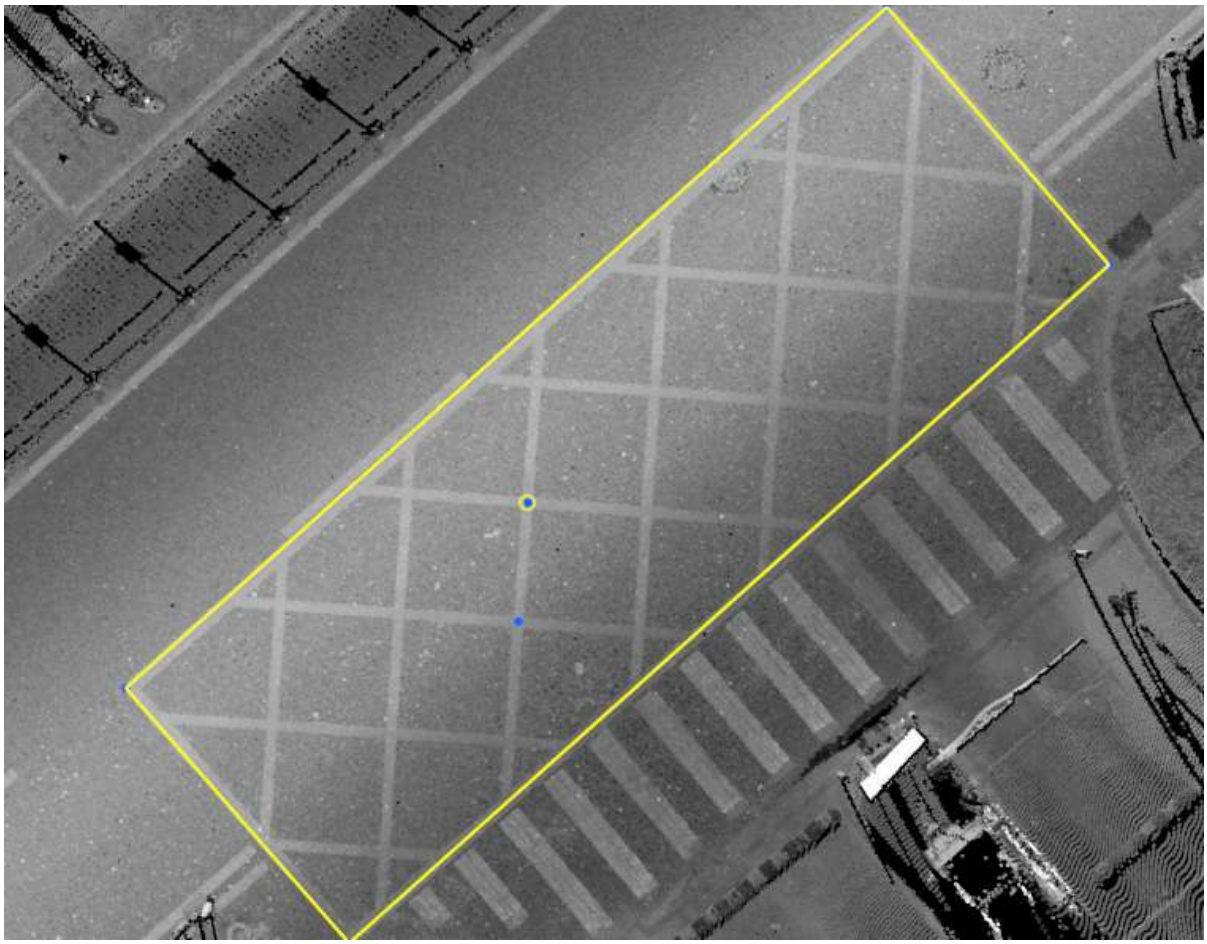
Third key point

5.Move the mouse to view the **template** and **actual point cloud** fit, **left mouse button click the fourth key point**, the details are as follows, **yellow point** is the **fourth key point** in the icon.



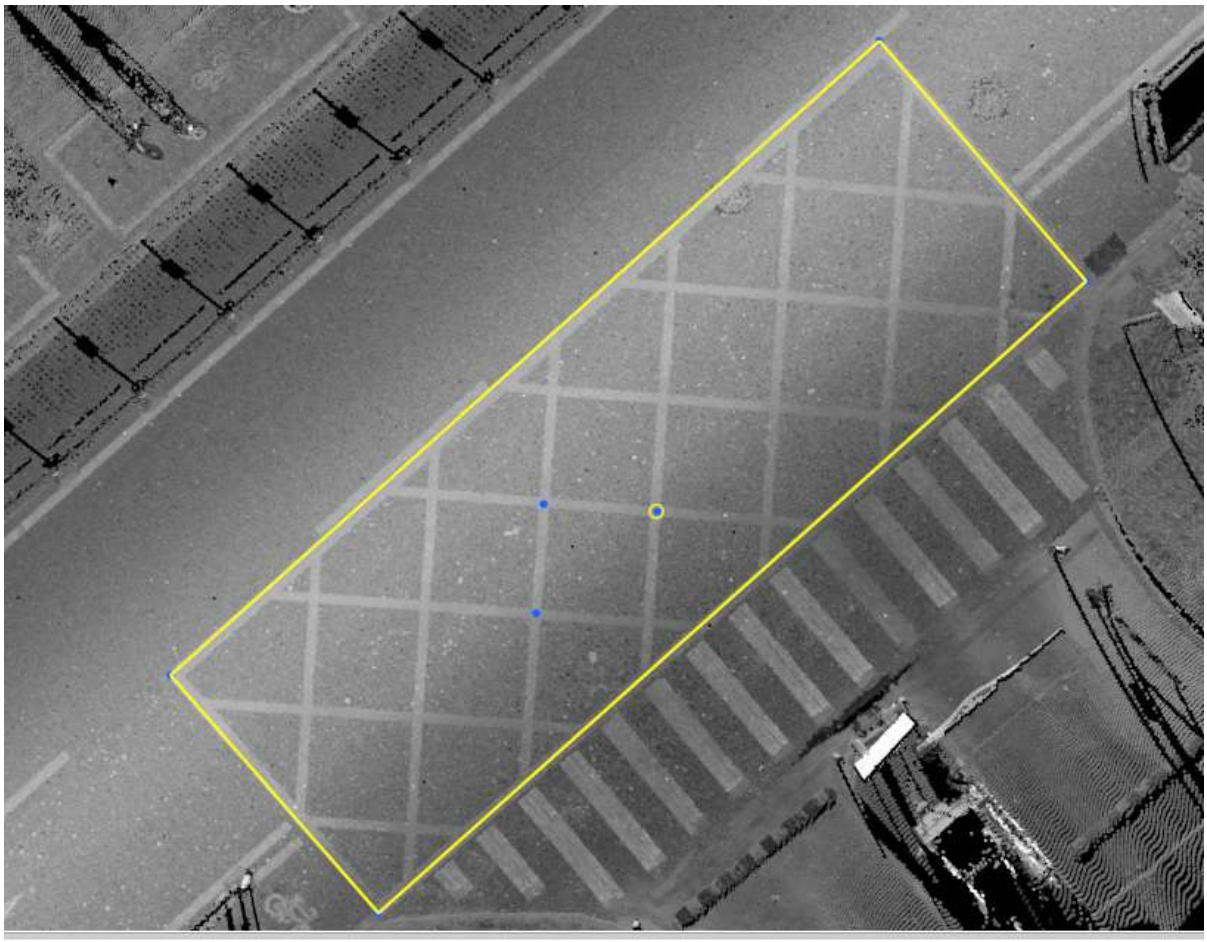
Fourth key point

6.Move the mouse to view the **template** and **actual point cloud** fit, **left mouse button click the fifth key point**, the details are as follows, **yellow point** that is the **fifth key point** in the icon.

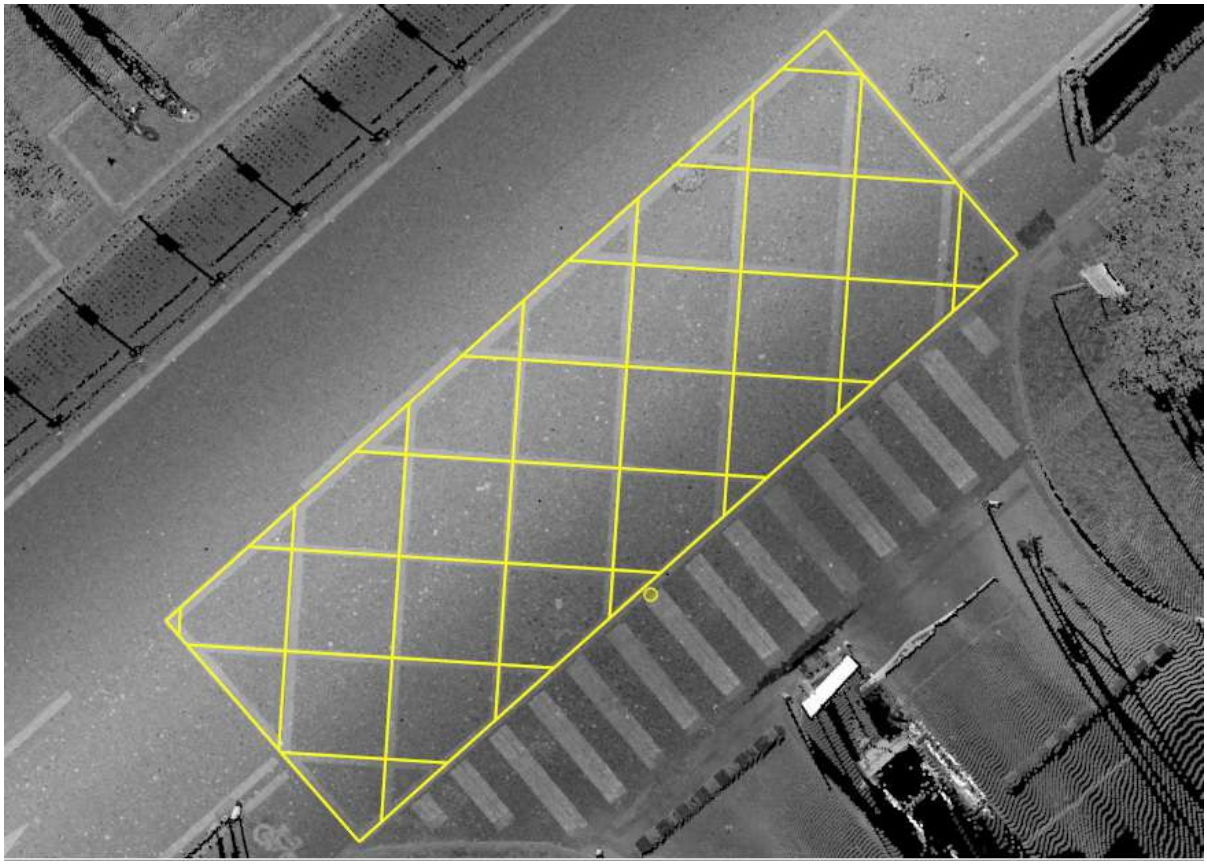


Fifth key point

7.Move the mouse to see **template** and **actual point cloud** fit, **left mouse button click the sixth key point** to complete the production, the details are as follows, **yellow point** that is, the icon in **the sixth key point**.



Sixth key point



No-ParkingArea Drawing

Add BicycleLane

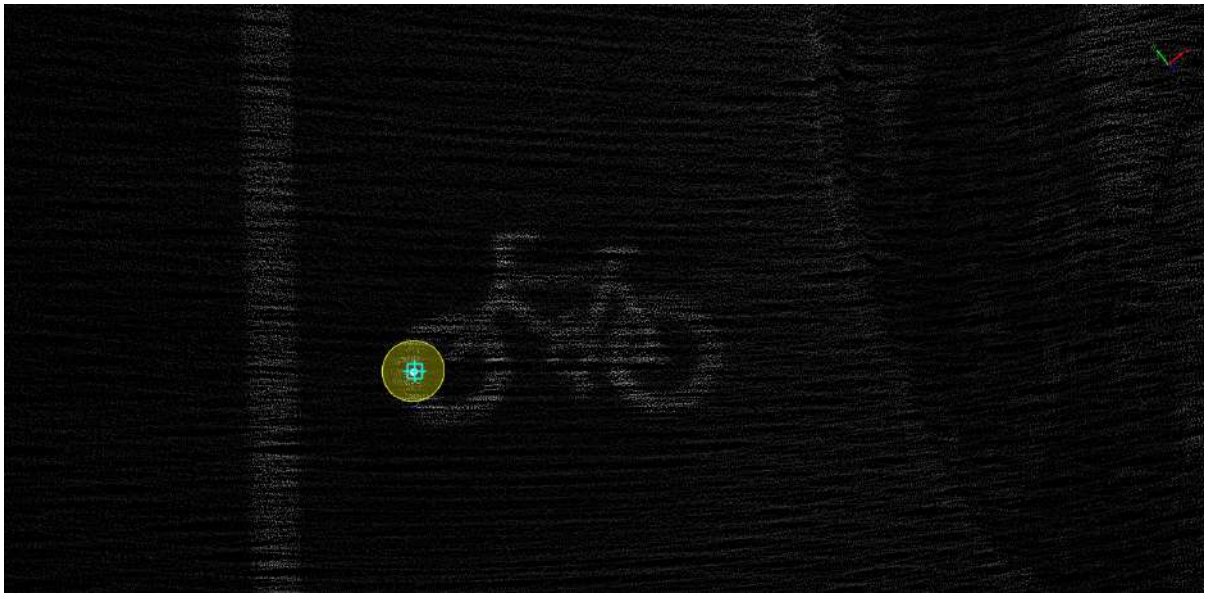
Function Description： The bike lane marker indicates that the current lane is a non-motor vehicle lane. This tool is used to manually click on the point cloud to create vectorized bike lane markers.

Steps

1.Click the **Add BicycleLane** button in the Road Surface panel.

> - The **red point** in the arrow icon is the key point, which needs to be aligned with the actual point cloud.
> - The **red point** position is the **left click** position.

2.**Left mouse click** to select the key point of the point cloud, the details are as follows, the position of the **yellow point** is the **first key point** in the icon.



First key point

3.Move the mouse to check the fit of the **template** and the **actual point cloud**, **left click the second key point** to complete the production, the details are as follows, **Yellow point** is the **second key point** in the icon.

Second key point

Bike lane marker drawing

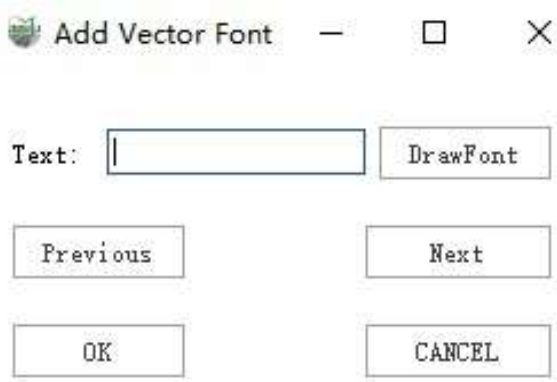
Add Font

Function Description：This tool can be used to add text to vectors, including the vectorized production of ground vector text, numbers and English characters.

> Starting from the first text, until clicking OK is a group of operations, and the vector result of a group of operations is an object. For example, "bus lane", 4 words for a group of operations, the result is an object.

Steps

1.Click the **Add Font**  button in the Road Surface panel to open the **Add Vector Text dialog box**.



Add text dialog box

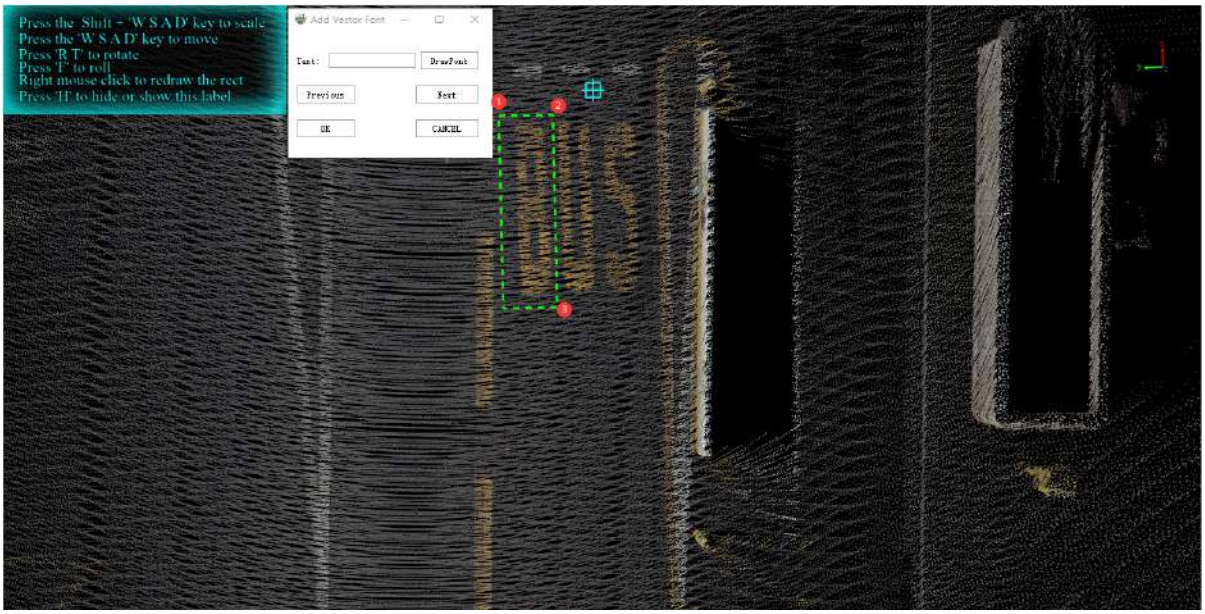
Parameter Descriptions

- **Text**：The text to be entered, users can enter a single letter or a single number.
- **DrawFont**：Display the entered text in a rectangular area of the point cloud.
- **Previous**：Use the last vector text as the current processing unit to perform operations such as rotation, translation, zooming, etc.

  > Starting from the first text, until clicking OK is a group of operations, the previous and the next refer to the current operation.

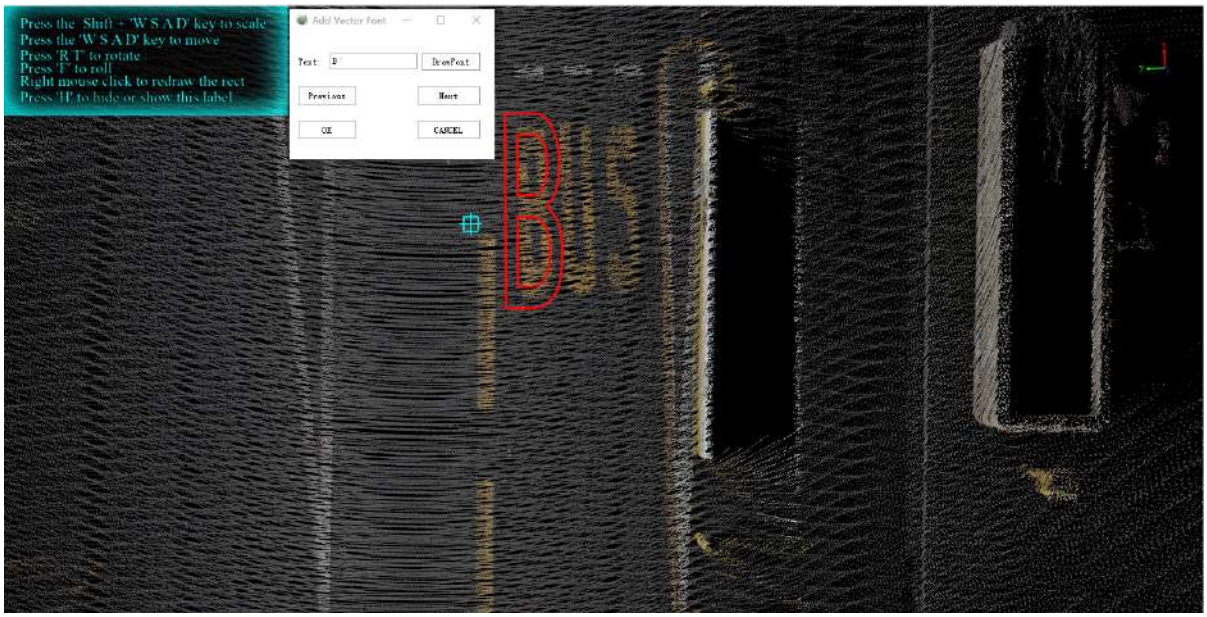
- **Next**： Save this vector result and start the next operation.
- **OK**：End this group of operations and save the vector results.
- **CANCEL**：End this group of operations and discard the vector results of this group of operations.

2.**Left mouse click** three points, determine the position of the current text, the direction of the pull box is the initial direction of the text drawing.



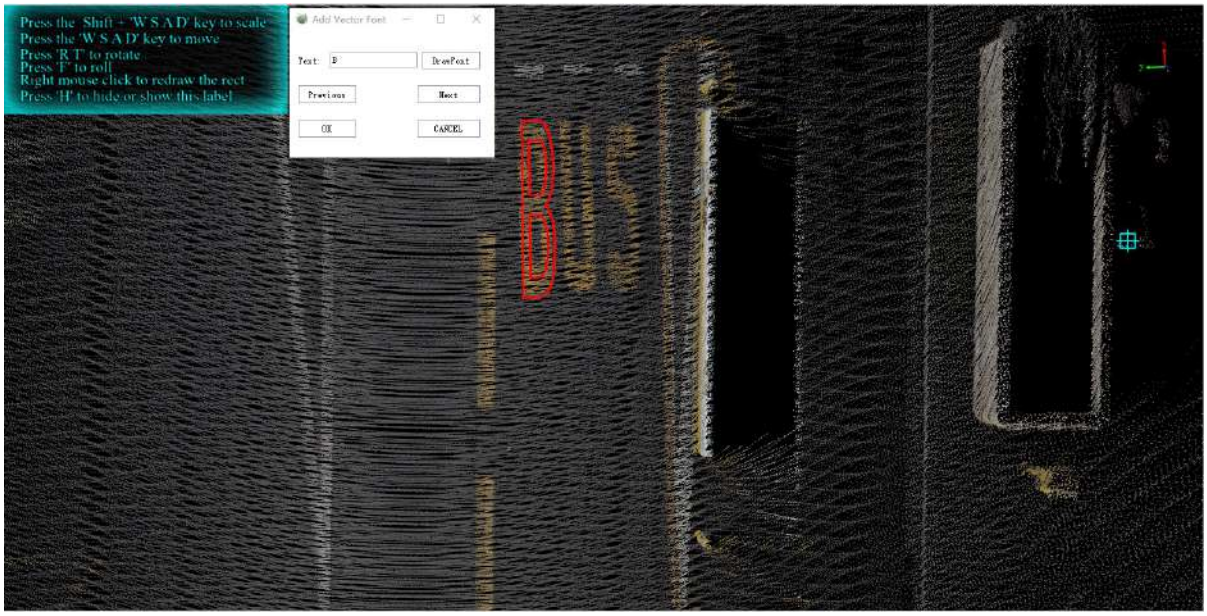
Text position

3.Enter text in the dialog box, such as in the figure: B, and click the DrawFont button to actually display it on the point cloud.



Show actual vector

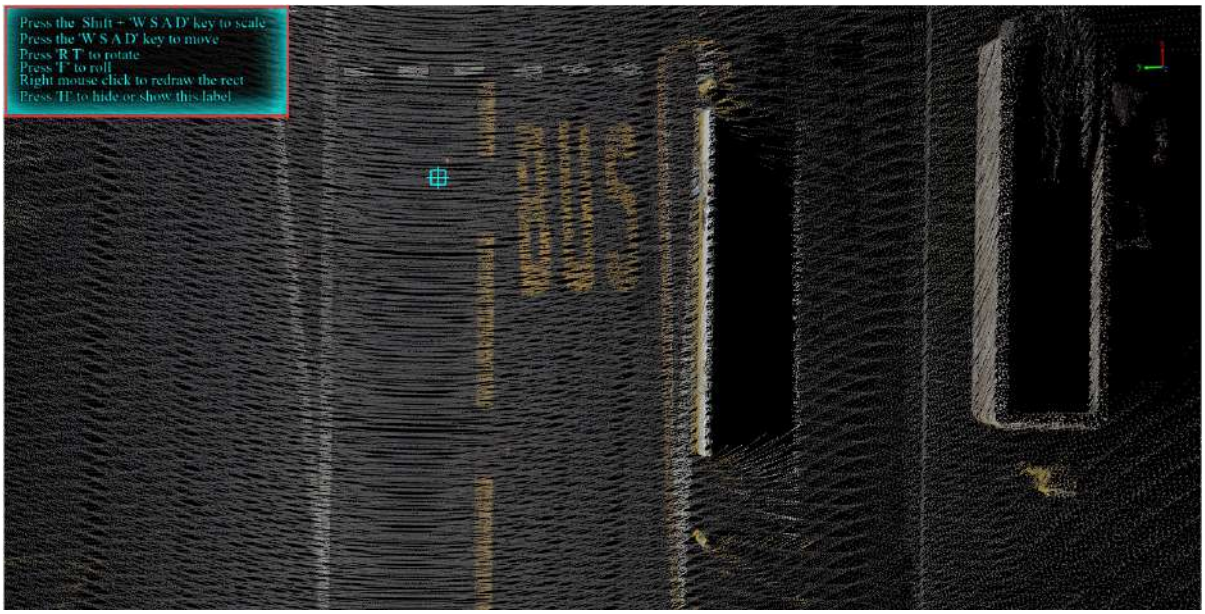
4.Use shortcut keys to perform operations such as rotation, translation, zooming, etc. Making the vector fit in to the point cloud as smooth as possible.



Editing

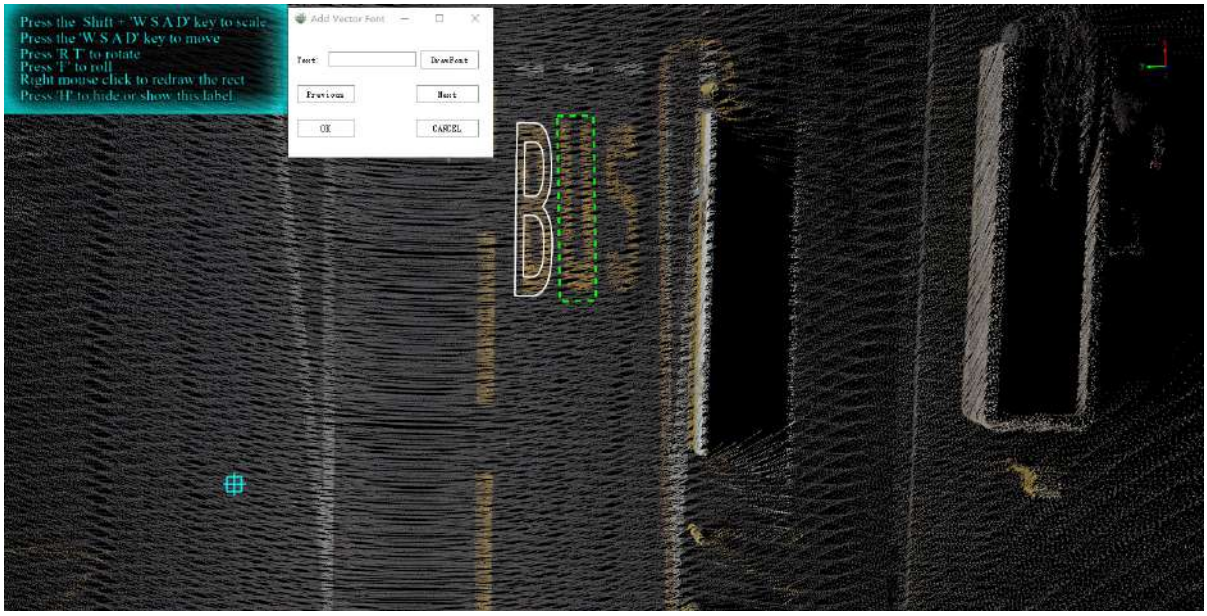
Shortcut names	Effect
Shift + A	Make the current text wider
Shift + D	Narrow the current text
Shift + W	Lengthen the current text
Shift + S	Shorten current text
A	Move the current text to the left
D	Move the current text to the right
W	Move up the current text
S	Move down the current text
R	Rotate the current text counterclockwise
T	Rotate the current text clockwise
F	Flip the current text left and right

> **Shortcut key display**：The shortcut key tip is in the upper left corner of the 3D window.



Shortcut key display

5.Click **Next** to perform this group of operations (current object) to draw the next text.



Next operation

6.Follow the steps 2 ~ 4 to make until the last text of the current object , click OK to automatically save the current object.

Complete drawing process

Template matching

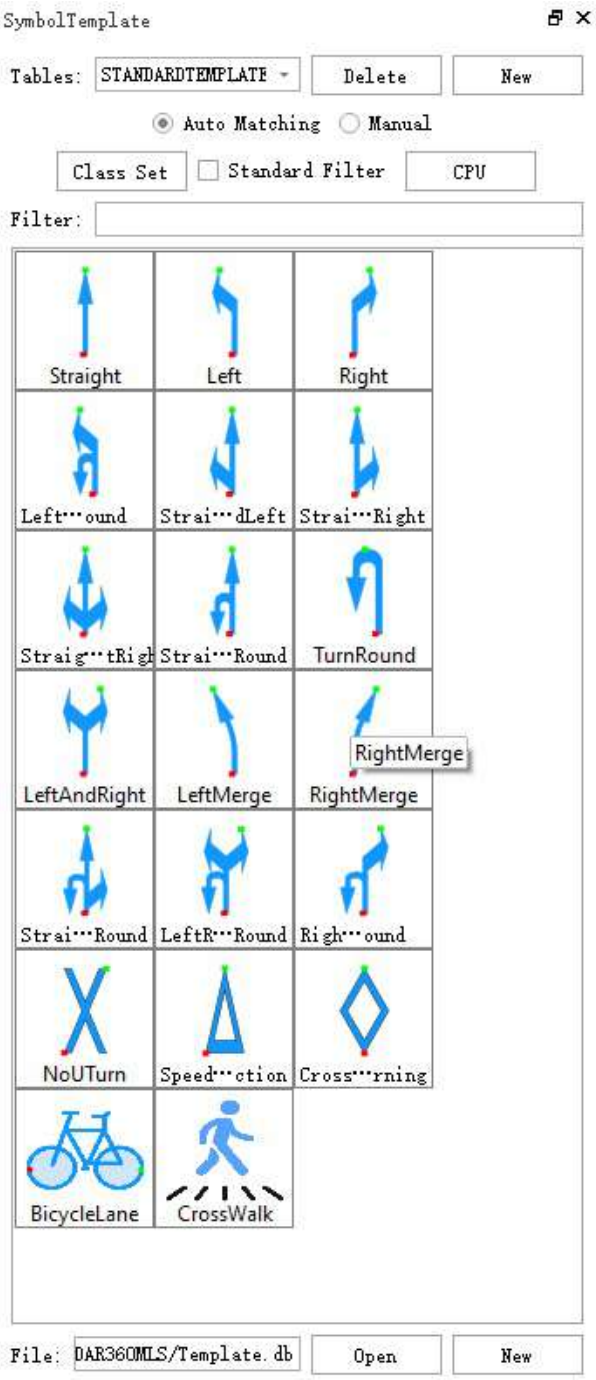
Function Description:This tool can be used to apply a symbol from the SymbolTemplate.
1.Automatic matching: 1.built-in arrow template of Chinese national standard size, as well as ground signs such as crosswalk warning, deceleration sign, and no U-turn of Chinese national standard size, three keys can be used click the box to select the point cloud to perform automatic template matching, and directly get the vectorized result obtained by fitting the point cloud.

2.Manual addition: For different forms of ground marks, manual drawing function can be performed, and the point cloud can be manually described and saved as a custom template. The custom template can be manually added or used for automatic matching.

Steps

1.Click **Template Matching** in the Road Surface panel  button.

2.The **Template Matching** sidebar should pop on the right side of the software, as shown in the figure below, for button meanings see Detailed explanation of template matching sidebar parameters.



Sidebar preview

3.**Custom Tables** (optional): Users can add or delete new templates by themselves, see Template Matching Custom Tables.

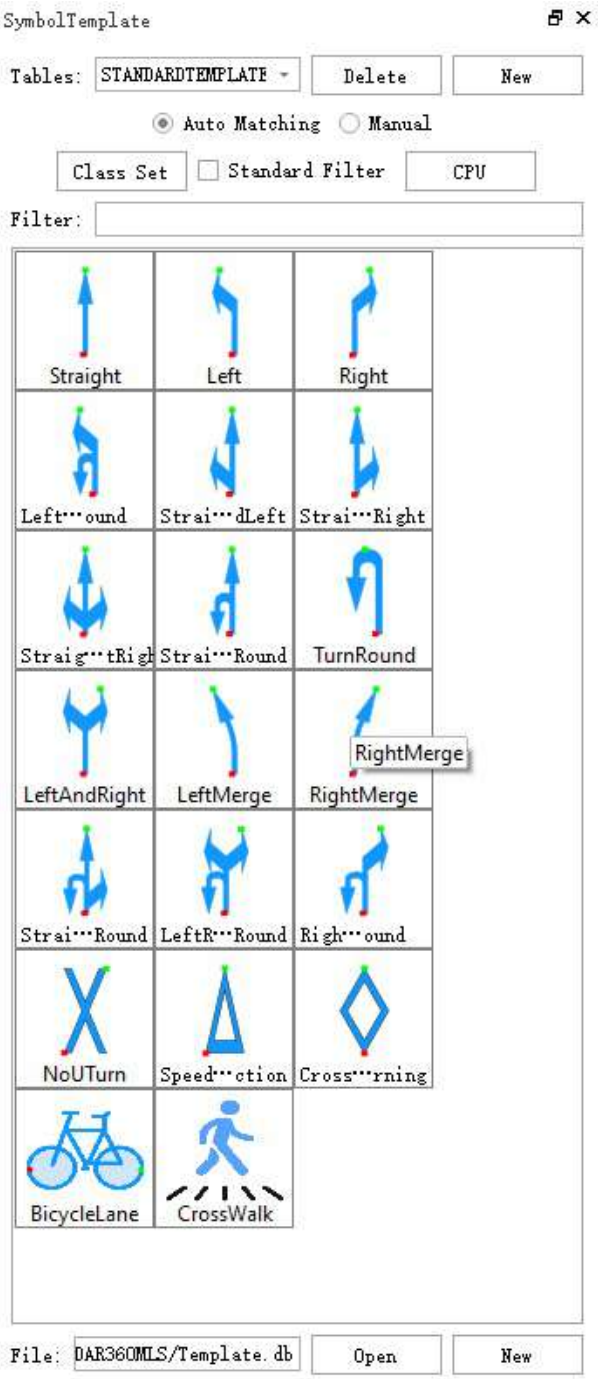
4.**Auto Matching:**: Use the currently activated form and use the three-key point method to perform automatic template matching, see Template Automatic Matching.

5.**Manual**: Use the currently activated form to make manual vector, see Template Manual Making.

Template Matching Para

Function Description: Describes the template matching sidebar parameters in detail.

Sidebar



Sidebar

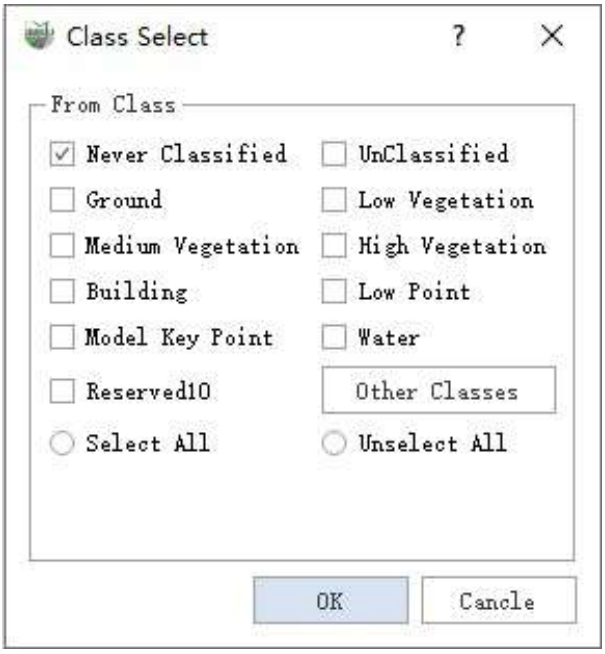
Parameter Description

- **Tables**: The SymbolTemplate module uses a database to store various template tables, as well as user-defined tables.

- Custom tables: Users can add and delete new templates by themselves, see Template Matching Custom Tables.
- The name of the built-in template table is STANDARDTEMPLATE, which cannot be deleted and does not support adding custom templates.

> Built-in tables cannot be deleted or modified.

- **Delete, New**: Used to create and delete custom tables, see Template Matching Custom Tables.
- **Filter**: Enter the template name, perform a quick search, and the preview window below will display real-time linkage.
- Auto Matching: Use the currently activated table and use the three-key point method to perform automatic template matching, see Template Automatic Matching.
- **Manual creation**: Use the currently activated table to make manual vector creation, see Template Manual Production.
- **Class Select**: Set the point cloud type used for detection, all are selected by default. If the point cloud data has been classified, only the category of the ground is selected, which can speed up the calculation efficiency and reduce the influence of other categories of noise.



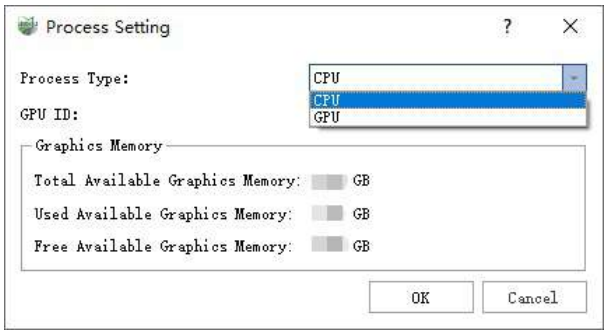
Class select

- **Standard Filter**: Unchecked by default, if checked, the recognition results will be filtered according to the Chinese national standard size, and the results that are too small or too large will be filtered out.

> The length of ground signs on domestic urban roads is not less than 3m, and the length of ground signs on domestic expressways is not less than 6m. If standard filter is enabled, results with a length of less than 3m will be filtered out.

- **Mode: CPU, Mode: GPU**: Set the operation mode, users can choose CPU or GPU settings, running in GPU mode can improve the operation efficiency to a certain extent.

> GPU mode requires computer hardware and software environment support, if the conditions are not met, even if checked, it will still run in CPU mode.



Processing settings

> The three functions of class select, standard filter, and CPU/GPU are only available when **the current table is the built-in standard table of the software and it is in automatic matching mode**.

- **File**: The currently opened database file. For details, see TemplateMatchingDataBase.
- **Open, New**: Open other database files and create new database files. For details, see TemplateMatchingDataBase.
- **Strength adjustment**: It is used to manually select the maximum and minimum values to stretch the strength in places where the strength is not easy to distinguish. Available only when the current table is a custom table.

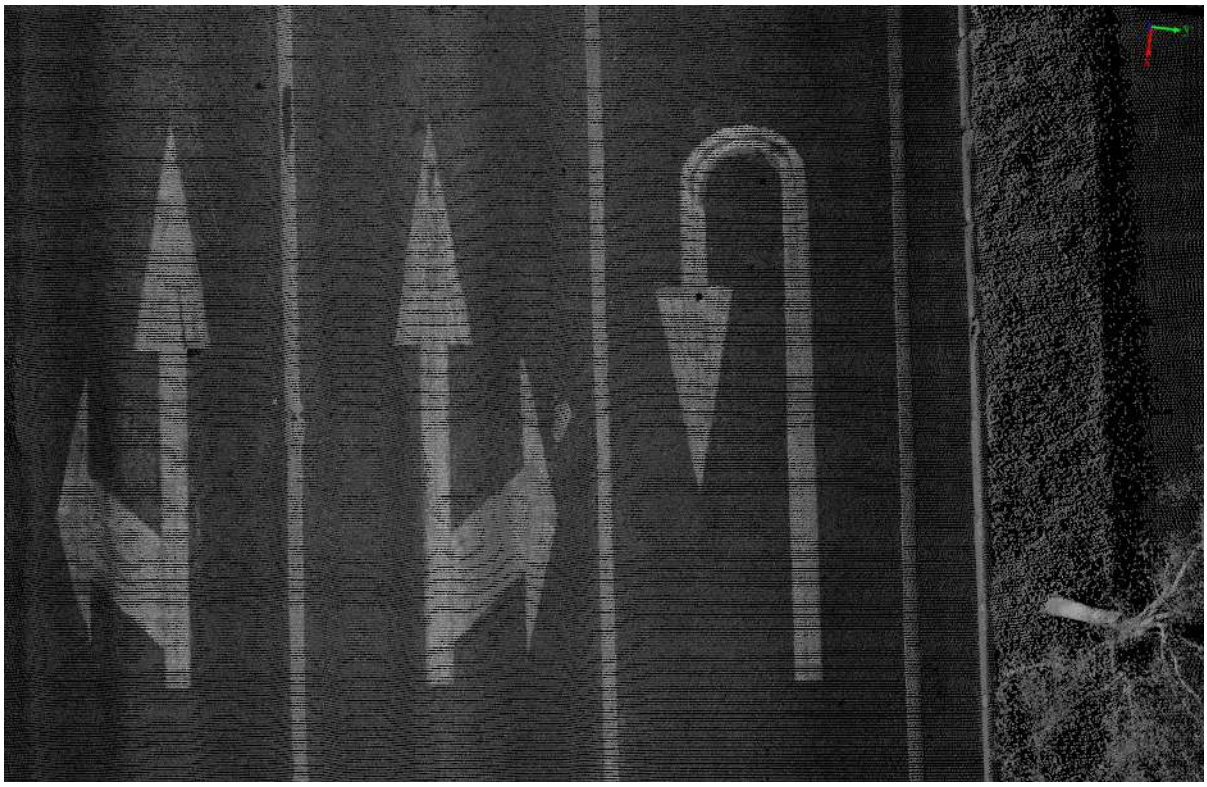
Dialog

- **PickPoint**: The left is the minimum intensity, and the right is the maximum intensity. After checking the box, users can click the point cloud with the left mouse button to determine the maximum and minimum intensity respectively.

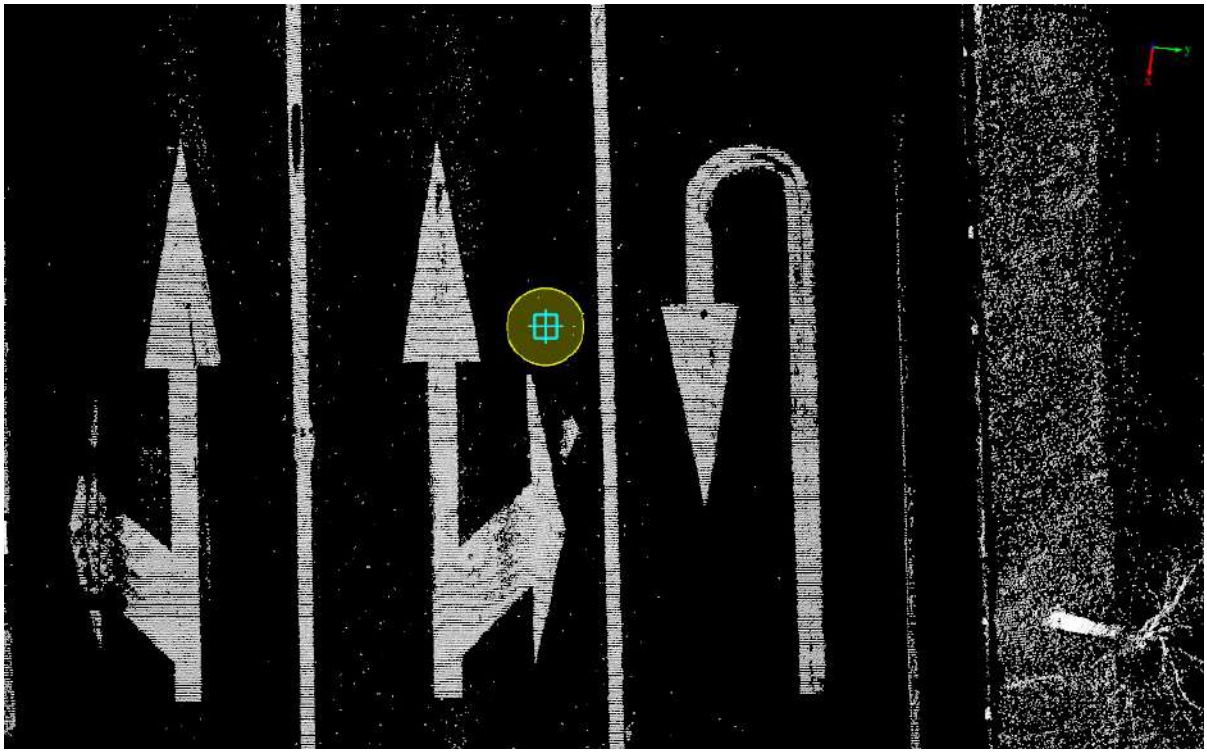
  > Scroll bar, value box, click box, the three are linked together, the user can choose one of the operation methods to adjust the strength, and the point cloud display will also be linked in real time (stretch the strength of the overall point cloud).

- **default**: Restore the original maximum and minimum intensity, and the point cloud display will also be linked in real time.

  Intensity adjustment effect



Original point cloud screenshot

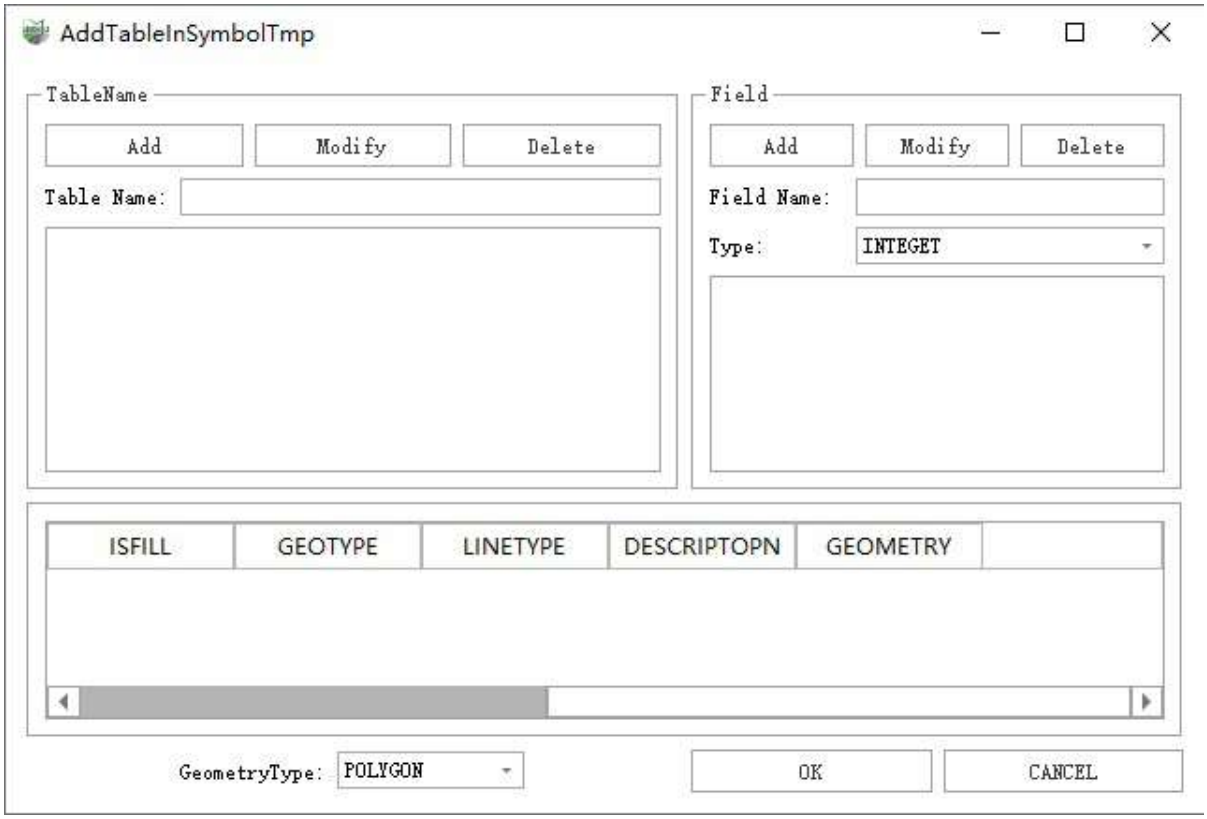


The minimum value is adjusted to 14, the maximum value is adjusted to 18, and the box is not selected to hide

Template Matching Custom Tables

Function Description: This tool can be used to create a custom table, create a custom template.

Dialog



Dialog

Parameter Description

- **Table Name**
  - **Table Name**: Enter the table name.
  - **Add**: Confirm to add table.
  - **Modify**: Modify the added table name and field.
  - **Delete**: Delete the selected added table.
- **Field**: Add and delete fields to the selected table.

  > Each new table has built-in fields, which cannot be deleted or changed.

  - **Field Name**: the newly added field name.
  - **Type**: field type.
    - REAL: decimal.
    - INTEGET: integer.
    - TEXT: text.
  - **Add**: Confirm to add field.
  - **Modify**: Modify the added field name and type.
  - **Delete**: Delete the selected added field.

- **Table Preview**: Real-time preview of fields in the table.
- **GeometryType**: The geometry type stored in the current table.
  - POINT: Point (not commonly used).
  - LINE: Line (not commonly used).
  - POLYGON: Single contour (commonly used).
  - MULTIPOLYGON: Multi-contour, Chinese, English and other texts or other ground signs with multiple contours, use this type (commonly used).
  - TEXT: Text type (only for backup).

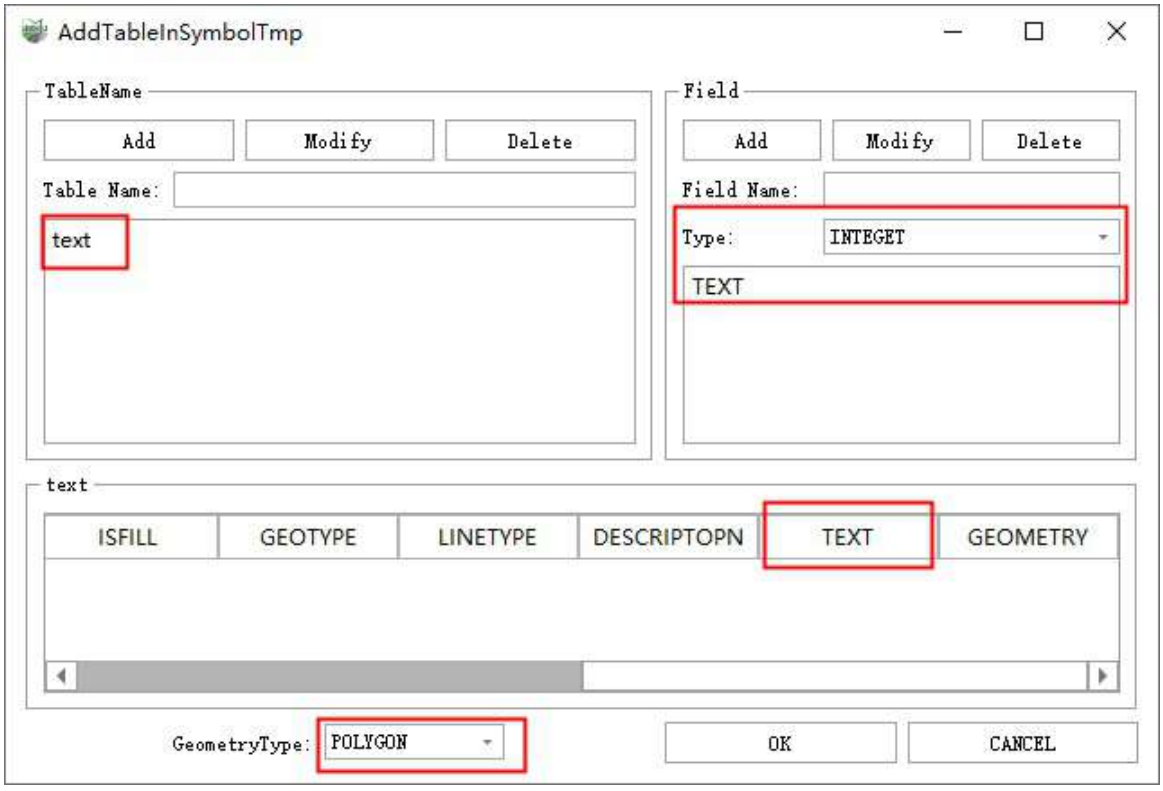
Steps

1.Click the **Delete** button to delete the current table.

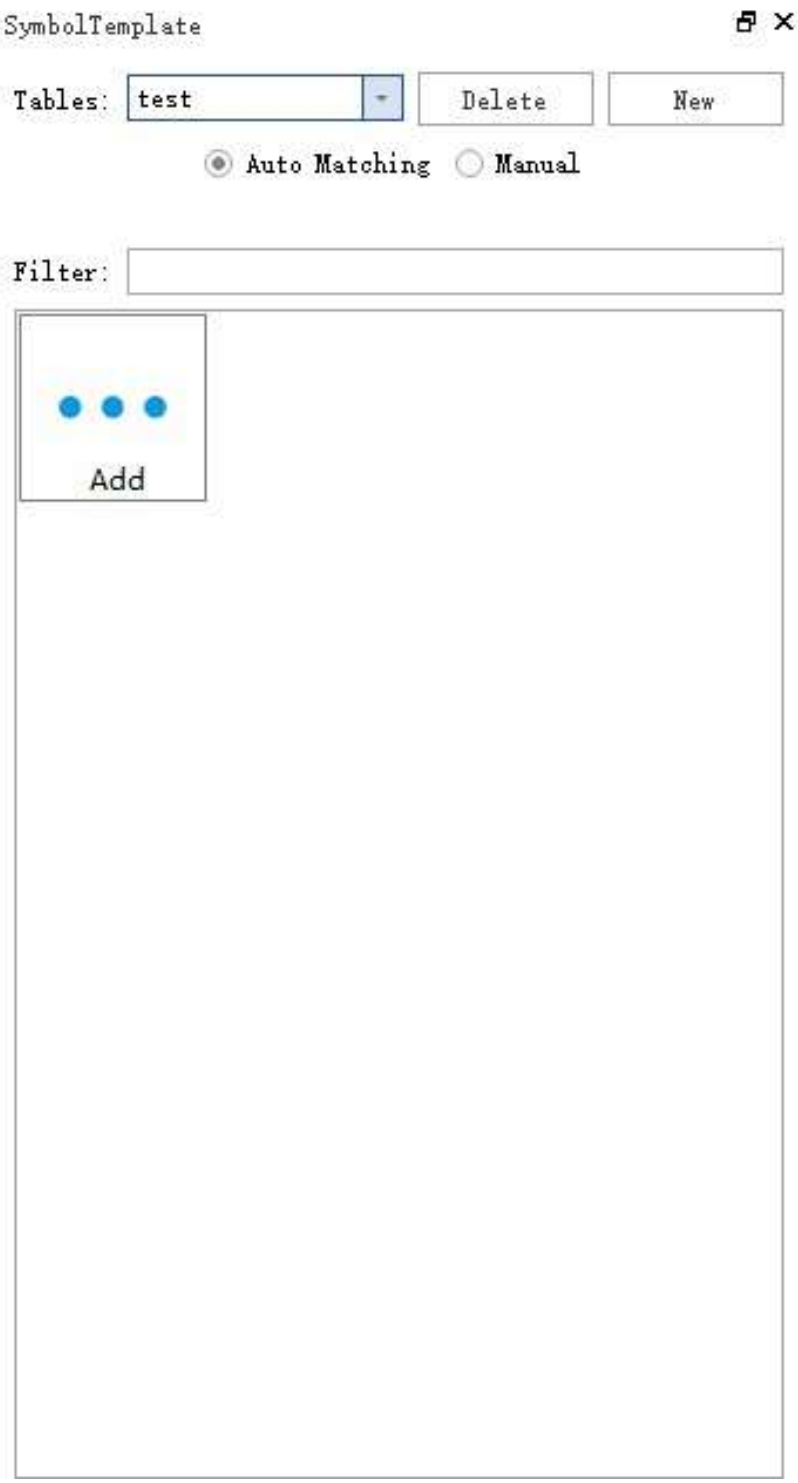
> Built-in table, cannot be deleted.

2.Create a new table:

- Enter the table name (required) and click the Add button.
- Add a new field (optional), enter the field name, select the type, click the Add button.
- Set the table type (required).
- Click the OK button.



New table



Add results

3.Create a new template, see Template Matching Custom Template.

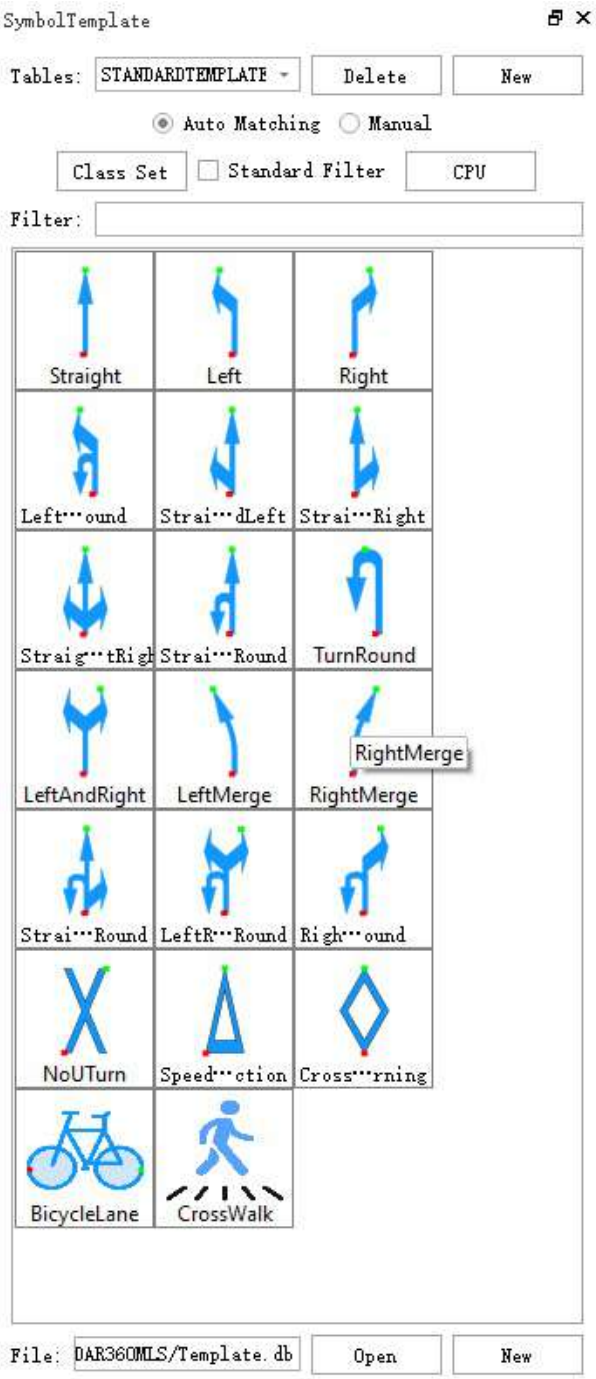
> Built-in tables, cannot add new templates.

Template Automatic Matching

Function Description: This tool can be used to select the target point cloud in a box, and automatically detect and vectorize the ground marks displayed in the template file.

> The built-in templates cannot be added, deleted, or modified. The built-in templates can be selected at one time. Multiple templates can be detected at the same time, and multiple results can be generated at the same time, such as generating arrows and deceleration signs at the same time.
>
> Custom template file, can be added, deleted and modified, custom template **only supports** one frame selection, a single template is detected, **only** generates a single result.

Parameters dialog



Parameter Description:

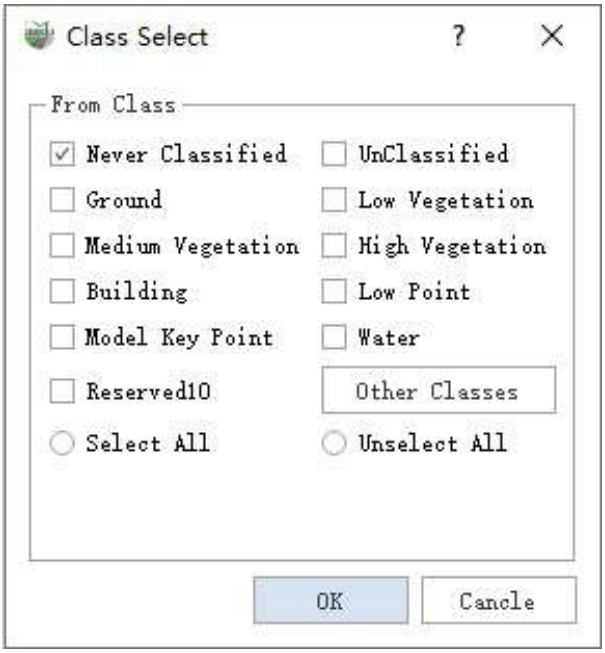
- **Tables**: Used to select the template table to be used, the main table below displays the content of the currently selected template table in the form of graphics + name.

  > There are only built-in standard templates by default, the user can view the supported types of built-in templates in the main table below.

- **Delete**: Used to delete the currently selected table, among which, the built-in template table cannot be deleted.
- **New**: Used to create a custom template table.
- **Class Select**: Appears if and only if the currently used template table is a built-in template table and the mode is auto-matching, it is used to select the class for calculation, multiple selections are possible.

> **Strongly recommended**: only use the class of the ground, if the point cloud has not been classified, strongly recommended: when selecting the point cloud, click on the ground point.

- **Other Classes**: Used to display and select more classes: used to display and select more classes.
- **Select All**: Check all classes.
- **Uncheck All**: Uncheck all classes.
- **OK**: Complete the classes settings.
- **Cancel**: Cancel classes settings.



- **Standard Filter**: Appears if and only if the currently used template table is the built-in template table and the mode is automatic matching, simple filtering is performed using the size of the ground signs of urban roads and expressways in China, and the filter is too large or too small result.

  > If the user is not familiar with the national standard size of ground signs, or the size of the current road ground signs is not standard, or the current road ground signs have various shapes, **not recommended** to check this option.

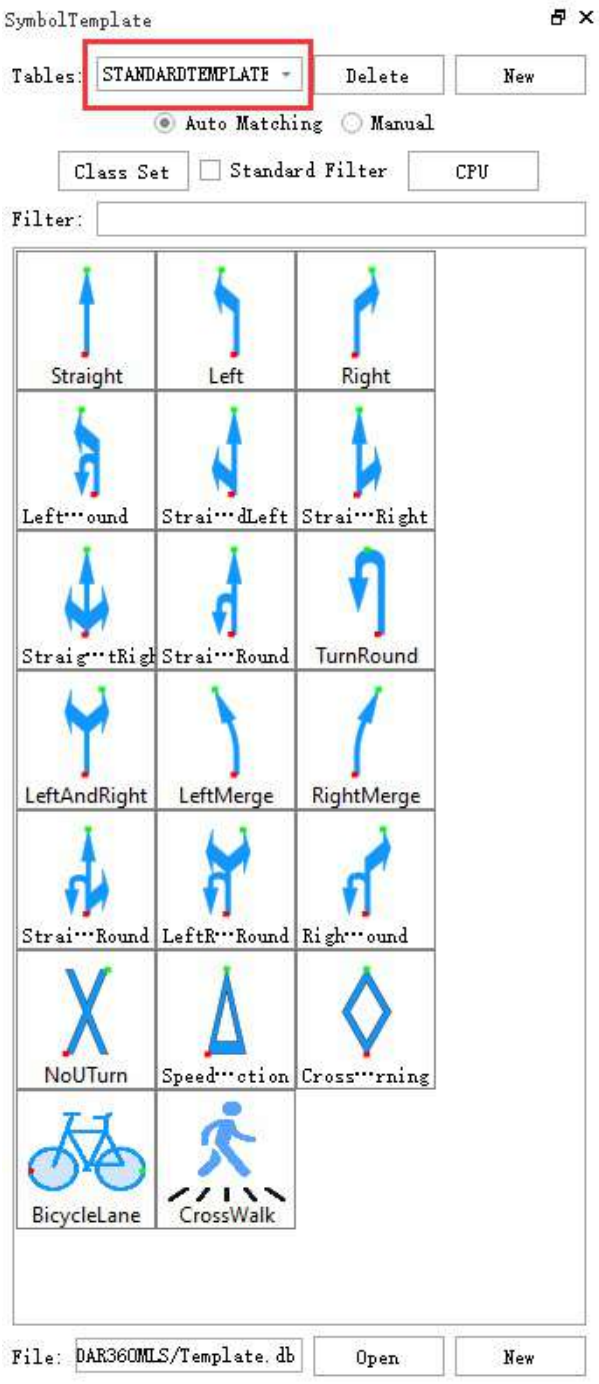
- **Mode: CPU, Mode: GPU**: Appears if and only if the currently used template table is the built-in template table and the mode is auto match, and is used to select the operation mode of the function.

  > The default is **CPU** mode. When the computer has a **GPU** environment, the user can check this option, which can slightly improve the efficiency of a single detection.

Built-in Standard Template Automatic Matching Steps

The built-in standard template can be selected at one time to detect multiple targets.

1.Select the built-in standard template table.

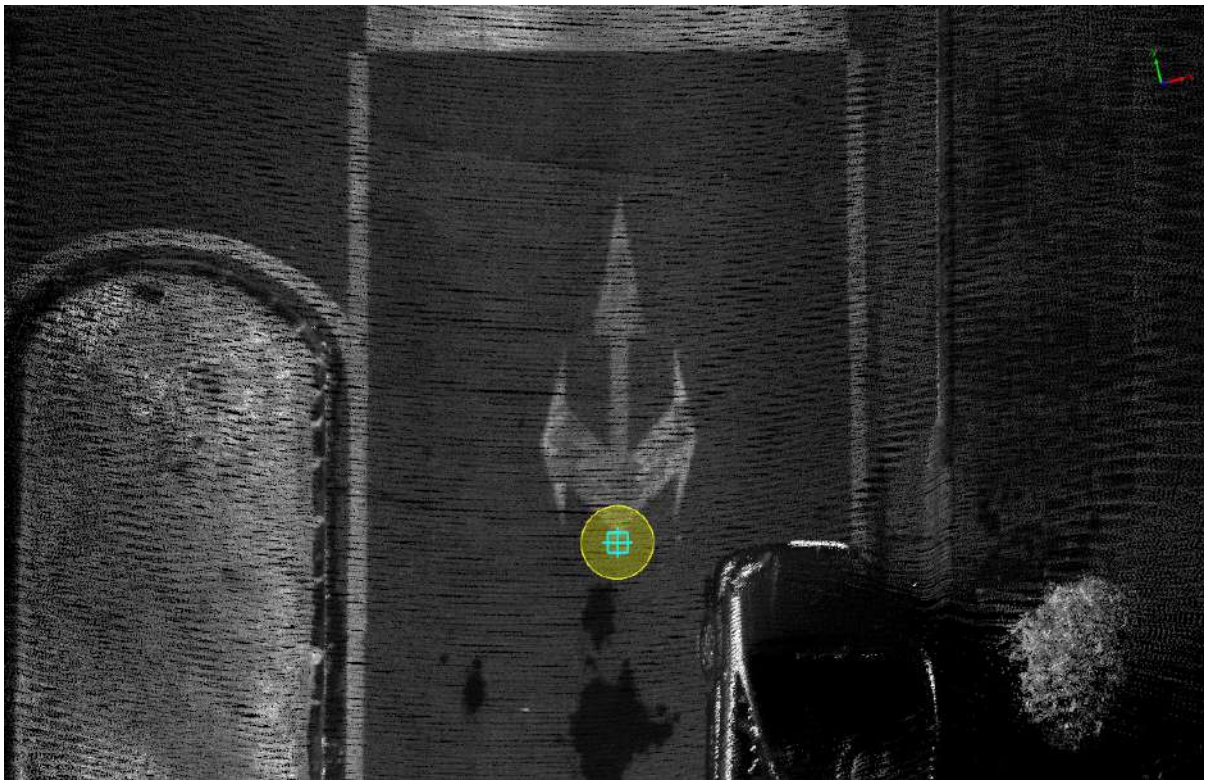


Select table

2.Left mouse button, click on the first key point of the target area.

(1)If the point cloud data has classification information, it is strongly recommended that in the class selection, set the input class to the class of the ground point.

(2) If the point cloud data has no classification information, or use all classes as input classes, must select the first key point and the second key point on the ground, and the third key point can be arbitrarily selected according to the size of the pull box.



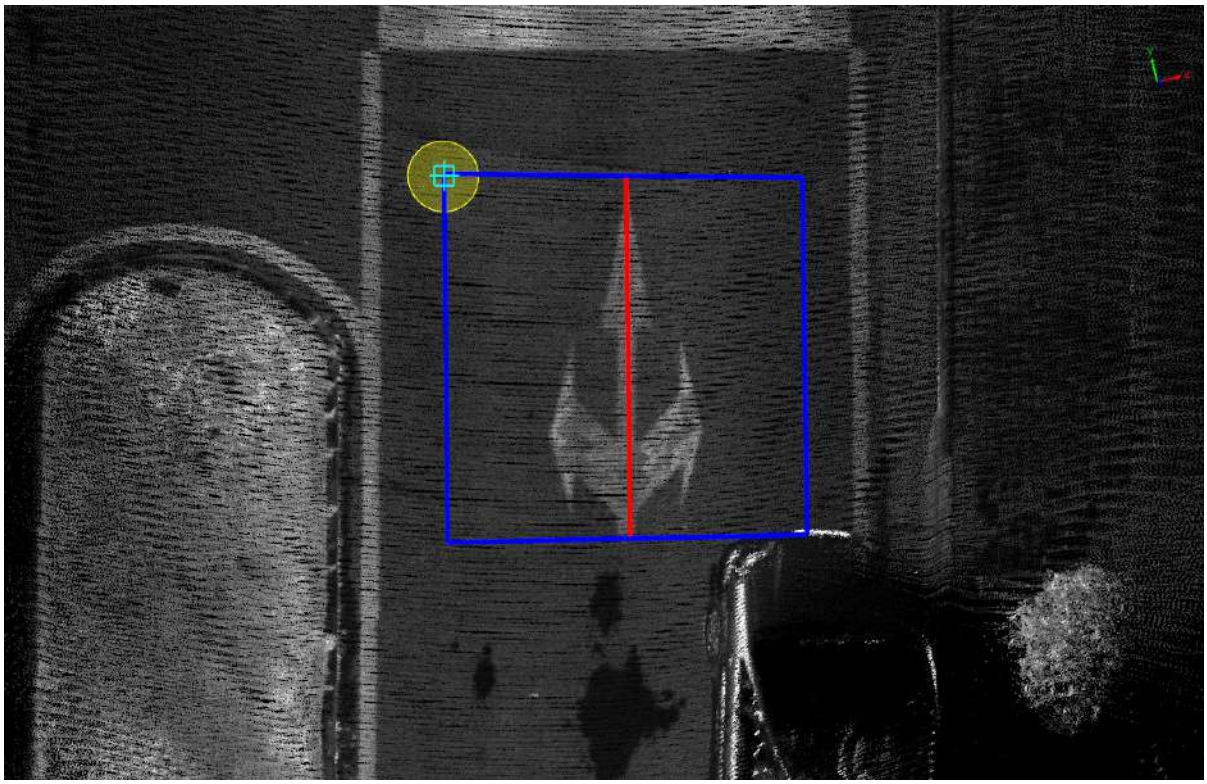
Key point drawing

3.Left mouse button, click the second key point.

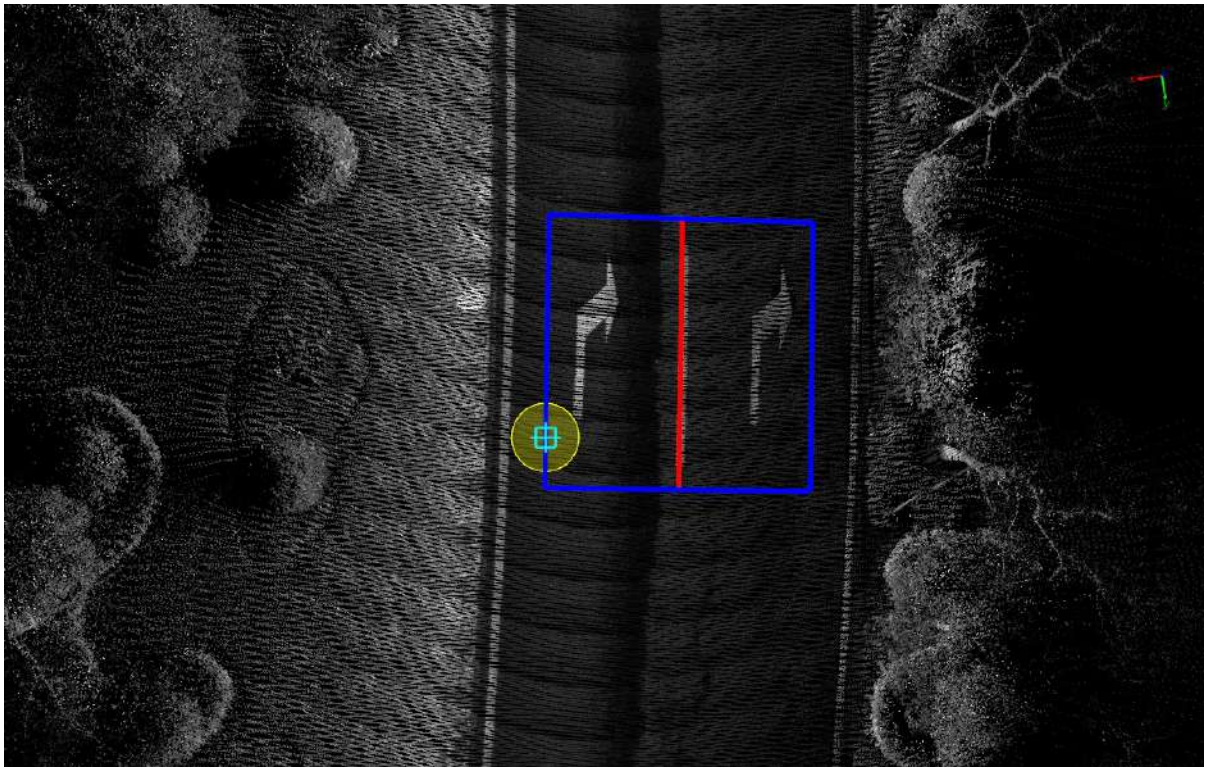
Key point drawing

4.Left mouse button, click the third key point, and then automatically start detection.

> In order to ensure the detection efficiency and detection accuracy, the size of the pull box is **recommended** slightly larger than the ground mark.



Recommended size of standard template table pull box

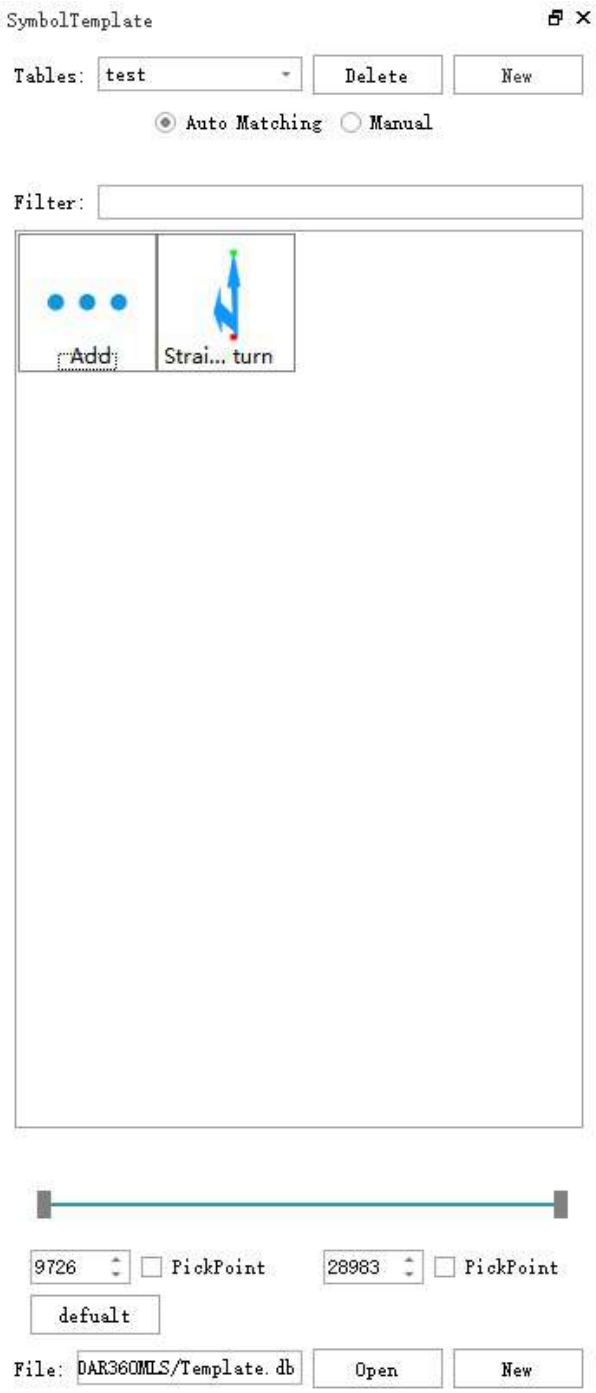


Detection results

Custom Template Automatic Matching Steps

The custom template can only be selected once, and one target can be detected.

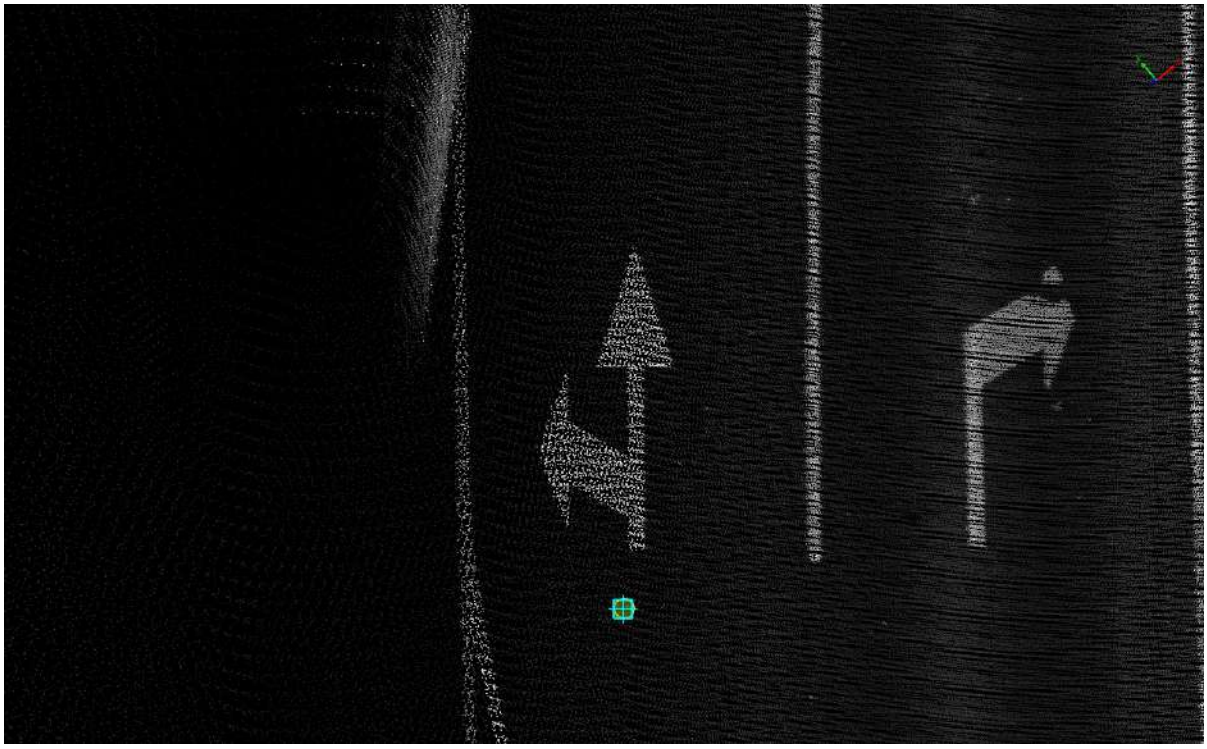
1.Select custom template table.



Select table

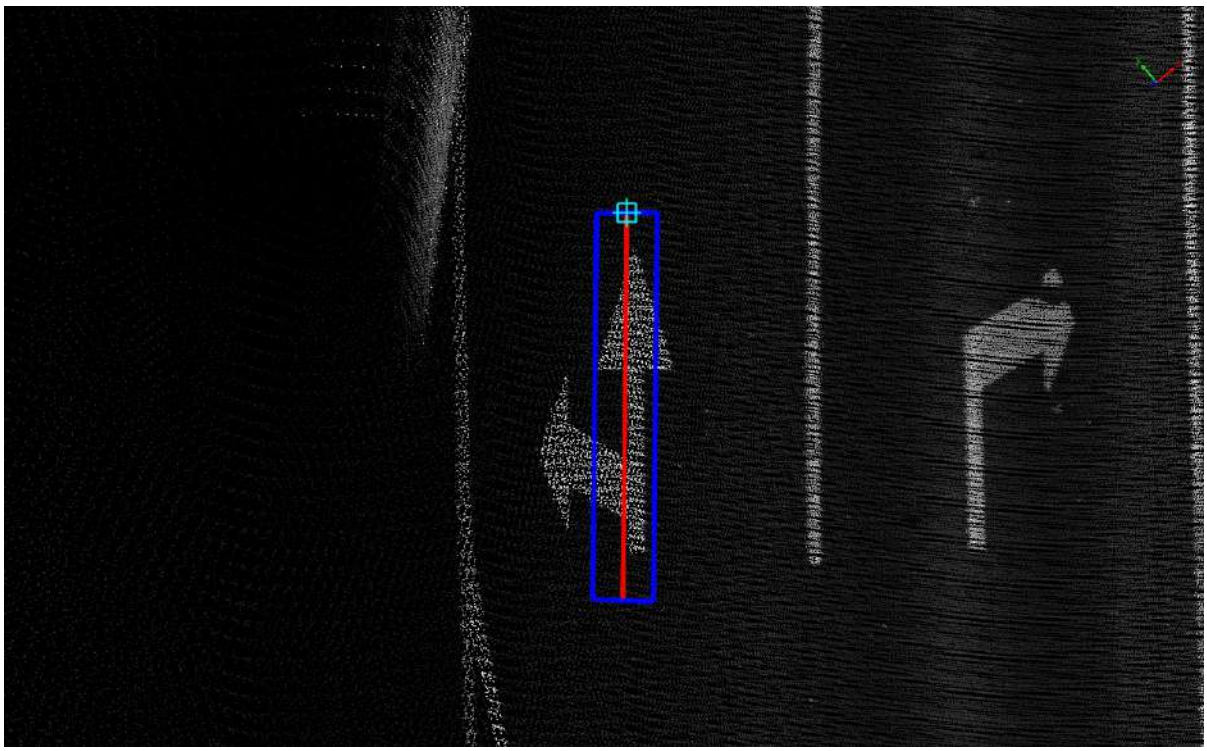
2.Left mouse button, click on the first key point of the target area.

> If the point cloud data has classification information, it is strongly recommended that in the class selection, set the input class to the class of the ground point.
>
> If the point cloud data has no classification information, or use all classes as input classes, **must** select the first key point and the second key point on the ground, and the third key point can be arbitrarily selected according to the size of the pull box.



Key point drawing

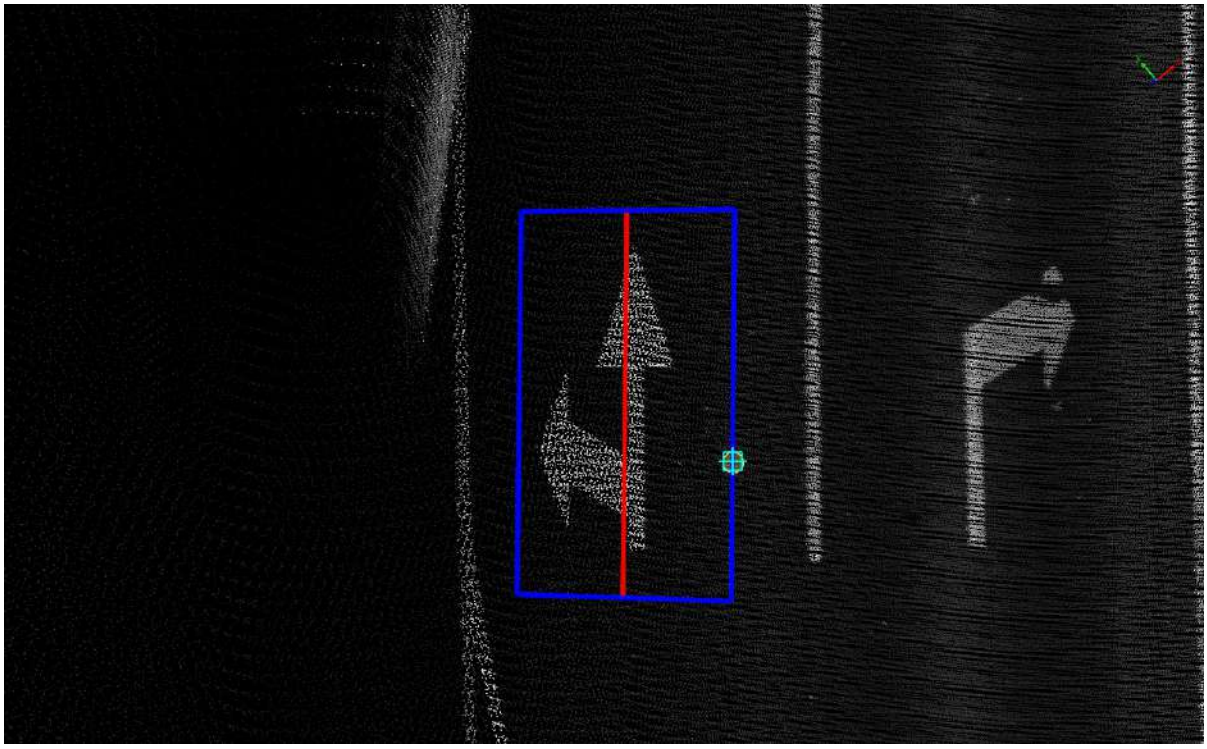
3.Left mouse button, click the second key point.



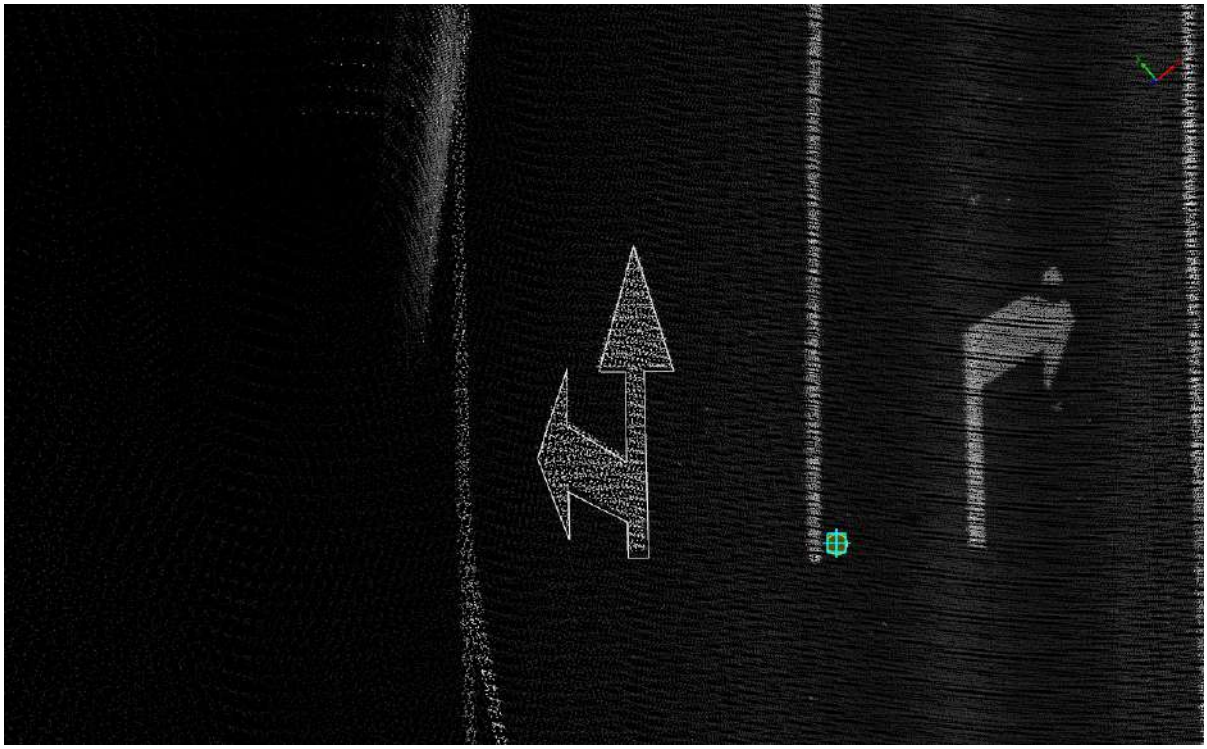
Key point drawing

4.Left mouse button, click the third key point, and then automatically start detection.

> In order to ensure the detection efficiency and detection accuracy, the size of the pull box is **recommended** slightly larger than the ground mark.



Recommended size of standard template table pull box



Detection results

Template Matching Manual

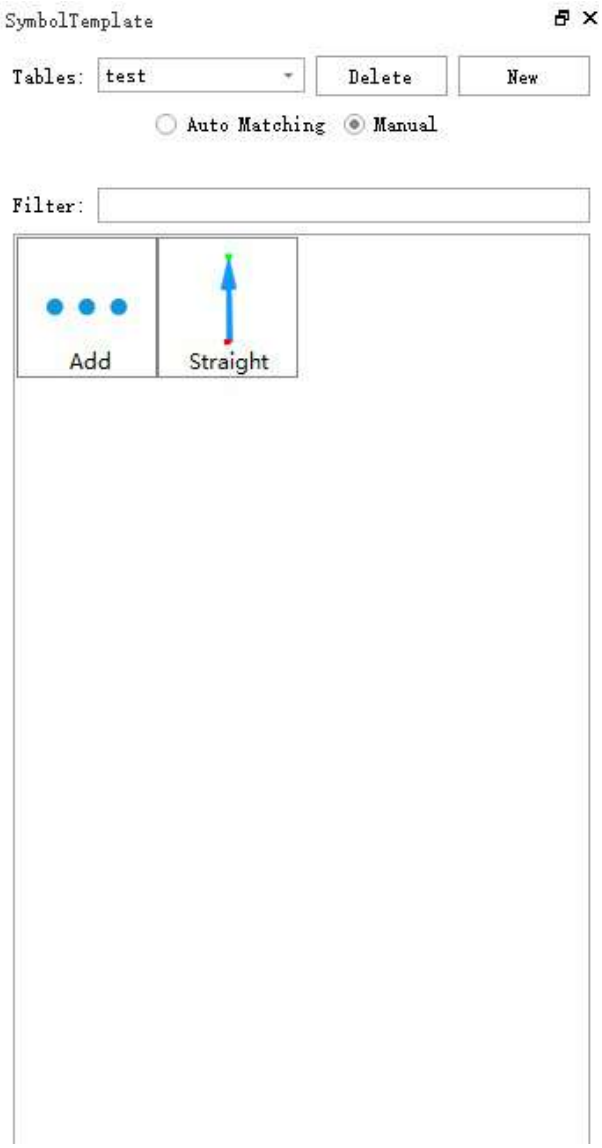
Function Description: This tool can be used to make a new template, or manually add vector results from an existing template.

New template creation

To make a new template, see Template Matching--New Template.

Template based, manually drawn

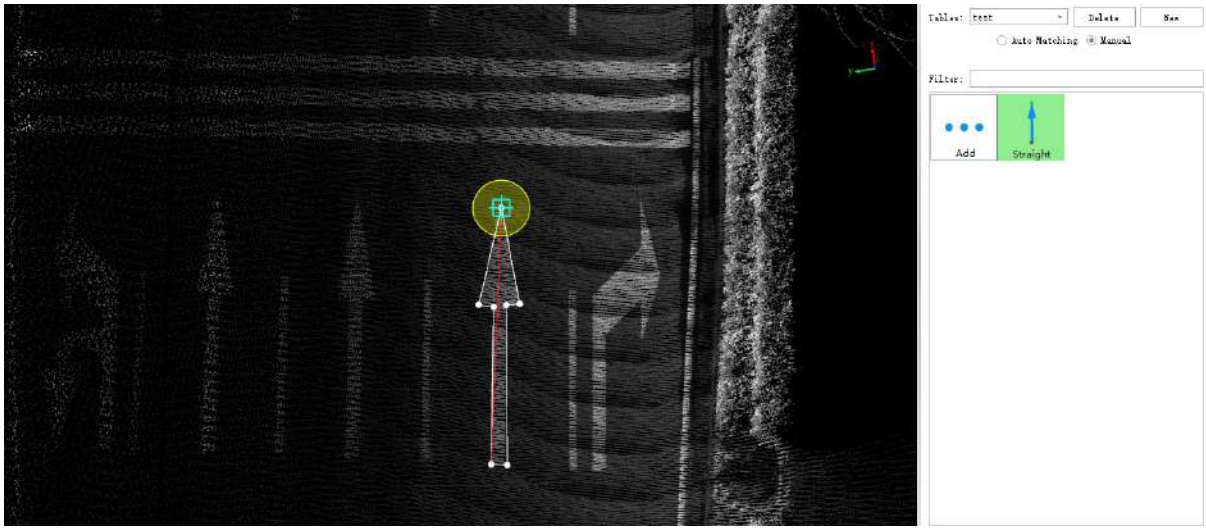
1.Left mouse button in the preview window, select the template to be added manually.



Table

> In the preview icon of the template, the red point is the initial point, and the green point is the direction point.

2.Manual addition, operation method, refer to StraightArrow.


Tables:
test
Delete
New
Auto Matching
Manual
Filter:
Add
Straight

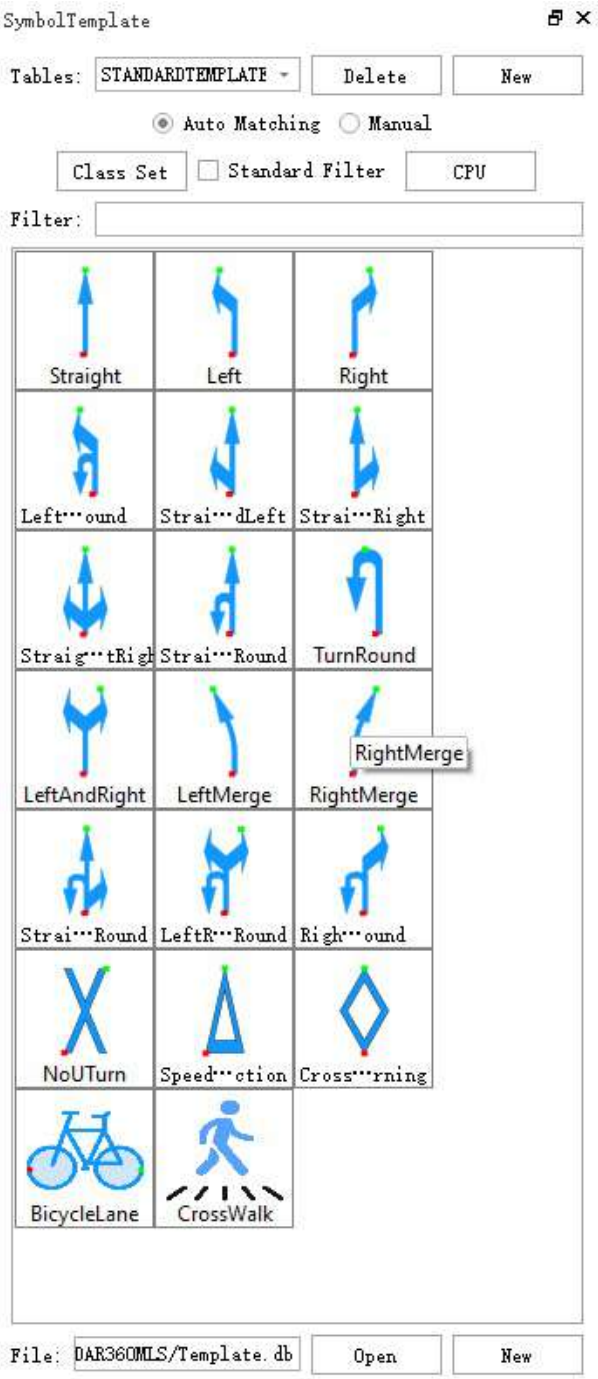
Rendering

Template matching database file

Function Description: This tool can be used to create and open database files.

> Open built-in database file by default.

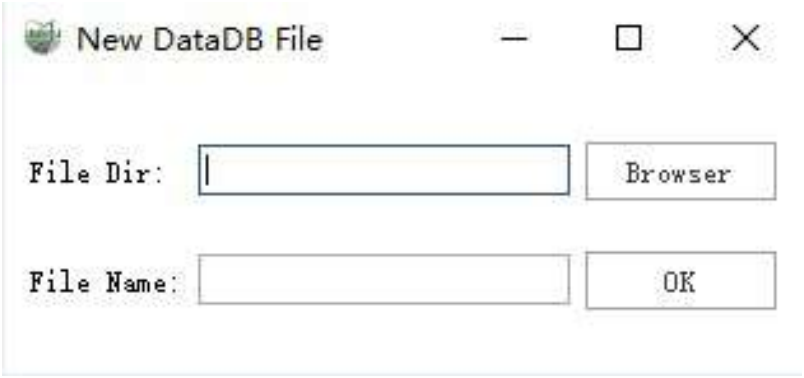
Steps



Shortcut hints

1.Click the **Open** button to open the **Select File dialog box** and select the database file.

2.Click the **New** button to open the **New DataDB File dialog box**.



3.Click the Browser button corresponding to the folder path to set the location of the new database file.

4.Enter a database file name.

5.Click the **OK** button to complete the new creation.

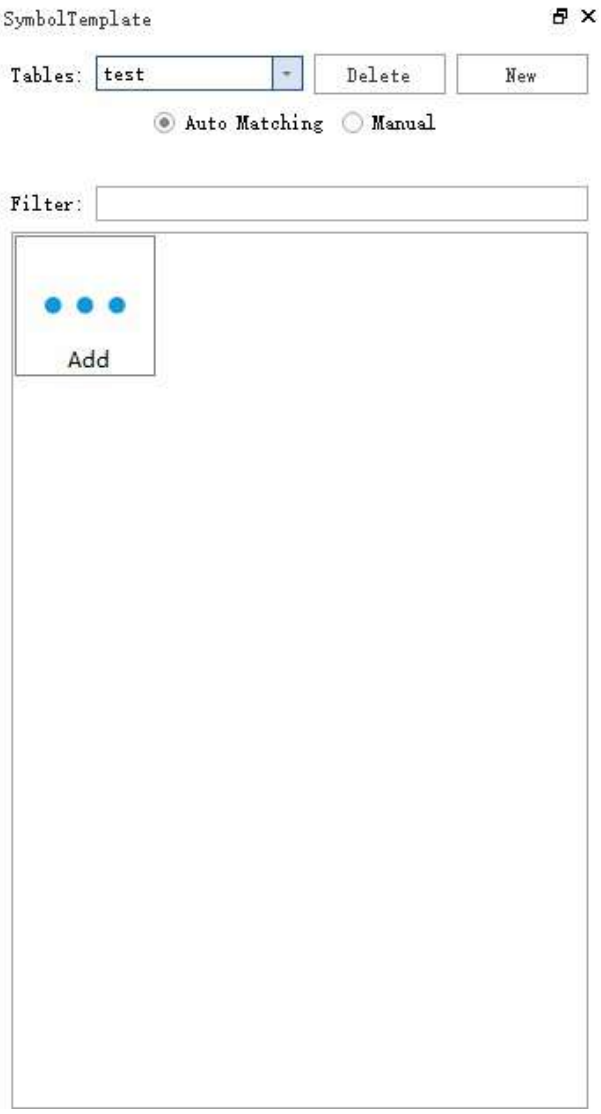
Template Matching New template

Function Description: In the custom table, this tool can be used to create a new custom template.

> New templates can only be created in custom table.

Steps

1.Select a custom table.

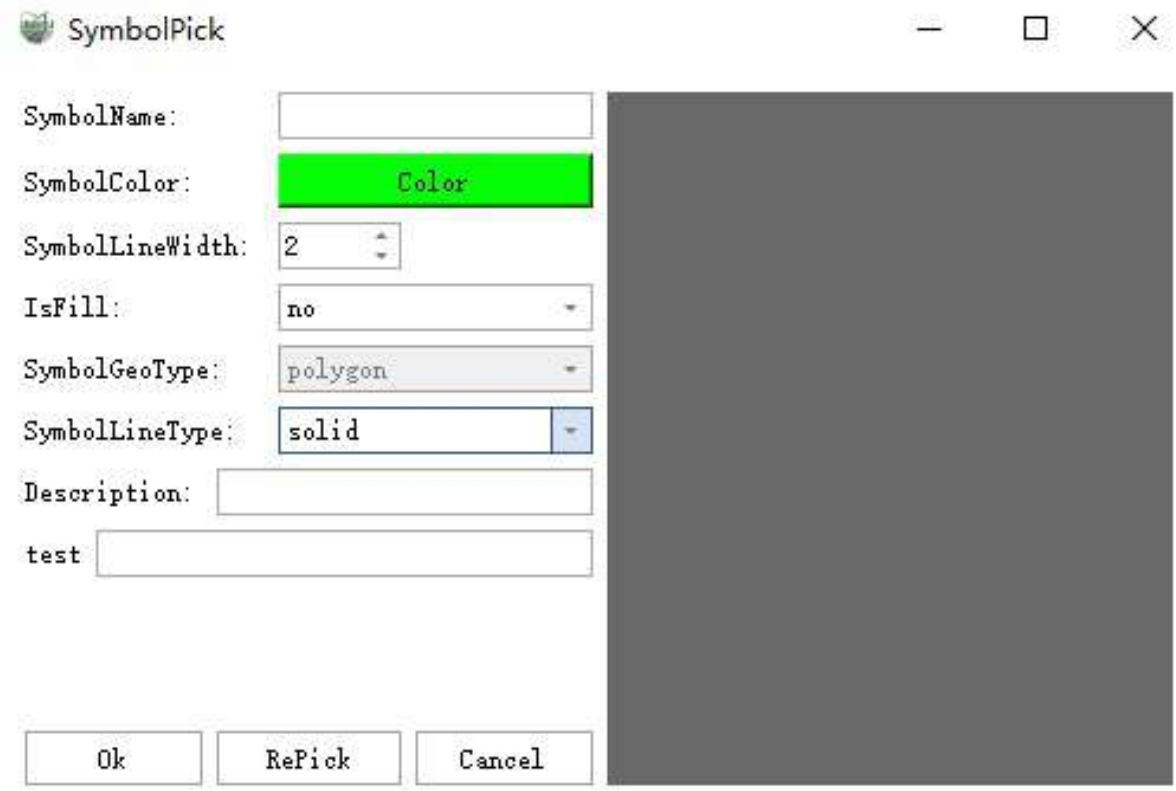


Custom form

2.Click the **Manual creation** checkbox.

3.Click the **Add** icon in the preview.

4.The **SymbolPick** dialog box should pop up.



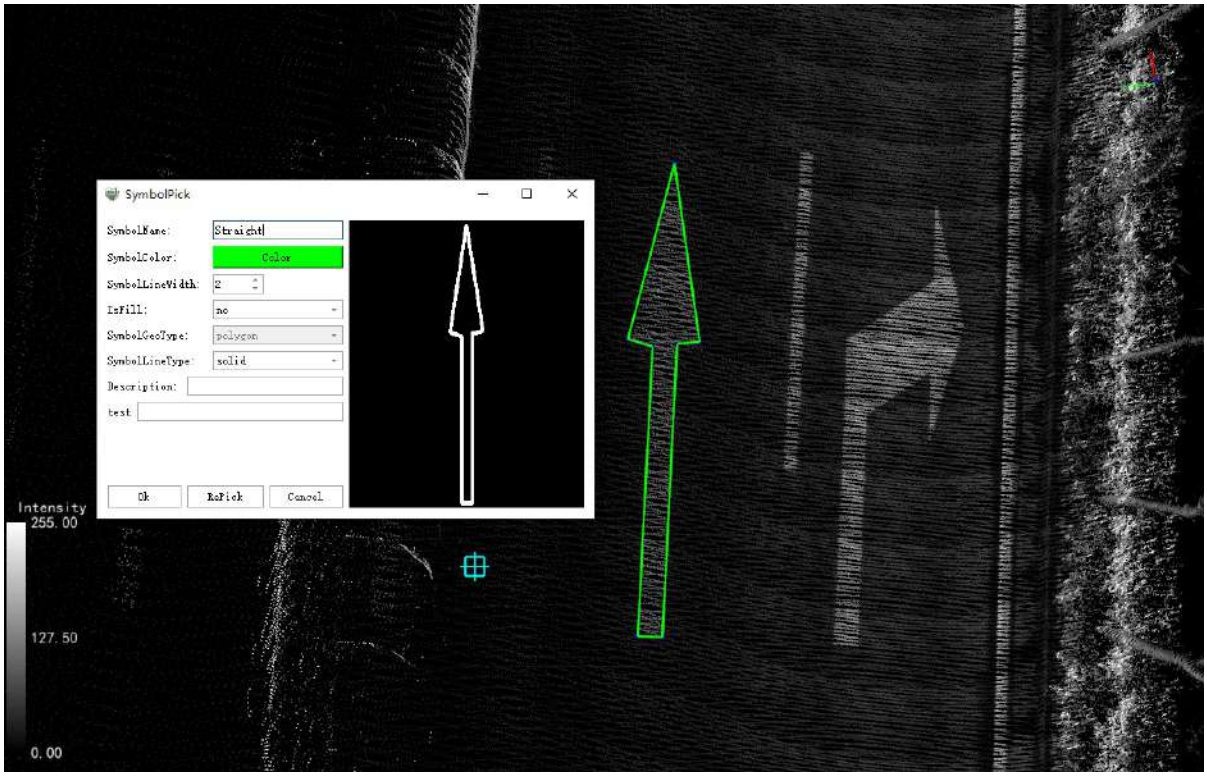
Dialog

Parameter Description

- **SymbolName**: Template name.
- **SymbolColor**: Set the color of the template.
- **SymbolLineWidth**: Set the template line width.
- **Isfill**: Whether to fill the template.
- **SymbolGeoType**: Set the geometry type of the template.
- **Description**: Text description.
- **Real-time preview**: Real-time preview of the template clicked by the mouse.

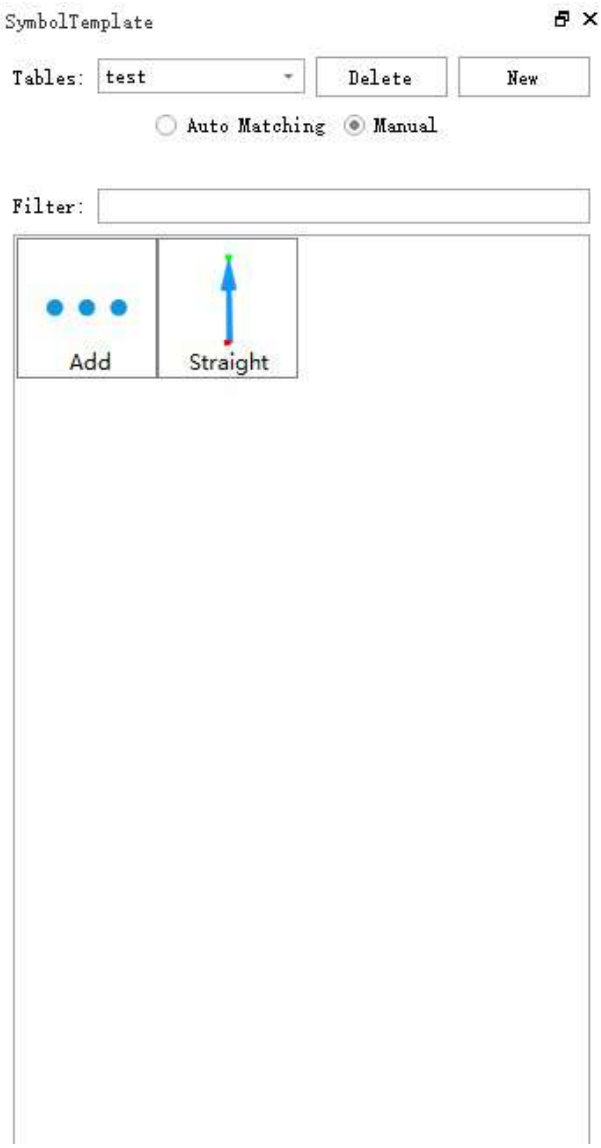
5.Click the left mouse button to select the inner and outer contours of the target point cloud consecutively.

> It is necessary to ensure that the head and tail are connected, that is, the last point coincides with the first point.



New template rendering and real-time preview

6.Click the **OK** button to save the template.



New template result

7.Click the **Reselect** button to redraw.

Road Marking Batch Process

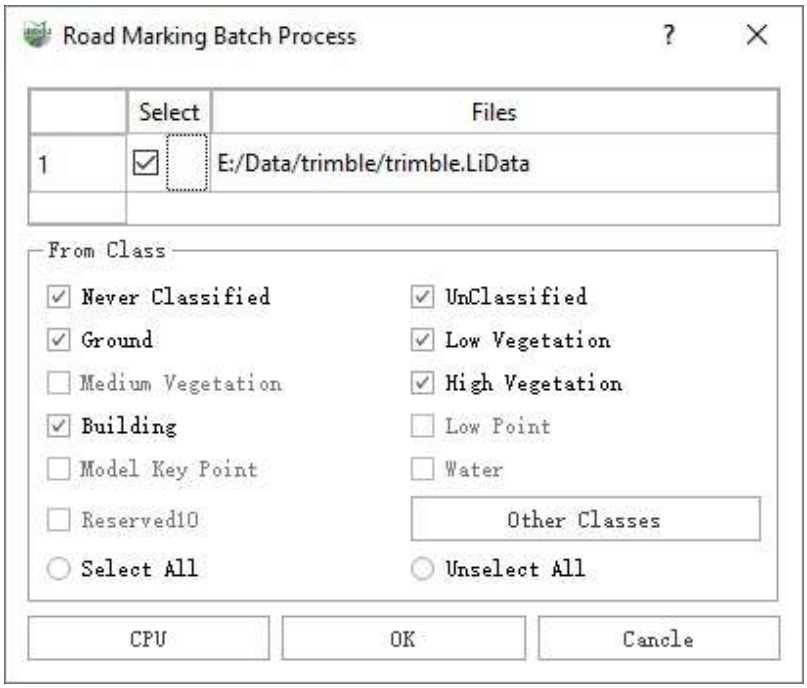
Function Description: This tool enables automatic, deep learning-assisted ground identification extraction for a large amount of point cloud data.

> **Strongly recommended**: Before using ground identification batch processing, perform point cloud classification, and only select the category where the ground point is located as the input class, which can effectively improve processing efficiency and detection result accuracy.

Steps

1.Click **Road Marking Batch Process** in the Map Elements panel  button.

2.The **Parameter setting dialog** box should pop up.



Road marking batch process

Parameter Description

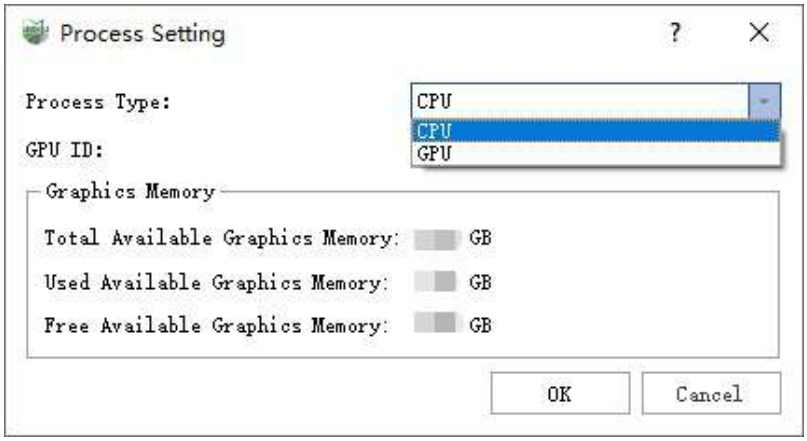
- **Select**: Check/uncheck the point cloud data to be processed.
- **From Class**: Select the class point that needs to be calculated and users can select multiple.
  - **Other Classes**: Click to show other classes.
  - **Select All**: Click to select all classes.
  - **Unselect All**: Click to cancel all selections.

> If the current point cloud data is not classified, only the "Create point, unclassified" option is available;
>
> **Strongly recommended**: Before using ground identification batch processing, perform point cloud classification, and only select the class where the ground point is located as the input class, which can effectively improve processing efficiency and detection result accuracy.

- **Mode: CPU, Mode: GPU**: Set the operation mode and users can choose CPU or GPU settings, running in

GPU mode can improve the operation efficiency to a certain extent.

> GPU mode requires computer hardware and software environment support, if the conditions are not met, even if checked, it will still run in CPU mode.



Processing settings

- **OK**: After setting the corresponding parameters, click the **OK** button to start automatic detection.
- **Cancel**: Cancel and exit the function.

3.Click **OK** to enable automatic identification of road surface identifiers and vectorize them automatically. Crosswalk signs will be stored in the software's built-in crosswalk layer, and other signs will be stored in the built-in road surface signs.

Facility

Road Facilities

The road facility module is used for **vectorization and manual production production** of **road appurtenances**.

It includes the following functions：

- Detect Pole
- Detect PowerLine
- Add TrafficSign
- Add Parking Spot
- Add Rain/Strainer
- Add Manhole

Detect Pole

Distinguished by **operating mode** and is divided into four modes:

- Single Point mode
- Two Point mode
- Polygon mode
- Grip Ground mode

Single Point Mode

Function Description: The Single Point Mode for pole detection. To apply, manually click on the top of the pole as the key point, and automatically generate the rod by setting the search radius.

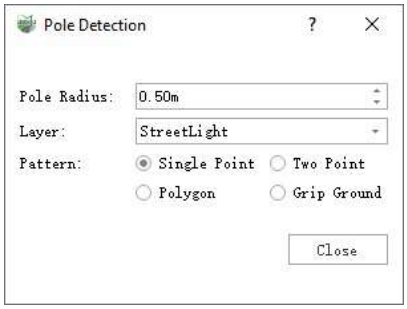
> The generated rod is **precisely perpendicular** to the ground, while the real rod is **not necessarily perpendicular** to the ground due to the **instrument scanning unilateral, tilt itself, thick at the bottom and thin at the top**, etc.

Steps

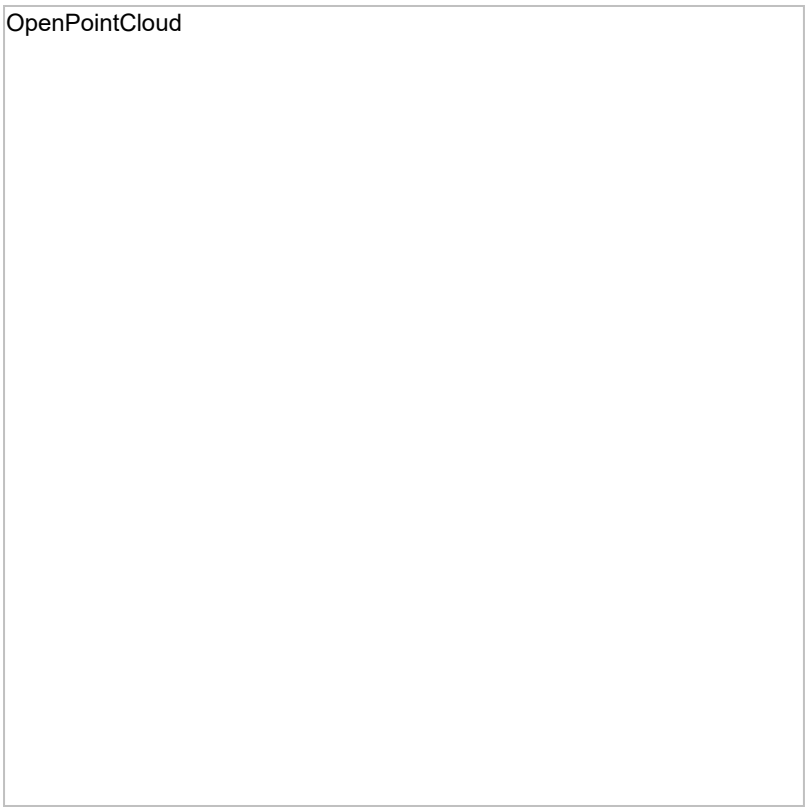
1.In the Road Facilities panel, click on the drop-down logo on the right and select **Detect Pole**  button.

2.**Pole Detection** setting dialog box should pop up.

- Single point mode.



Single Point Mode Dialog

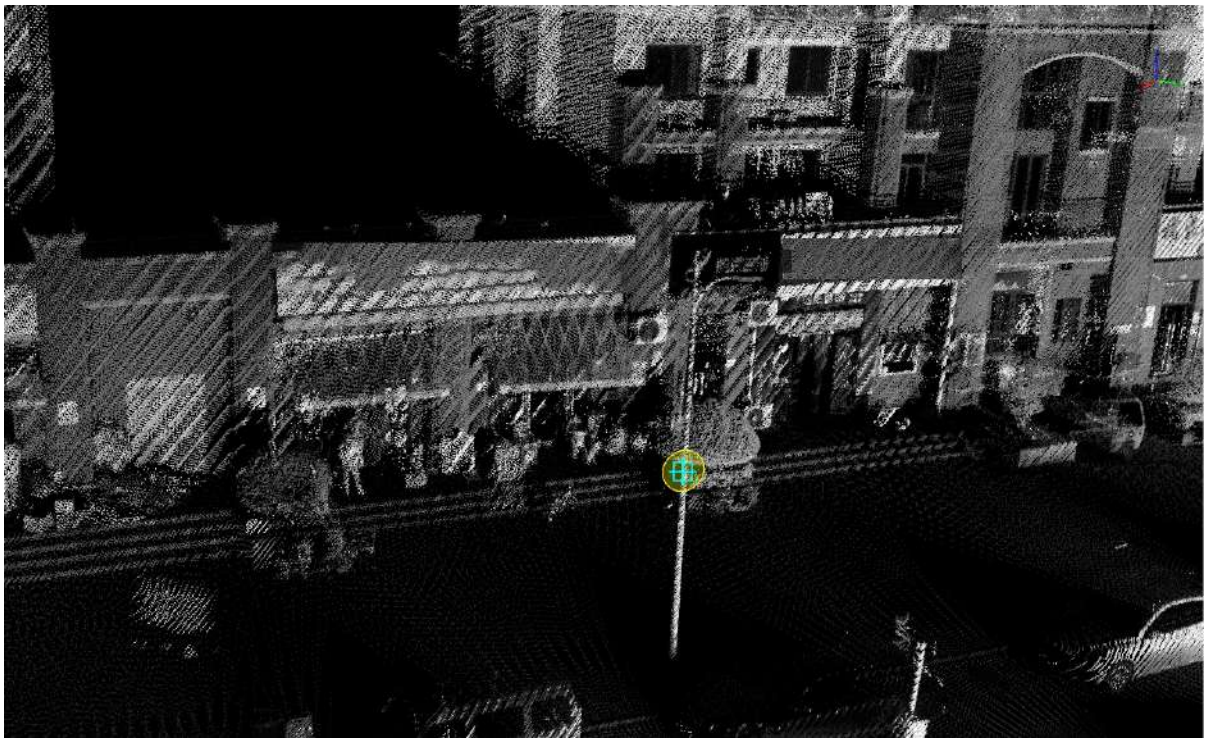


Single Point Mode Dialog Custom Line Layer

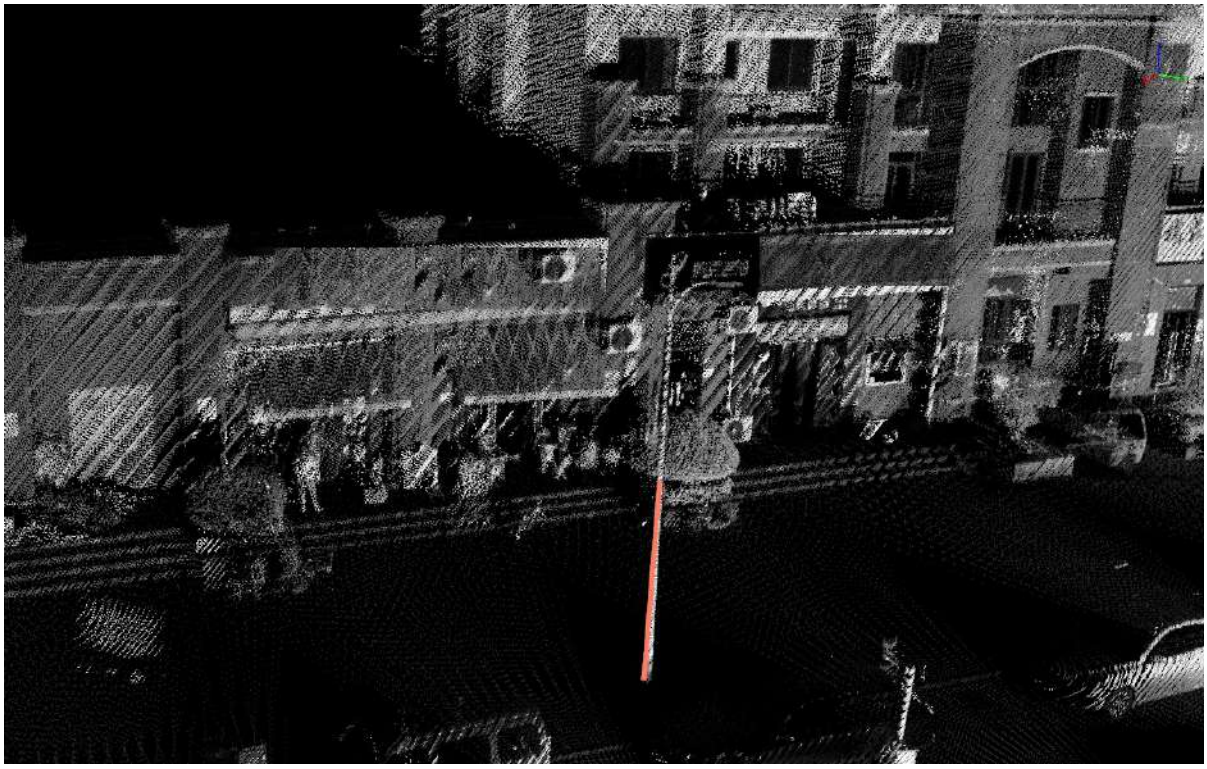
Parameter description：

- **Rod Radius**: The default is 0.5 meters, which means that the self-clicked **key point** is the reference point, and the radius of the selected box **rod is the radius, and the cylinder** is **vertically downward**.
- **Layer**: The default is street light, the options are: **streetlights, utility poles**, and also custom layers to add a line layer, indicating the layer where the vectorization results are deposited.
- **Mode**: The default is **single point mode**, which means the operation mode of **clicking on key points**.

3.**Click on the top** of the pole **with the left mouse button to automatically generate a single pole**. The details are shown in the figure. The yellow dot **is the key point** clicked by the mouse. It is recommended to click on the top position of the rod.



Click on the key point



Single point mode renderings

Two Point Mode

Function Description: The two-point mode of pole detection. To apply, manually click on the top of the rod and the bottom of the pole as the key points to generate the pole vectorization results.

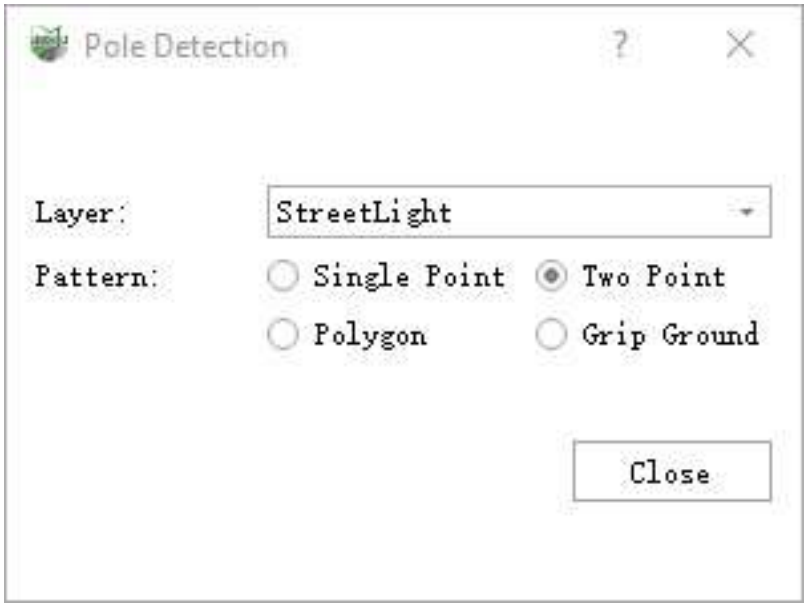
> The generated bar is **exactly in line with the two points clicked**.

Steps

1.In the Road Facilities panel click on the drop-down logo on the right and select **Detect Pole**  button.

2.The **Pole Detection** setting dialog box should pop up.

- Two Point mode.



Two-point modal dialog box

Parameter description：

- **Layer**: Default is street light, and the options are: **street light, utility pole**, Users can also customize the layer to add a line layer, indicating the layer where the vectorization results are deposited.
- **Mode**: The default is **Two Point**, which means the operation mode of **clicking on key points**.

3.**Clicktop of the rod with the left mouse button**, the details are as shown in the figure, and **yellow point** is the key point selected by mouse.

Click on the key point at the top of the pole

4.**Click the bottom of the rod with left mouse button**, the details are shown in the figure, **yellow point** is the key point of the mouse click.

Two-point mode renderings

> Note: The two-point mode with the profile drawing effect will be better and the efficiency will be faster.

Polygon Mode

Function Description: The polygon mode of pole detection can be used to select multiple rods at once. To apply, manually click on the polygonal area where the poles are located as the key point to automatically extract the vectorization results of the rods.

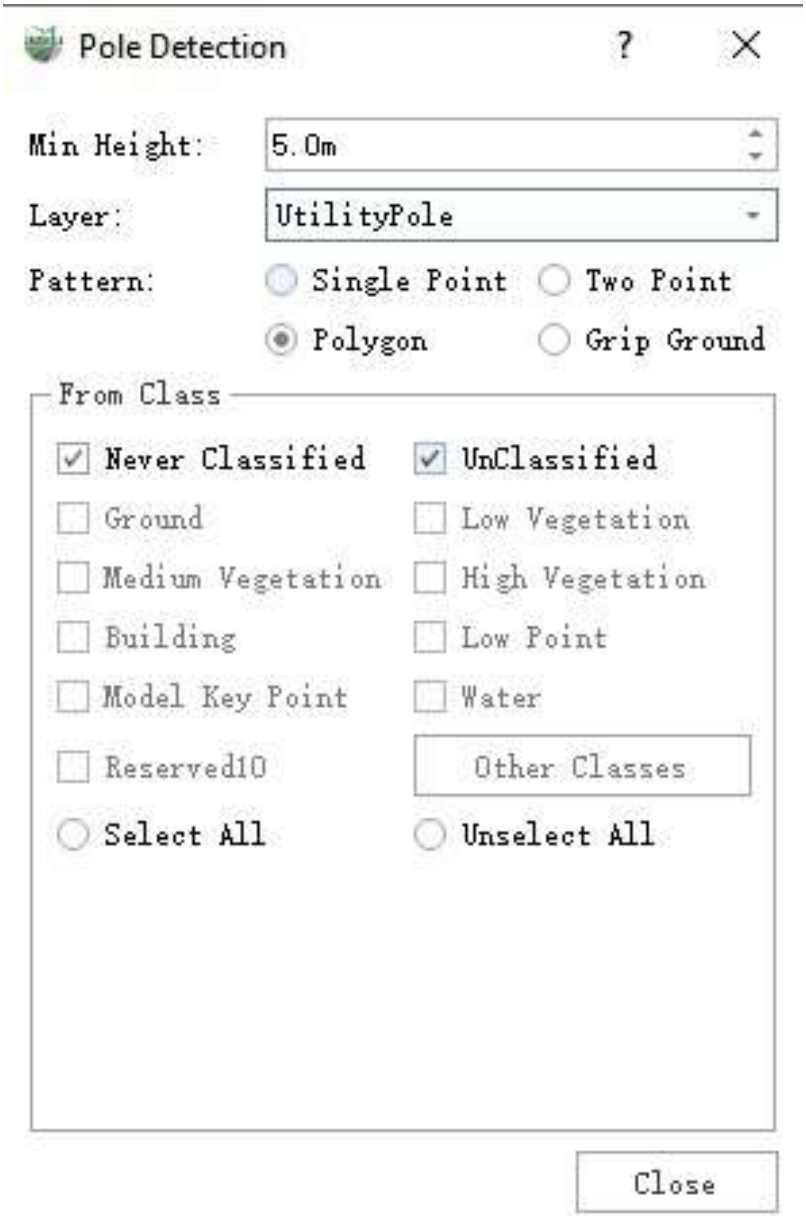
> The generated rod is **precisely perpendicular** to the ground, while the real rod is **not necessarily perpendicular** to the ground due to the **instrument scanning unilateral, tilt itself, thick at the bottom and thin at the top**, etc.

Steps

1.In the Road Facilities panel, click on the drop-down logo on the right and select **Detect Pole**  button.

2.The **Pole Detection** setting dialog box should pop up:

- Polygon mode.



Polygon Mode Dialog box

Parameter description：

- **Min Height**: The default is 5.0 meters, pole-shaped objects **below** set height **will not** be extracted, at the same time, if the height is set too low , **it is easy to cause the shape of trees similar to the rod mistakenly extracted, recommended 5.0 meters**.
- **Layer**: The default is street light, the options are: **Street light, telephone pole**, and also custom layers to add a line layer, indicating the layer where the vectorization results are deposited.
- **Mode**: Default is **Single Point Mode**, which indicates the operation mode of **Tap Key Point**.
- **Source category**：Set the input category for category filtering, if the data has been classified, you can select only the point cloud category where the rod is located to improve detection accuracy.

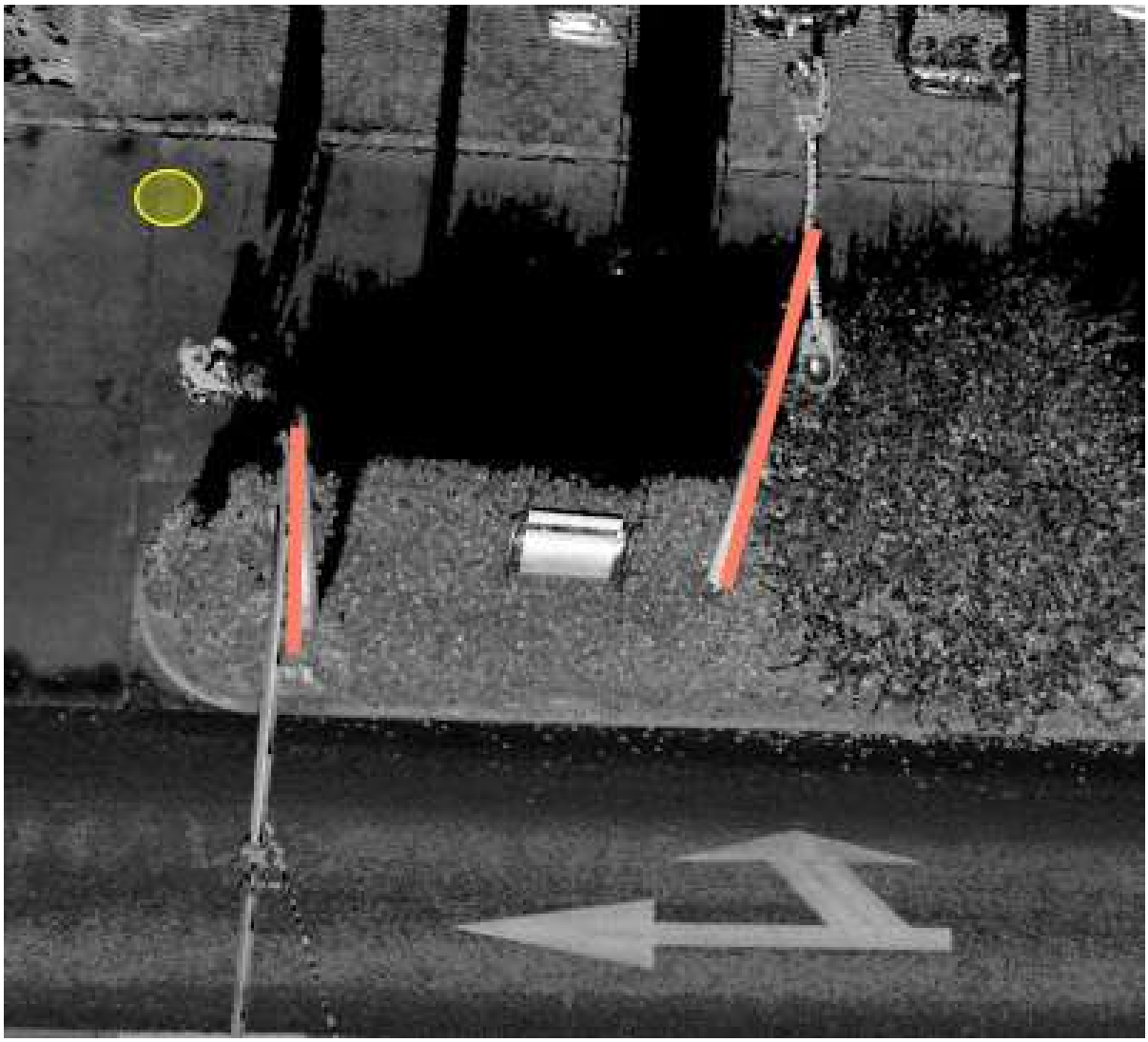
3.**Click the left mouse button in order to select the polygon area where the rod is located, double click the left mouse button to end** polygon area box selection, and automatically start **multiple rod detection**, the details are as shown in the figure, and the **yellow point** is the **key point** of the mouse click.

> **When the box is selected, try to narrow down unnecessary areas to speed up the extraction process**.

Click on the key point

4.Wait for the **detection completion pop-up window** or **detection failure pop-up window** pop-up, that is, complete **polygon mode rod automatic extraction**.

> When the boxed polygon area, **no** or **not detected** any rod, the **Detection Failure** pop-up window will appear, and the **Detection Complete** pop-up window will appear for the rest.



Detection completed

Polygon mode effect

Grip Ground Mode

Function Description: The grip point mode of pole detection. To apply, manually click on any point on the pole, and then click on the ground point, according to the set height of the rod, the rod vectorization result is generated.

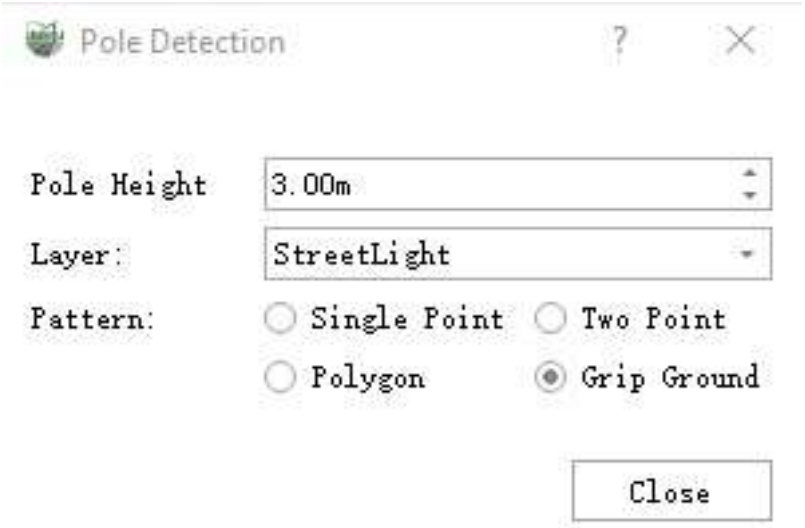
> The generated pole **rises from the clicked ground point, with a precisely set height**.

Steps

1.In the Road Facilities panel click on the drop-down logo on the right and select **Detect Pole**  button.

2.The **Pole Detection** setting dialog box should pop up.

- Grip mode



Grip mode

Parameter description：

- **Pole Height**: The height of the generated pole. In grip mode, the height is set, and the subsequent pole heights are generated according to that height.
- **Layer**: The default is street light, the options are: **streetlights, telephone poles**, and also custom layers to add a line layer, indicating the layer where the vectorization results are deposited.
- **Mode**: The default is **single point mode**, which means the operation mode of **Clicking on key points**.

3.**Click any point on the pole with left mouse button**, the details are as shown in the figure, the **yellow point** is the **key point** of mouse click.

Click on the key point of the lever

4.Click **Surrounding ground points with left mouse button**, the details are shown in the figure, **Yellow point** is the key point of the mouse click.After clicking, a three-dimensional vector line will be generated according to the height of the pole set in advance.

Click on the ground point

Renderings

Detect PowerLine

Function Description: Multiple power lines are clicked at once, and they grow automatically at the same time to generate power line vectors.

> Mainly used for simultaneous growth of multiple parallel power lines.

Steps

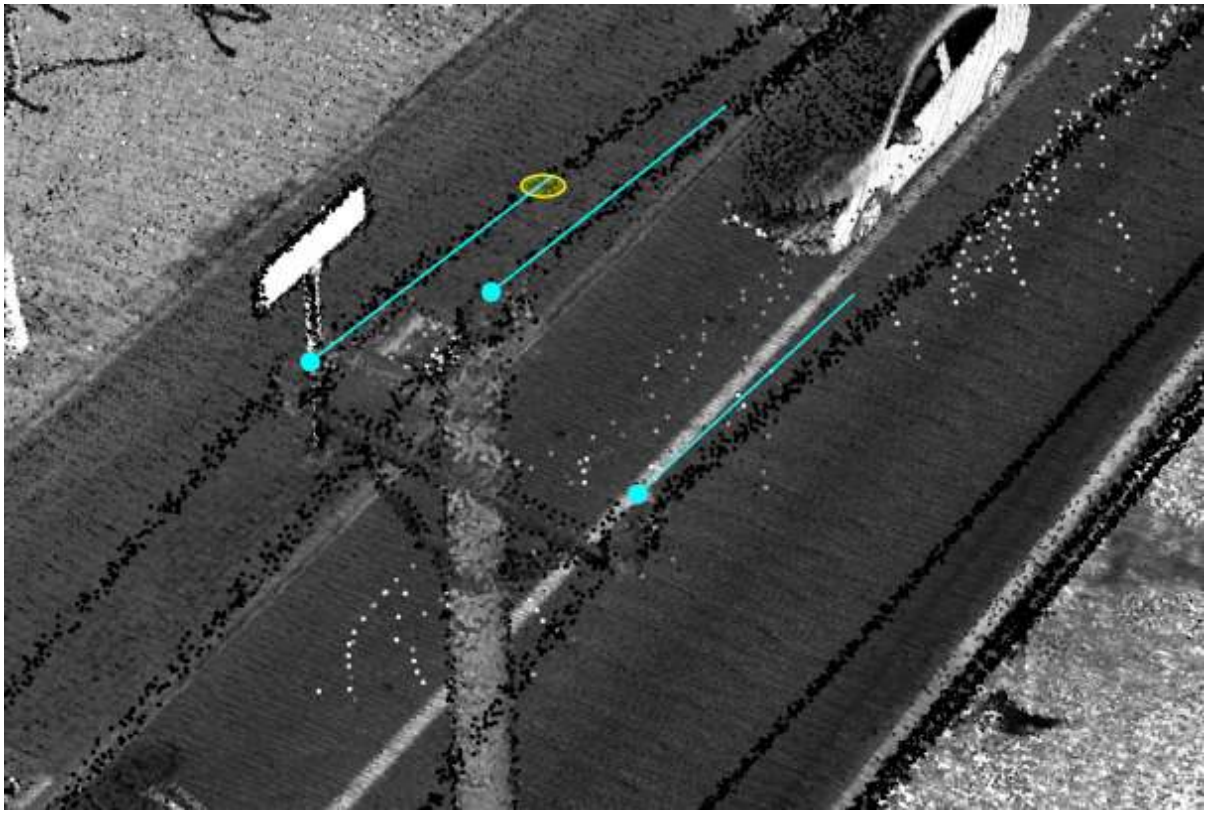
1.In the Road Facilities panel click on the drop-down logo on the right and select **Detect PowerLine**  button.

2.Click **The starting point of the growth of each power line with left mouse button**, the details are shown in the figure, double-click at the starting point of the last power line to end the click.

Click on the key point

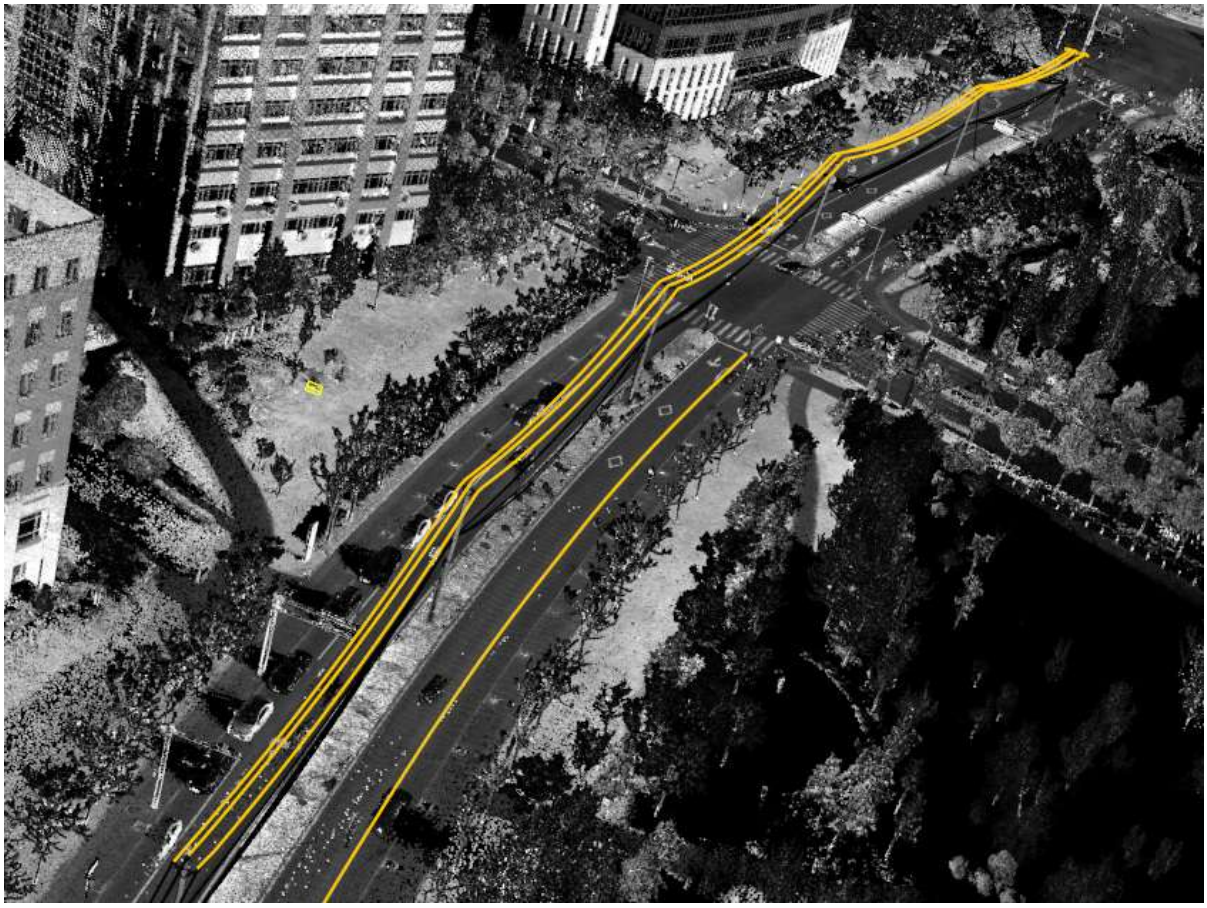
3.Click the mouse to get the growth direction of the power line.

> Just click on the direction of the first power line, and the directions of the other power lines will be automatically parallel.



Click on the growth direction

4.After clicking on the growth direction, it will automatically start to grow



Click on the growth direction

Add TrafficSign

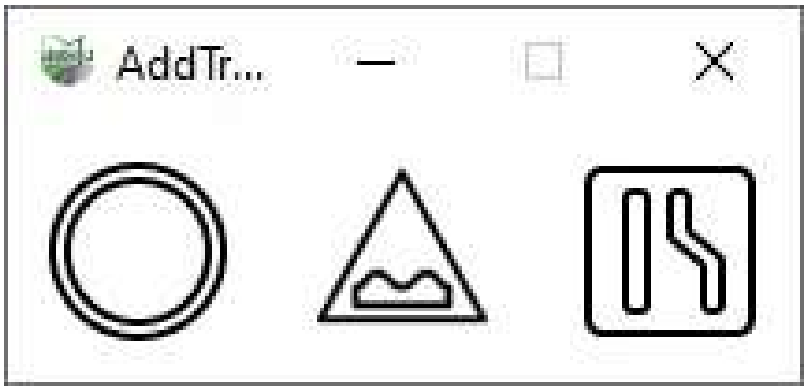
Function Description: This tool can be used to manually add traffic signs.

> In order to obtain the plane position correctly, users need to turn on the mouse hover.

Steps

1.In the Road Facilities panel, click on the drop-down sign on the right and select **Add TrafficSign**  button.

2.The **Settings dialog box** should pop up. Select the shape the user wants to draw:



Dialog box

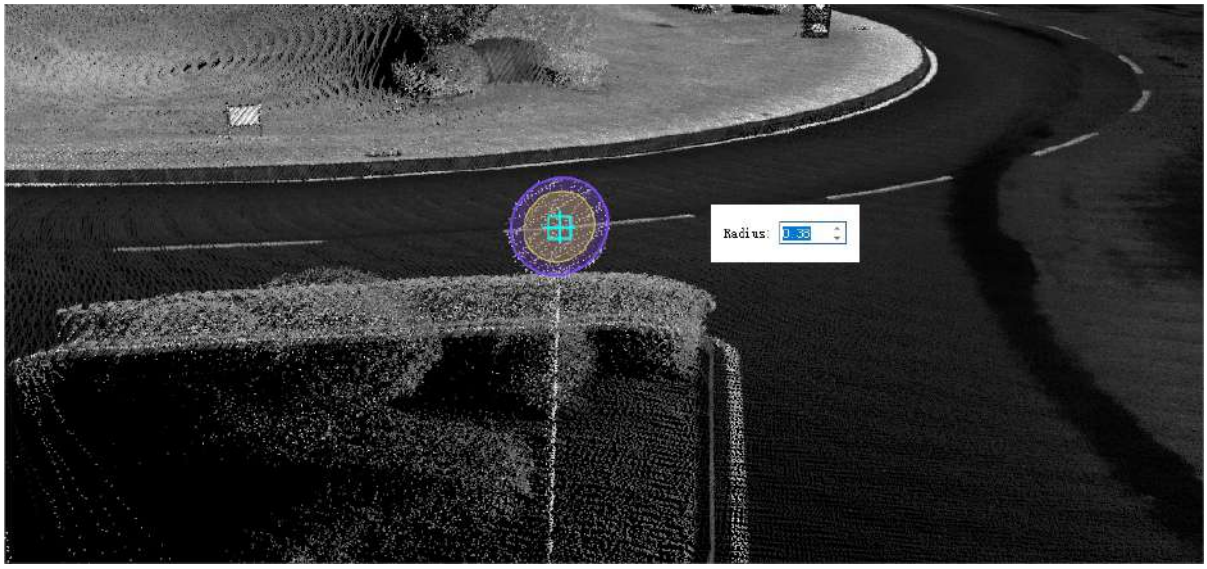
Parameter description：

- In order: round signs, triangular signs, rectangular signs.

3.Circle and triangle sign drawing: After selecting the shape to be drawn, the graphics to be drawn will be displayed on the mouse. Place the mouse on the center of the sign point cloud, use the shortcut keys to rotate, use the right mouse button to bring up the dialog box to adjust the size, **Double-click** to determine the current sign.

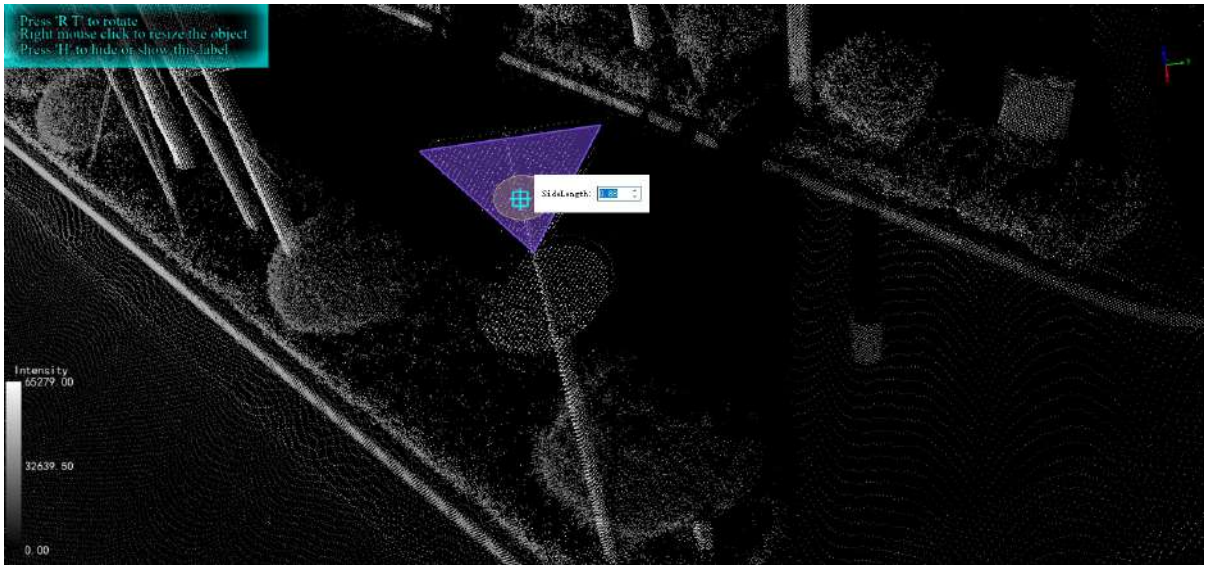
Shortcut key name	Function
r	Rotate counterclockwise
q	Rotate clockwise
Right mouse button	Change the size of the vector template

Round sign



Round

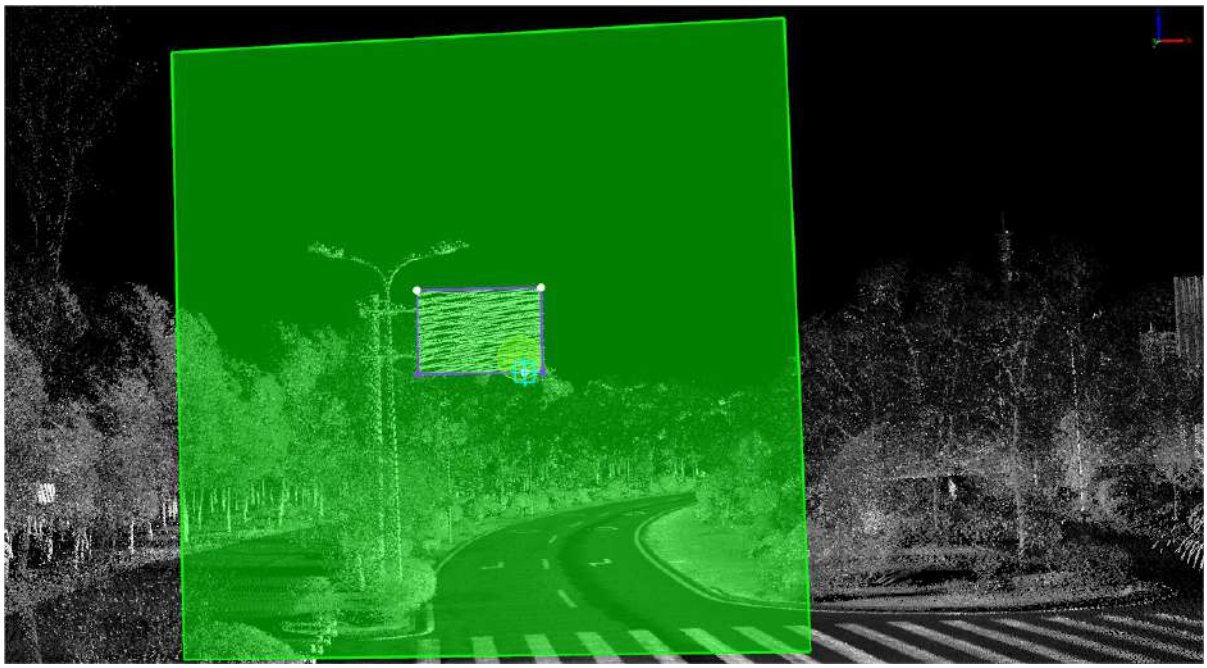
Triangle sign



Triangle

4.Rectangular sign drawing: Turn on the point cloud fitting function, click with the left mouse button on the point cloud plane of the road sign to generate a fitting plane, and continue to use the left mouse button to draw a rectangle with three points on the plane to draw a rectangle.

Rectangular sign



Rectangle

Add Parking Spot

Function Description: Manual generation of parking space vectors in two modes with and without common edges.

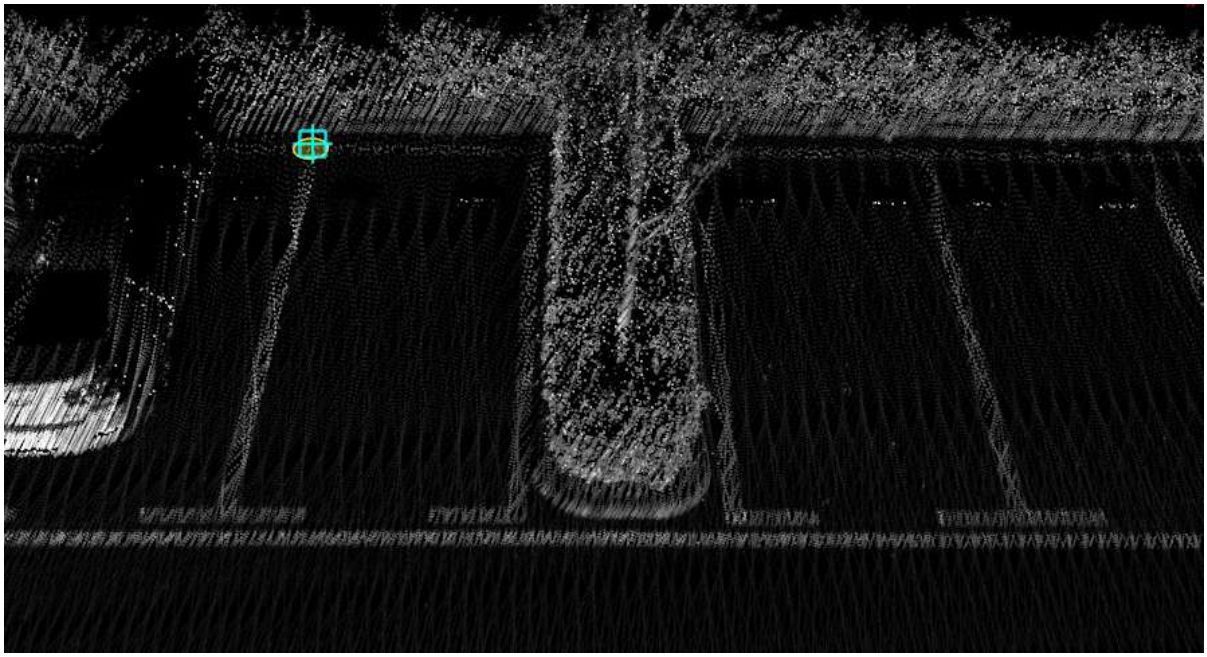
> For parking spaces with a communal edge, use a 4-point sub-pattern; for parking spaces without a communal edge, use a 5-point sub-pattern.

Steps

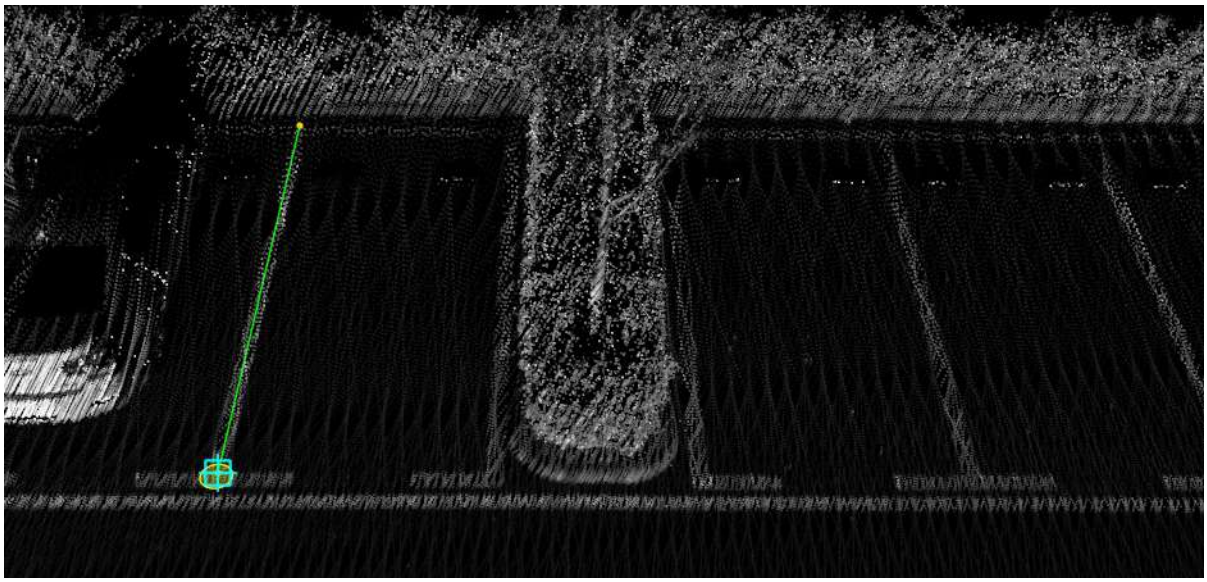
1.In the Road Facilities panel click on the drop-down sign on the right and select **Add Parking Spot**  button.

No communal side parking：

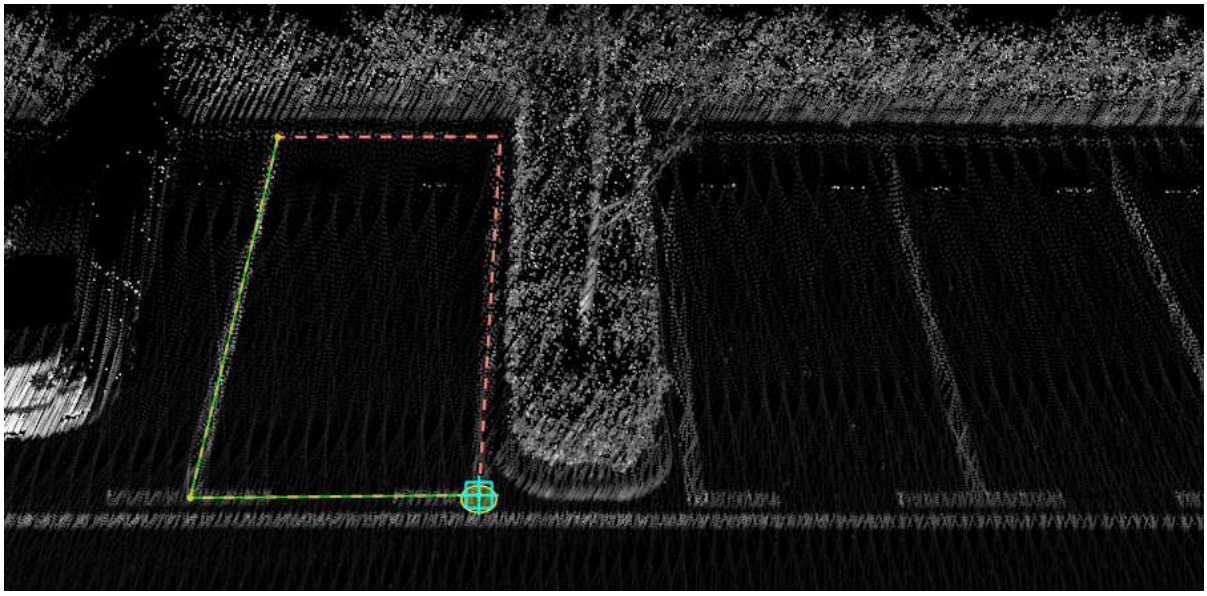
2.The function is activated and the default is 5-point mode, i.e. no common edges by default. **left mouse button** to tap the three corner points of a single parking space in turn for determining the size of the parking space



Key points

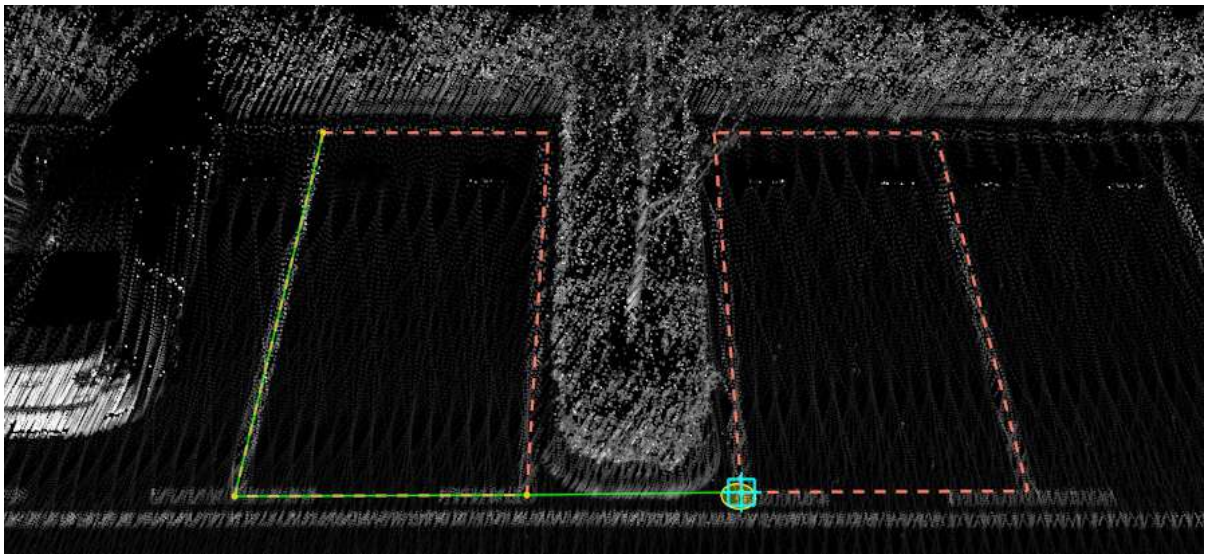


Key points



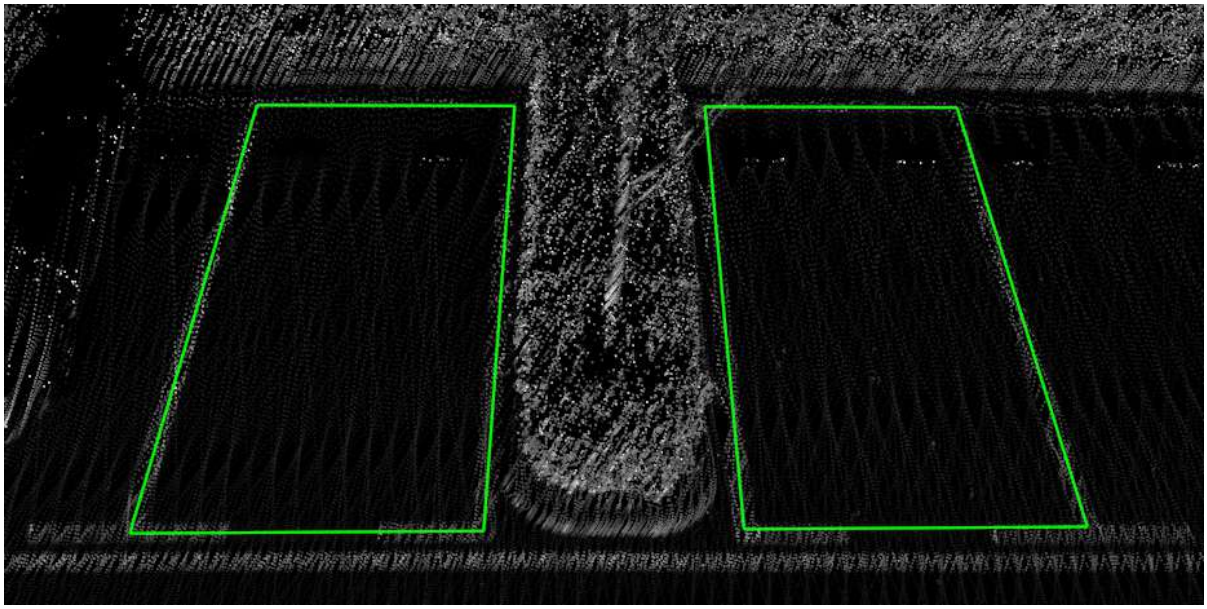
Key points

3.Move the mouse to see the second parking space move with the mouse, **left mouse button** tap the corner of the next parking space to determine the distance between the two parking spaces, special attention needs to be paid to the location of the tap



Key points

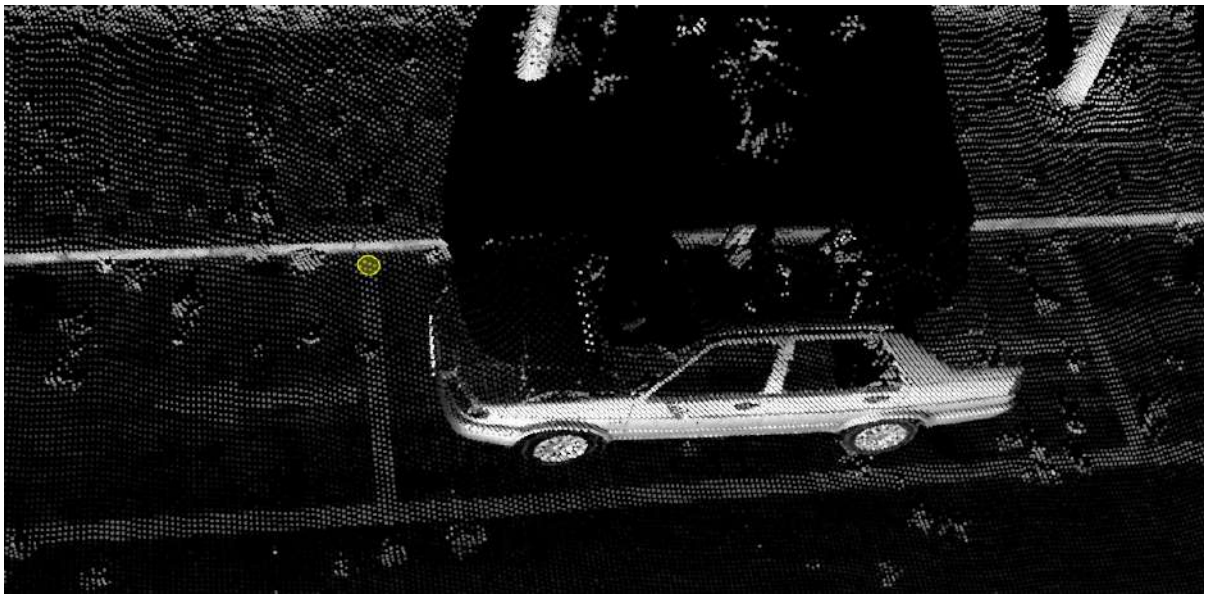
4.Move the mouse to automatically update the number of parking spaces in real time according to the size of the individual parking spaces and the spacing between them **double click** the fifth key point to complete the production



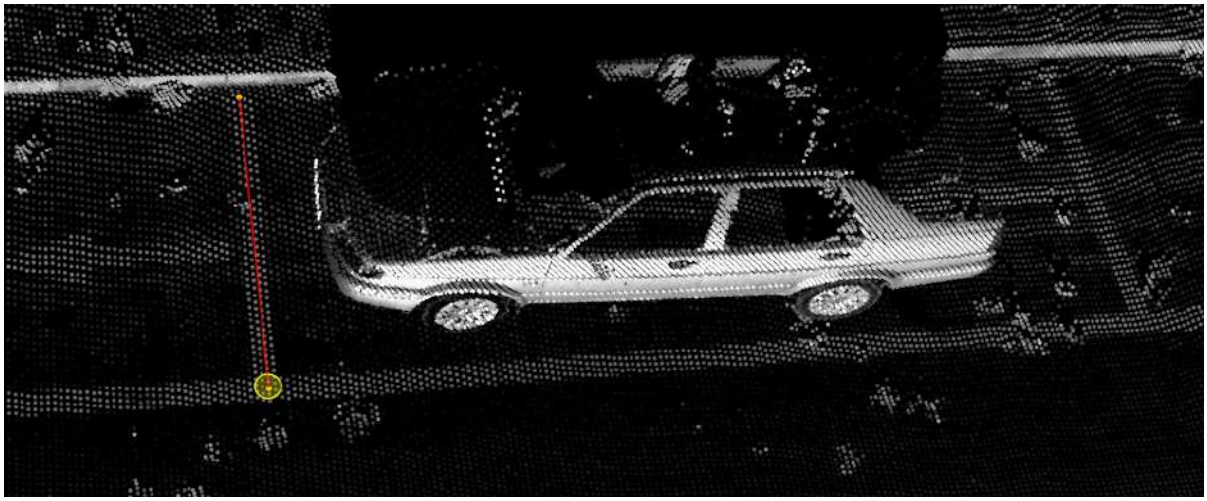
Key points

Communal side parking available

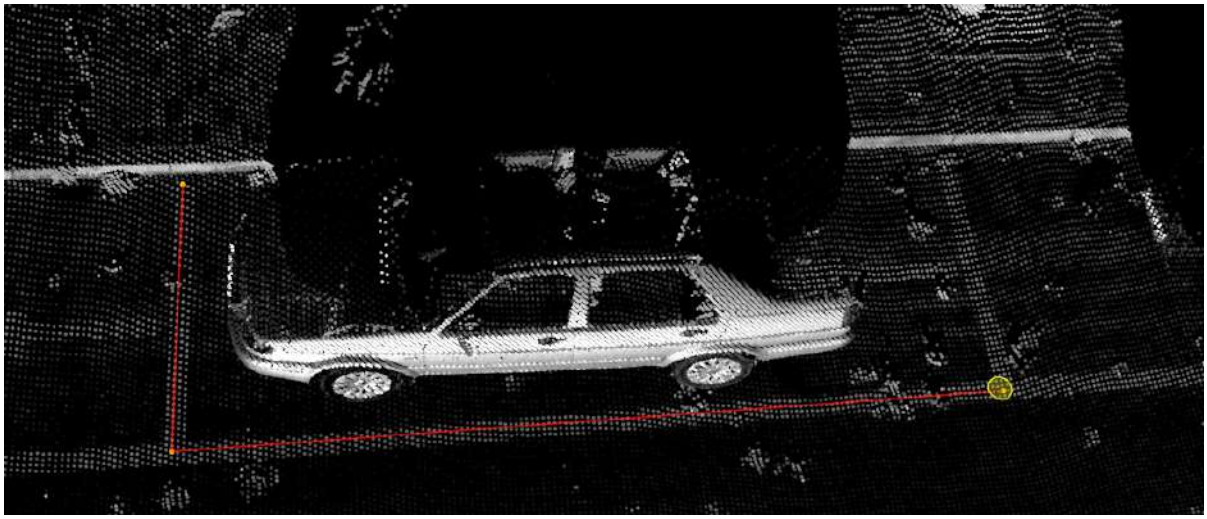
2.**Left mouse button** to select each of the three corner points of a single parking space



Key points

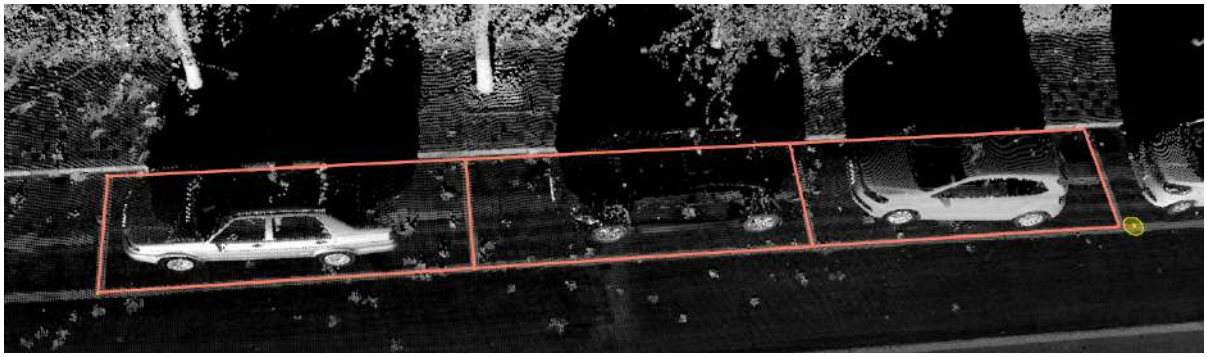


Key points



Key points

3.After three clicks to identify a single parking space, move the mouse and the number of spaces will be automatically updated in real time according to the length of the second key point and the fourth key point, **double click** the fourth key point to complete the production



Renderings

Add Rain/Strainer

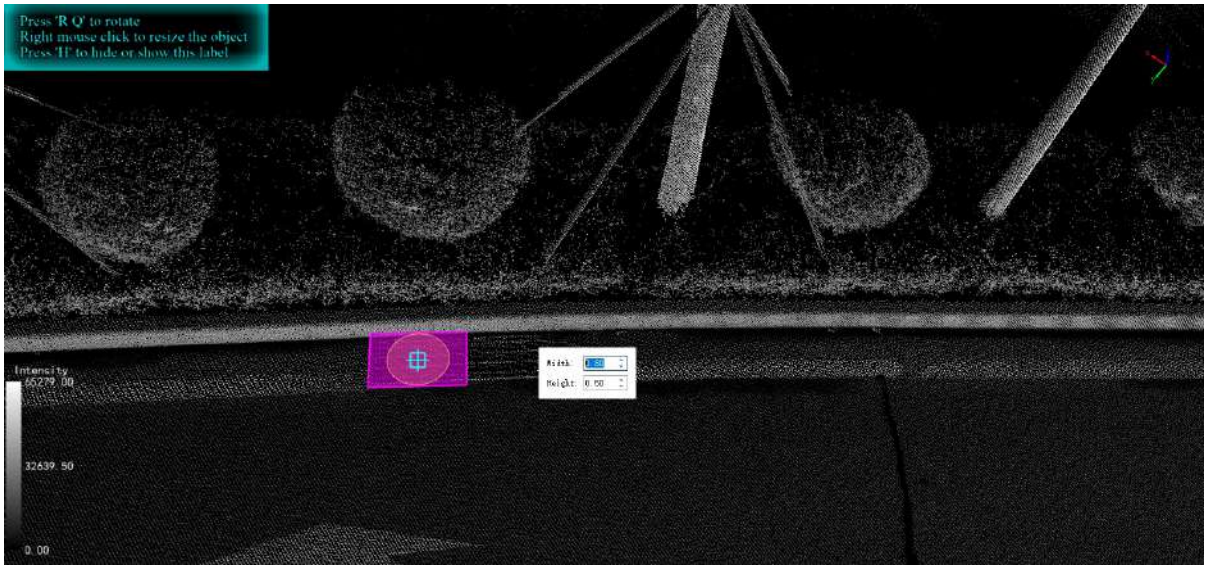
Function Description: This tool can be used to manually add a drain grate.

> In order to obtain the plane position correctly, users need to turn on the mouse hover.

Steps

1.In the Road Facilities panel click on the drop-down sign on the right and select **Add Rain/Strainer**  button.

2.The graphics to be drawn will be displayed on the mouse in real time. Place the mouse at the center of the point cloud, use the shortcut key to rotate, use the right mouse button to bring up the dialog box to adjust the size, and **Double-click** to determine the current vector.



Renderings

> Note: Shortcuts:

Shortcut key name	Function
r	Rotate counterclockwise
q	Rotate clockwise
Right mouse button	Change the size of the vector template

Add Manhole

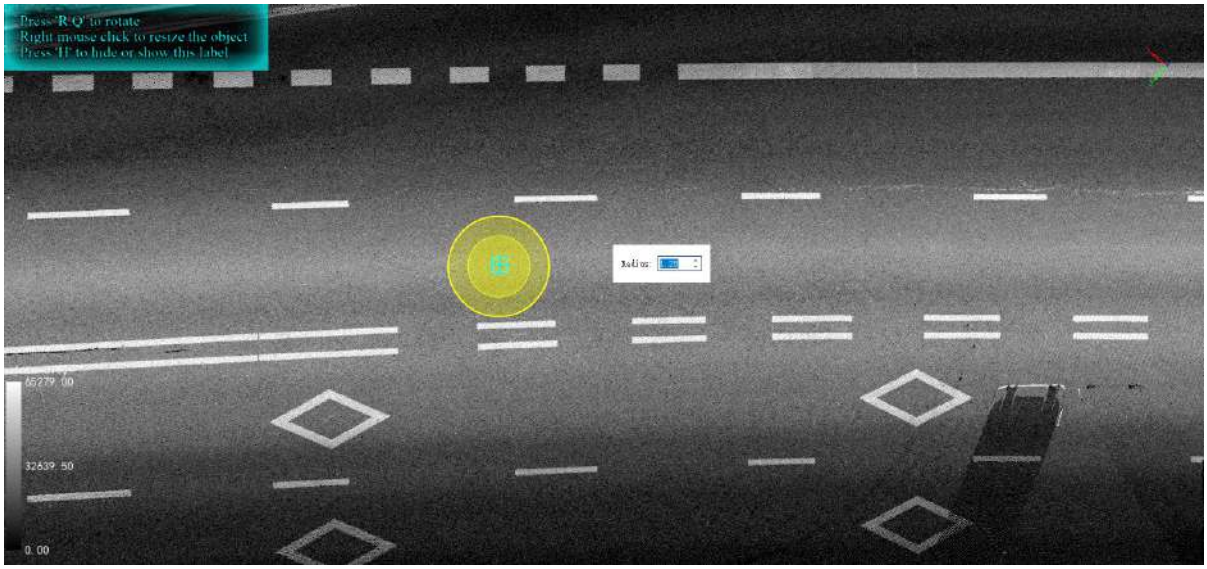
Function Description: This tool enables manually add manhole covers.

> In order to obtain the plane position correctly, users need to turn on the mouse hover.

Steps

1.In the Road Facilities panel click on the drop down sign on the right and select **Add Manhole**  button.

2.The graphics to be drawn will be displayed on the mouse in real time. Place the mouse at the center of the point cloud, use the shortcut key to rotate, use the right mouse button to bring up the dialog box to adjust the size, **Double-click** To determine the current vector.



Renderings

> Note: Shortcuts:

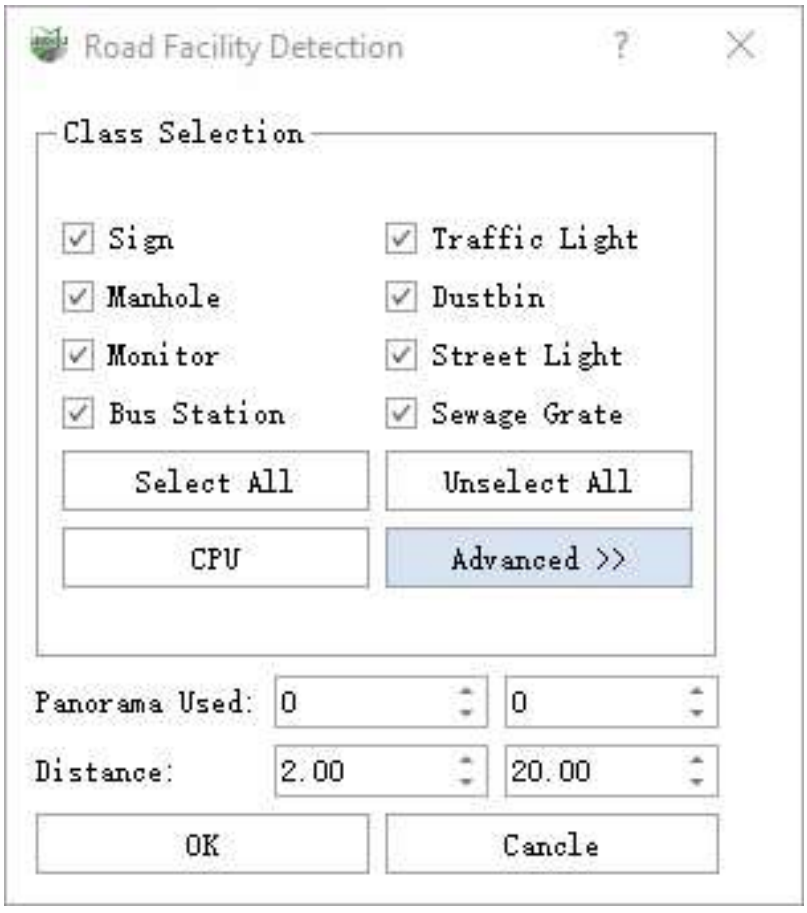
Shortcut key name	Function
r	Rotate counterclockwise
q	Rotate clockwise
Right mouse button	Change the size of the vector template

Batch processing of road facility recognition based on panoramic images

Function Description: This tool enables the use of panoramic images to assist in the automatic identification of road facilities. Among them, **traffic signs** reflect the results with plane-like elements (plane-like vectors), and **The rest** reflect the results with point elements (point-like vectors).

> Note: This function is only valid for engineering data with panoramic images.

1.Click **Road Facility Detection**  button and the parameter dialog box should pop up.



Parameter dialog box

Parameter description:

- **Category selection**: Displays the extraction categories supported by the current function, and users can check/uncheck the extraction categories as needed.

  > The ticking options are proportional to the time-consuming, that is, the more the user checks, the longer the entire processing process will take. Please tick them according to the actual situation.

- **Select all/cancel all selections**: Click the button to check all supported categories/uncheck all.
- **Pattern：CPU、Pattern：GPU：** For details, please refer to Classify by Deeplearning to set the operation mode.

> GPU mode requires computer hardware and software environment support, if the conditions are not met, even if the checkbox is checked, it will still run in CPU mode.

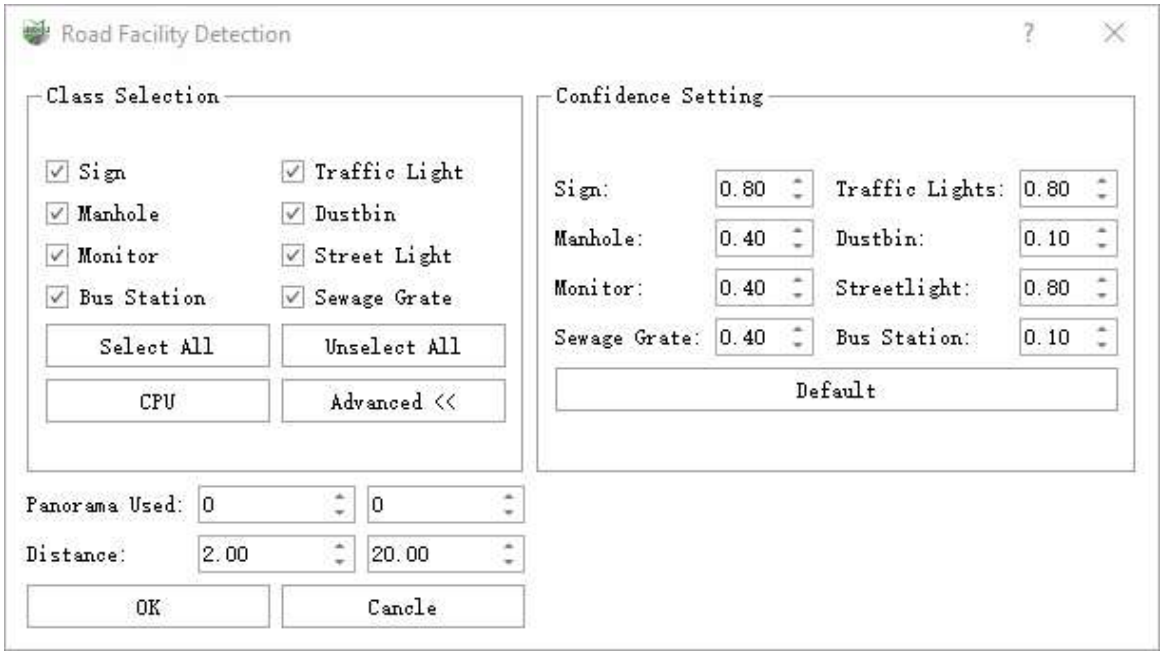
- **Use image**: Used to determine the picture number used in this processing, the default starts from 0, and the ending with the last picture in the project.

> Can be used in conjunction with the selection frame, fill in the start and end numbers of the image where the area of interest is located, and the precise start and end image numbers will greatly reduce useless calculations.

- **Distance**: The distance between the target and the center of the camera, less than the shortest distance or greater than the maximum distance, will not be extracted, **the default minimum distance is 2.0 meters, and the maximum distance is 20.0 meters**.

> The appropriate distance can help reduce incorrect extraction. It is recommended to use the default value.

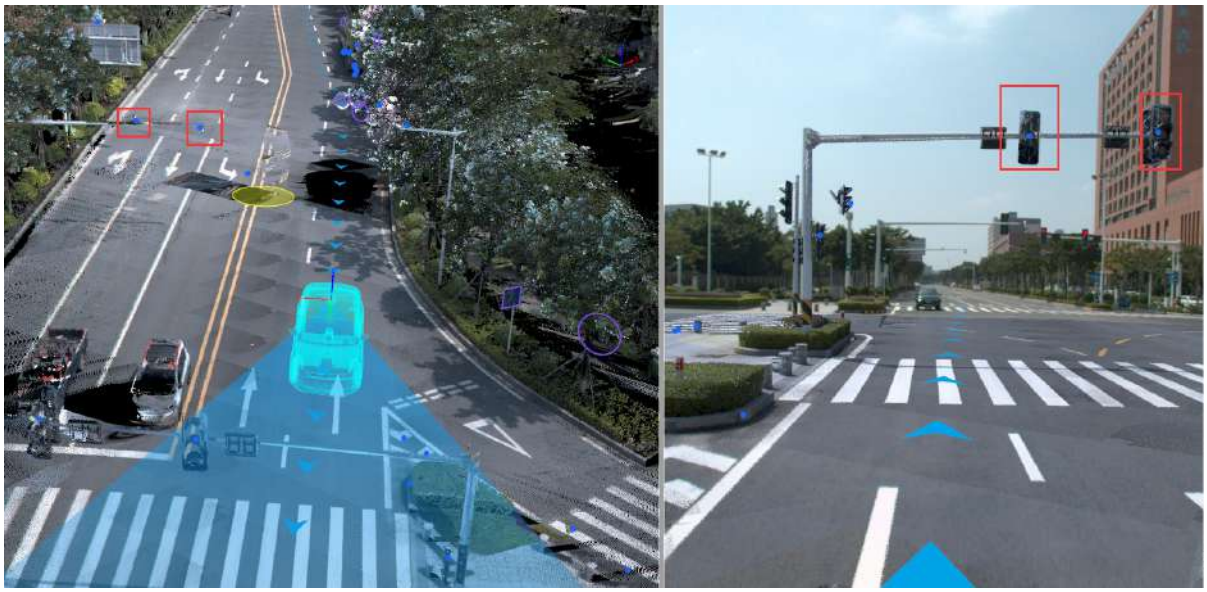
- **More**: Show/close the confidence setting window



More dialogs

  - **Confidence Setting**: For the categories supported by the current function, set the corresponding confidence level separately. And the targets below the set confidence level will be ignored.
  - **Default**: This button is used to set the confidence level of all categories to the default value.
- **OK**: After the parameters are set, click the OK button to start automatic detection.
- **Cancel**: Exit function.

2.Click OK for automatic detection.



Detection results

Linear Facility Extraction

Function Description: This tool can be used to create and edit a template. Strip object template extraction is divided into template selection and editing, and banded object extraction. The following will be described in detail.

Description

- **Template selection and editing**: Including template selection, template point cloud, key point editing and template saving.
- **Strip object extraction**: Draw a path based on the selected template, and then perform detection and extraction use manual or semi- automatic methods.

Template editing and selection operations

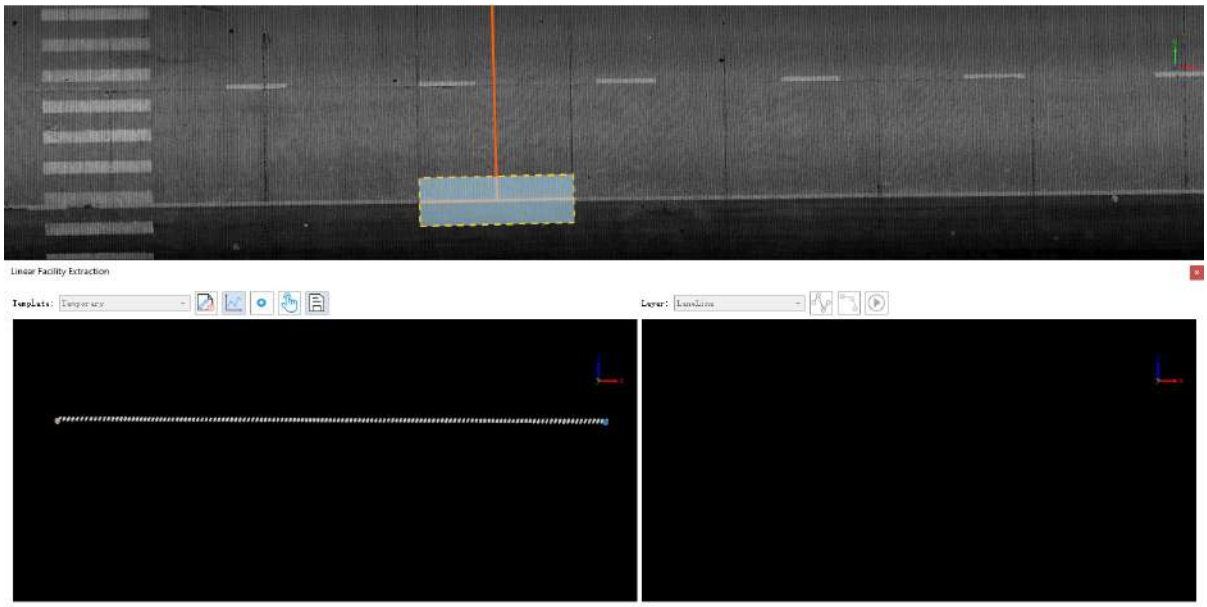
- **Template view**: The template view is located on the left and is used to display the currently applied template point cloud and key points.
- **Template selection**: When the function starts, the program automatically loads the template file located in the installation directory and places it in the template selection box. The user selects the template that matches the detected object. (pay attention to the direction of the template point cloud). After the template is set, the software will automatically copy the template file to the user data folder (AppData), and then read the template data from this folder.
- **Edit**: Enable and disable template editing. After template editing is enabled, the detection function button cannot be used, and it will be restored after template editing is turned off.
- **Create a template**: The interaction logic is the same as the profile, and a template point cloud with the appropriate thickness is drawn perpendicular to the direction of the strip object. The drawing result will be displayed in the template view on the left.
- **Add key points**: After creating the template point cloud, select the key points on the template point cloud as the growth points of the banded object vector line.
- **Select key points**: Used for key point editing. After selecting, use the "Delete" key to delete the key points, and long press and drag the left button to modify the position of the key points.
- **Save template**: The temporarily drawn template is placed in the "temporary" when it is temporarily drawn, and the template can be used the next time it is extracted after it is saved.

Operation steps：

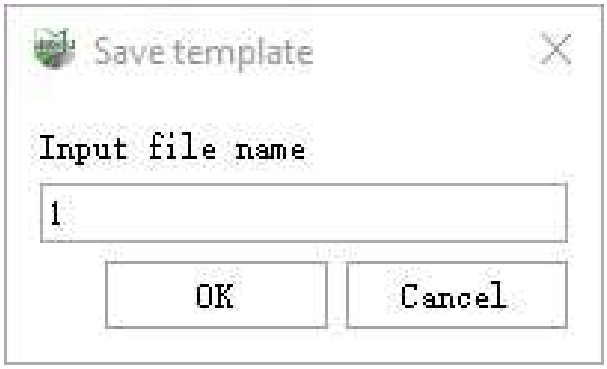
1.In road facilities panel, click **Linear Facility Extraction**  button, open the editing mode.

2.Click **To edit the template**  button, open template editing.

3.Click **To draw a template**  button, select the profile to be drawn in the point cloud, and add key point 1, key point 2.



4.Click **To save the template** button and the save template dialog box should pop up, set the file name, and click OK.



5.Click **To complete the edit** button to close editing.

Strip extraction operation

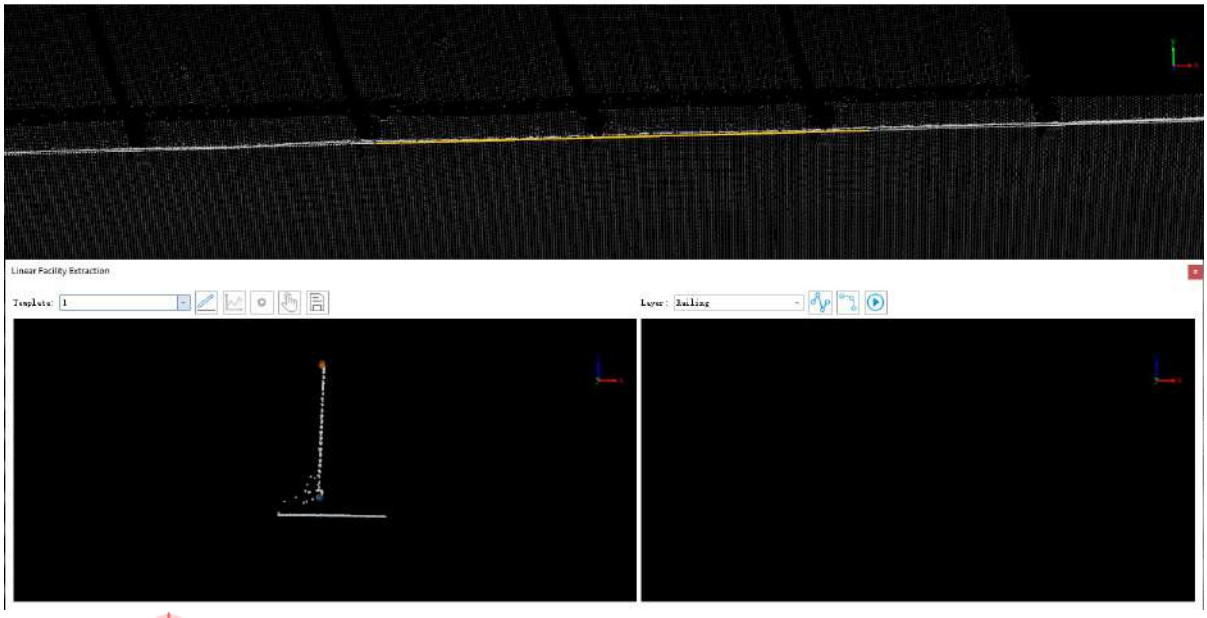
- **Result view**: The result view is located on the right, which is used to display the results of each extraction, and supports adjustments to the key points of the extraction results.
- **Layer selection**: Select the result to save the layer (line layer).
- **Draw a straight line**: After activation, draw a straight line along the strip-like object in the 3D view, the first line is used to provide the initial detection direction, and extract it from the second point. The extraction result is displayed in the result view on the right.
- **Draw an arc**: Draw an arc path, suitable for curved objects. Support ‘S’ shortcut key to switch between the two shapes of “straight line” and “arc”, and right-click to return to the previous point.
- **Automatic detection**: After completing the drawing of a line segment path (two points), automatic detection can be carried out, and detection can be carried out every 1.5 seconds. If the detection result does not meet the requirements, users can use the spacebar to stop the detection, and use the right mouse button to back the point that does not meet the requirements, and click the spacebar again to continue the forward detection. After the probe is complete, it can be terminated with Enter..

Operation steps：

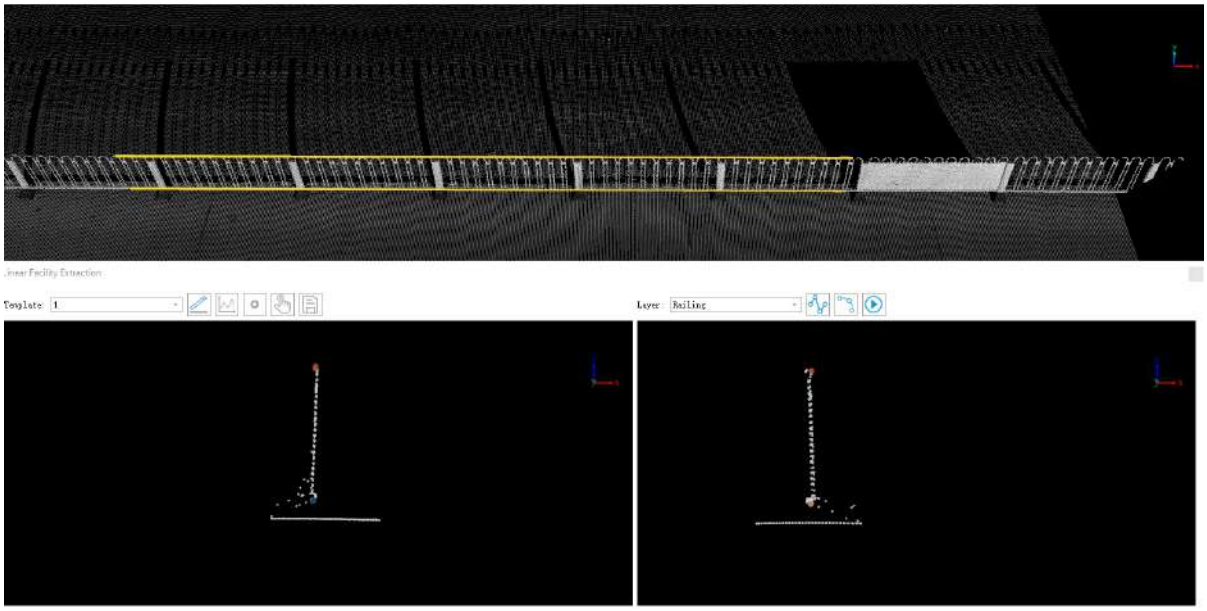
1.Select the template in the template drop-down box on the left side of the strip object extraction, and select the layer the user needs in the layer on the right.

2.Click **line** Button, draw the vector line of the object to be extracted in the point cloud, and the right mouse button can fall back one node.Click auto detect button, it can be automatically detected.

> Users can also choose according to actual needs **Arc** button.



3.Click **stop** button to end the automatic detection.Double-click with the left mouse button to end the editing.



Strip extract results

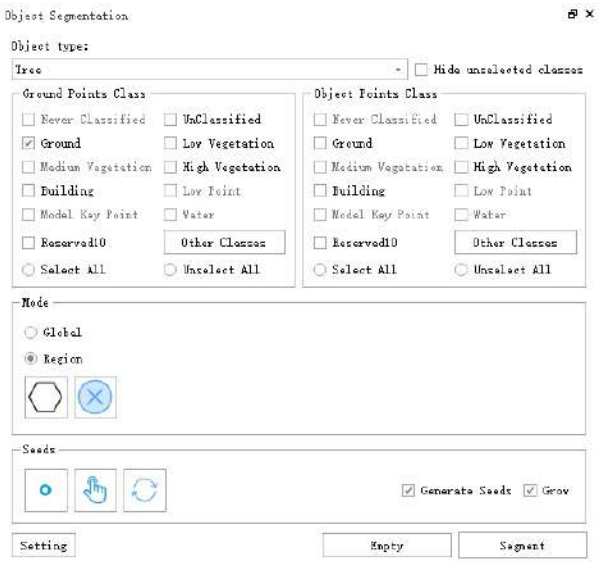
Figure objects

Object Segmentation

Function Description: This tool enables the segmentation and removal of two types of monomers, tree and rod, which will be described in detail below.

Steps

1.Click **Object Segmentation**  button and the following box should pop up：



Monomer segmentation

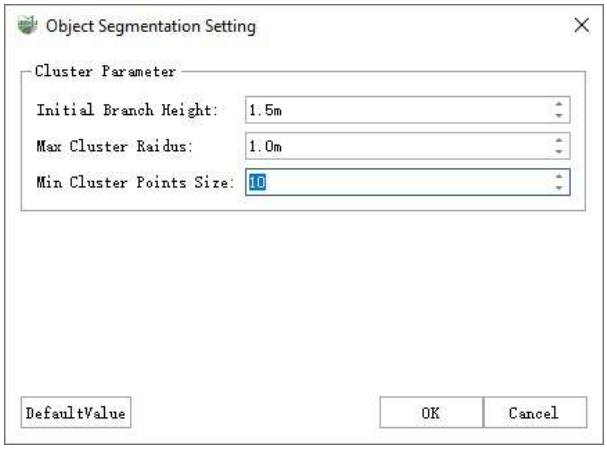
- **Object type**: Both tree and rod types can be supported currently.
- **Hide unselected classes**: After checking, the unselected category points will be hidden and can be used with any rendering method to improve the convenience of seed point operation and point cloud observation.
- **Ground Point Class**: The category where the ground point is located (check the category where the ground point is located to assist in segmentation and improve the robustness of the algorithm).
- **Object Point Class**: The category of the monomer to be divided (for example, when the tree is divided, check the category of the tree point).
- **Mode**: It is divided into global and local modes. The global mode is to divide all point clouds under the project, and the local mode is to divide the point clouds in the selected area.
- **Seeds**：
  - Click the Seed Points button to add seed points to the point cloud.
  - Click the Move button to move the added or generated seed points.
  - Update button:
    - When the vector result does not contain a single body already in the point cloud, the single body points are generated and the relevant parameters are updated.

- When the vector result contains a single point already in the point cloud, only the relevant parameters are updated.
- No operation is performed when the vector result contains a single point that does not exist in the point cloud.

> Note: All three operations are for single points (points with values in the Object ID field), and will not operate on general vector points added through the vector editing tool

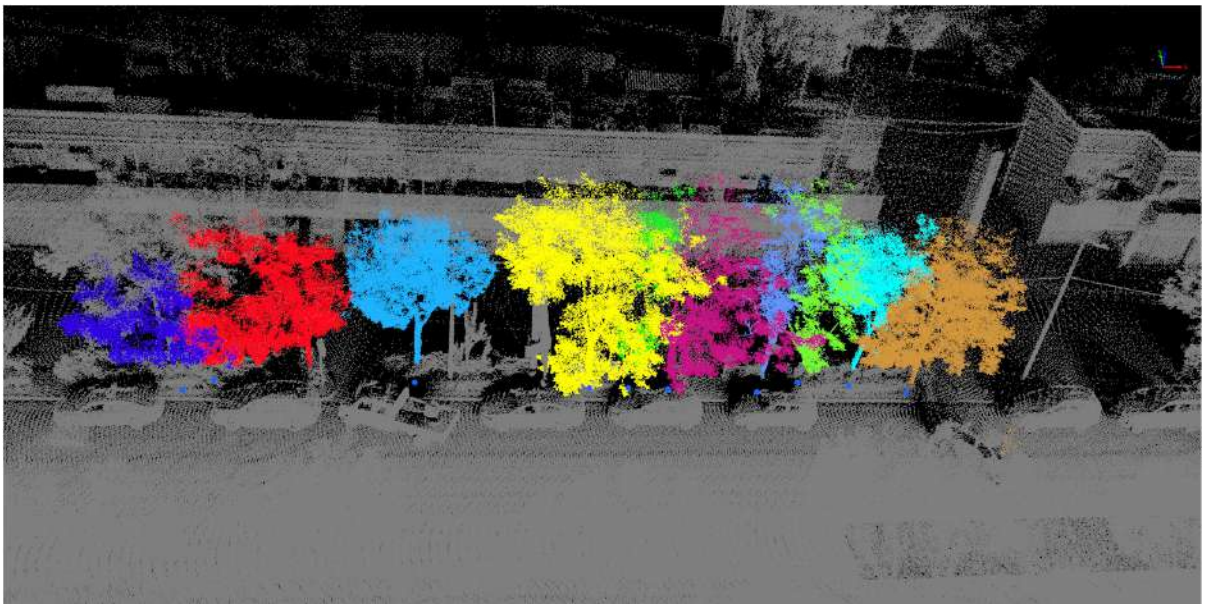
> Tip: When adding and moving, users can use it in combination with [Hide unselected categories] and [Quick horizontal section] for easier operation.

- **Segmentation**: Segmentation is divided into two steps: seed point generation and growth from seed point. The calculation process is controlled by checking.
  - If only [Generate seed points] is checked, after the segmentation is over, there will only be seed points in the result.At this time, it is used to manually add seed points or modify the location of seed points, which is suitable for scenes where the user wants to edit the seed points twice.
  - After the seed point is determined, only check [Growth]. After the segmentation is over, the final segmentation result will be generated to ensure the correctness of the number of trees.The segmentation results are displayed in the 3D view, attribute table, and segmentation result table.
- **Mode**: Divided into global and local modes
  - Global mode: During operations such as seed point generation, segmentation based on seed points, or clearing segmentation, all point clouds of the set single point type are operated;
  - Local mode: Using the polygon tool, the analysis and processing area can be customized. The operation process of seed point generation, segmentation based on seed points, or clearing segmentation is only carried out in the selected area.
- **Settings**: Used to set clustering parameters.



  - Initial Branch Height (only valid for tree segmentation), set to the minimum trunk height in the area to be divided.
  - Max Cluste Raidus: the minimum distance between clusters.
  - Min Cluster Points Size: minimum number of points for a single cluster.

2.After clicking **Split**, the split result is shown in the figure：



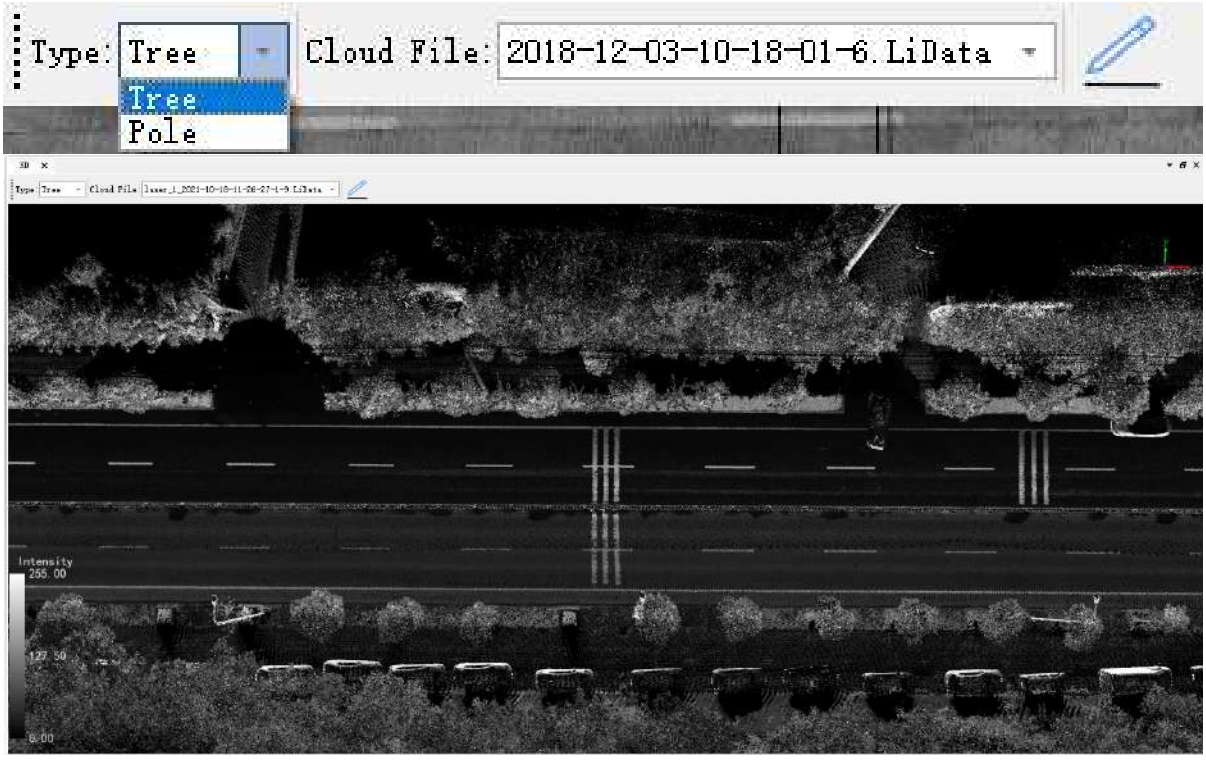
3.Click **Clear**, all objects of the specified type in the target area will be cleared, and the point object ID will be set to zero.

Object Edit

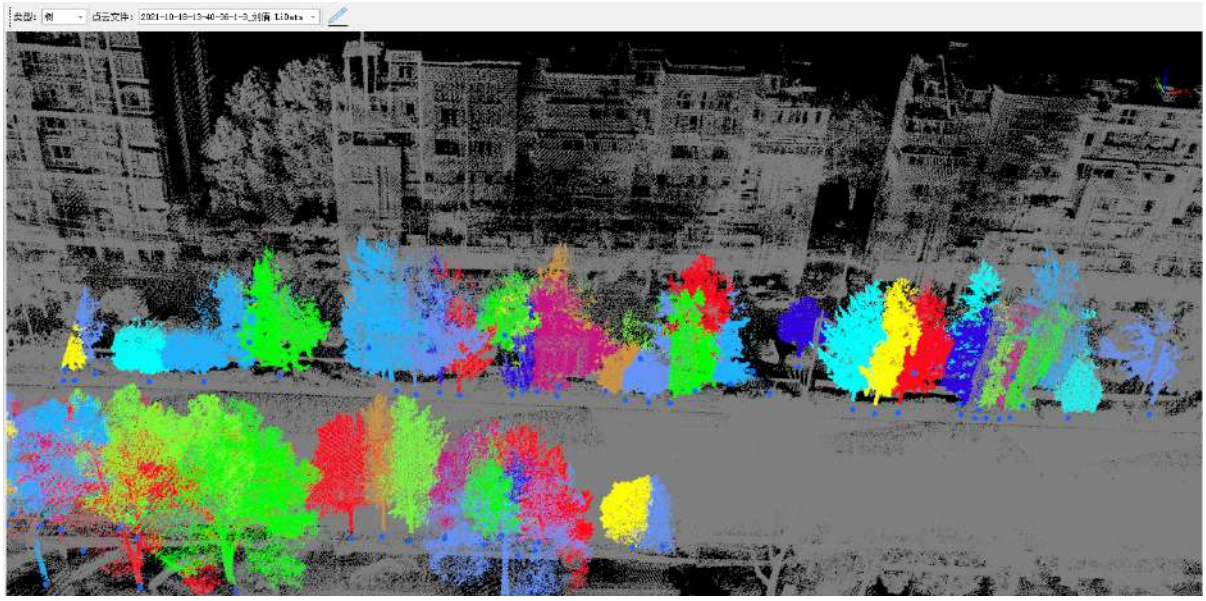
Function Description: This tool enables the manual editing of tree and Pole monomers, including the functions of creating, deleting, and setting monomers. The following will be described in detail.

Steps

1.Click **Object Edit**  button.After selecting the category and point cloud file to be edited, click **Start Editing**  button. In this case, the 3D window point cloud will be displayed according to the selected monomer type ID.



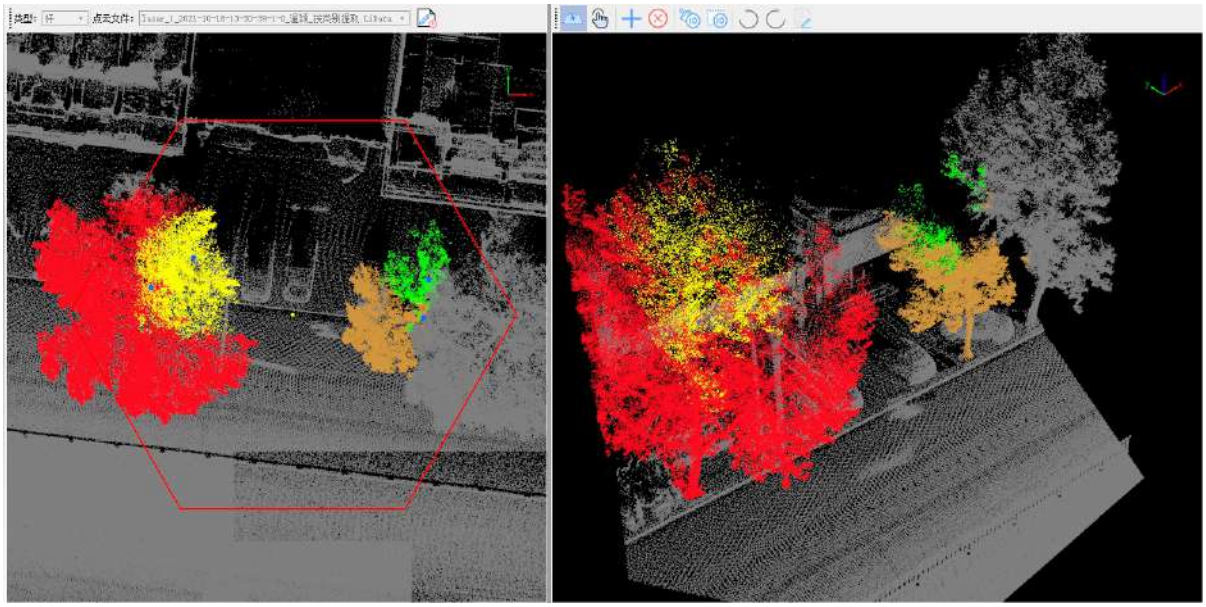
Before user editing



After user editing on trees

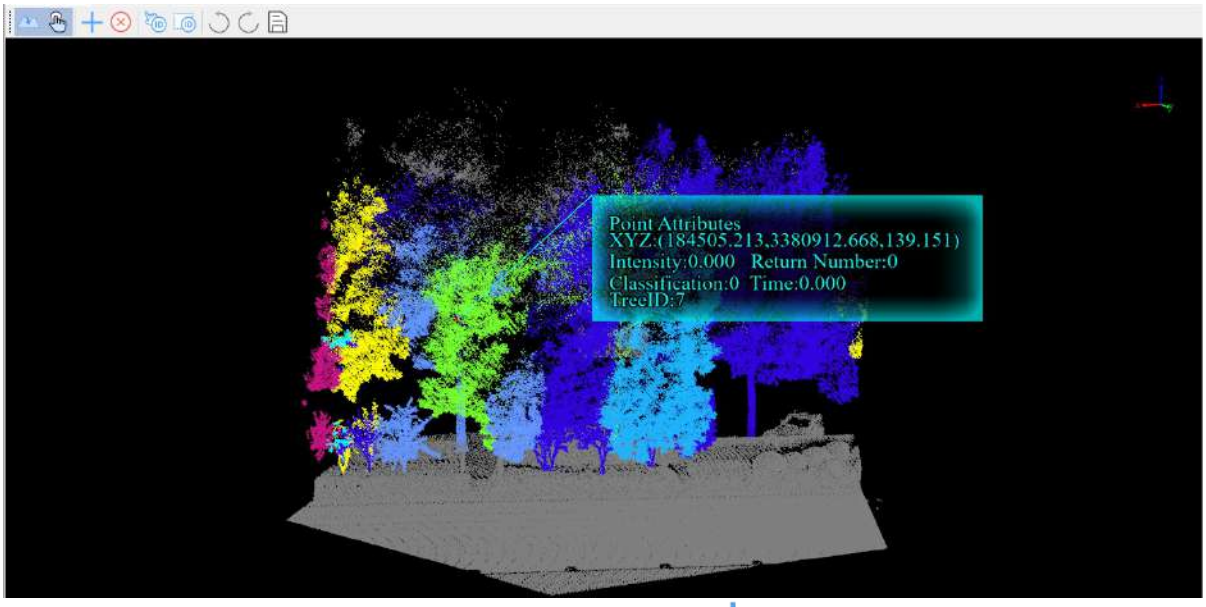
2.After starting editing, a single body editing window will pop up automatically, in which users can add, delete and reduce single body.

3.In the 3D window, click the left mouse button to draw a hexagonal area to be edited, at this time, the points in the area will be displayed in the single edit window.

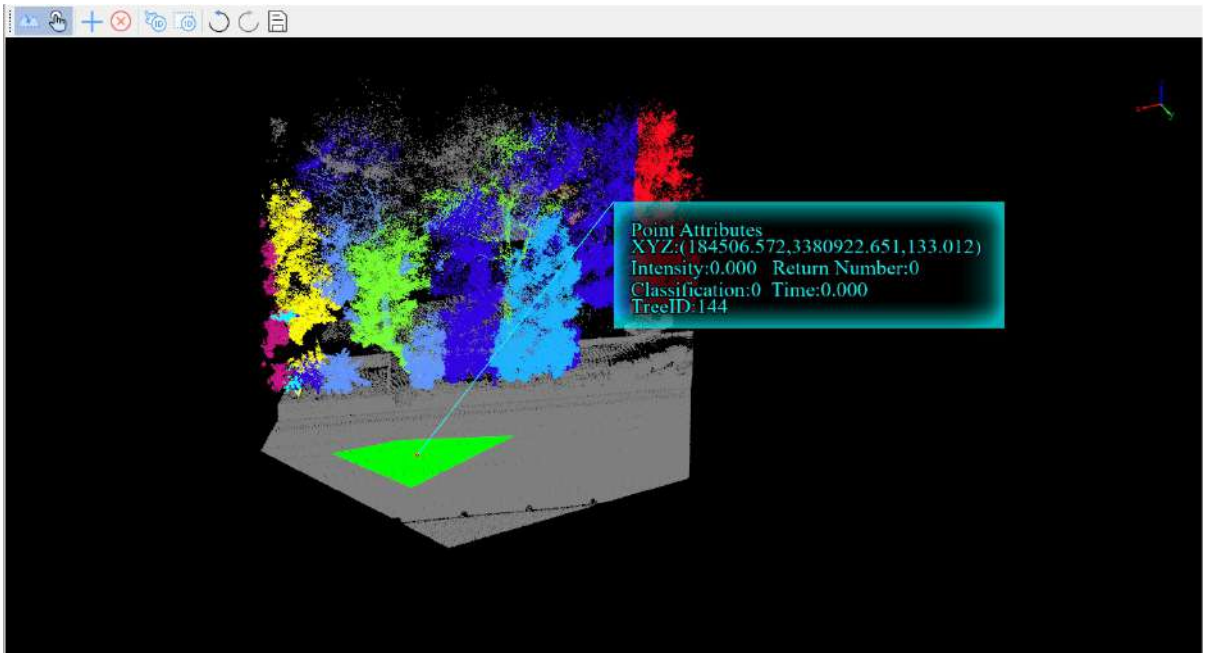


4.**Load Edit Area**: In the Single Edit window, Click **Load Edit Area** button. Add the point cloud of the area to be edited into the memory for editing. Editing is not supported if it is not loaded.

5.**Pick Object**: In the Single Edit window, Click **Pick Object** button. Users can view the property information of the selected point, including 3D coordinates, GPS time, intensity, RGB, object ID and other information.



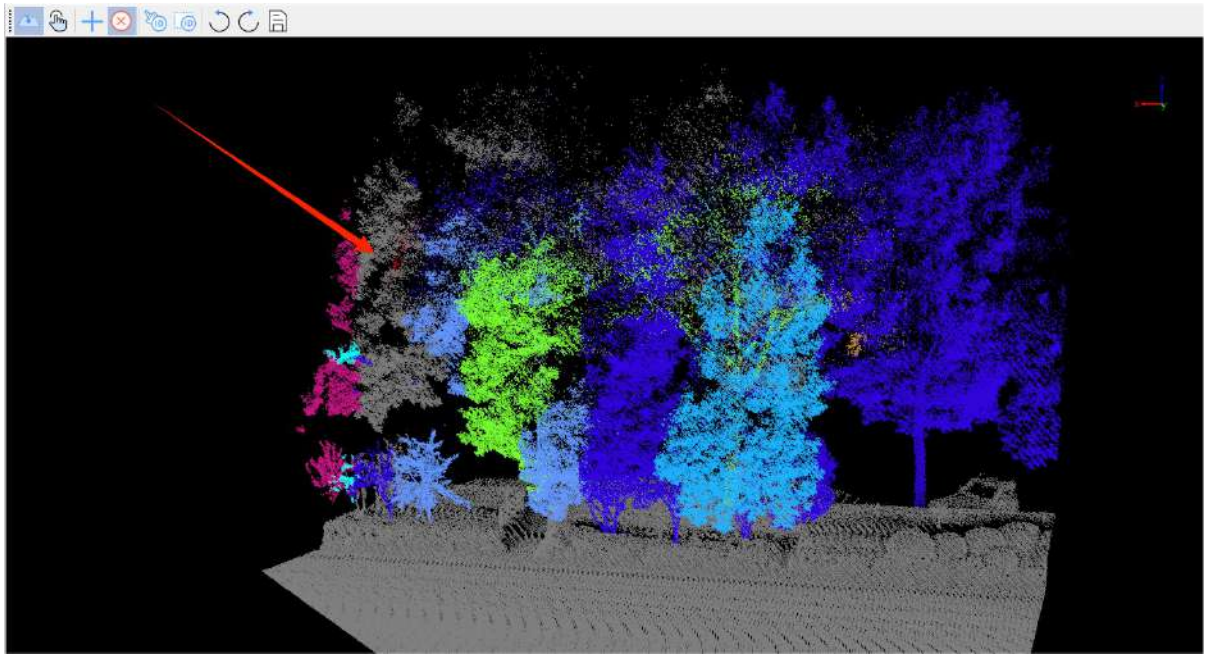
6.**Create Object**：In the Single Edit window, Click **Create Object** button. By boxing the point cloud of the single object is located to complete the creation of a single object, the point object ID in the boxed area will be set to the new object ID, and an object record is automatically added to the property table. To fine-tune the point cloud, users can use the [Point Set ID] and [Box Set ID] operations.



Newly built monoliths

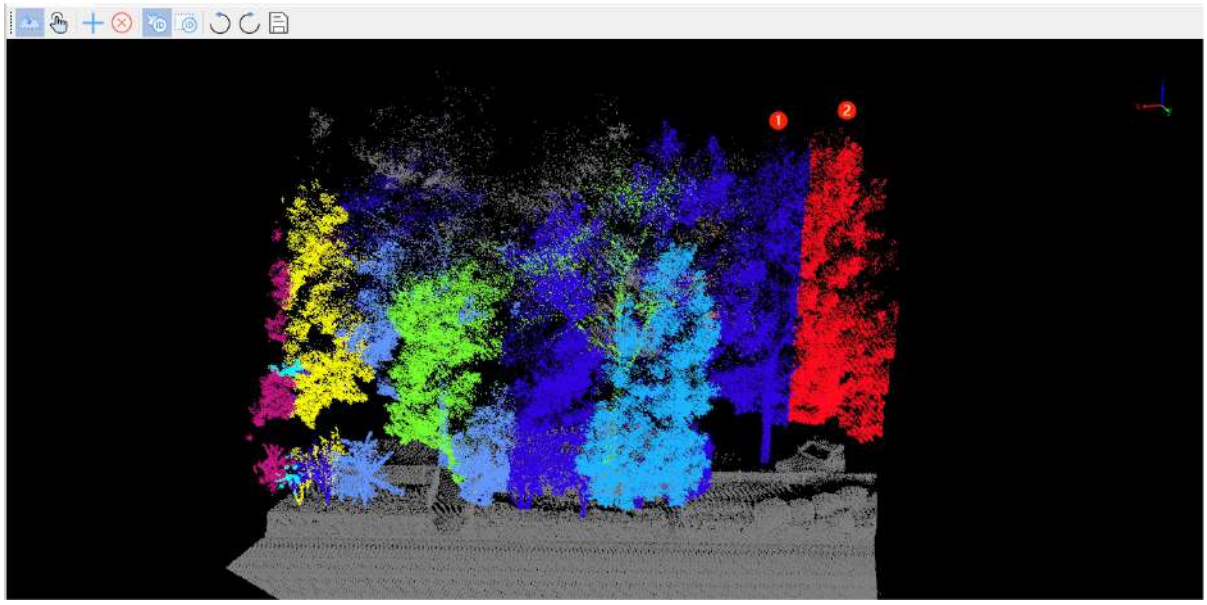
7.**Settings**：In the Single Edit window, Click **Settings** button.In the setting window, users can check the box to display only the points to be edited from two dimensions, category and single ID, to reduce the interference of the background point cloud to the editing process and improve the convenience of operation.

8.**Detele Object**：In the Single Edit window, Click **Detele Object** button. When a single object is deleted by point-and-click, the point cloud object ID of the single object in the edit area is set to zero, and the object record in the property sheet is deleted.

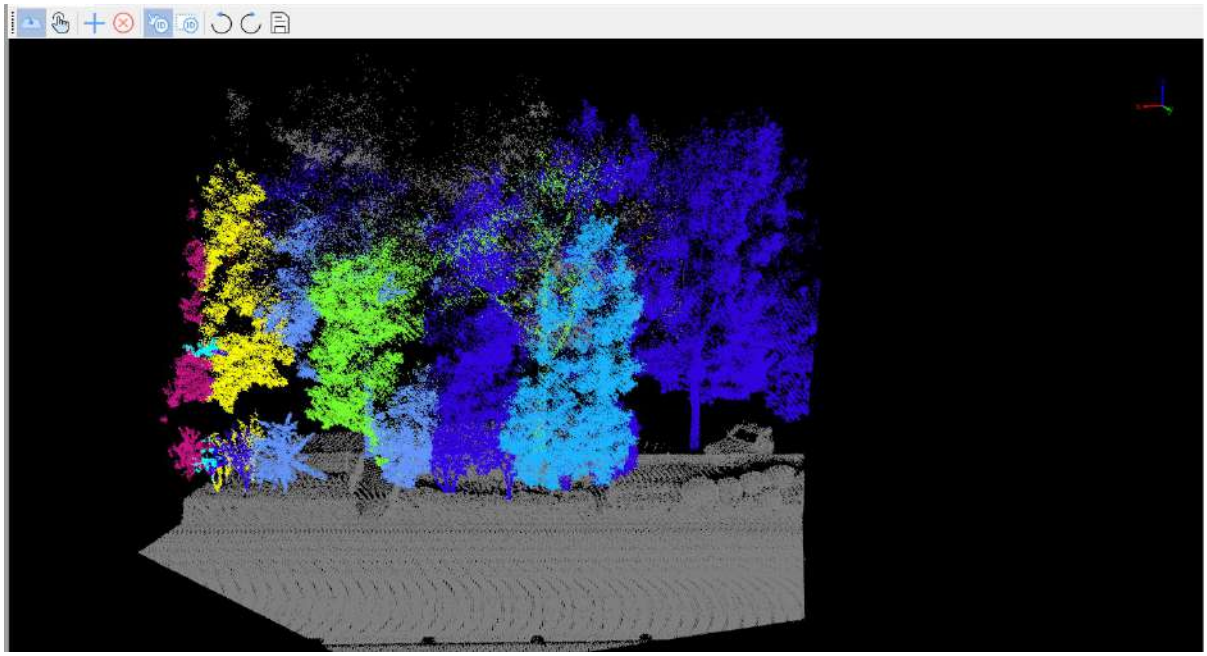


Delete Object

9.**Set ID By Pick**：In the Single Edit window, Click **Set ID By Pick** button. Click to select two single points, the first click of the single object ID will be set to the second click of the single object ID. It can be used to merge two single point clouds in the editing area ,and delete a single point cloud.

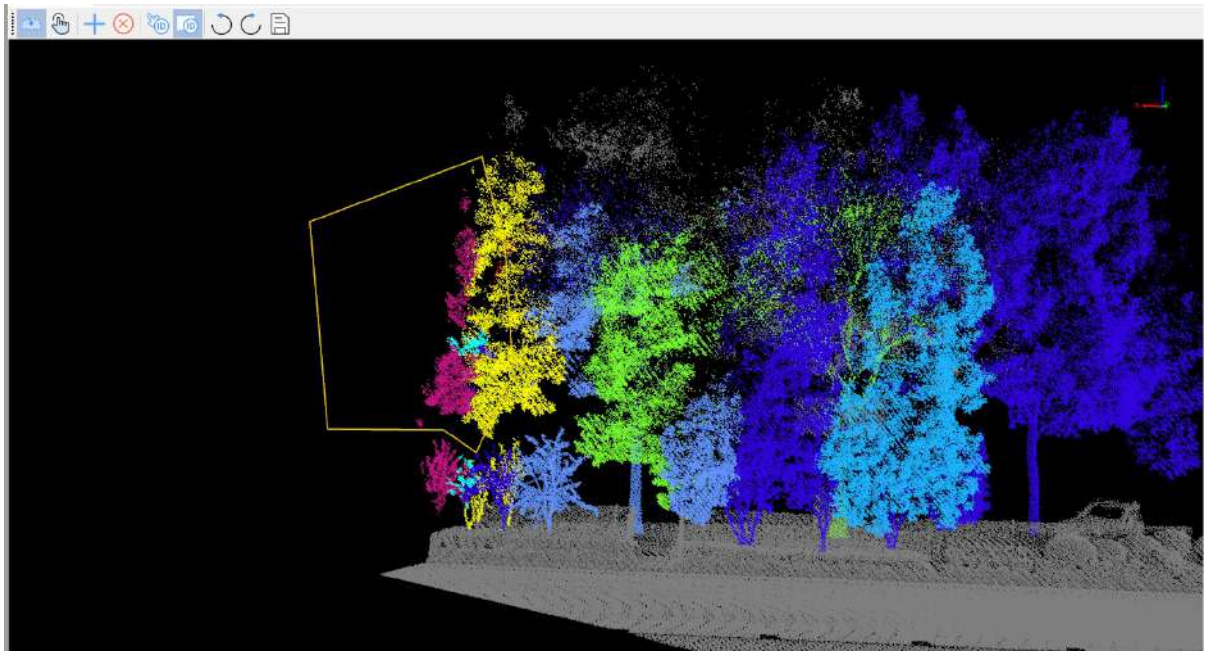


Before tapping Set ID

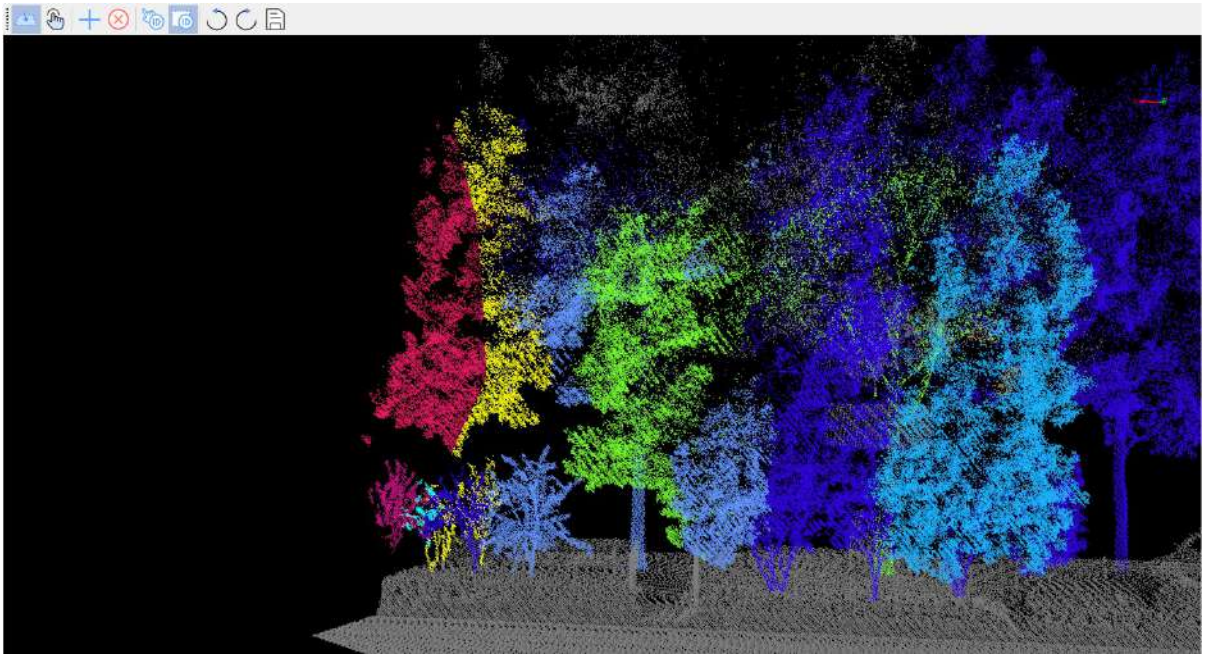


After tapping Set ID

10.**Set ID By Polygon**：In the Single Edit window, Click **Set ID By Polygon** button. First box a part of the point, and then click a single (or non-single point), the box point object ID will be set to click the single object ID. can be used to achieve single point refinement, box single point deletion.



Before setting ID in the Polygon



After checking the Set ID Polygon

11.**Undo**: In the Single Edit window, Click **Undo** button. Restore to the previous step, with support for the "Ctrl+Z" shortcut keys.

12.**Redo**: In the Single Edit window, Click **Redo** button. Resume to the next step, with support for the "Ctrl+Y" shortcut keys.

13.**Save**: In the Single Edit window, Click **Save** button. Save the edit to the file, the left 3D view will refresh the result, with support for the "Ctrl+Shift+S" shortcut keys.

Vertical Feature Parameter

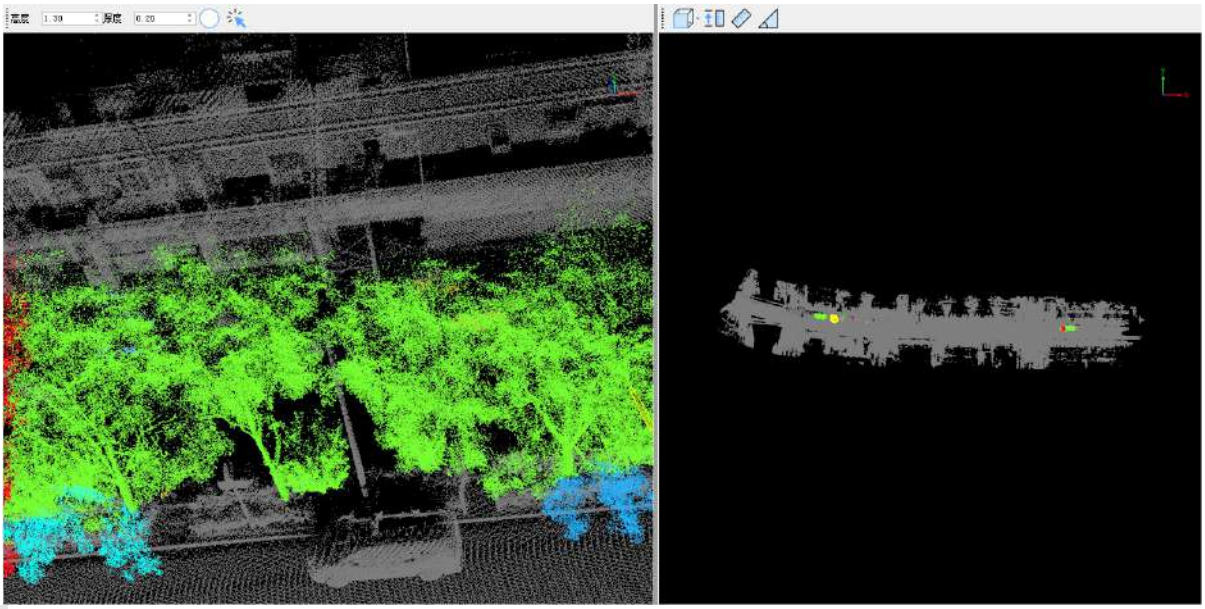
Function Description: This tool enables the modification of the vertical object parameters of chest diameter, height, angle, and crown width.

Steps

1.Select one or more rows in the parameter extraction table (hold down ctrl to select multiple rows), click the **Calculate** button to start calculating the parameters, and the calculation results of the parameters are displayed in the table in real time, as shown in the figure below：

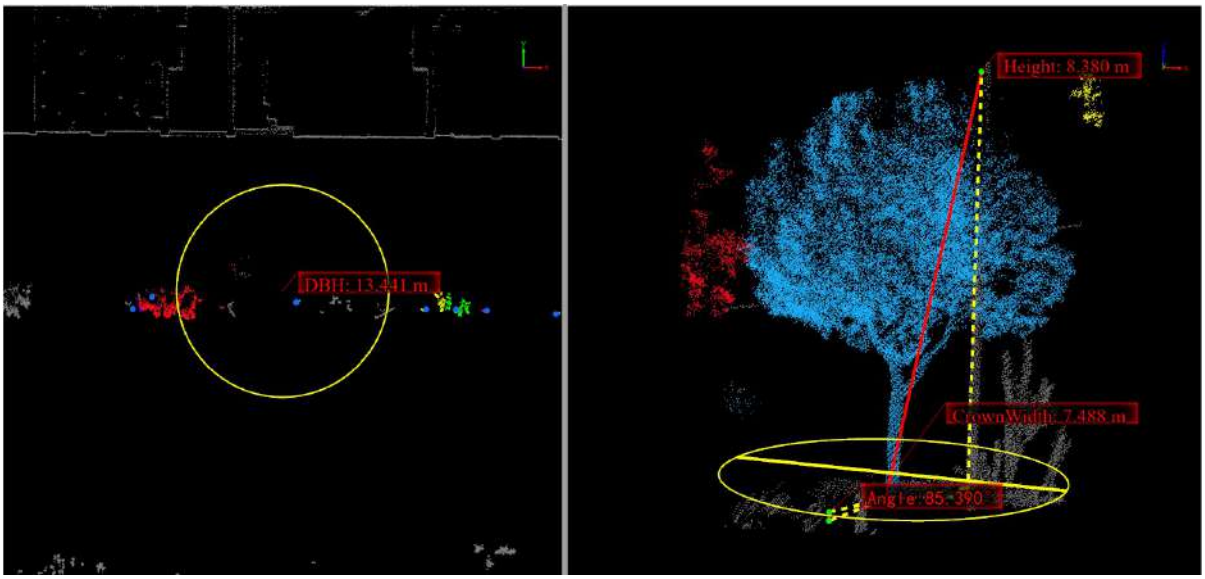
Note: Clicking on the white box in the upper left corner is the full menu body ID.

2.Click **Vertical Feature Parameter**  button, enter the function interface：



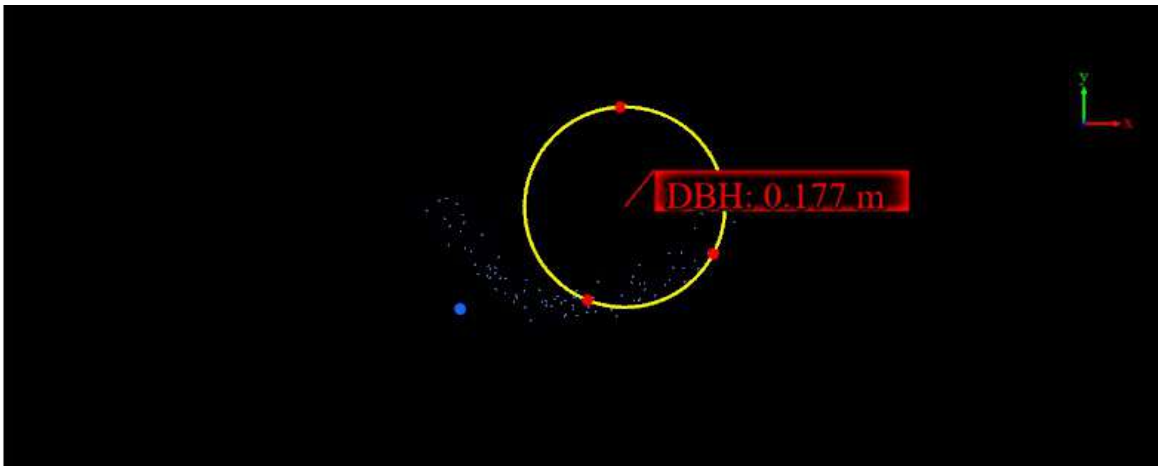
Note: You must first complete the monomer segmentation and editing operations.

3.After the parameters are calculated, select a row in the table, the 3Dview view on the left will show the DBH parameters, and the Object View window on the right will show the height, crown width, angle and other parameters. Among them, the left view toolbar has two parameters, slice height and slice thickness, indicating the points involved in the calculation of diameter at breast height, which are 1.3m and 0.2m by default.

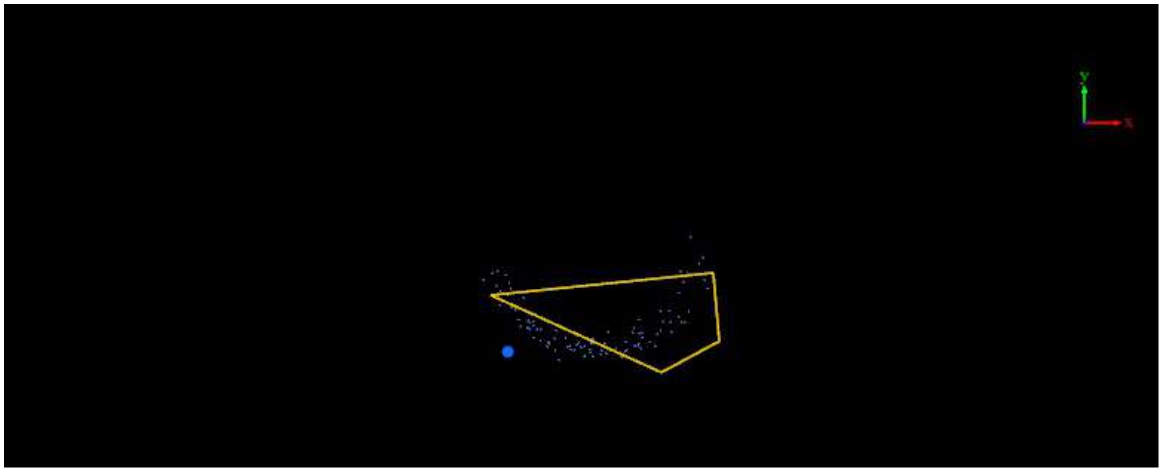


4.**DBH three-point fit**: There are two ways to refit the DBH:

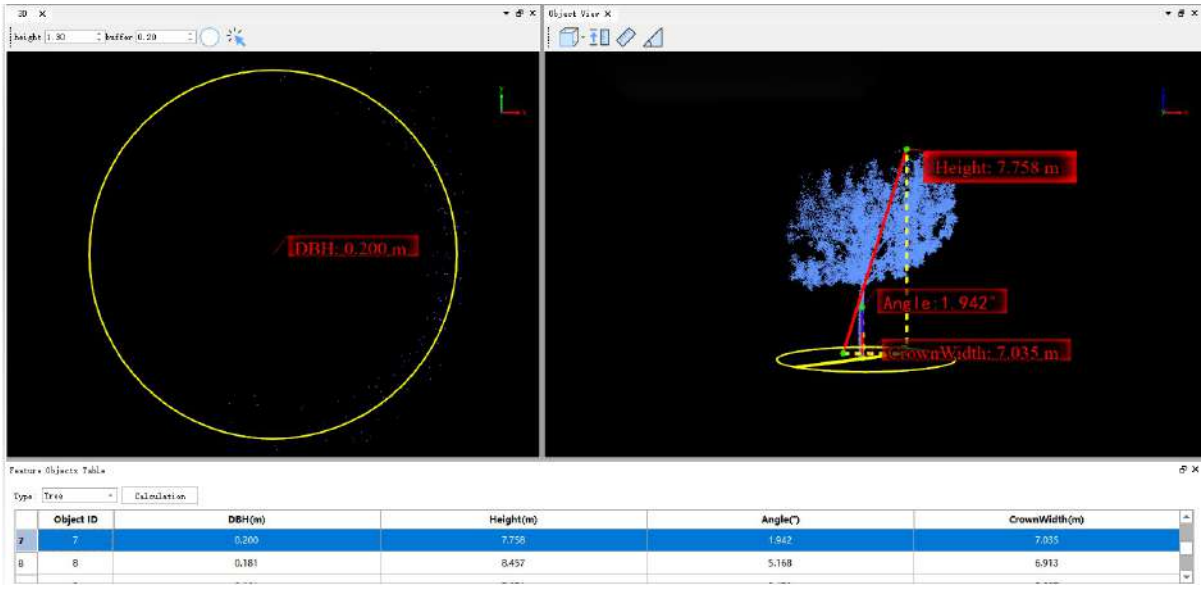
- **3DView on the left**: Display DBH parameters: First click on the monomer ID the user wants to view, and then click **DBH three-point fit** button, click three points with the left mouse button to determine the fitting surface the users wants to view, as shown in the figure:



- **DBH box to select**:Also after switching the monomer, click on the **DBH box in the 3D window to select the fit** button. Box the point cloud that needs to be refitted with DBH, as shown in the figure.

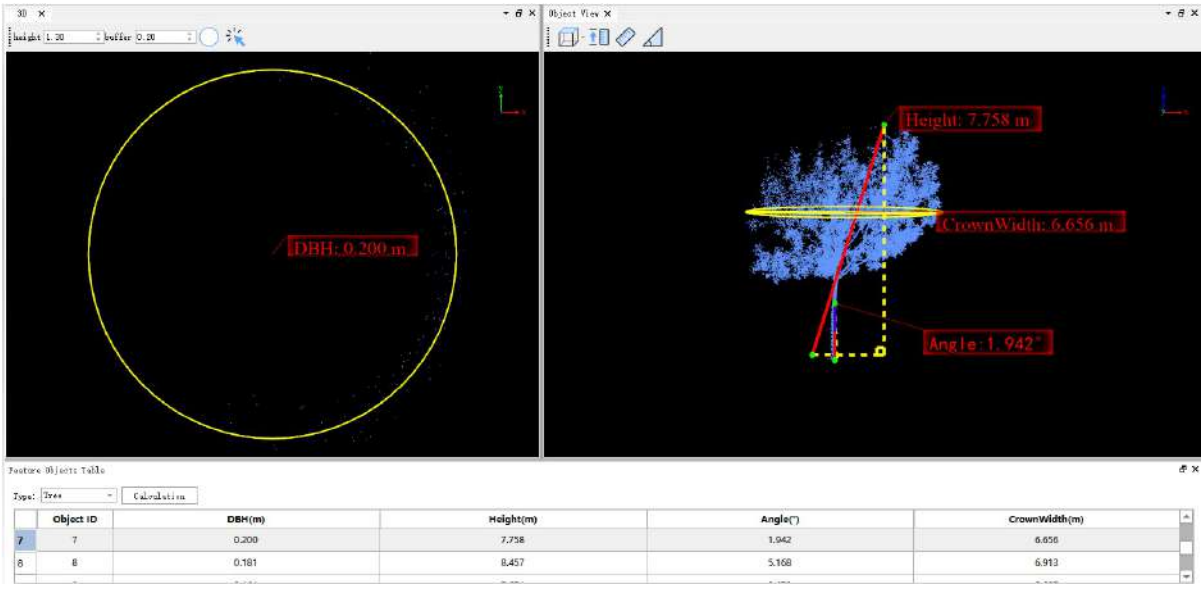


5.**Height measurement**: In the Object View window, click **Height Measurement** button, Using a two-point approach, the height of the monolith is re-measured on the point cloud and the new measurements are updated in real time in the corresponding list of monolith tables.



Value after height re-measurement

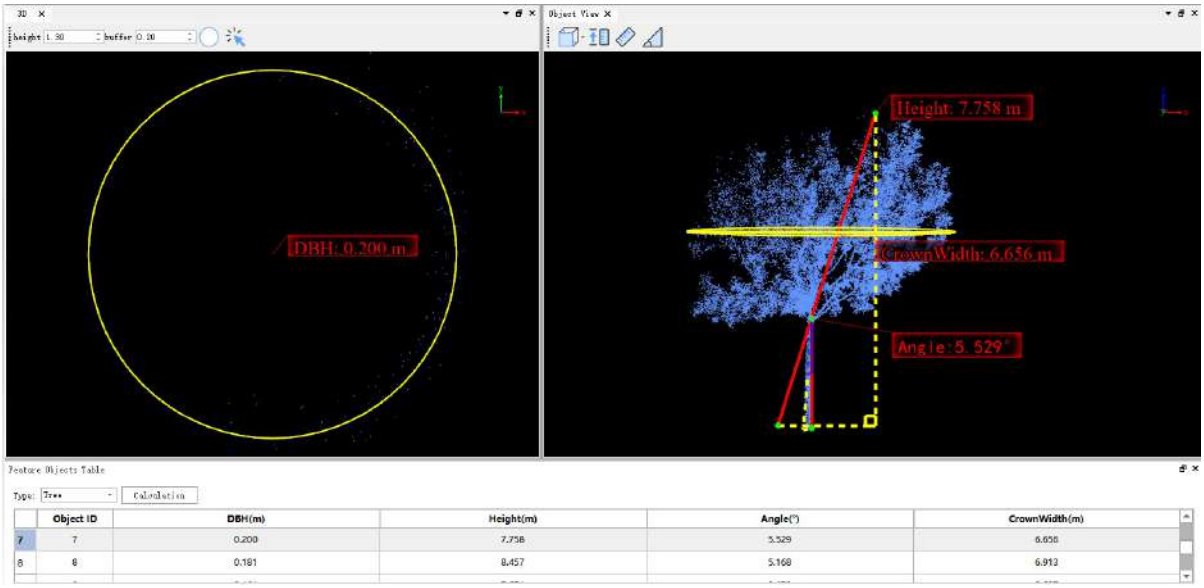
6.**Crown width measurement**: In the Object View window, click **Crown Measurement** button. Switching through views in **Top View** button, Switch the point cloud in the Object View window to the top view display, then using a two-point circle, re-measure the crown width and update the results of the crown width in the parameter table.



Value after re-measurement of crown width

7.**Angle measurement**:In the Object View window, use the switch button of the view(Top View 、Front View 、Back View 、Left View 、Right View)After switching the point cloud to the appropriate angle, click **Angle Measurement** button, Re-measure the angle and update the result of the angle in the

parameter table.



Value after re-measuring the angle

Vector Editor

Vector Editor allows the user to edit vector data through modules such as layer management, vector drawing, element editor, and edit vertices.

Draw Vector

Element Editor

Edit Vertices

Drawing

The function of the drawing module is mainly to add shape vectors such as **point**, **line**, **surface**, **circle**, and **arc**.

> Note: The geometric characteristics of the layer must be consistent with the geometric characteristics of the vector result. For example, the point-like vector result must be added to the point-like layer, otherwise it will be prompted that it cannot be drawn. See the specific Layer Management details.

Drawing Elements

- Add Point
- Add Multi-point
- Add Line
- Add Line Segment
- Add Arc
- Add Polygon
- Add Rectangle
- Add Parallelogram
- Addw Circle
- Add Hexagon
- Add Octagon

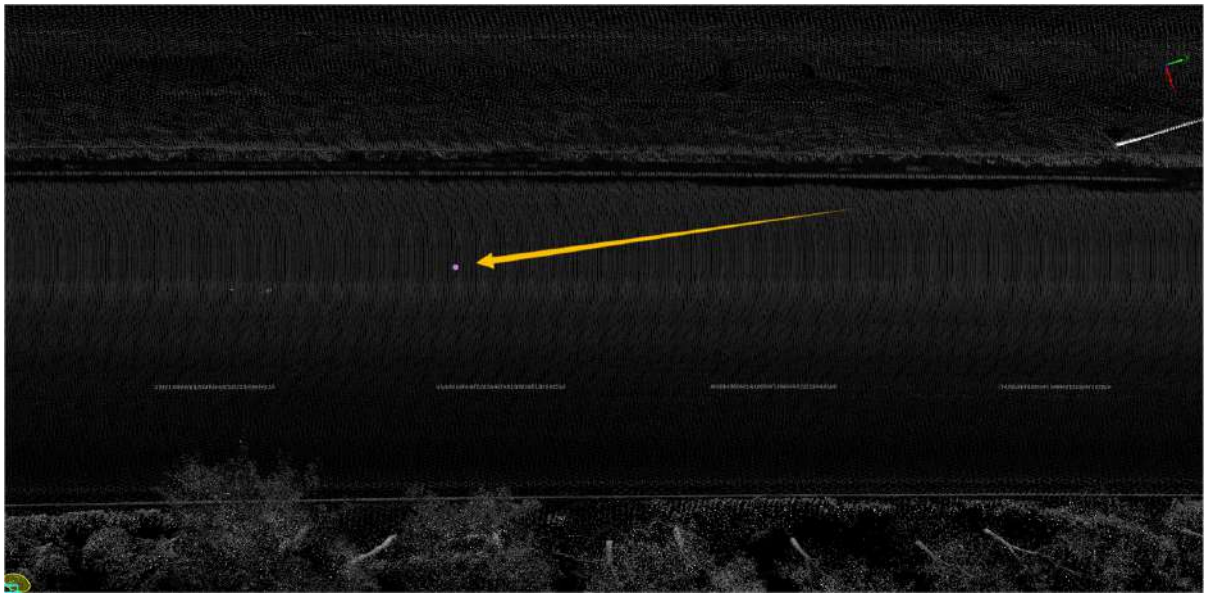
Add Point

Function Description： This tool allows the user to add vector points to objects.

Steps

1.Select a point layer, such as a surveillance camera point layer.

2.Click **Add Point** button in the Vector Editor, use the mouse to select a point in the point cloud, and a dot will be drawn at that position according to the current layer.



Draw point interface

> Note: The geometric characteristics of the vector result must be consistent with the geometric characteristics of the layer to be added. For example, the point-like vector result must be added to the point-like layer, otherwise it will be prompted that it cannot be drawn. For the description of layers, see Layer Management

Add Multi-point

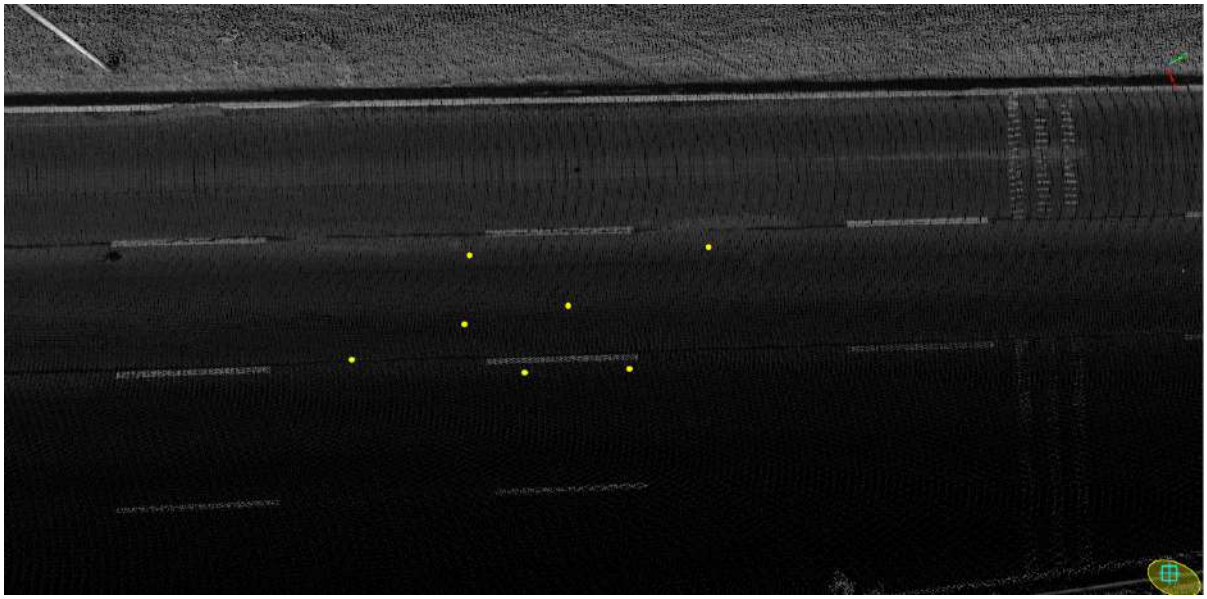
Function Description： This tool allows the user to add multiple vector points to objects.

Steps

1.Select a point layer, such as a surveillance camera point layer.

2.Click **Add Multi-point** button in the Vector Editor, use the mouse to continuously select multiple points in the point cloud, and quickly double-click the mouse to end the drawing.

> Note: After drawing, multiple points are stored as an object.



Draw multiple point interface

> Note: The geometric characteristics of the vector result must be consistent with the geometric characteristics of the layer to be added. For example, the point-like vector result must be added to the point-like layer, otherwise it will be prompted that it cannot be drawn. For the description of layers, see Layer Management

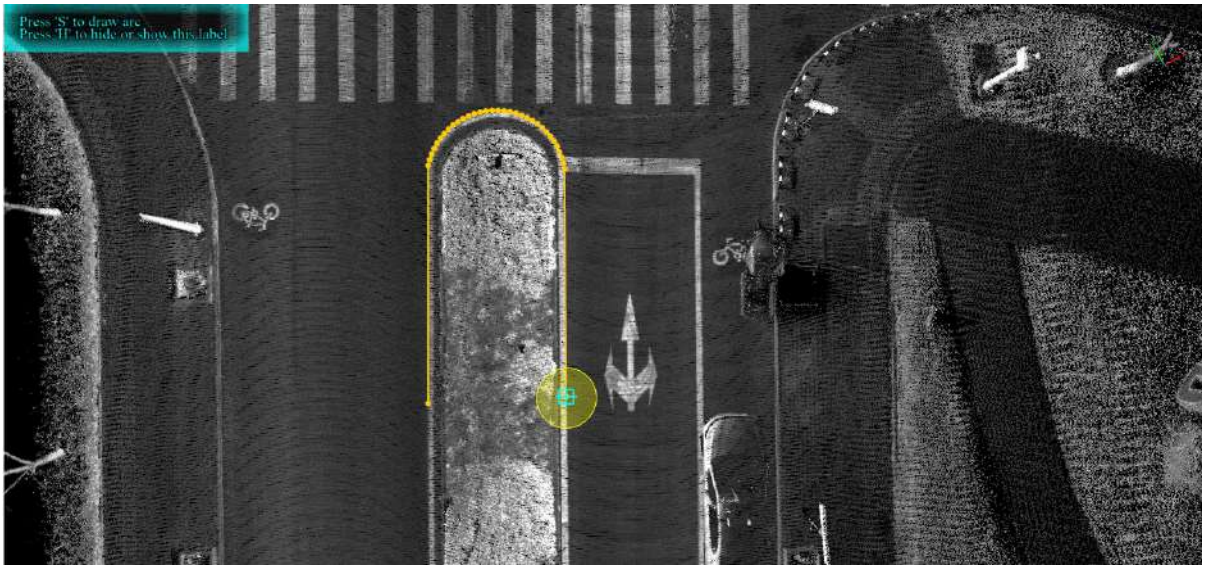
Add Line

Function Description： This tool allows the user to add vector lines to objects.

Steps

1.Select a linear layer, such as a lane line layer.

2.Click **Add Line**  button in the Vector Editor. Use the mouse to select the starting point of the line in the point cloud, click the left mouse button continuously to confirm multiple nodes of the line, and double-click the left mouse button to end the drawing of the current line.



Draw line interface

> Note:
>
> - The geometric characteristics of the vector result must be consistent with the geometric characteristics of the layer to be added. For example, the point-like vector result must be added to the point-like layer, otherwise it will be prompted that it cannot be drawn. For the description of layers, see Layer Management.
> - During the drawing process, the right mouse button and the shortcut key are operated in the same map elements as the Road Lane line.

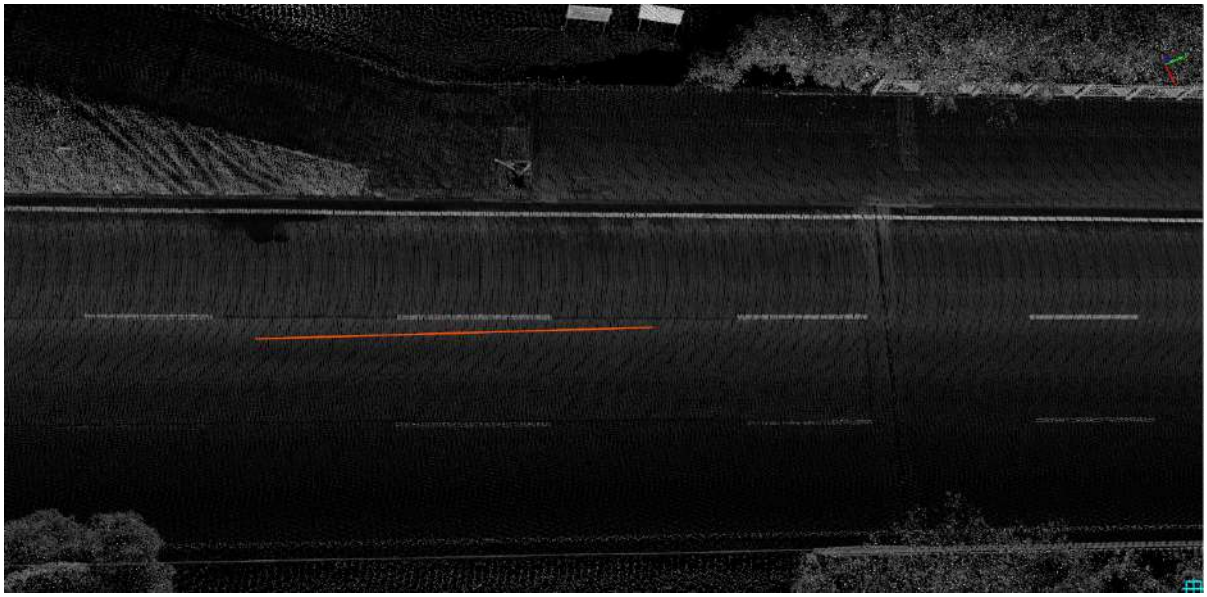
Add Line Segment

Function Description： This tool allows the user to add a vector line segments to objects.

Steps

1.Select a linear layer, such as a lane line layer.

2.Click **Add Line Segment**  button in the Vector Editor. Use the mouse to select the starting point of the selection in the point cloud, move the mouse, and click the left mouse button again to confirm the other end of the line.



Draw Line segment interface

> Note: The geometric characteristics of the vector result must be consistent with the geometric characteristics of the layer to be added. For example, the point-like vector result must be added to the point-like layer, otherwise it will be prompted that it cannot be drawn. For the description of layers, see Layer Management

Add Arc

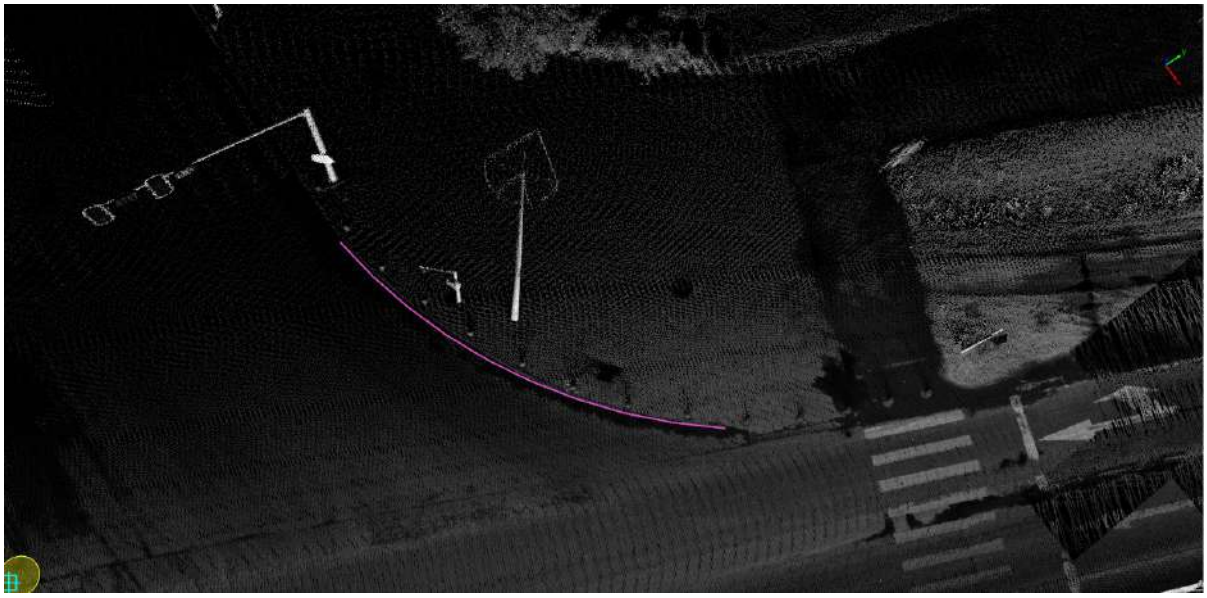
Function Description： This tool allows the user to use the three key point method to add arc-shaped vector objects.

Steps

1.Select a surface layer, such as a lane centerline layer.

2.Click **Add Arc**  button in the Vector Editor:

- Left click the point cloud to select the first key point.
- Left click the point cloud to select the second key point.
- After clicking two key points, an arc-shaped real-time display box will appear. Left-click on the point cloud and select the third key point to determine an arc-shaped vector object.



Draw arc interface

> Note: The geometric characteristics of the vector result must be consistent with the geometric characteristics of the layer to be added. For example, the point-like vector result must be added to the point-like layer, otherwise it will be prompted that it cannot be drawn. For the description of layers, see Layer Management.

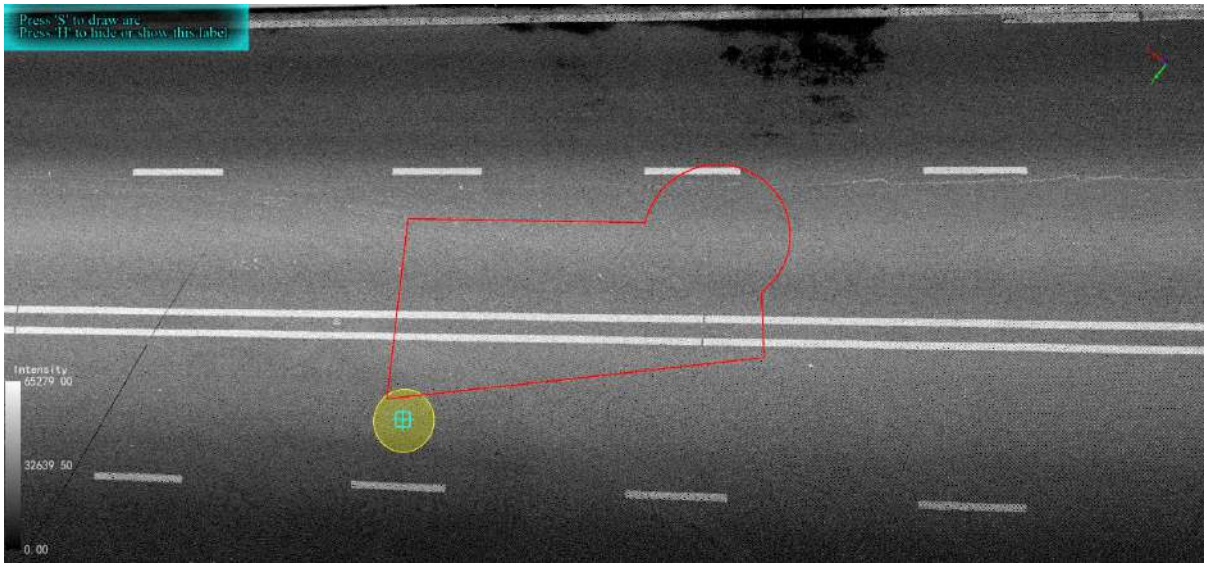
Add Polygon

Function Description： This tool allows the user to add vector polygons to objects.

Steps

1.Select a surface layer, such as a surface facility layer.

2.After click **Add Polygon**  button in the Vector Editor, use the mouse to click multiple points in succession to confirm the nodes of the polygon, The shortcut key B can be used to roll back the drawn node, and the keyboard key s can be used to switch the mode of drawing the arc in the polygon. The current point is the first point of the arc, the second point is the end point of the arc, and the third point determines the radian of the arc, which can complete the arc drawing. During the drawing process, the line and arc drawing can be switched several times, and the last point can be double-clicked to complete the drawing.



Draw polygon interface

> Note:
>
> - The geometric characteristics of the vector result must be consistent with the geometric characteristics of the layer to be added. For example, the point-like vector result must be added to the point-like layer, otherwise it will be prompted that it cannot be drawn. For the description of layers, see Layer Management.
> - The right mouse button and shortcut key are the same as those of map elements during drawing，see Road Lane line.

Add Rectangle

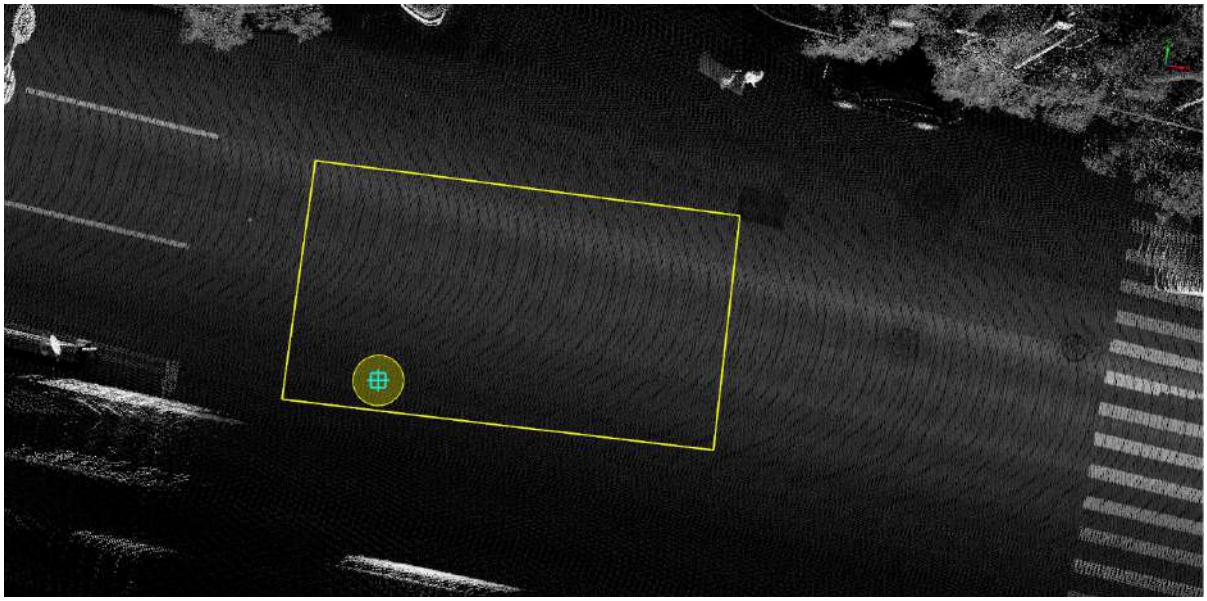
Function Description： This tool allows the user to use the three key point method to add vector rectangular objects.

Steps

1.Select a surface layer, such as a surface facility layer.

2.Click **Add Rectangle**  button in the Vector Editor:

- Left click the point cloud to select the first key point as the starting point of the rectangle.
- Left click the point cloud to select the second key point, and determine the width of the rectangle.
- Left click the point cloud to select the third key point, determine the length of the rectangle, and complete the drawing of the rectangle object.



Draw rectangle interface

> Note: The geometric characteristics of the vector result must be consistent with the geometric characteristics of the layer to be added. For example, the point-like vector result must be added to the point-like layer, otherwise it will be prompted that it cannot be drawn. For the description of layers, see Layer Management

Add Parallelogram

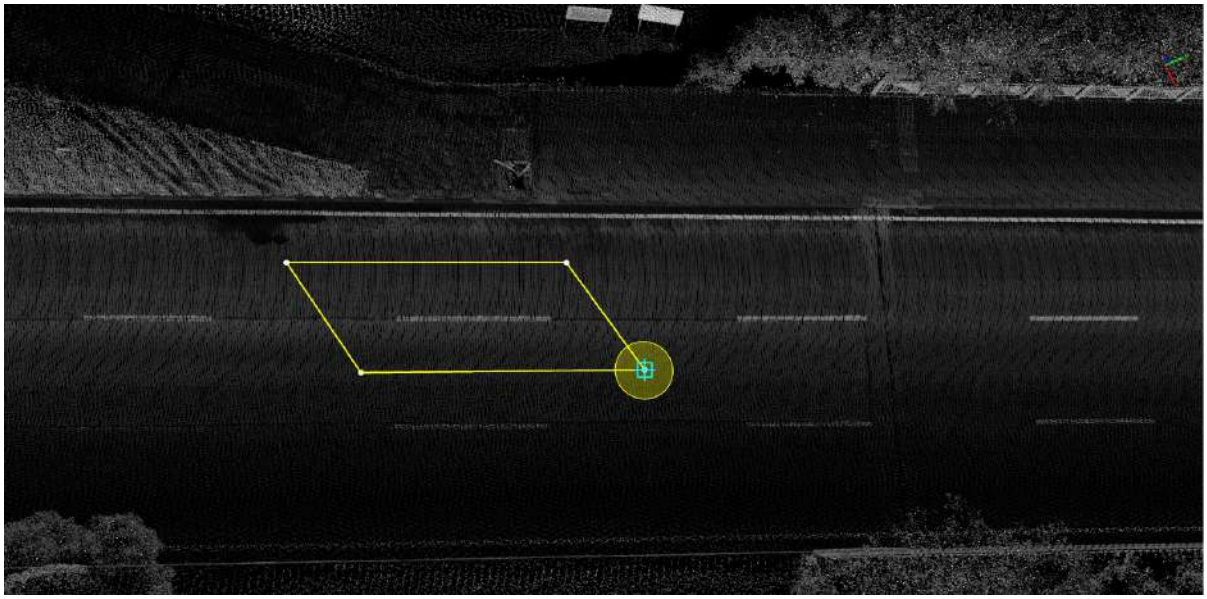
Function Description： This tool allows the user to use the three key point method to add a parallelogram vector objects.

Steps

1.Select a surface layer, such as a surface facility layer.

2.Click **Add Parallelogram**  button in the Vector Editor:

- Left click the point cloud to select the first key point as the starting point of the parallelogram.
- Left click the point cloud to select the second key point, and determine one side of the parallelogram.
- Left click the point cloud, select the third key point, confirm the other side of the parallelogram, and then complete the parallelogram object drawing.



Draw parallelogram interface

> Note: The geometric characteristics of the vector result must be consistent with the geometric characteristics of the layer to be added. For example, the point-like vector result must be added to the point-like layer, otherwise it will be prompted that it cannot be drawn. For the description of layers, see Layer Management

Add Circle

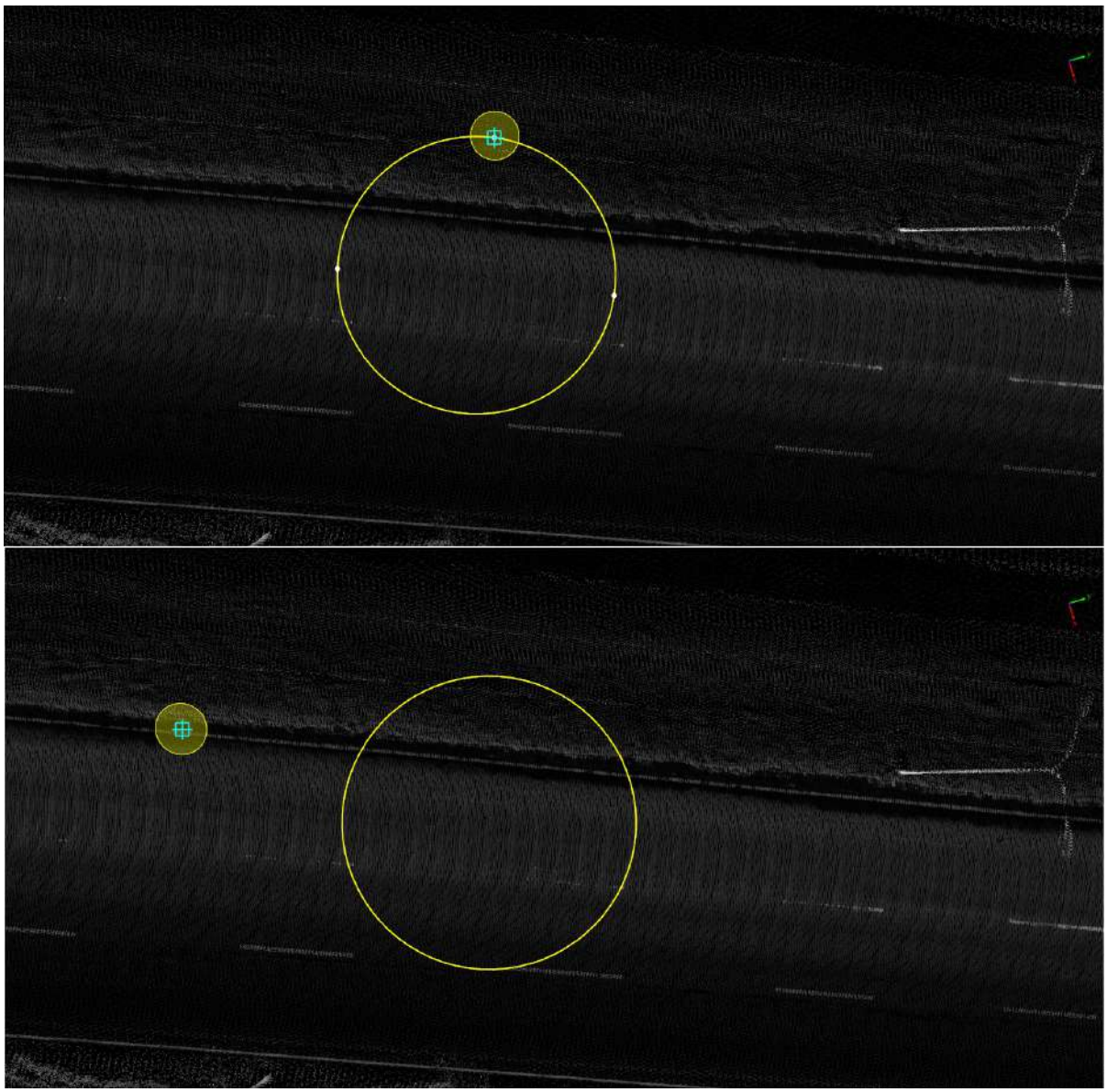
Function Description： This tool allows the user to use the three key point method to add circle-shaped vector objects.

Steps

1.Select a surface layer, such as a surface facility layer.

2.Click **Add Circle**  button in the Vector Editor:

- Left click the point cloud to select the first key point.
- Left click the point cloud to select the second key point.
- After clicking two key points, an arc-shaped real-time display box will appear. Left click on the point cloud and select the third key point to determine a circle-shaped vector object.



Draw circle interface

> Note: The geometric characteristics of the vector result must be consistent with the geometric characteristics of the layer to be added. For example, the point-like vector result must be added to the point-like layer, otherwise it will be prompted that it cannot be drawn. For the description of layers, see Layer Management

Add Hexagon

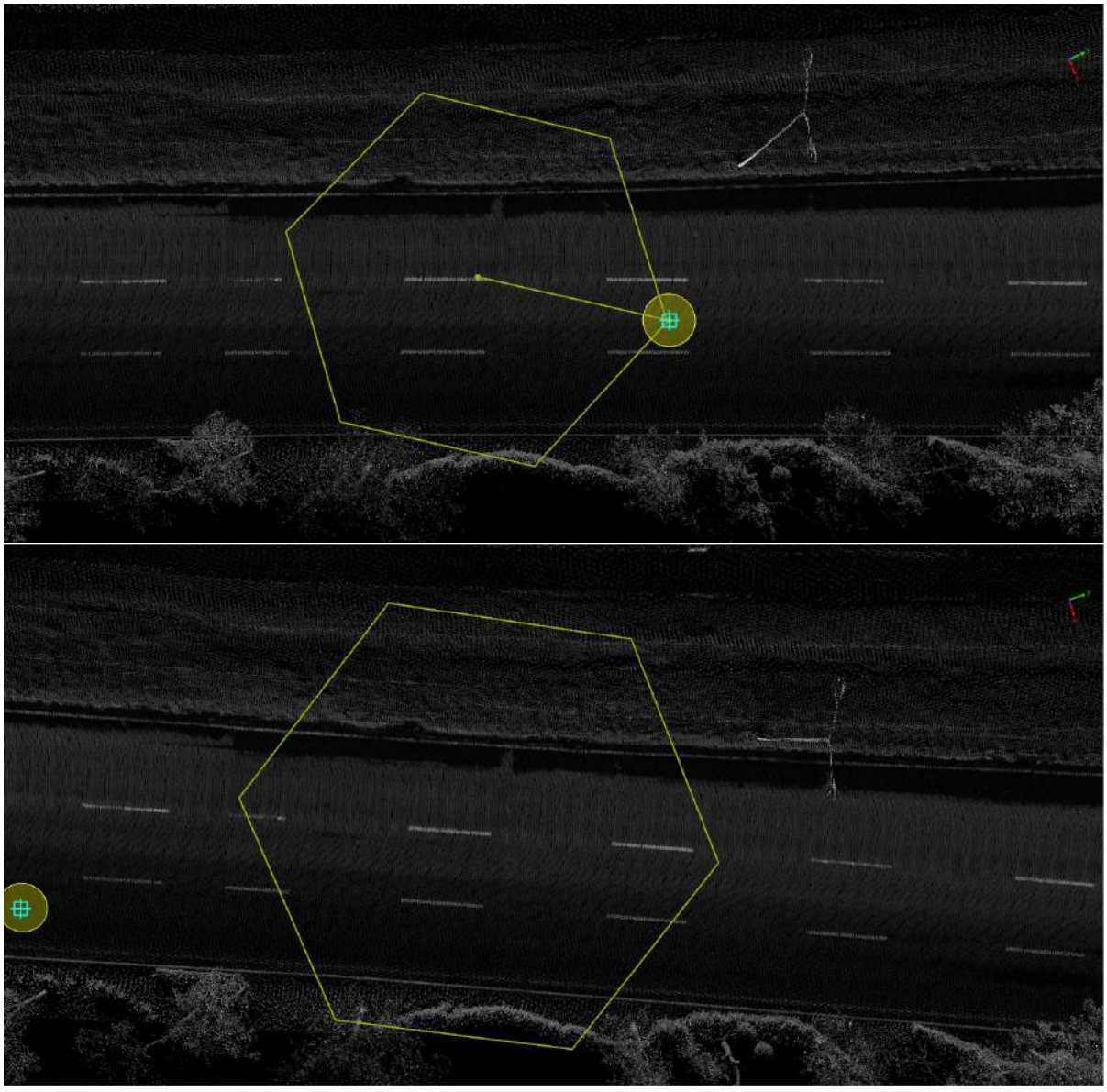
Function Description: This tool allows the user to add a hexagonal vector object by using the center point and dragging the mouse.

Steps

1.Select a surface layer, such as a surface facility layer

2.Click **Add Hexagon** button in the Vector Editor:

- Click the point cloud with the left mouse button, select the center point of the hexagon, and then drag the mouse, a real-time display box of the hexagon will appear. After determining the size and direction, click the point cloud with the left mouse button to determine a hexagon vector object.



Draw hexagonal interface

> Note: The geometric features of the vector result must be consistent with the geometric features of the layer to be added. For example, the point-like vector result must be added to the point-like layer, otherwise it will prompt that it cannot be drawn. For the description of layers, see Layer Management

Add octagon

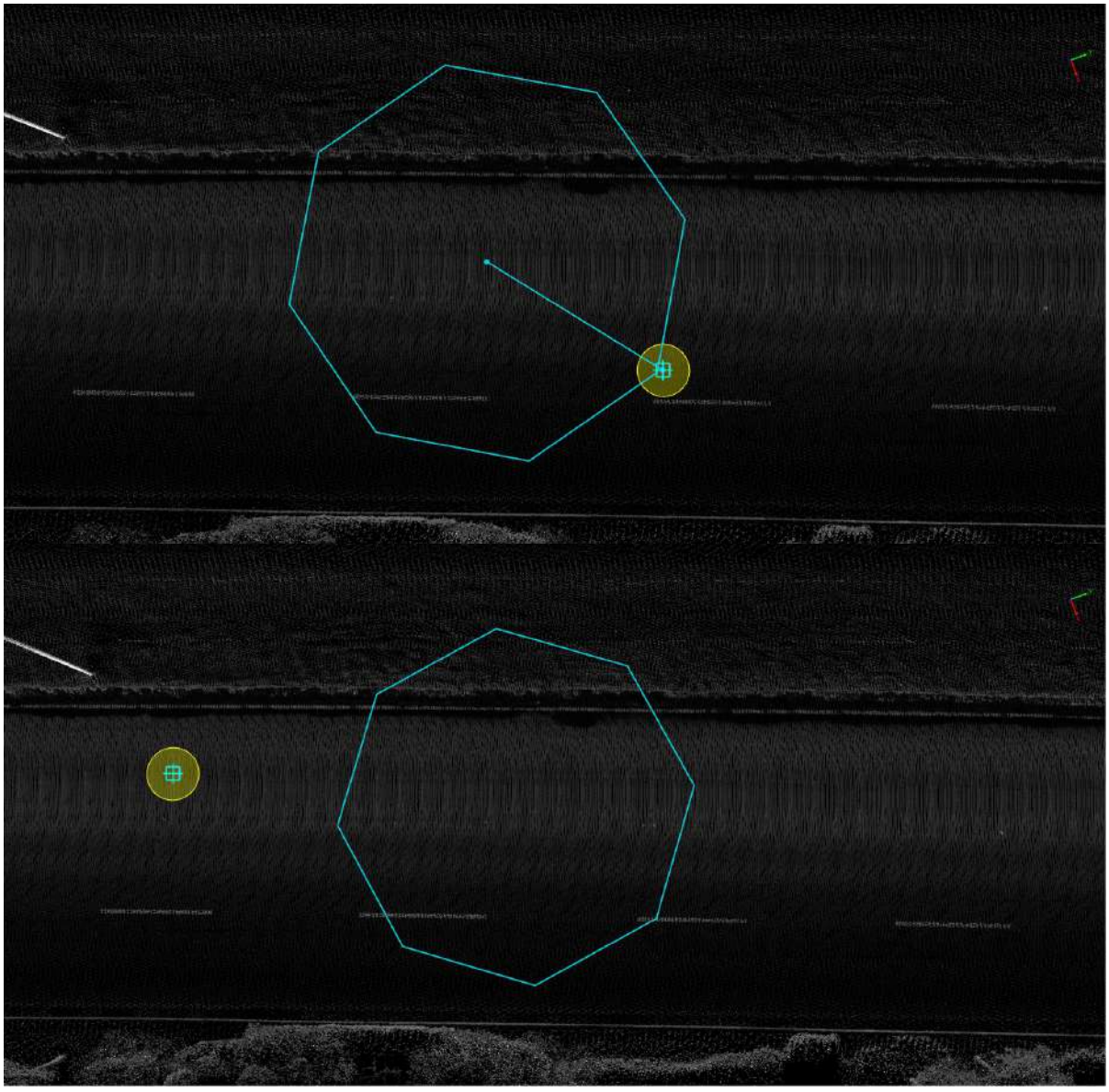
Function Description: This tool allows the user to add an octagon vector object by using the center point and dragging the mouse.

Steps

1.Select a surface layer, such as a surface facility layer.

2.Click **Add Octagon**  button in the Vector Editor:

- Click the point cloud with the left mouse button, select the center point, and then drag the mouse, an octagonal real-time display box will appear. After determining the size and direction, click the left mouse button on the point cloud to determine an octagonal vector object.



Drawing octagonal interface

> Note: The geometric features of the vector result must be consistent with the geometric features of the layer to be added. For example, the point-like vector result must be added to the point-like layer, otherwise it will prompt that it cannot be drawn. For the description of layers, see Layer Management

Element Editor

The Element Editor module mainly performs operations such as **moving**, **copying**, **deleting**, and **editing** on vector objects to improve the efficiency of user drawing. It mainly includes the following specific functions:

- Select
  - Pick
  - Rectangle
  - Line
  - Polygon
  - Lasso
  - Circle
  - Box
  - Sphere
  - Cylinder
- Interrupt
  - Point Split Line
  - Line Split Line
  - Line Split Polygon
  - Line Split Objects
- Alignment
  - Move
  - Parrallel Copy
  - Rotate
  - Scale Polygon
  - Mirror
- Reshape
  - Reshape Line
  - Reshape Rect
  - Merge Lines
  - Break Line
  - Split Line
  - Extend or Trim

    - Split or Merge Polygon
    - Link Line
    - Extend Line
    - Fillet
    - Split Mulitpolygon or Mulitpoint
    - Combine Polygons or Points
    - Smooth
- Auxiliary
    - Drape
    - Add Trace Line
    - Add Trace Polygon
    - Draw By Intersection
    - Draw By Perpendicular

Select

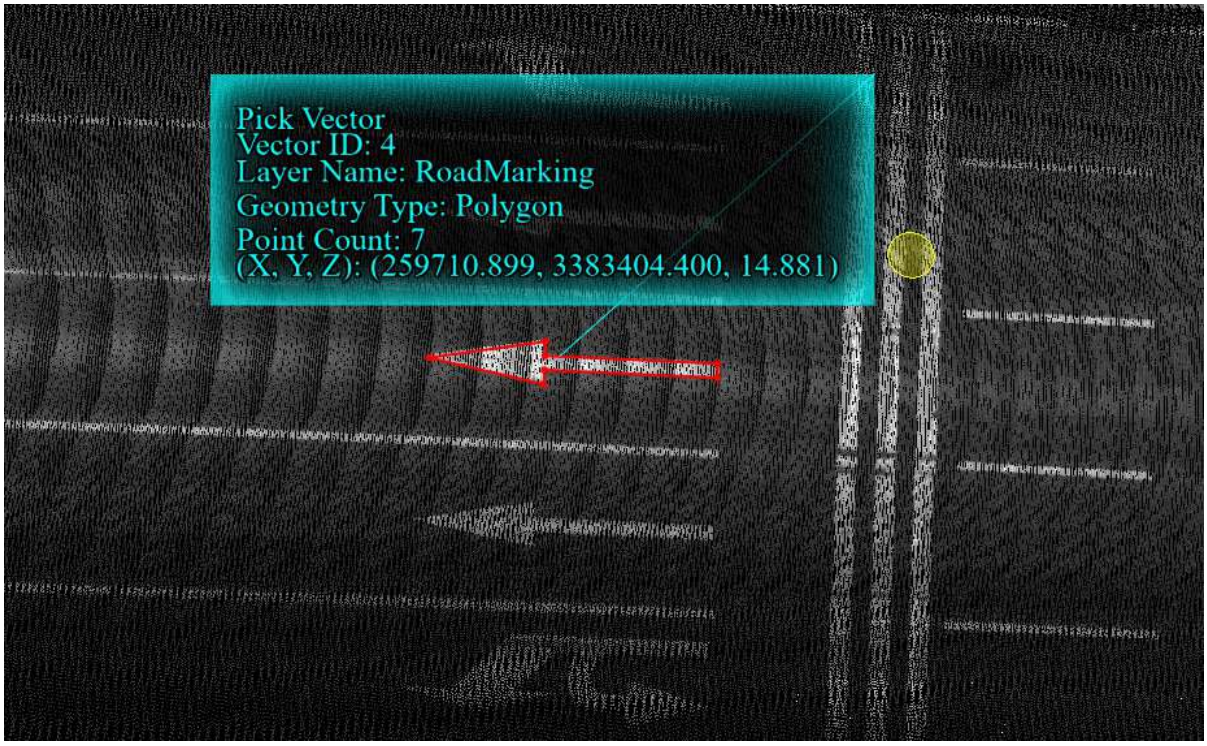
- Pick
- Rectangle
- Line
- Polygon
- Lasso
- Circle
- Box
- Sphere
- Cylinder

Pick

Function Description： The select vector function allows the user to select a vector object and obtain its detailed information.

Steps

1.After clicking the **Pick** button in the Vector Editor, move the mouse to the vector and click the left mouse button, the selected vector will be highlighted, and the main viewport will display the ID number of the current vector. Layer, geometry information and the attribute table window will locate to the row where the attribute field of the currently selected vector is located.



Select vector

2.After clicking, you can directly use the shortcut keys to copy, delete and other operations.

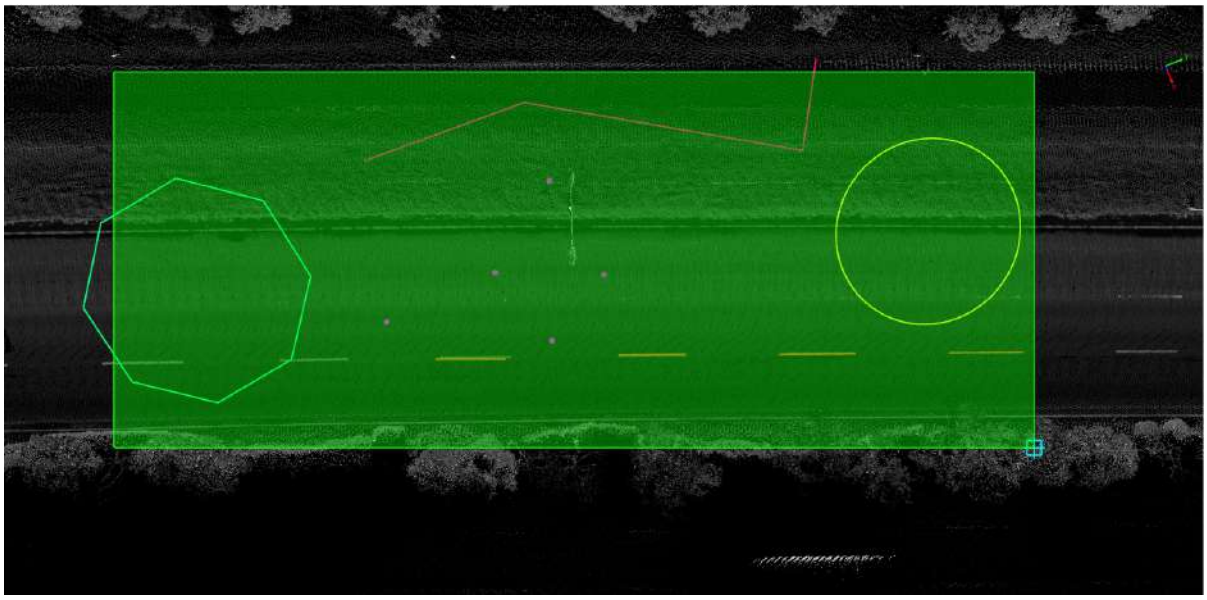
Shortcuts	Function
Ctrl+C	Copy
Ctrl+V	Copy to the same layer
Ctrl+Alt+V	Copy to any layer with the same geometry (pop-up layer is optional)
Delete	Delete

Rectangle

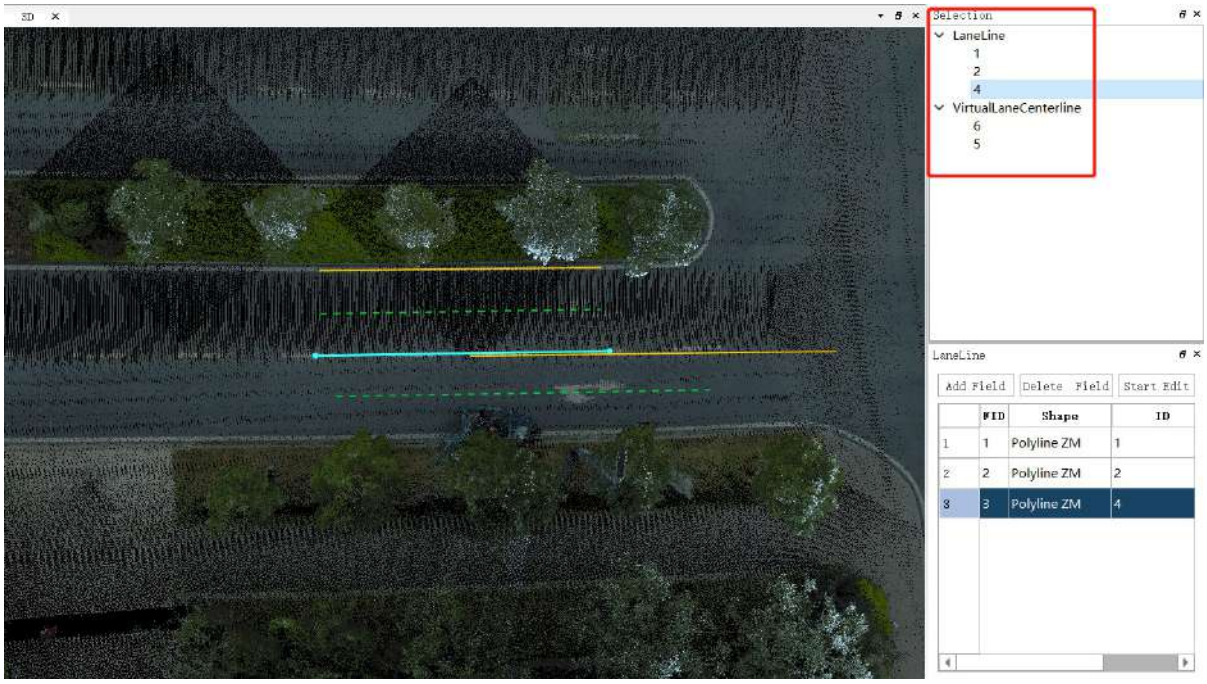
Function Description： The Rect Pickup Vector function allows users to select a series of vector objects by using a rectangle frame.

Steps

1.Click **Rectangle**  button in the Vector Editor, Left click for the first time, draw the upper left point of the frame selection rectangle, drag the mouse, draw the rectangle frame, left click again, the frame selection rectangle is drawn, and the polygon inside the frame and the polygon that intersects with the frame will be selected and highlighted show.



Section selection vector



Section selection vector - vector tree

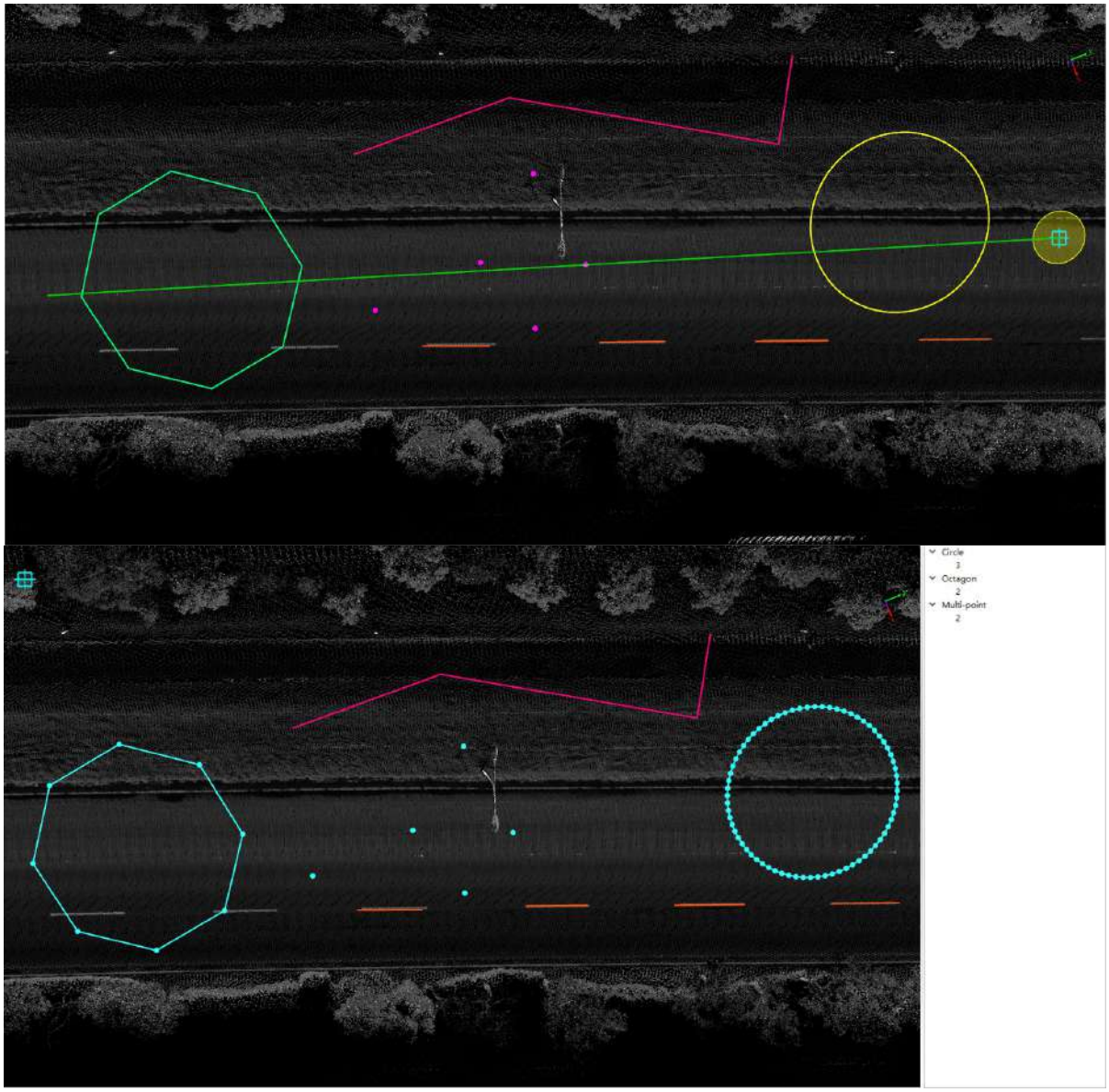
> Note: Select the shortcut key to copy, delete and other operations are the same as Pickup Vector.

Line

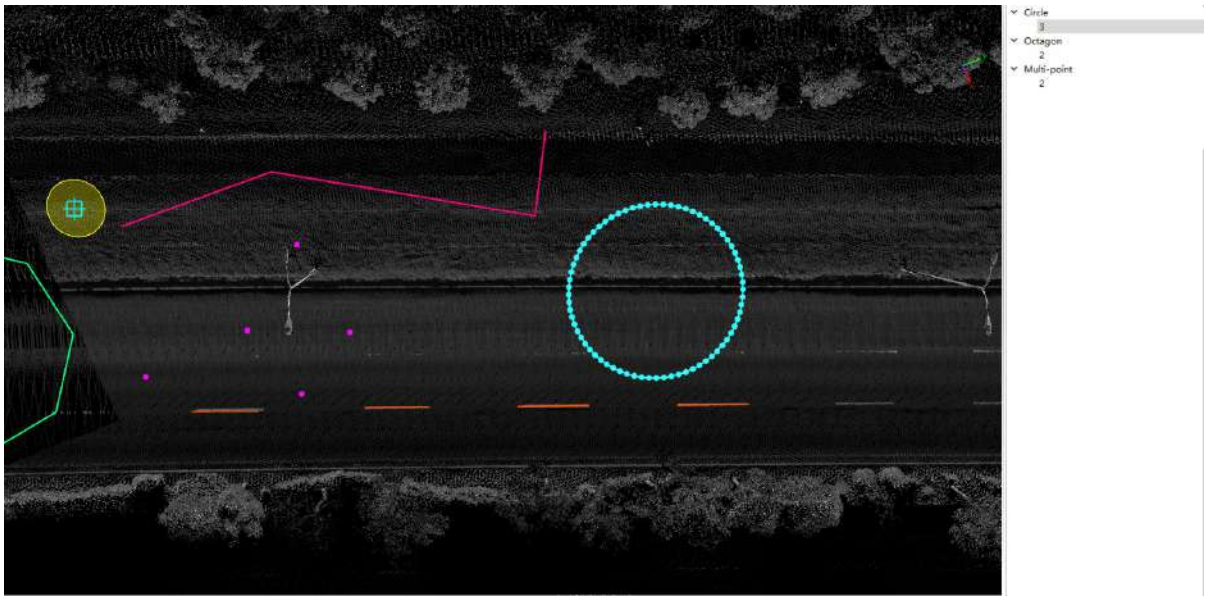
Function Description： The Pickup by Line function allows users to select multiple objects by drawing lines and operate them.

Steps

1.After clicking the **Line** button in the Vector Editor, click the left mouse button to draw a line segment, the vector objects intersecting with the line segment will be selected and highlighted, and the selected elements in the attribute table window will also be highlighted.



Line selection vector



Framed Vector - Vector Tree

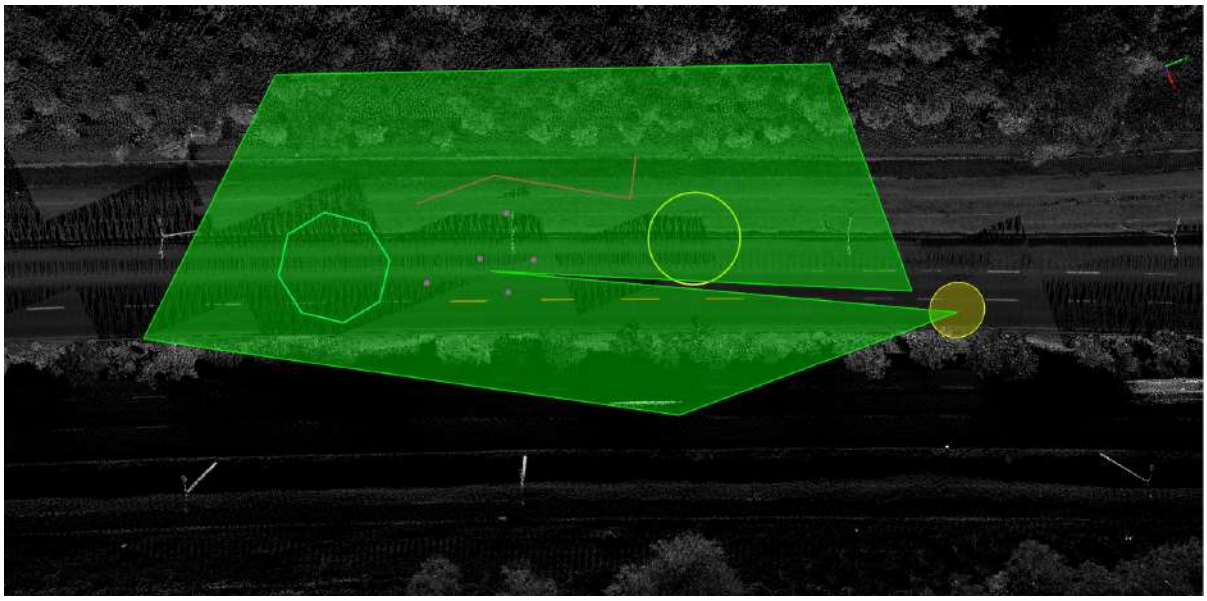
> Note: Select the shortcut key to copy, delete and other operations are the same as Pickup Vector.

Polygon

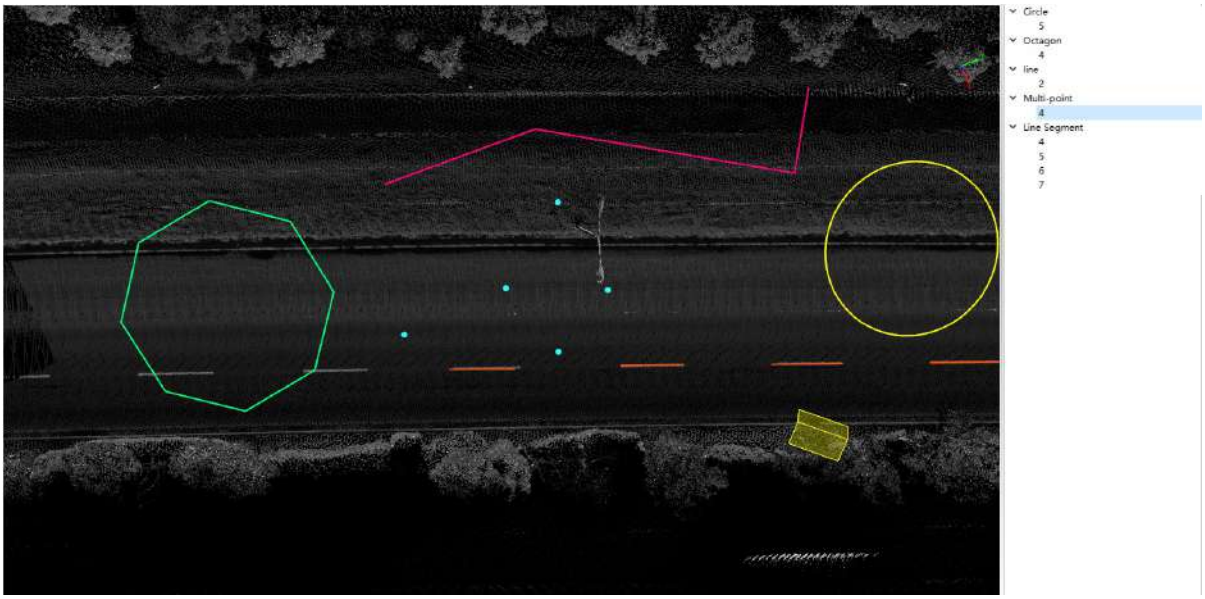
Function Description: The Pickup by Polygon function allows users to select a series of vector objects by using a polygon frame.

Steps

1.Click **Polygon**  button in the Vector Editor, use the mouse to continuously click multiple points to determine the nodes of the polygon box, double-click the left button of the mouse to select the polygon box and draw it, and the vectors in the box and intersecting with the box will be selected and highlighted. If multiple vectors are selected, a tree-structured dialog box will pop up on the right side of the view, including all currently selected vectors. Clicking the node with the left mouse button will deselect other vectors, and only select the corresponding vector and highlight it.



Polygon selection vector



Polygon frame vector - vector tree

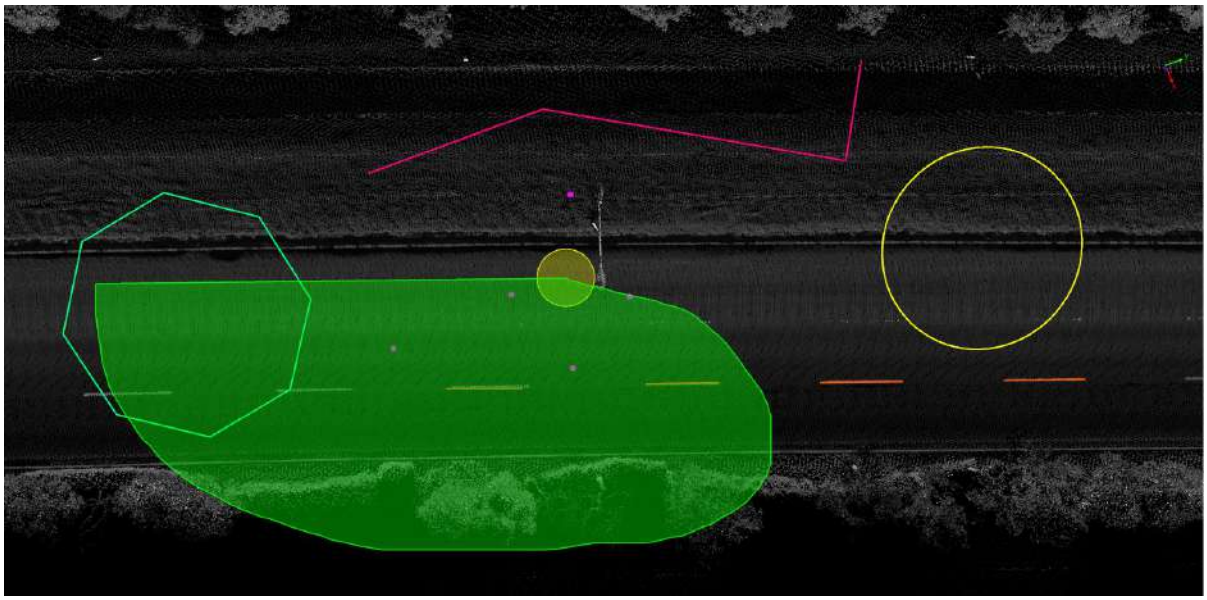
> Note: Select the shortcut key to copy, delete and other operations are the same as Pickup Vector.

Lasso

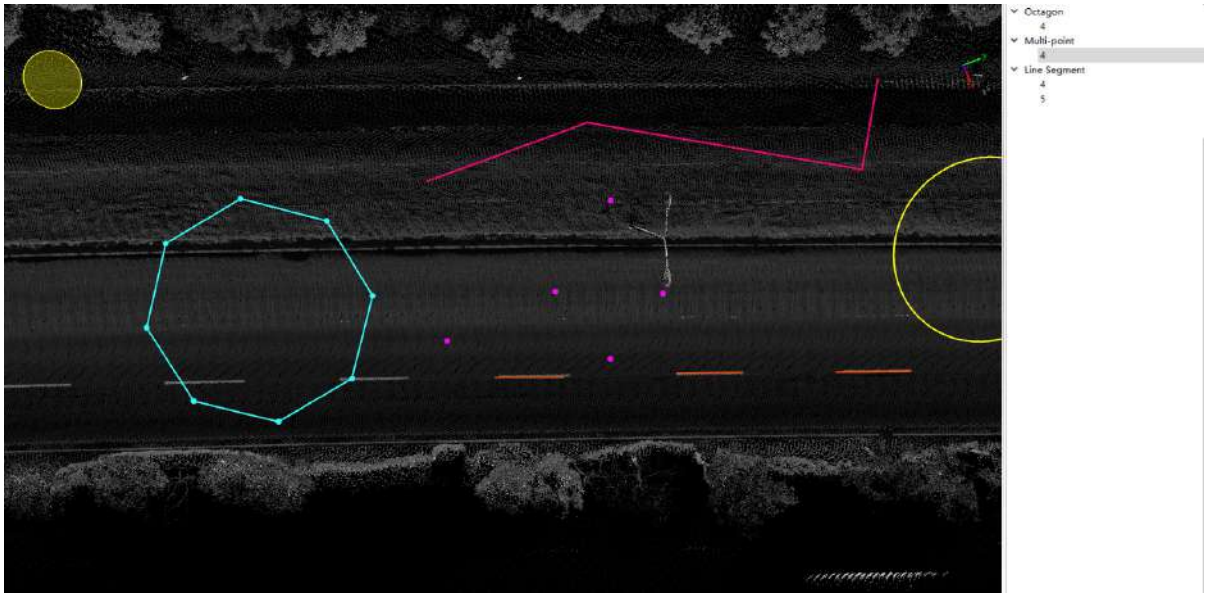
Function Description: The Pickup by Lasso function allows users to move the mouse sleeve to select a series of vector objects.

steps

1.Click **Lasso**  button in the Vector Editor, select a key point with the left mouse button, hold down the left mouse button and drag to form an area, and finally release the left mouse button, the vector in the area and its intersection will be selected and highlighted. If multiple vectors are selected, a tree-structured dialog box will pop up on the right side of the view, including all currently selected vectors. Clicking the node with the left mouse button will deselect other vectors, and only select the corresponding vector and highlight it.



Lasso selection vector



Lasso selection vector - vector tree

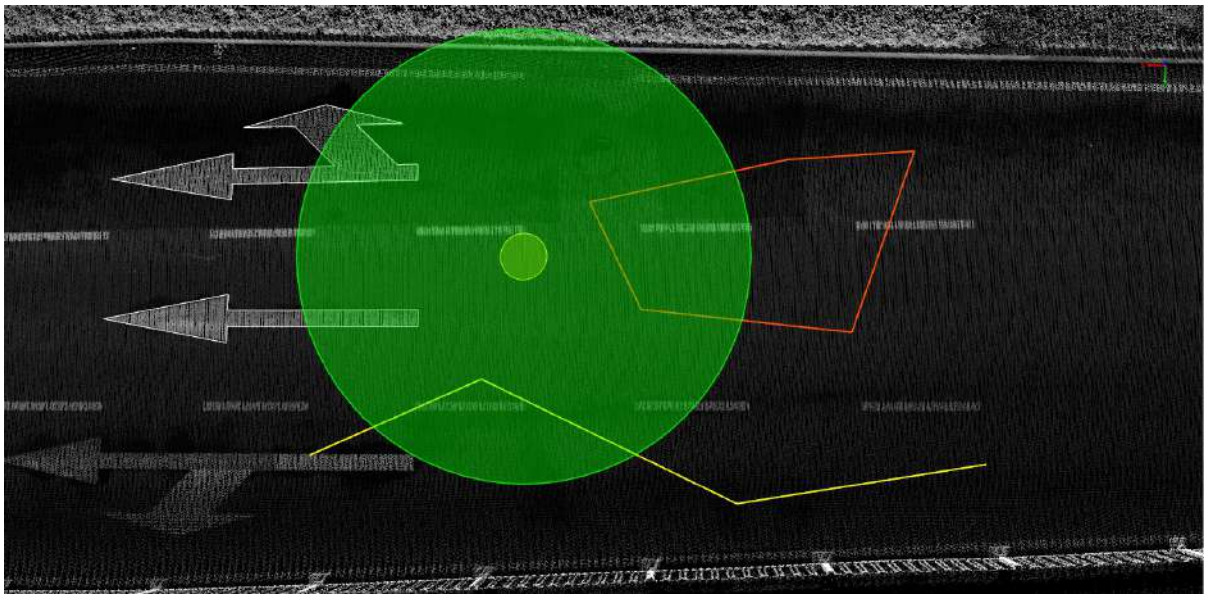
> Note: Select the shortcut key to copy, delete and other operations are the same as Pickup Vector.

Circle

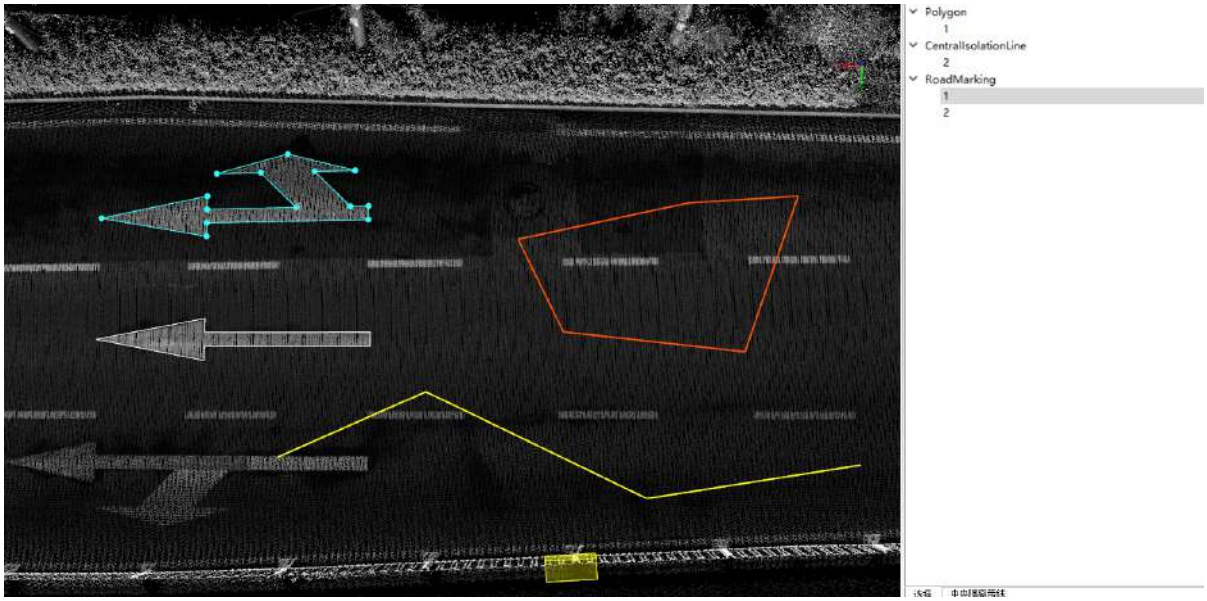
Function Description: The Pickup by Circle function allows users to use circles to select a series of vector objects.

steps

1.Click **Cicle**  button in the Vector Editor, select a key point with the left mouse button, hold down the left mouse button and drag to form a circular display box, and finally release the left mouse button, the vector within the circle and intersecting with the circle will be selected and highlighted. If multiple vectors are selected, a tree-structured dialog box will pop up on the right side of the view, including all currently selected vectors. Clicking the node with the left mouse button will deselect other vectors, and only select the corresponding vector and highlight it.



Circle selection vector



Circle selection vector-vector tree

> Note: Select the shortcut key to copy, delete and other operations are the same as Pickup Vector.

Box

Function description: The Pickup by Box function allows users to select a series of vector objects by using a rectangle.

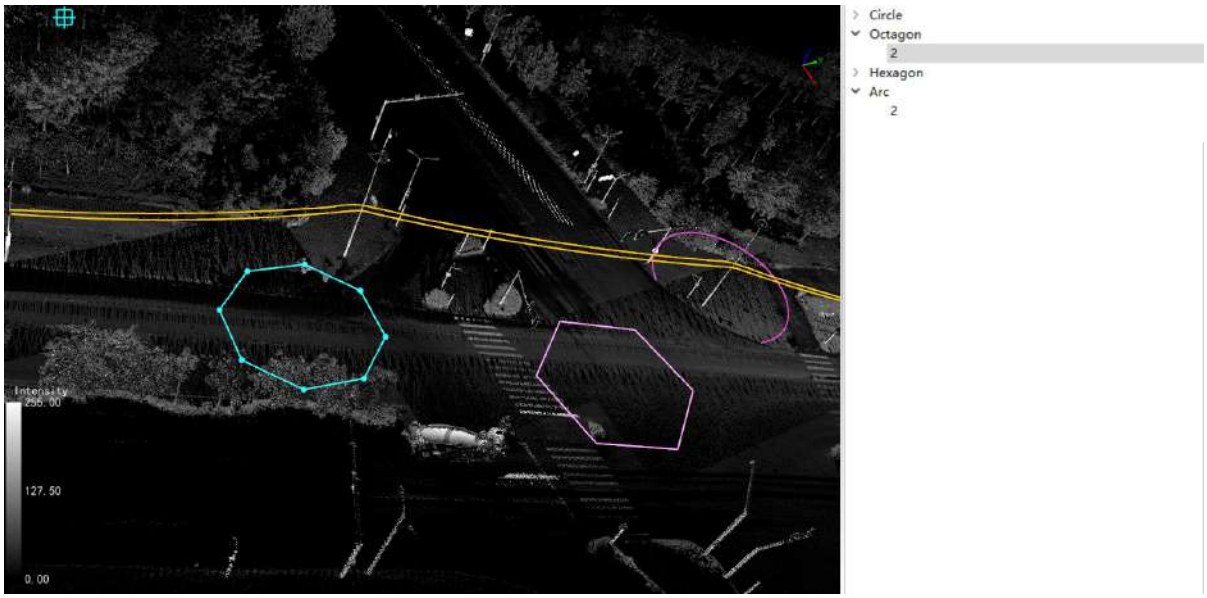
steps

1.Click **Box**  button in the Vector Editor:

- Click the point cloud with the left mouse button and select the first key point as the starting point of the cuboid.
- Click the point cloud with the left mouse button, select the second key point, and define an edge of the bottom surface of the box.
- Click the point cloud with the left mouse button, select the third key point, and determine the other side of the bottom of the box.
- Click the point cloud with the left mouse button, select the fourth key point, and determine the height of the cuboid to complete the drawing of the cuboid

Vectors inside and intersecting the bounding box will be selected and highlighted. If multiple vectors are selected, a tree-structured dialog box will pop up on the right side of the view, including all currently selected vectors. Clicking the node with the left mouse button will deselect other vectors, and only select the corresponding vector and highlight it.

Bounding box selection vector



Bounding box selection vector-vector tree

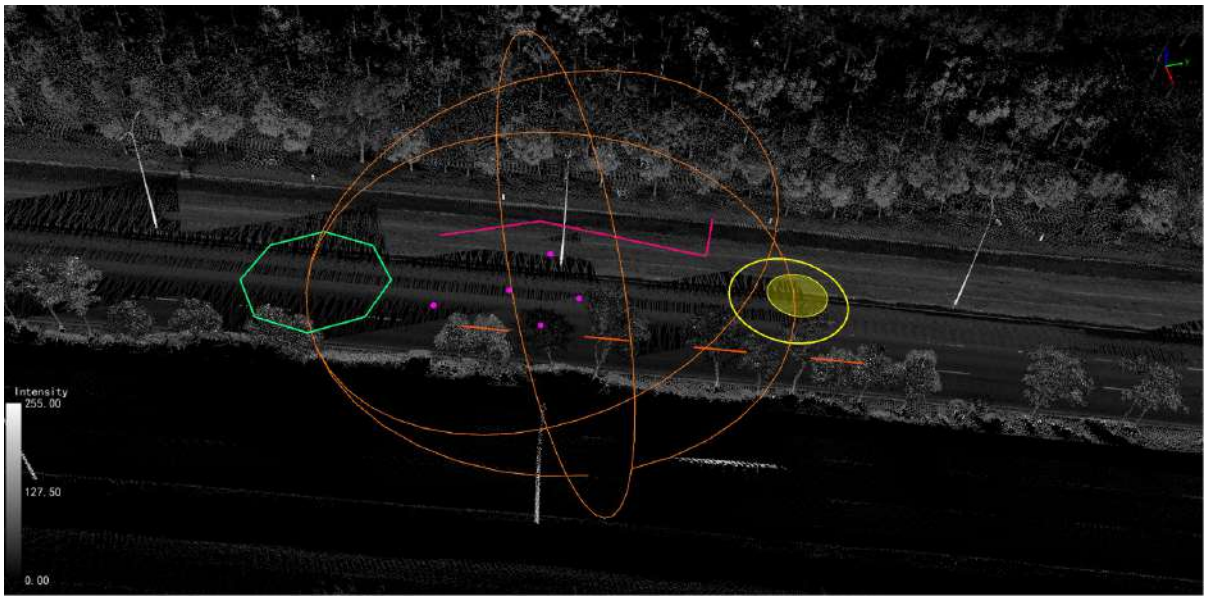
> Note: Select the shortcut key to copy, delete and other operations are the same as Pickup Vector.

Sphere

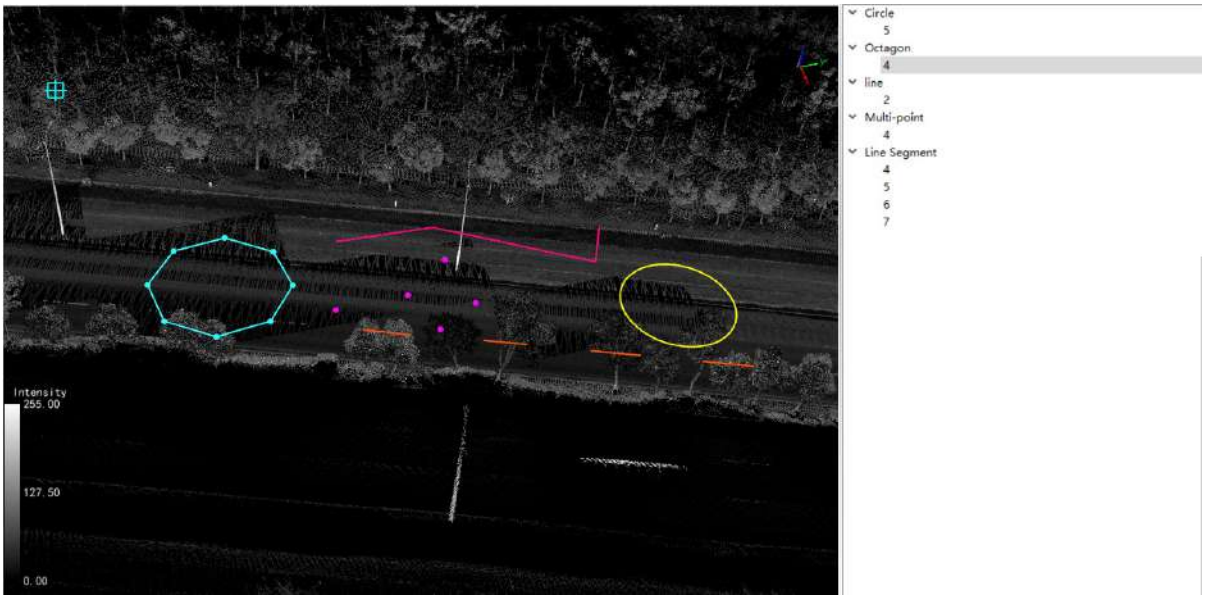
Function Description: The Pickup by Sphere function allows users to select a series of vector objects using spheres.

Steps

1.Click **Sphere**  button in the Vector Editor, click and drag the mouse with the left mouse button to display a spherical display. Finally, after clicking the left mouse button, the vectors within the sphere and intersecting with the ball will be selected and highlighted. If multiple vectors are selected, a tree-structured dialog box will pop up on the right side of the view, including all currently selected vectors. Clicking the node with the left mouse button will deselect other vectors, and only select the corresponding vector and highlight it.



Spherical selection vector



Spherical selection vector - vector tree

> Note: Select the shortcut key to copy, delete and other operations are the same as Pickup Vector.

Cylinder

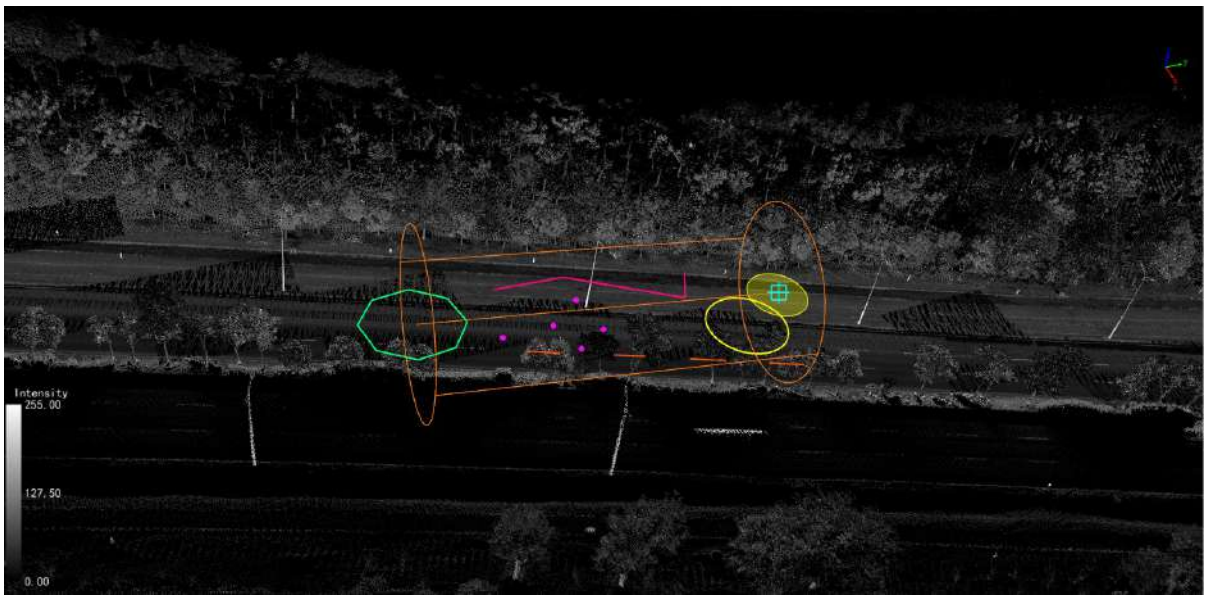
Function Description: The Pickup by Cylinde function allows users to select a series of vector objects using cylinders.

steps

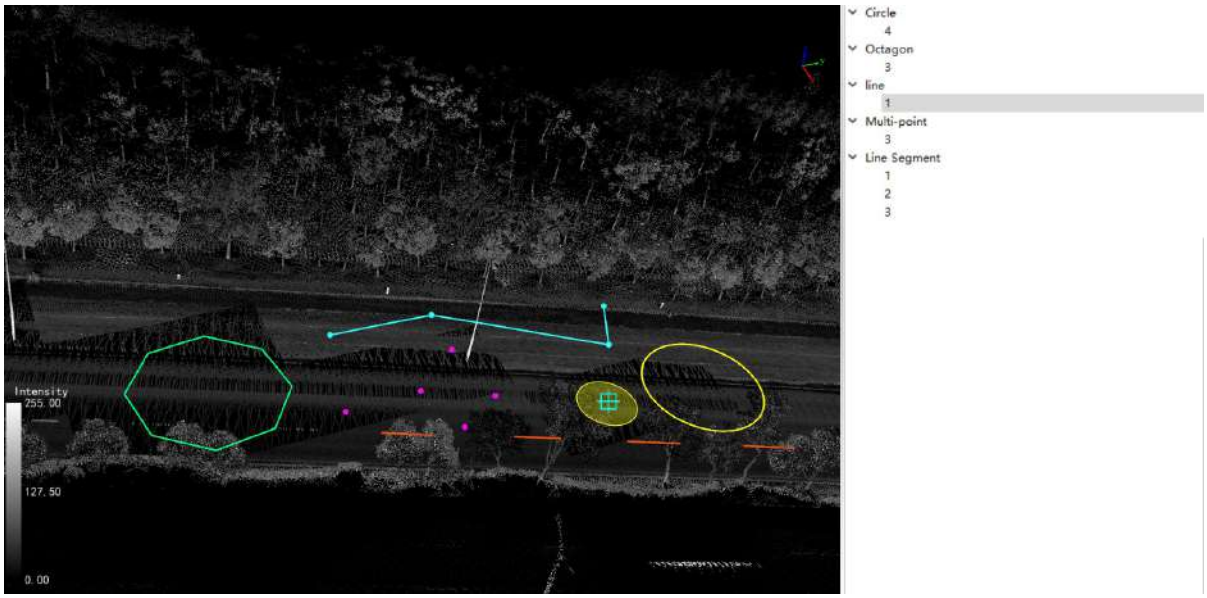
1.Click **Cylinder**  button in the Vector Editor:

- Click the point cloud with the left mouse button and select the first key point as the center point of the bottom surface of the cylinder.
- Click the point cloud with the left mouse button, select the second key point, and determine the center point of the other base circle of the cylinder.
- Click the point cloud with the left mouse button, select the third key point, and determine the size of the bottom circle of the cylinder to complete the drawing of the cylinder.

Select a key point with the left mouse button, hold down the left mouse button and drag to form a circular display box, and finally release the left mouse button, the vectors within the circle and intersecting with the circle will be selected and highlighted. If multiple vectors are selected, a tree-structured dialog box will pop up on the right side of the view, including all currently selected vectors. Clicking the node with the left mouse button will deselect other vectors, and only select the corresponding vector and highlight it.



Cylinder selection vector



Cylinder selection vector-vector tree

> Note: Select the shortcut key to copy, delete and other operations are the same as Pickup Vector.

Interrupt

- Point Split Line
- Line Split Line
- Line Split Polygon
- Line Split Objects

Point Split Line

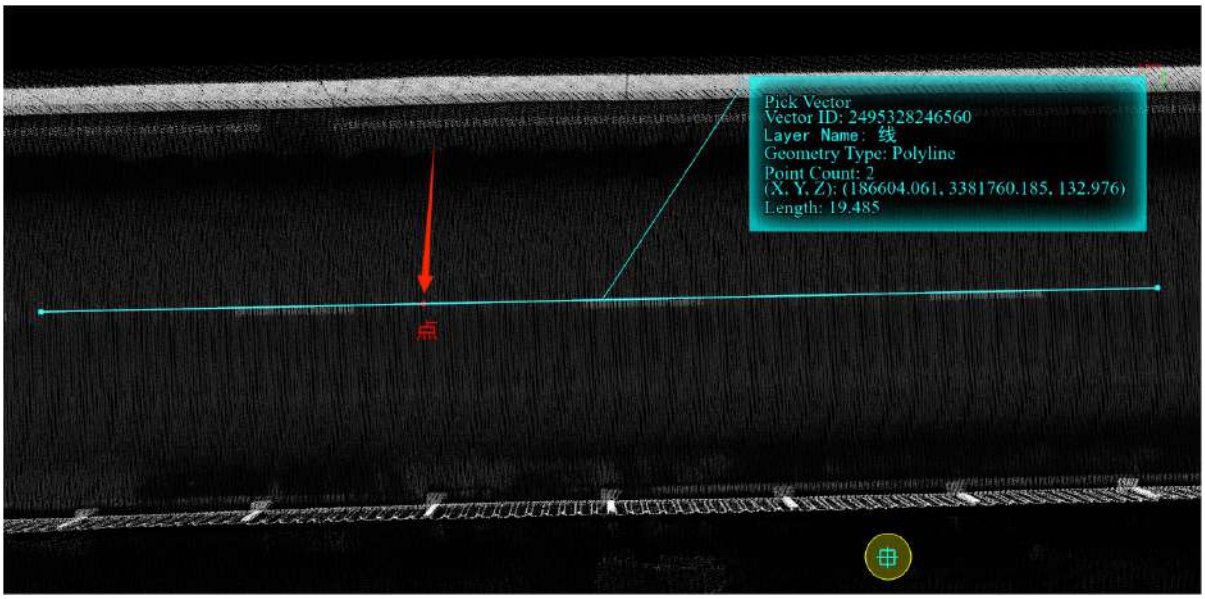
Function Description： The Point Split Line function allows the user to break a line object into two end-to-end line objects through a point on the line.

Steps

1.After clicking the **Point Split Line** button in the Vector Editor, move the mouse to click the point object on the line first, and then click the line object to be broken again, the line object will be broken into two line objects connected end to end.

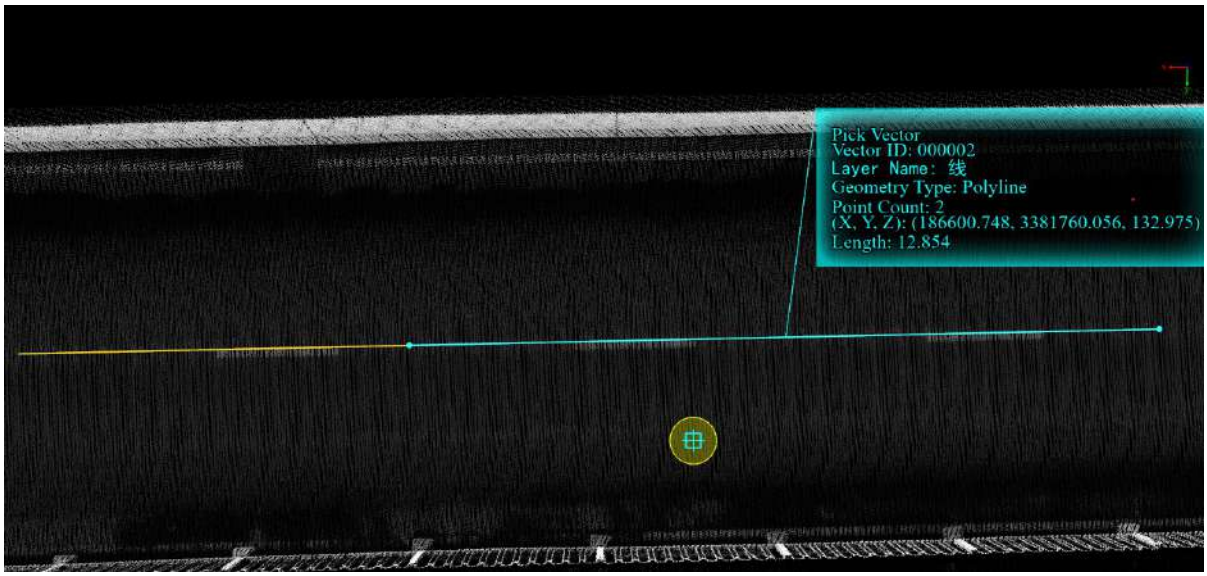
> If the point object is not on the line object, interruption is not allowed, and there will be a prompt in the output window.

- **Original line segment**: a single line object.



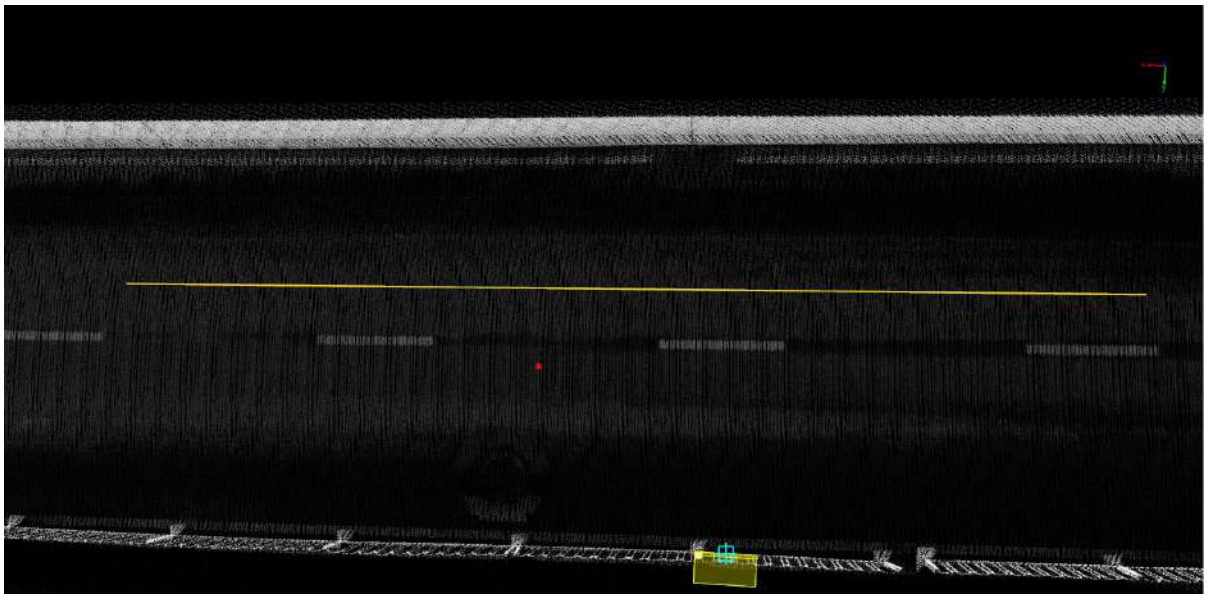
Original line segment

- **Break at the point object on the line segment**: The two lines after the break are connected end to end at the point object.

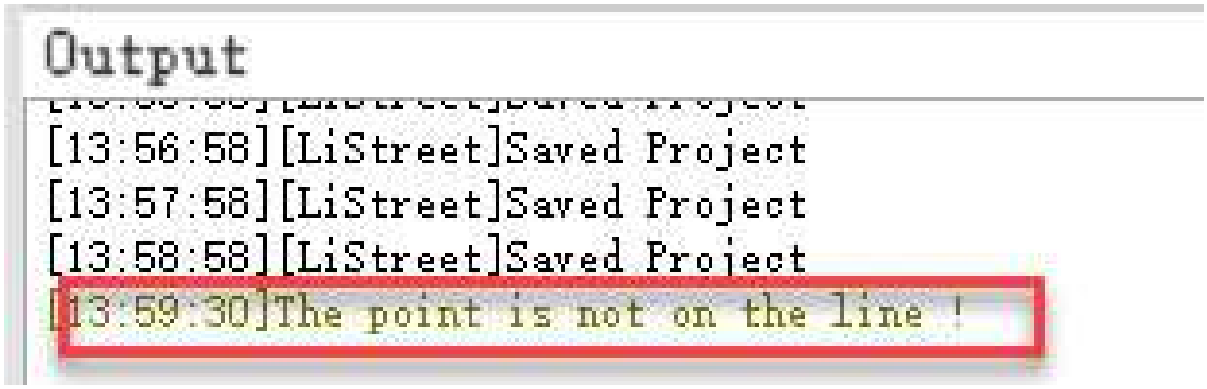


Line segment after break

- **The point is not on the line**: Break will not be allowed, and there will be a prompt in the output window.



Break point not on line



Error prompt

Line Split Line

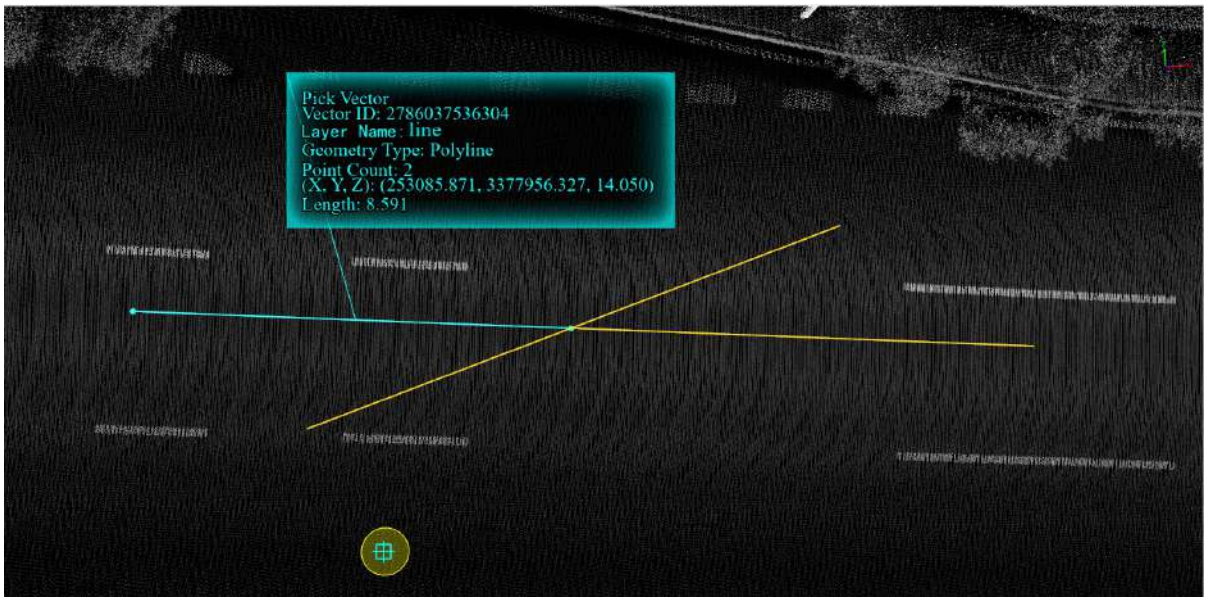
Function Description： The Line Split Line function supports by intersecting one of the two lines, the other line is broken into two lines connected end to end.

Steps

1.Click **Line Split Line**  button in the Vector Editor, move the mouse to click one of the intersecting line objects first, and then click the line object to be broken again, the line object will be broken into two line objects connected end to end.

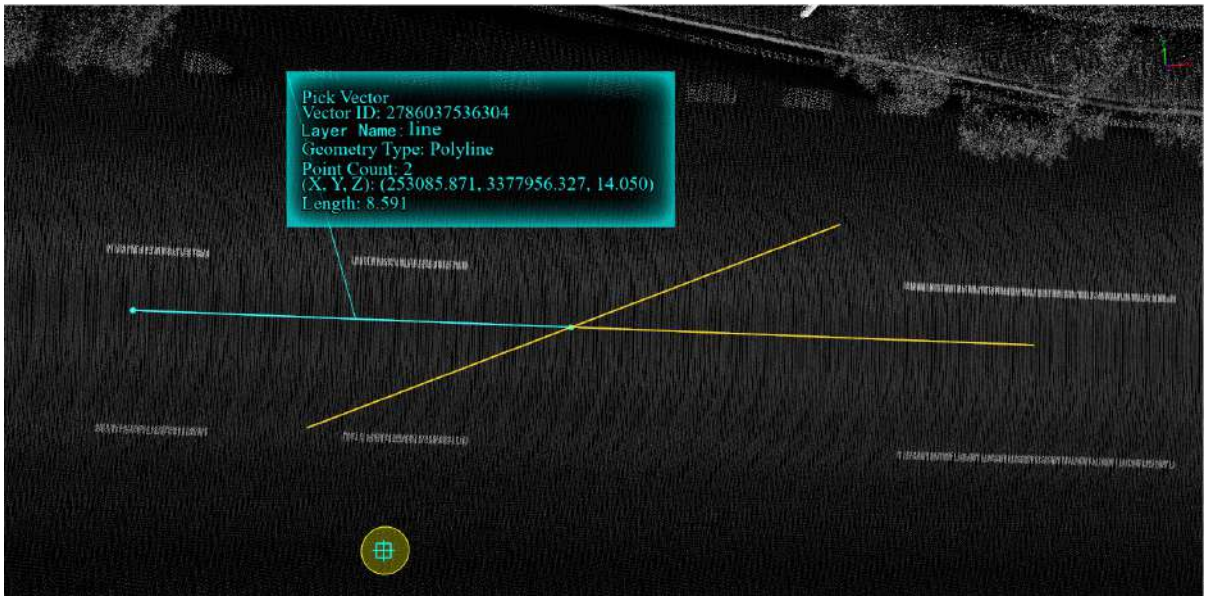
> If the two line objects do not intersect, line break will be invalid, and there will be a prompt in the output window.

- **Original intersecting lines**: Two intersecting line objects.



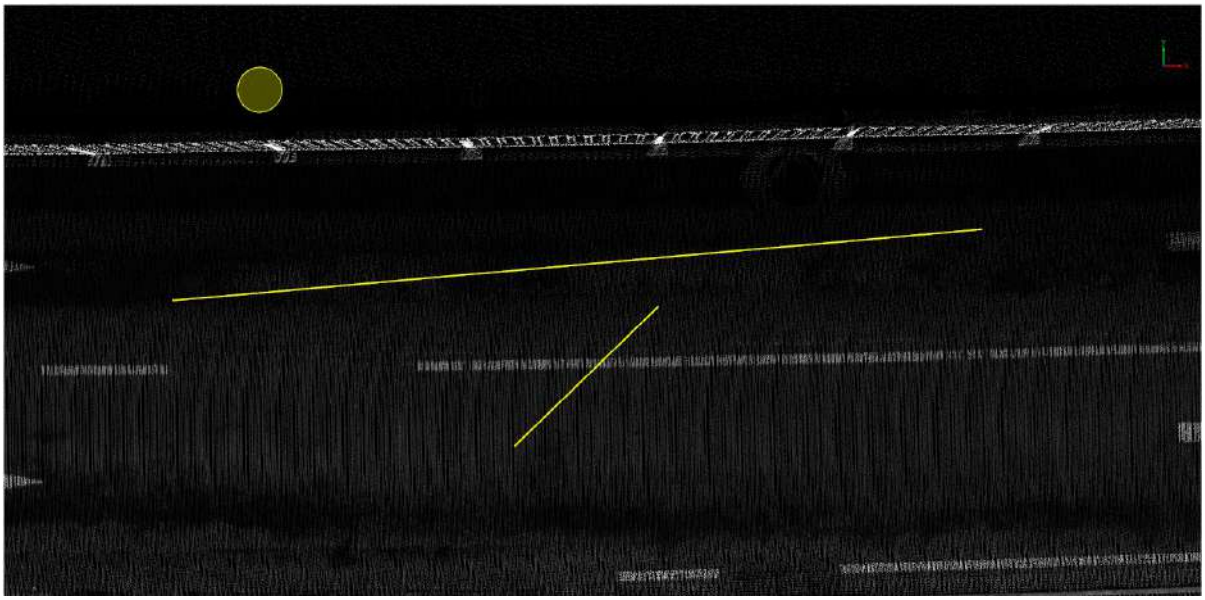
Original line

- **Break at intersection point**: The two lines after the break are connected end to end at the intersection point.

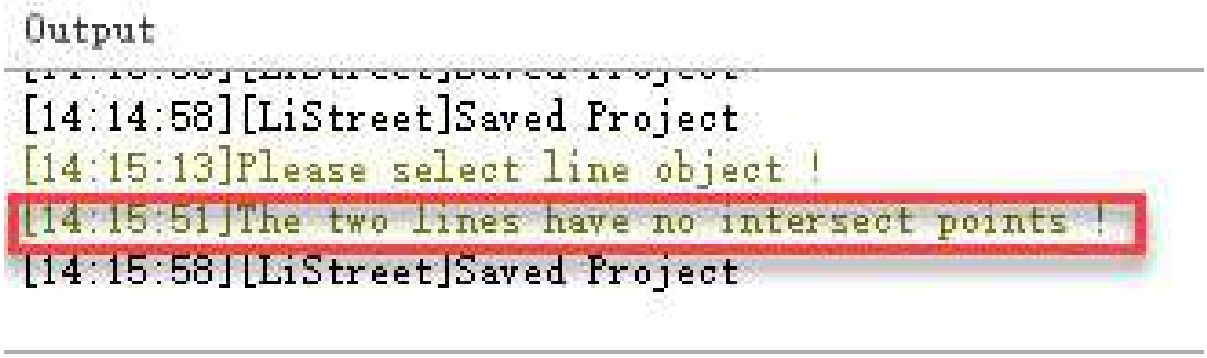


Line segment after break

- **Two line segments do not intersect**: line break is not allowed, and there will be a prompt in the output window.



Two line segments do not intersect



Error prompt

Line Split Polygon

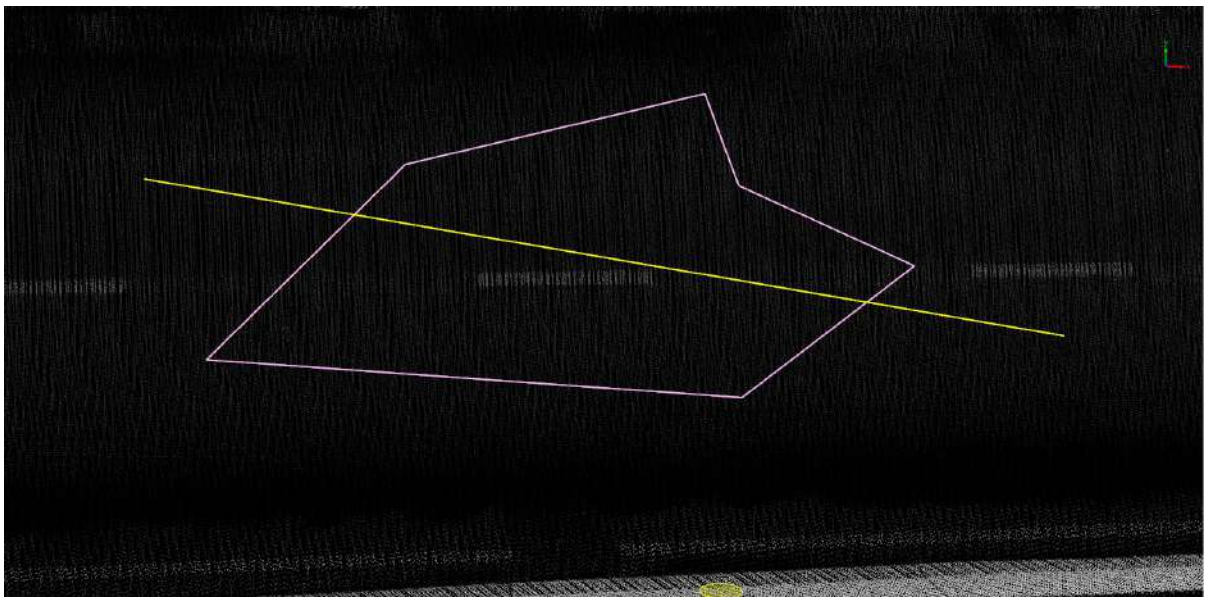
Function Description： The Line Split Polygon feature allows the user to break an area object into two area objects by intersecting a line in the area object.

Steps

1.After clicking the **Line Split Polygon** button in the Vector Editor, move the mouse to first click the line object in the intersecting object, and then click the surface object to be interrupted again, and the surface object will be broken into two connected surface objects.

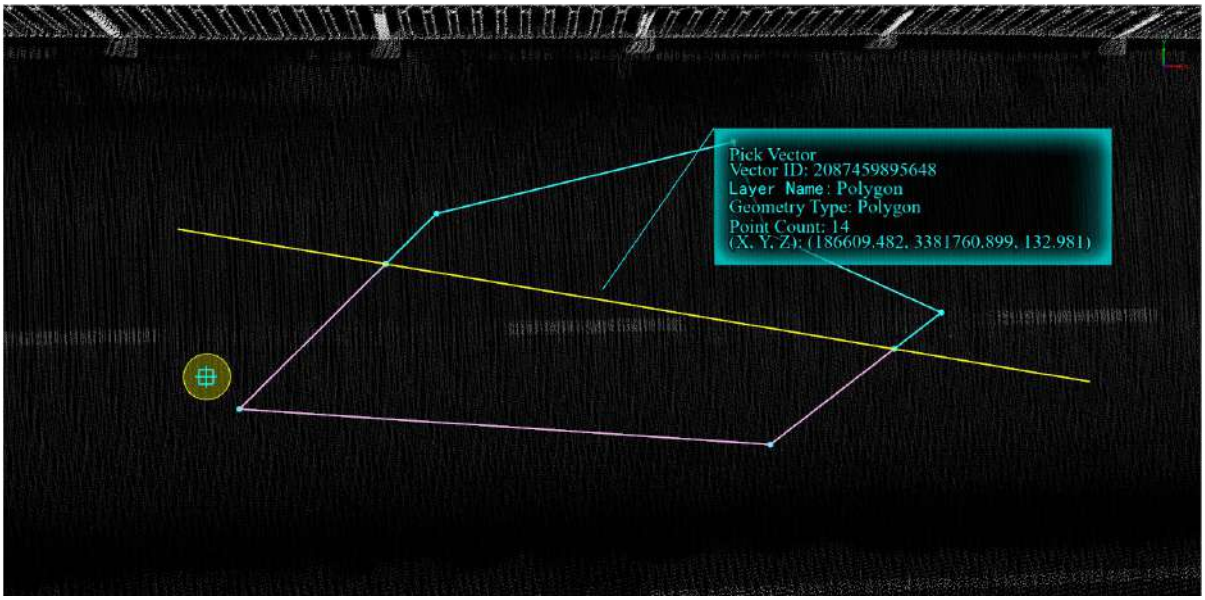
> If the intersection of the line and the polygon is less than two, line break will not be allowed, and there will be a prompt in the output window.

- **Original intersecting lines and polygon**: Two intersecting line and area objects.



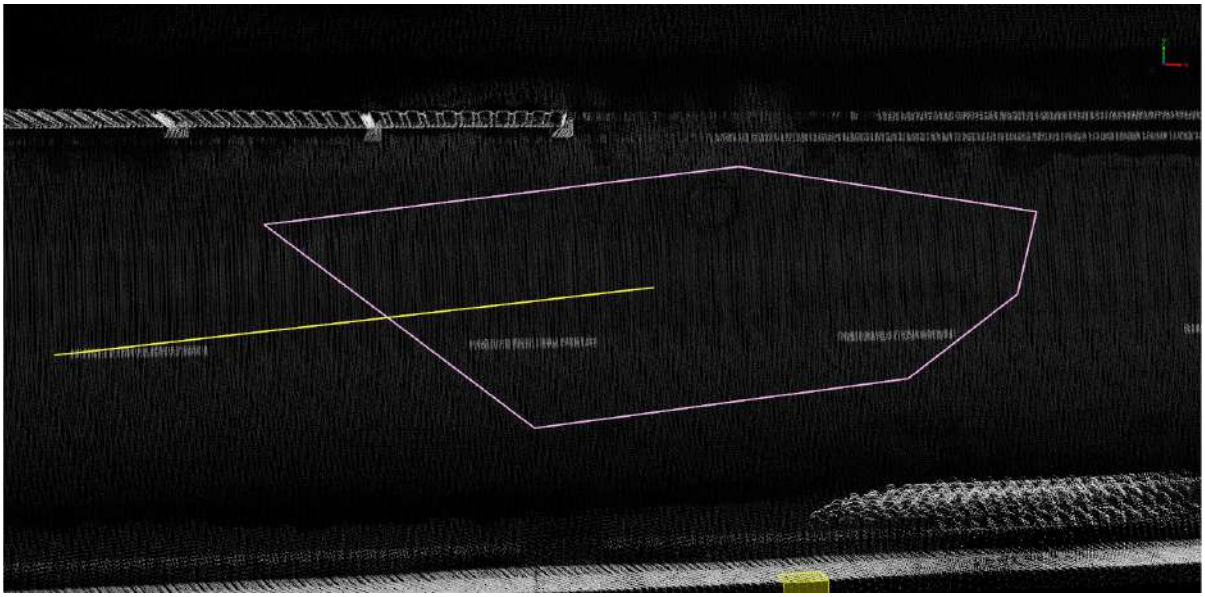
Original lines and polygon

- **Broken polygon**: Two polygons that are connected after being broken.

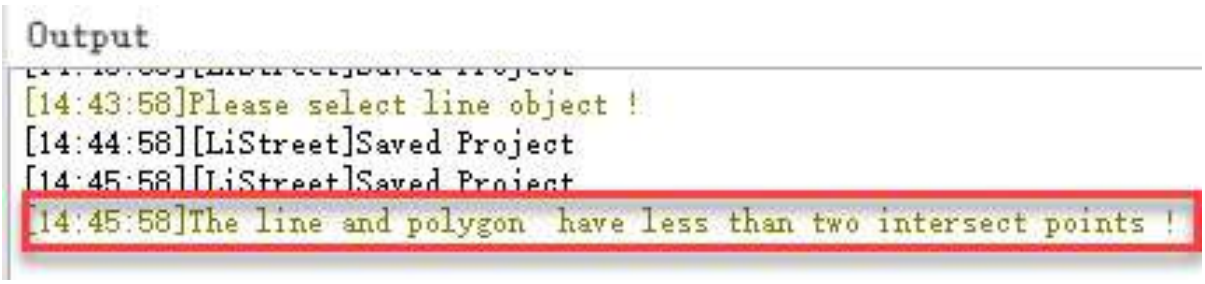


Two polygons connected after break

- **The break points of the polygon is less than 2**: No break is allowed, and there will be a prompt in the output window.



Less than 2 break points



Line-polygon intersection is less than two points

Line Split Objects

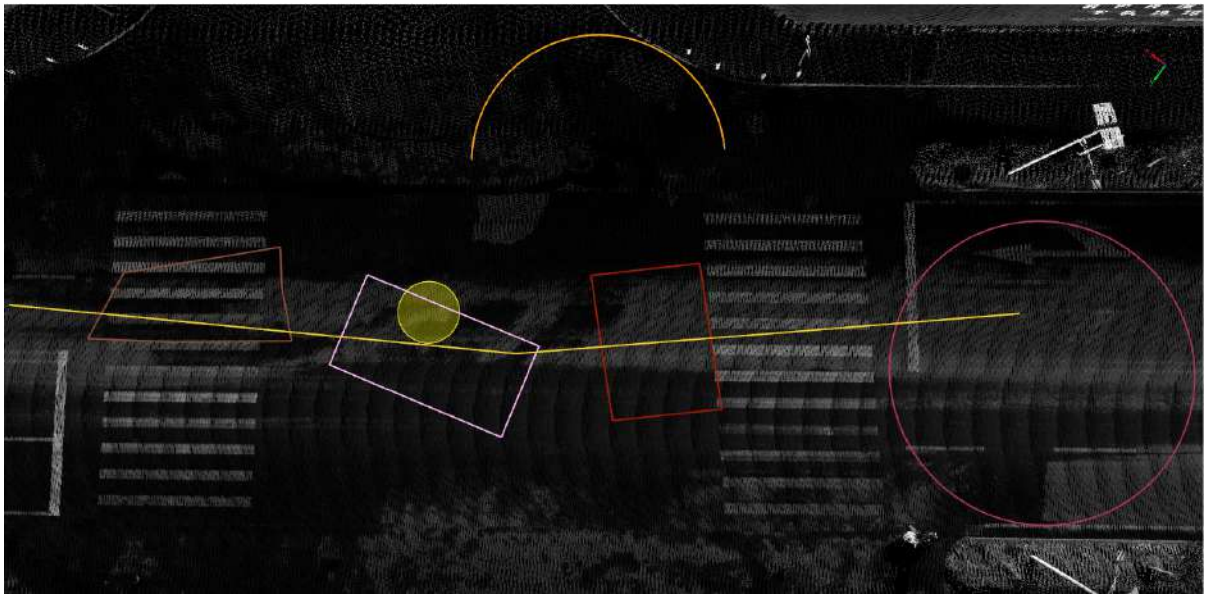
Function Description: This tool allows the user to break the intersecting objects through a line, and break a line or area object into two objects.

Steps

1.Click **Line Split Objects**  button in the Vector Editor, click the left mouse button for the first time, draw the upper left point of the rectangle, drag the mouse, determine the range, click the left mouse button again, the rectangle is drawn, and the objects inside the box and intersecting with the box will be selected and highlighted. Move the mouse and click the line object in the intersecting object first, then the object intersecting with the line object will be interrupted into two connected objects.

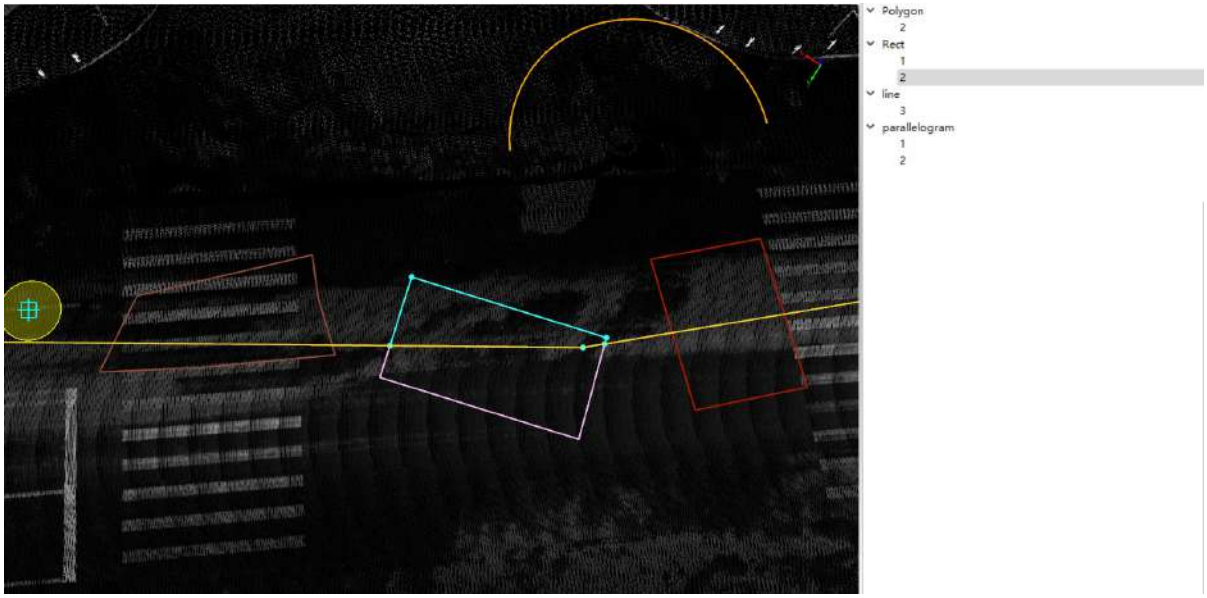
> If the intersection of the line and multiple objects is less than two, interruption is not allowed, and there will be a prompt in the output window.

- **Original intersected line with multiple objects**: Line object intersects with multiple objects.



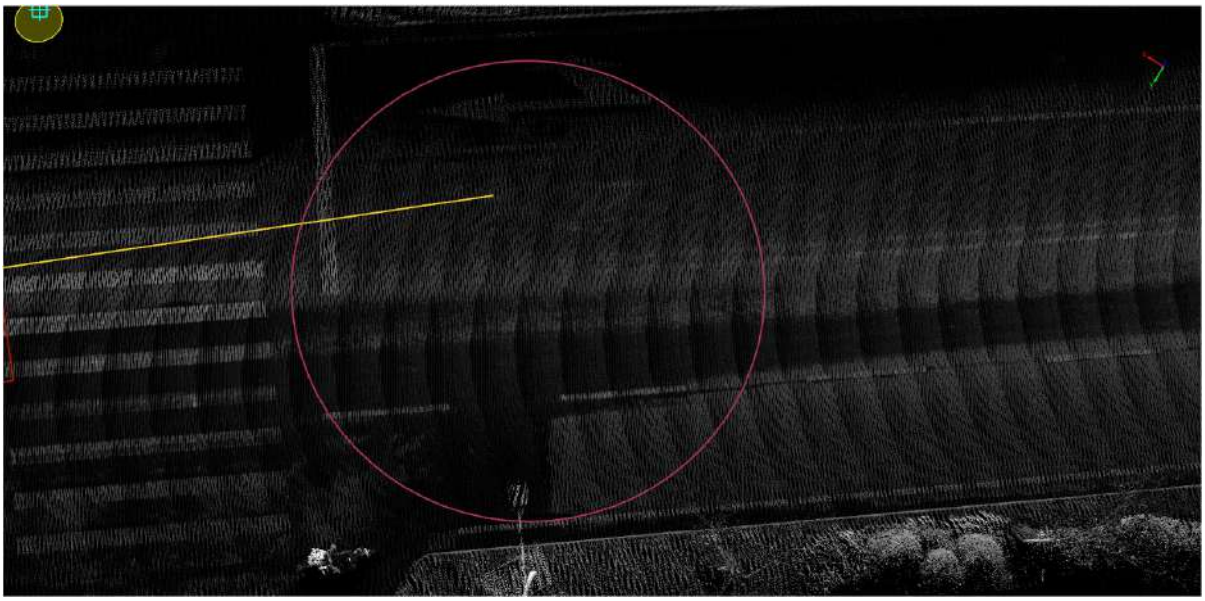
Original intersecting lines with multiple objects

- **Interrupted Objects**: Two objects connected after being interrupted. Use box selection to view the results.



Two objects connected after interruption

- **The intersection of the line and the object is less than two**: Interruption is not allowed, and there will be a prompt in the output window.



The intersection of the line and the object is less than two

The intersection of the line and the object is less than two prompts

Alignment

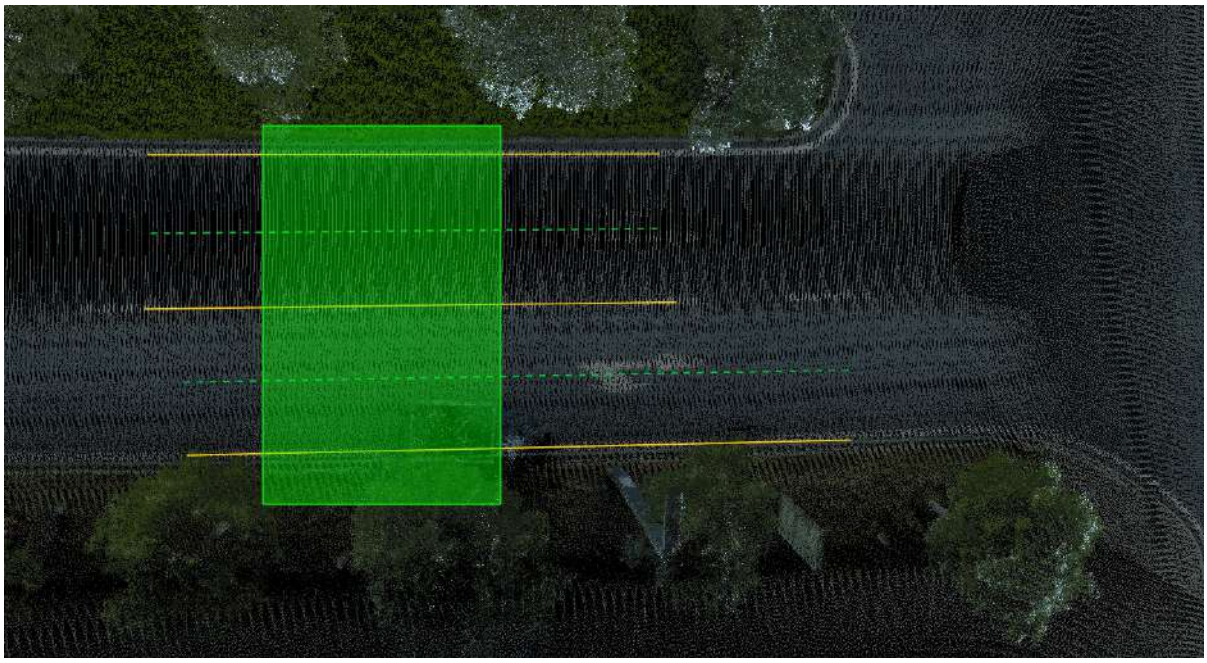
- Move
- Parrallel Copy
- Rotate
- Scale Polygon
- Mirror

Move

Function Description： The Move Vector function allows the user to move a vector object to a new position.

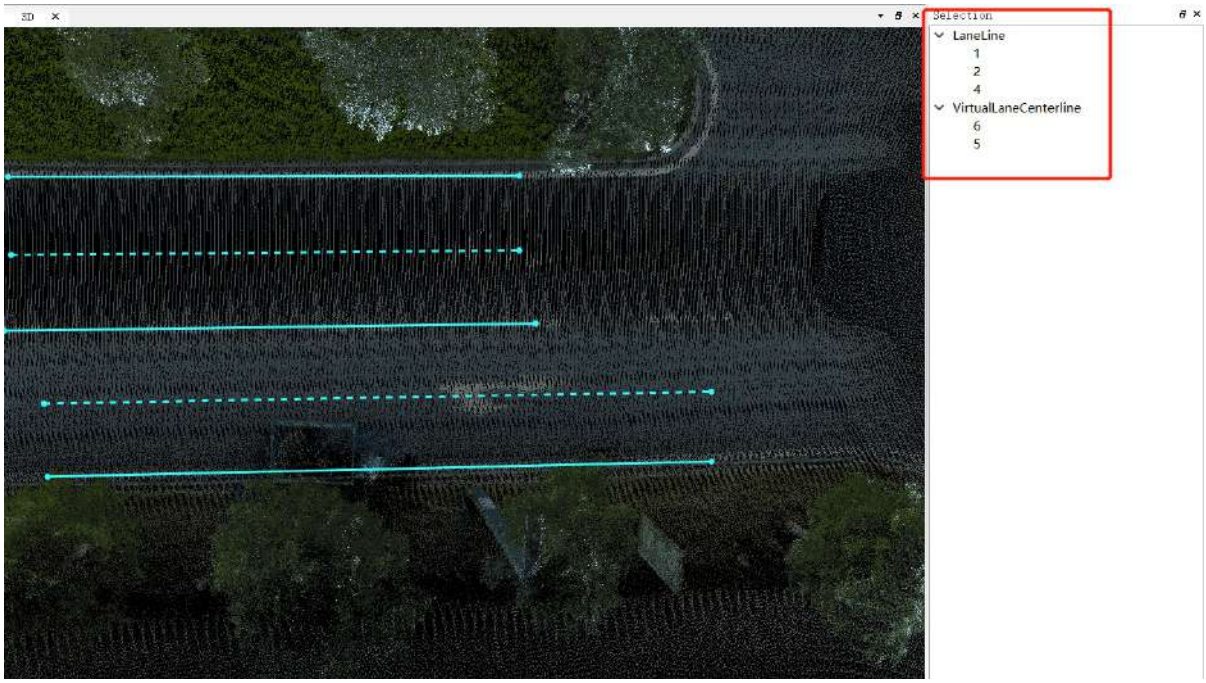
Steps

1.After clicking the **Move** button in the Vector Editor, frame the area where the vector to be moved is located.



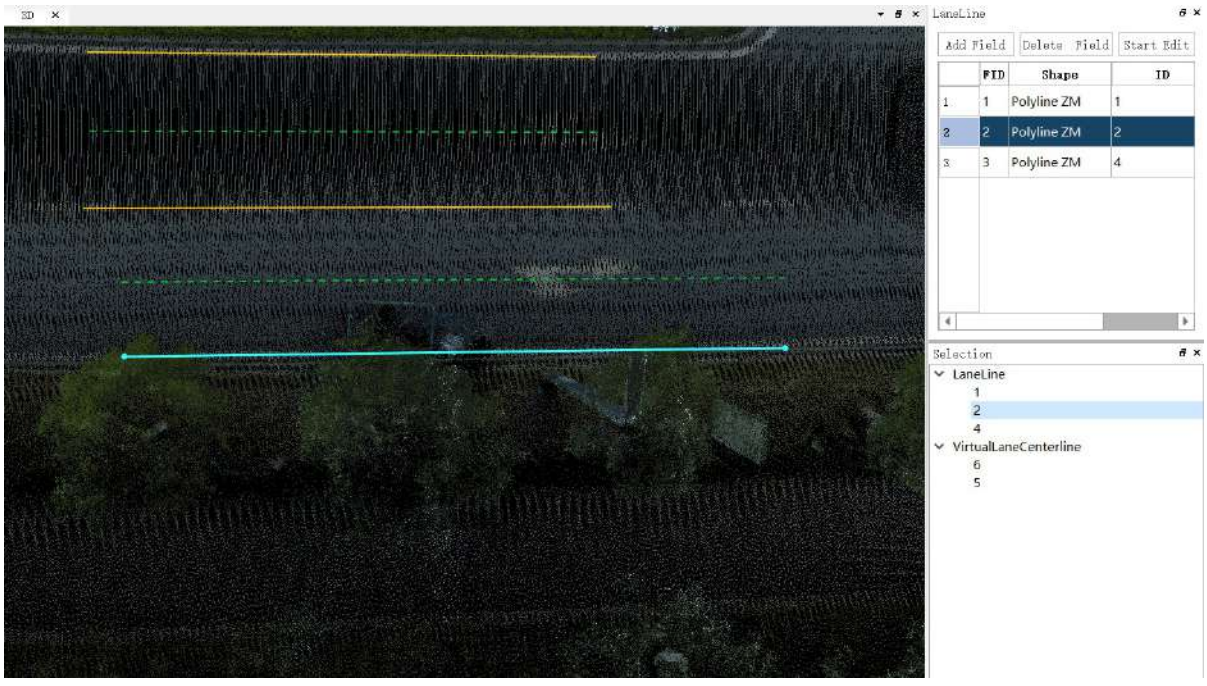
Frame selection of vectors

2.After the frame selection is completed, the selected vector will be highlighted, and the right side of the main window will display the detailed information of all the frame-selected vectors, including vector layers and IDs.



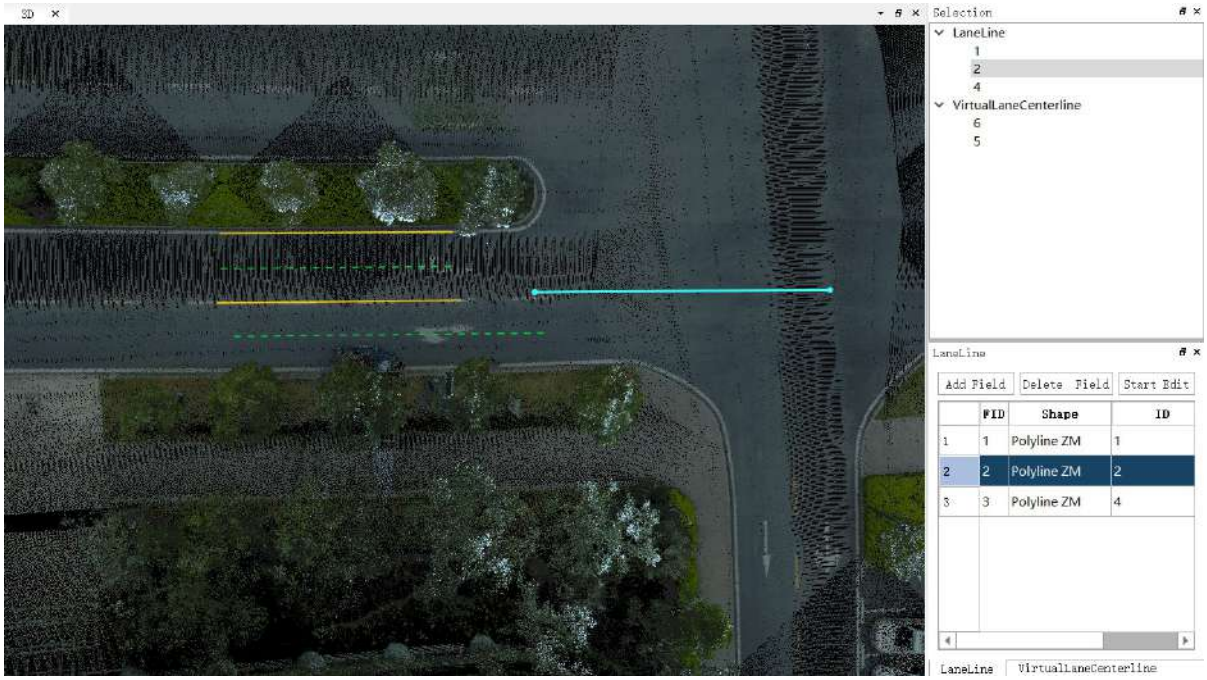
After frame selection of vectors

3.Click to select the details of a vector on the right, other vectors' highlights will be cleared, and the selected vector will be highlighted.



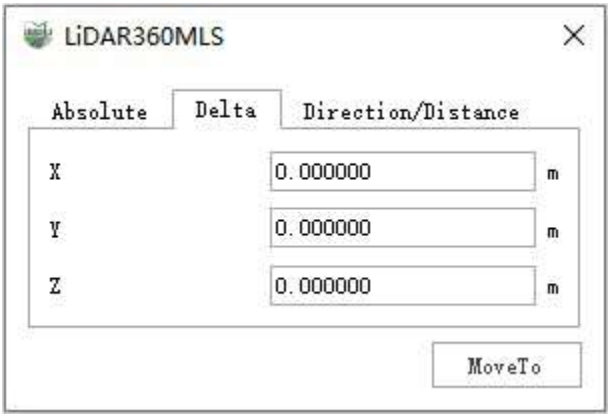
Select a vector

4.Click on the selected highlighted vector in the 3D window and hold down left click, drag the vector to the new position to be moved, and unclick, the vector will be moved to the new position.



After vector move

5.After the completion of frame selection, the selected vector will be highlighted. Click the right mouse button to pop up the drop-down box Move to. Click to the pop-up box to view, enter or adjust values.



- **Absolute**：Set the coordinate values of X, Y, X for absolute fixed-point movement.
- **Relative**：Customize the coordinate values of X, Y, X to move relative fixed-point.
- **Direction/Distance**：Set the direction angle, pitch angle and distance to move.

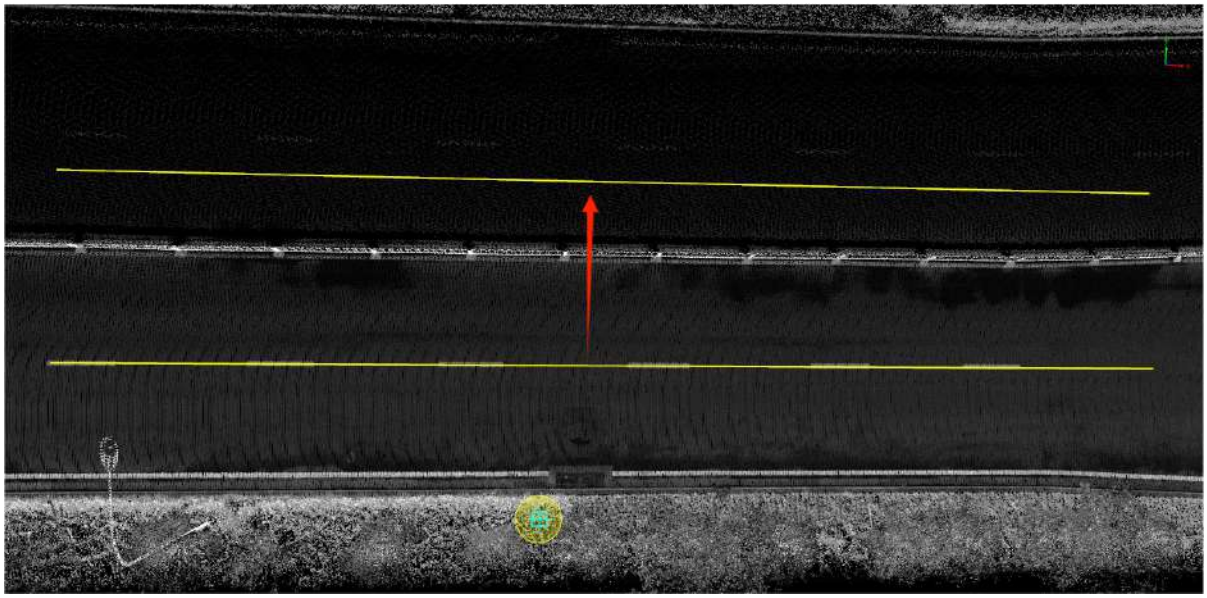
> Note: After you box selecting a vector object, the selected vector will be highlighted, and the details of all the selected vectors, including the vector layer and ID, will not be displayed on the right side of the main window at the same time.

Parallel Copy

Function Description： The Parallel Vector Copy function allows users to copy a vector object in parallel.

Steps

1.After clicking the **Parallel Copy** button in the Vector Editor, move the mouse to the vector and click and hold the left click, drag the mouse to copy the selected vector to the current mouse position.



Parallel vector copy

Rotate

Function Description： The Rotate function allows users to rotate vector objects with a certain point as the center.

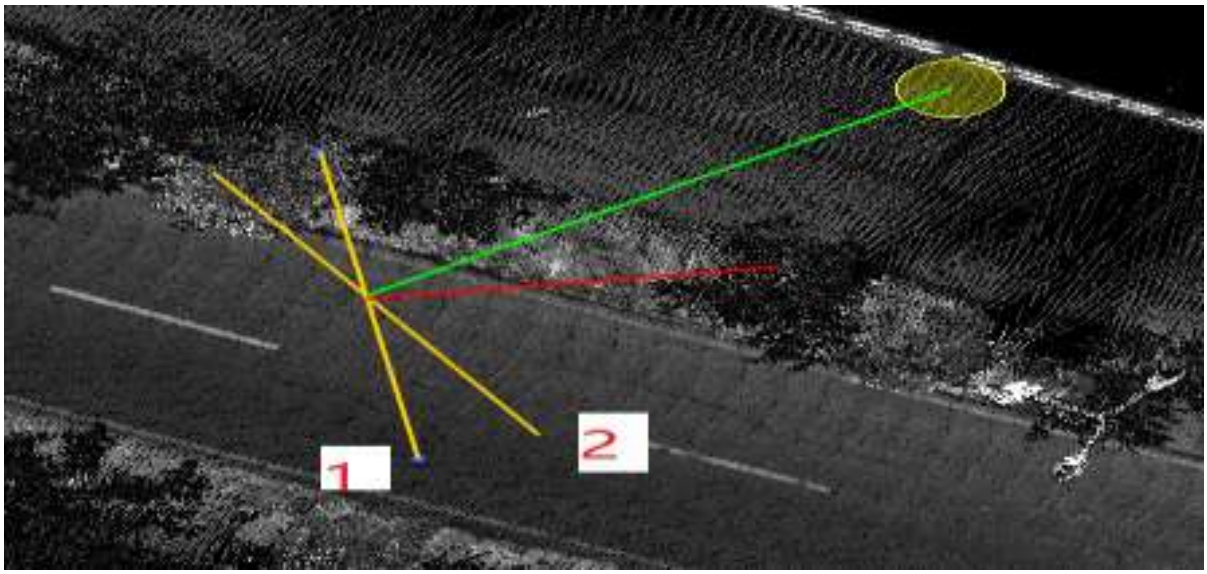
Steps

1.After clicking **Rotate**  button in Vector Editor, move the mouse to the vector and left click to confirm the center of rotation;

2.Left click again to select the second point to confirm the rotation auxiliary line, drag the mouse to rotate in the window, and the real-time preview will be at the position where the vector is rotated;

3.After the position is determined, finally left click, and the vector will be rotated to the position where it was previewed.

> In the figure below, the yellow line No. 1 is the initial vector object, the red line is the rotation auxiliary line determined by the second key point, the yellow line No.2 is the position after rotation, and the green line is the corresponding line of the red line after the rotation. The angle between the green line and the yellow line No.2 is equal to the angle between the red line and the yellow line No.1.



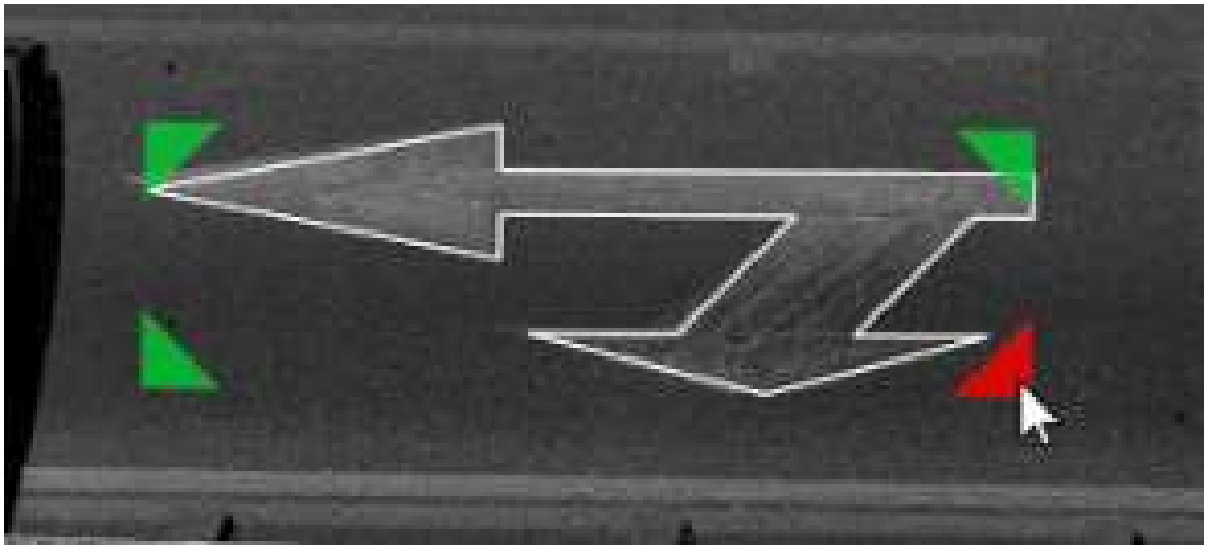
Rotate vector

Scale Polygon

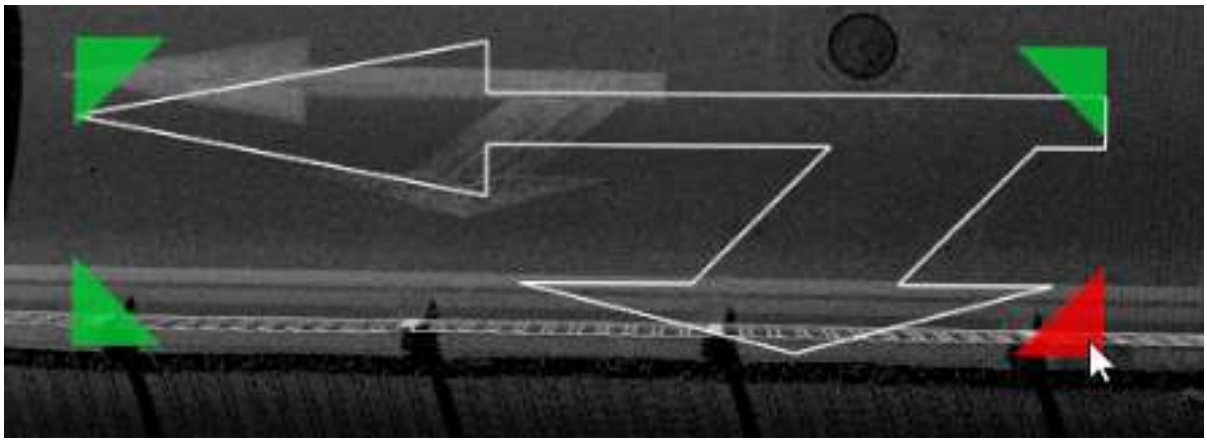
Function Description： The Scale Polygon function allows the user to scale polygons in equal proportions along the horizontal direction.

Steps

1.After clicking the **Scale Polygon** button in the Vector Editor, click the polygon to be scaled in the view, and a scale indicator will appear around the polygon. Hovering the mouse over the indicator will turn red, and drag it to the opposite corner. The vertices are anchor points to scale the current polygon.



Scale Polygon



Scale Polygon

Mirror

Function Description： The Mirror Vector function allows the user to move a vector object to a new position in a mirror-symmetrical manner.

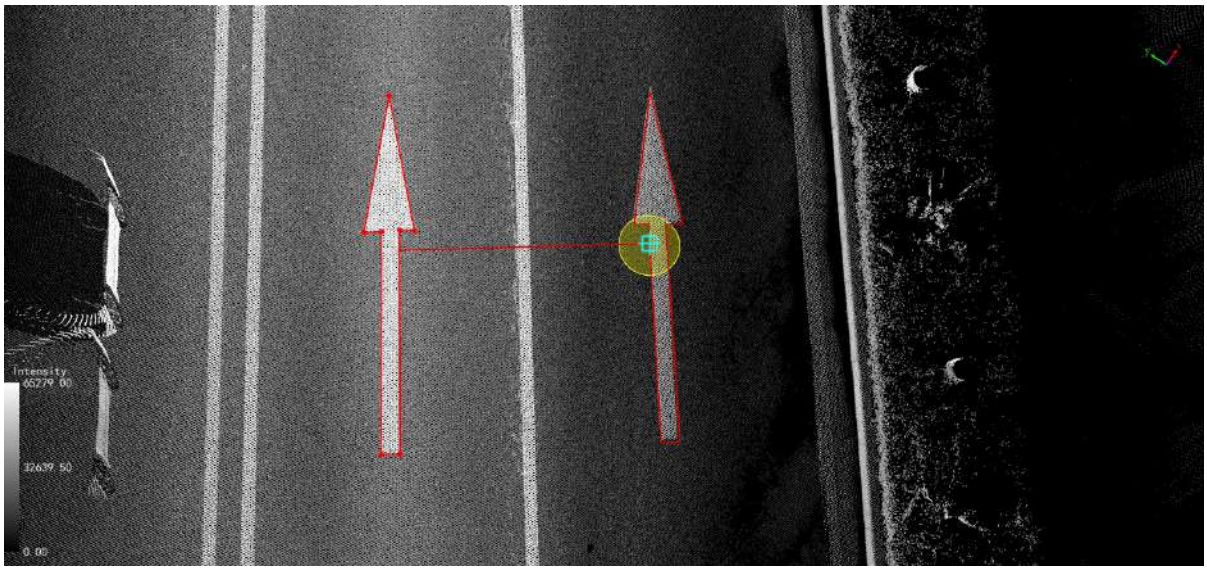
Steps

1.Click **Mirror**  button in the Vector Editor;

2.Move the mouse to the vector and click the left mouse button to determine the reference point for generating the mirror image;

3.Then move the mouse, a mirror preview will be generated at the position where the mouse moves;

4.After confirming the position where the vector will be mirrored, finally click the left mouse button, and the vector object will be mirrored and moved here.



Mirror Vector

Reshape

- Reshape Line
- Reshape Rect
- Merge Lines
- Break Line
- Split Line
- Extend or Trim
- Split or Merge Polygon
- Link Line
- Extend Line
- Fillet
- Split Mulitpolygon or Mulitpoint
- Combine Polygons or Points
- Smooth

ReShape Line

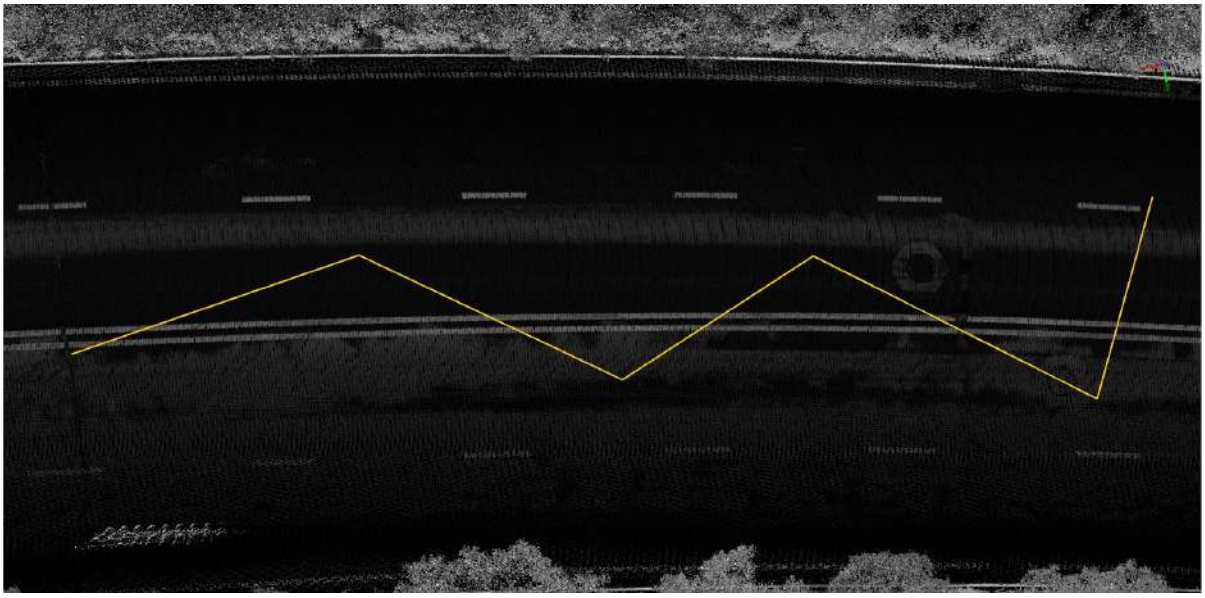
Function Description： ReShape Line allows the user to adjust and repaint of existing line objects.

Steps

1.After clicking the **ReShape Line**  button in the Vector Editor,move the mouse to the starting position of the line to be modified, left click to start redrawing, and double-click to end after drawing, and the line object will be redrawn.

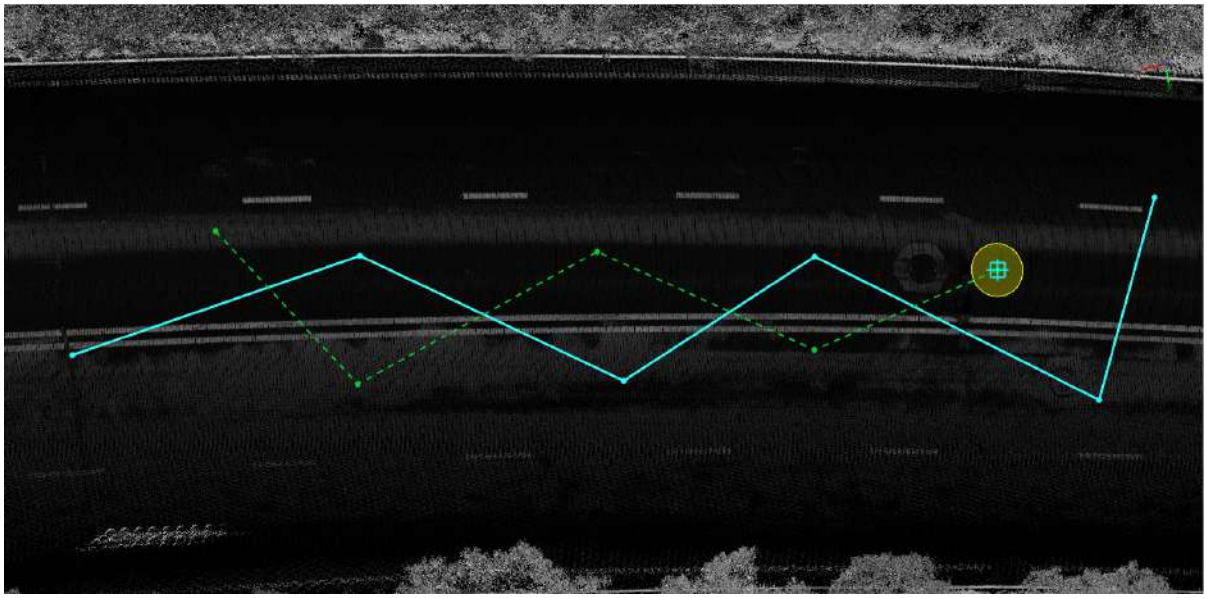
> Please ensure that the newly drawn polyline has more than two intersection points with the original line object, otherwise it will not be redrawn.

- **Original line object**: The object to be redrawn.



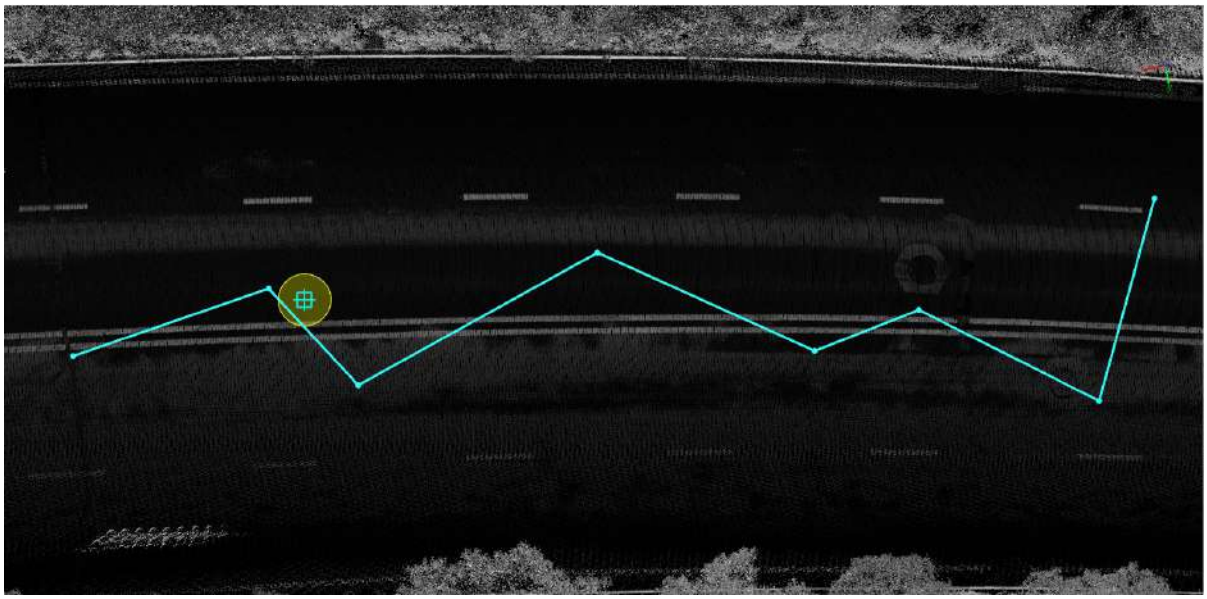
Original line object

- **Drawing process**: The same steps as drawing a polyline, consecutively click on each point to be redrawn , and double-click to end.



Reshape process

- **Reshaped object**: After the reshape is complete, the original line object will be regenerated based on the drawn line shape.



After reshape

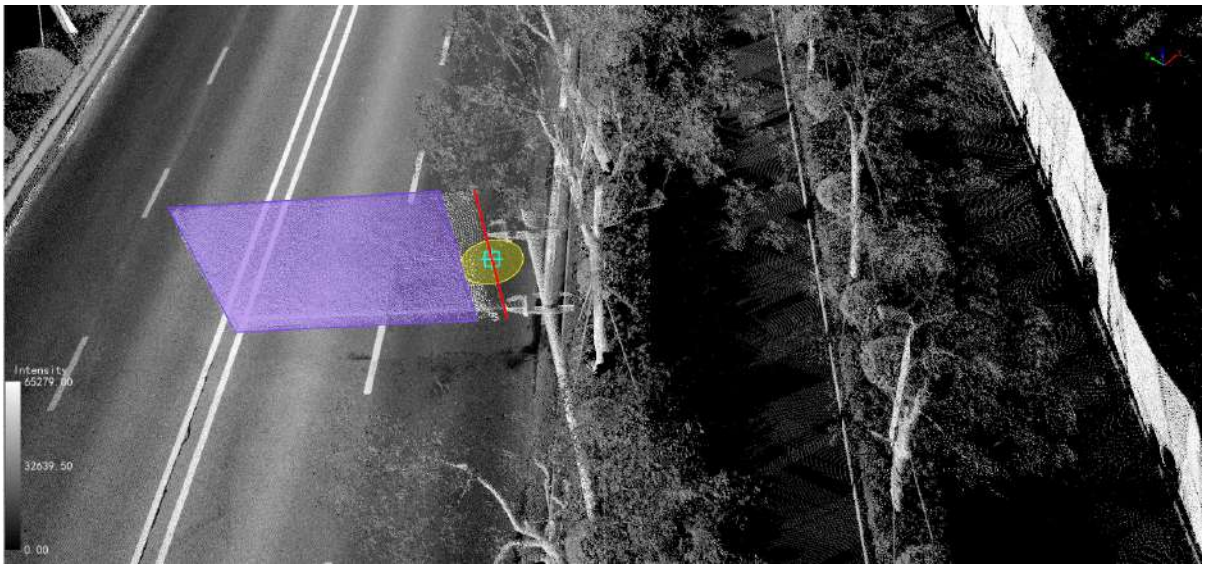
ReShape Rect

Function Description: The ReShape Rect function allows the user to move one edge of the quadrilateral object in parallel to the position of the mouse.

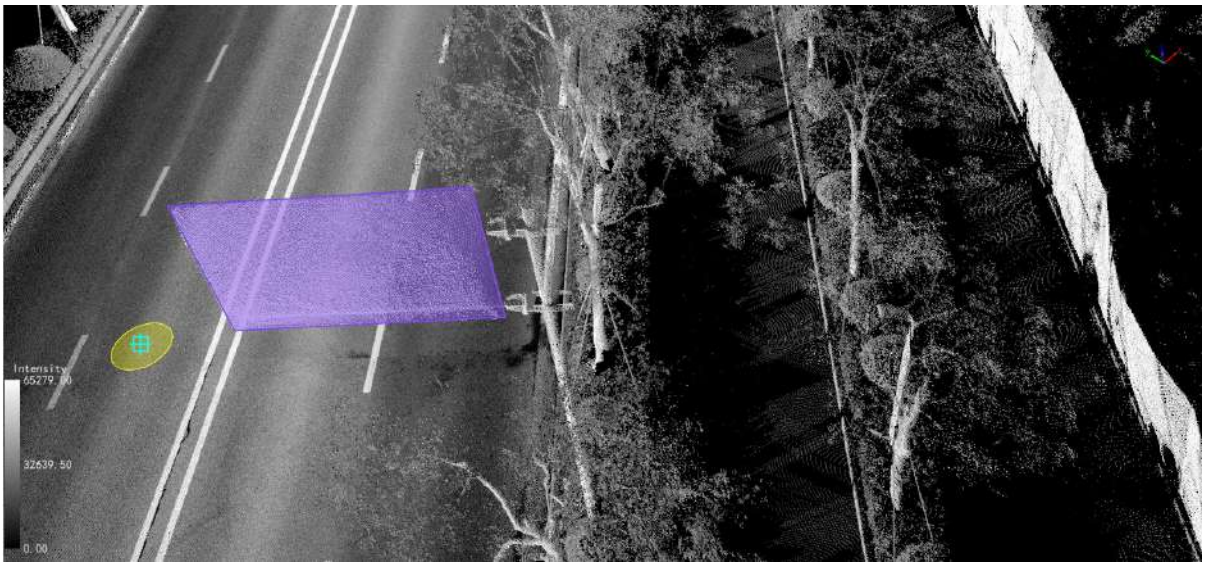
Steps

1.Click **ReShape Rect**  button in the Vector Editor;

2.Move the mouse to an edge of the face vector and click the left mouse button to determine the key edge;



3.Then move the mouse to the desired position, double-click the left button of the mouse to determine the edge position, and form a new vector, as shown in the following figure;



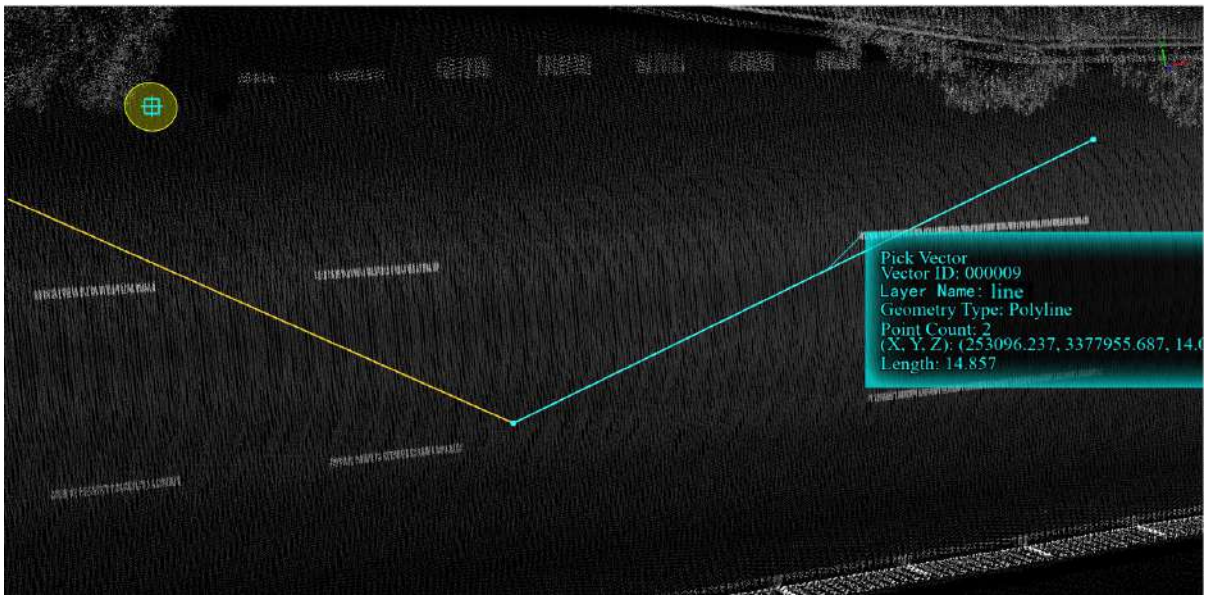
Merge Lines

Function Description： The Merge Lines function allows the user to merge two lines connected end to end into one line object.

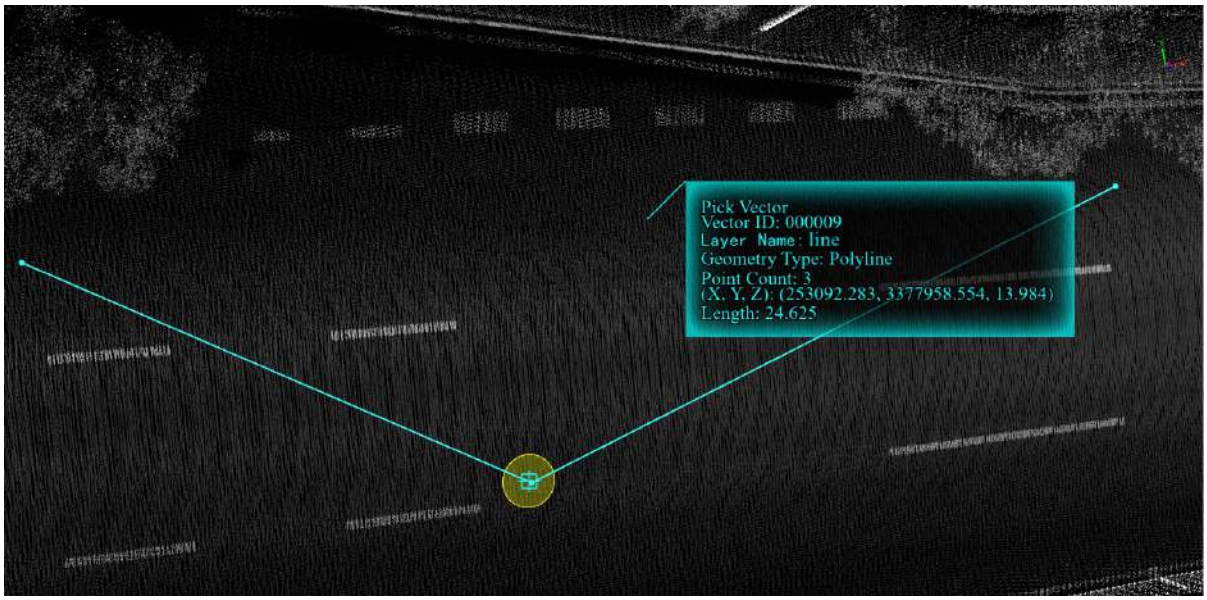
Steps

1.Click **Merge Lines** button in the Vector Editor, click the two lines that are connected end to end and need to be merged with the left mouse button.

> This function can only process two lines with the same point at the beginning and end. For example, the node 1-2 of line 1 and the node 2-3 of line 2 will be merged into a line string with nodes 1-2-3. The details are as follows:



Two lines to be connected



Two lines after connection

Break Line

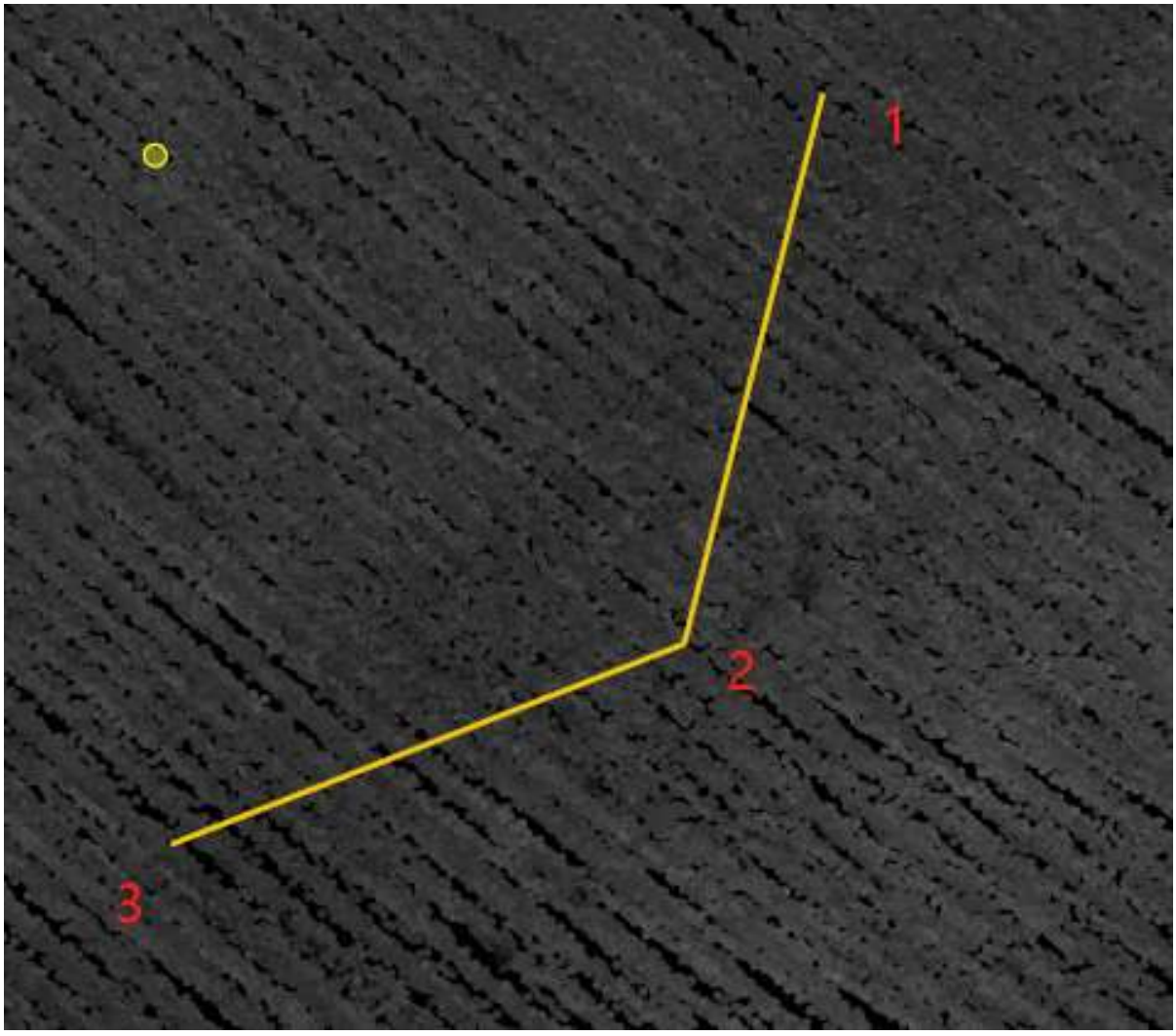
Function Description： The Break Line function allows the user to break a line object into two line objects connected end to end.

Steps

Click **Break Line**  button in the Vector Editor, move the mouse to the vector, select the place you want to break, and left click to complete the break of the line.

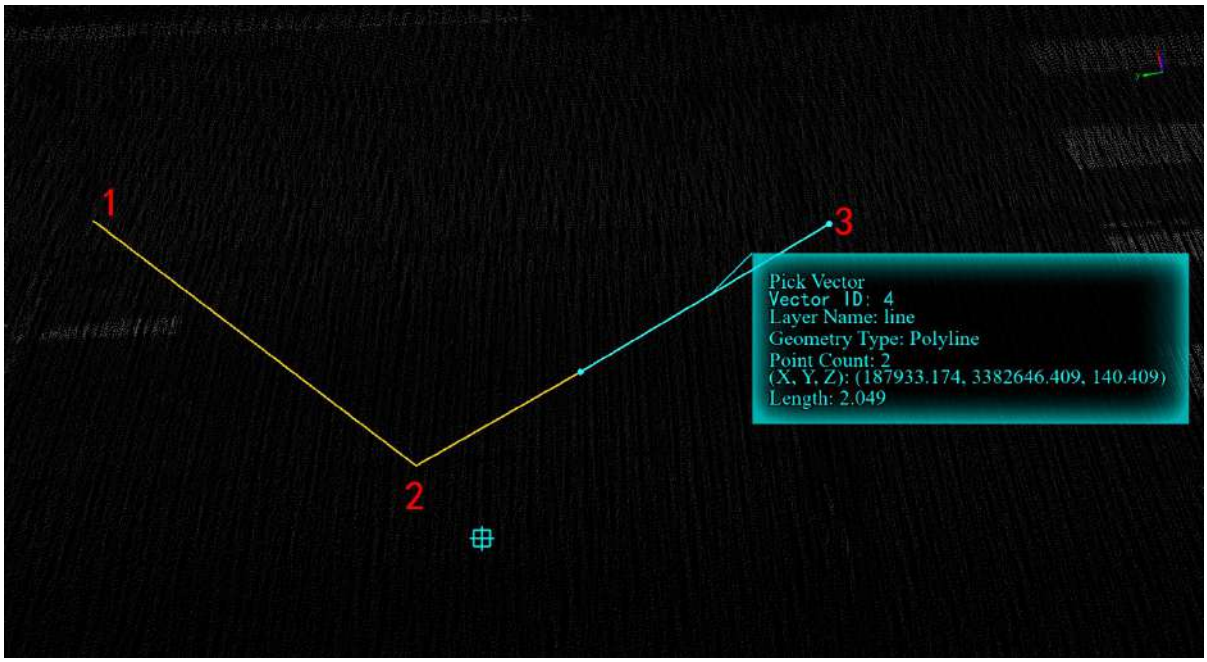
> If the node is selected, it will break from the node and change from a line string to two lines; if the mouse selected is not a node, insert a node at that position and break it into two lines.

- **Original polyline**: A polyline composed of three points.



- **Break at original node 2**: The two lines after the break are connected end to end at node 2.

- **Break at non-nodes**: Insert a node, and the two lines after the interruption are connected end to end at the inserted node.



Split Line

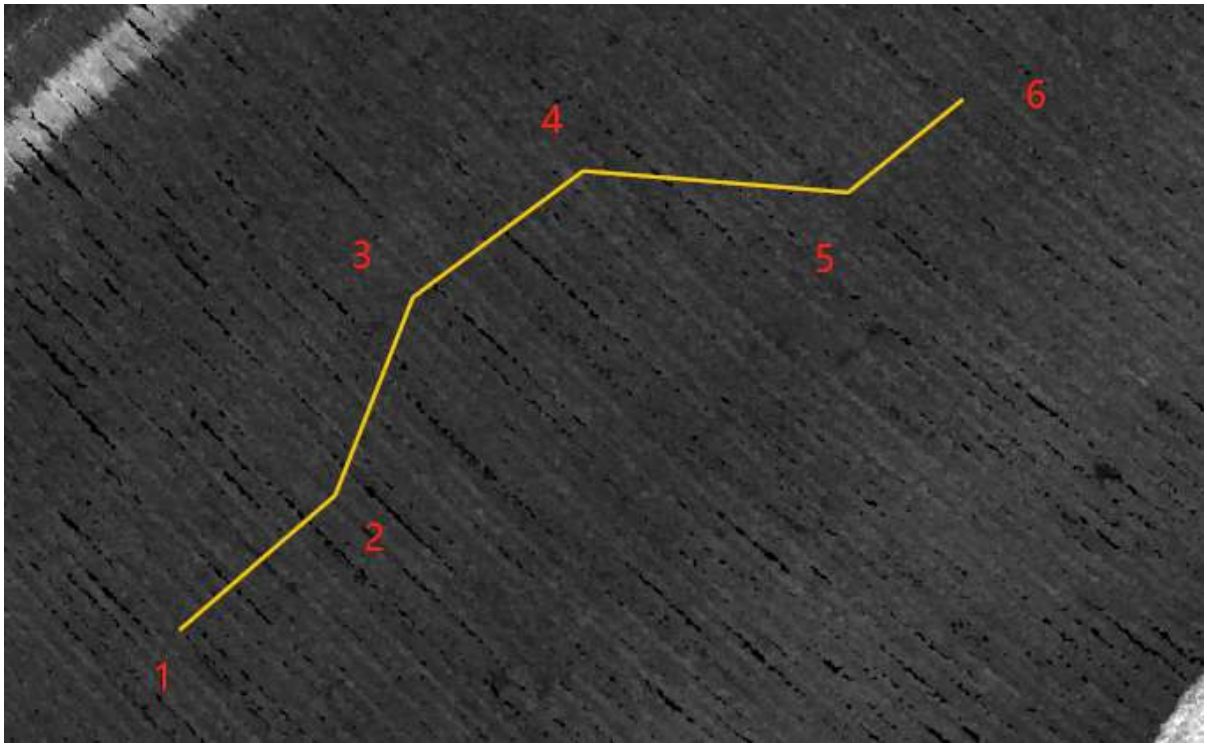
Function Description： The Split Line function allows the user to split a solid line object into a dashed line object. See the picture below for details.

Steps

1.Click **Split Line**  button in the Vector Editor.

2.Left click the line you want to split to complete the line segment splitting.

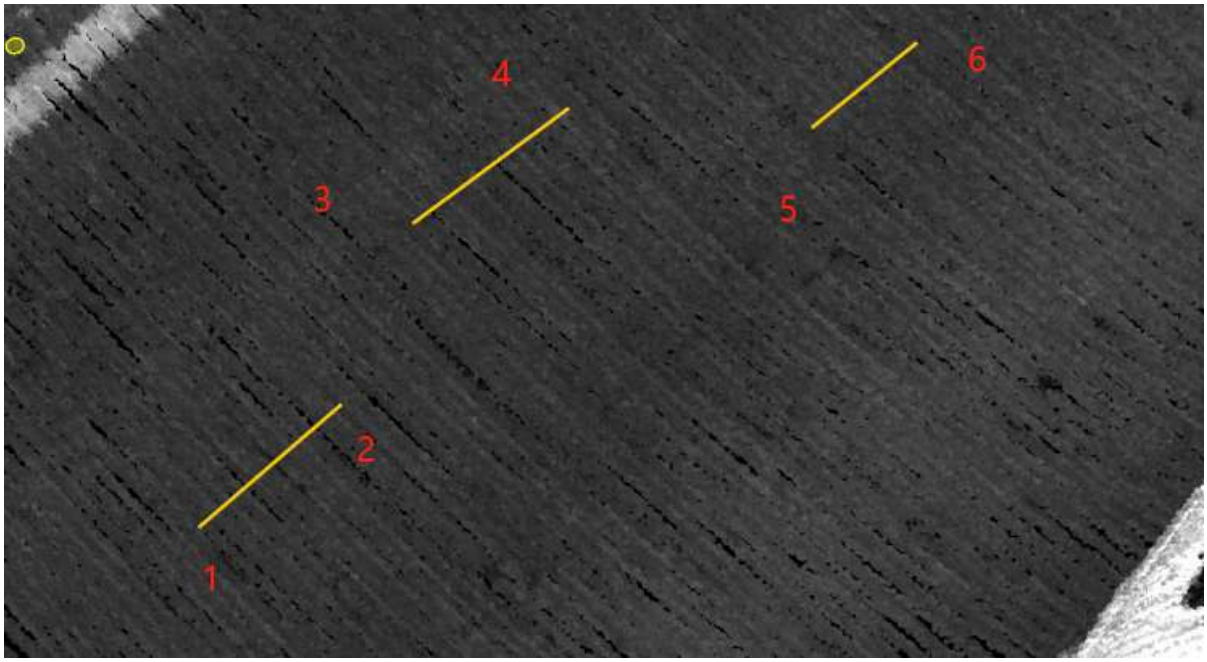
- **Before vector line splitting**: The original solid line consists of 6 nodes.



Original solid line

- **After the vector line is split**: The even-numbered node -> odd-numbered node part of the original solid line is interrupted and disappeared, thus realizing the function of splitting the original solid line into dashed lines.

> In this example: 1-2 nodes form a line segment, 3-4 nodes form a line segment, 2-3 parts are interrupted and disappear.



Split into segments

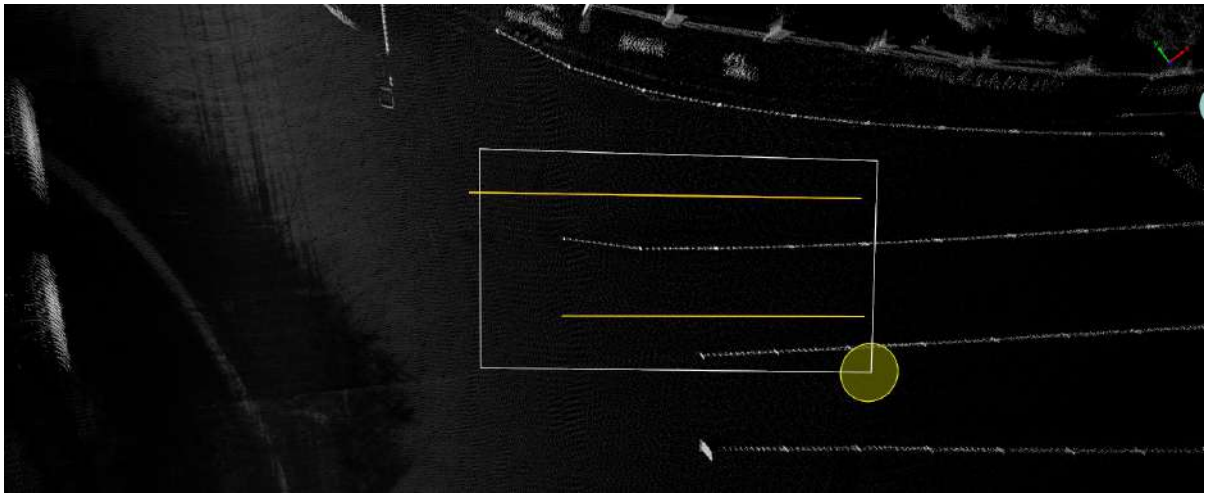
Extend or Trim

Function Description： Allows the user to modify a line, wherein a target line can be drawn by the user to stretch or cut the start or end of a series of lines.

Steps

1.Click **Extend or Trim**  button in the Vector Editor, the left mouse button uses a three-key point method on the screen to select the line object to be edited, as shown in the figure below.

> Note: The white quadrilateral is the selection box, and the line objects located inside the selection box or intersecting the selection box will be selected (highlighted in red).



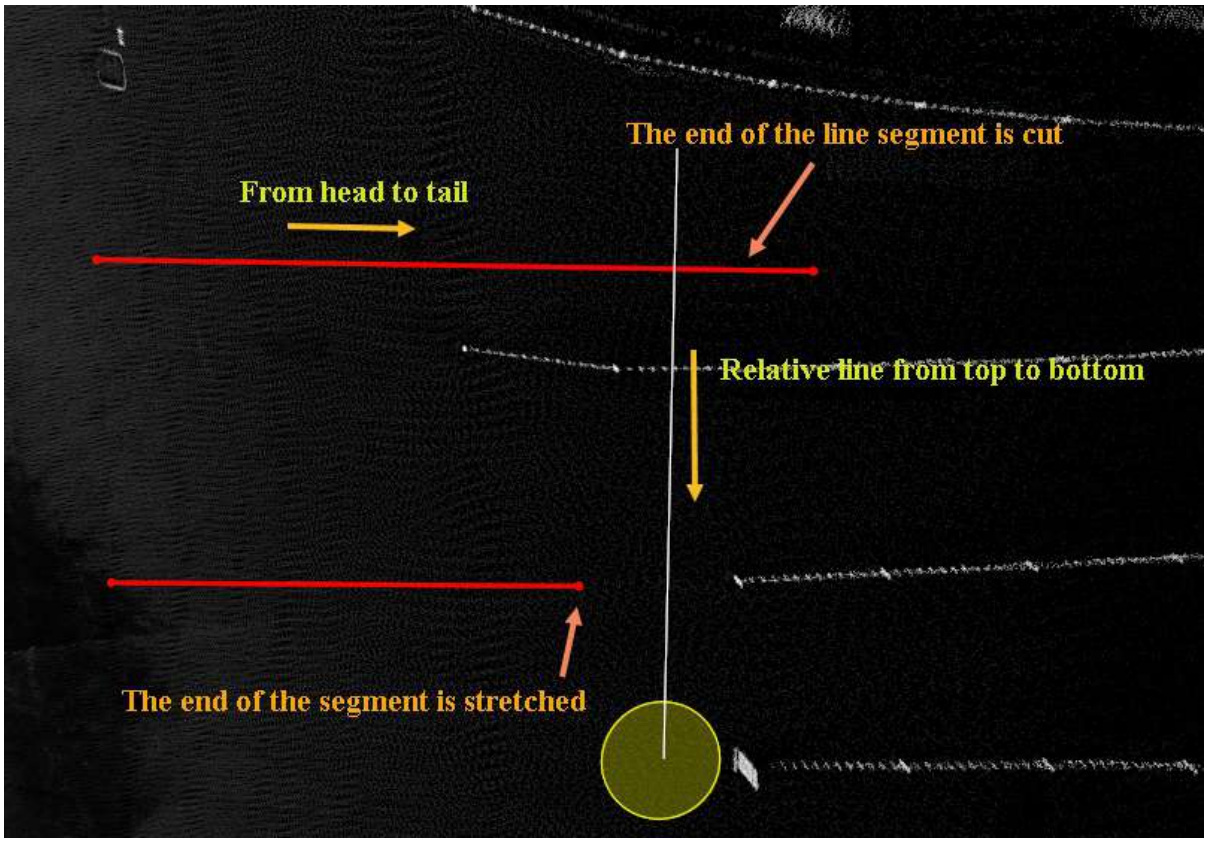
Frame selection of line objects

2.Right-click to select two points one after the other to form a **cutting/stretching** reference line to complete the cutting or stretching. Refer to the stretching/cutting principle below for details.

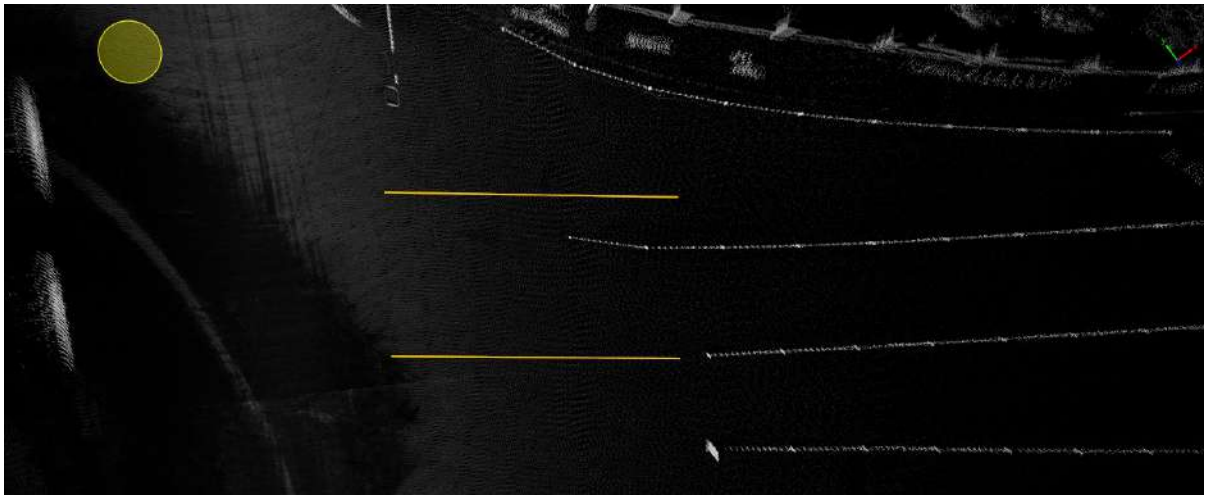
3.By clicking the shortcut key G and then performing the cutting/stretching operation, the line after cutting/stretching can be affixed to the ground.

Cutting/Stretching Principle

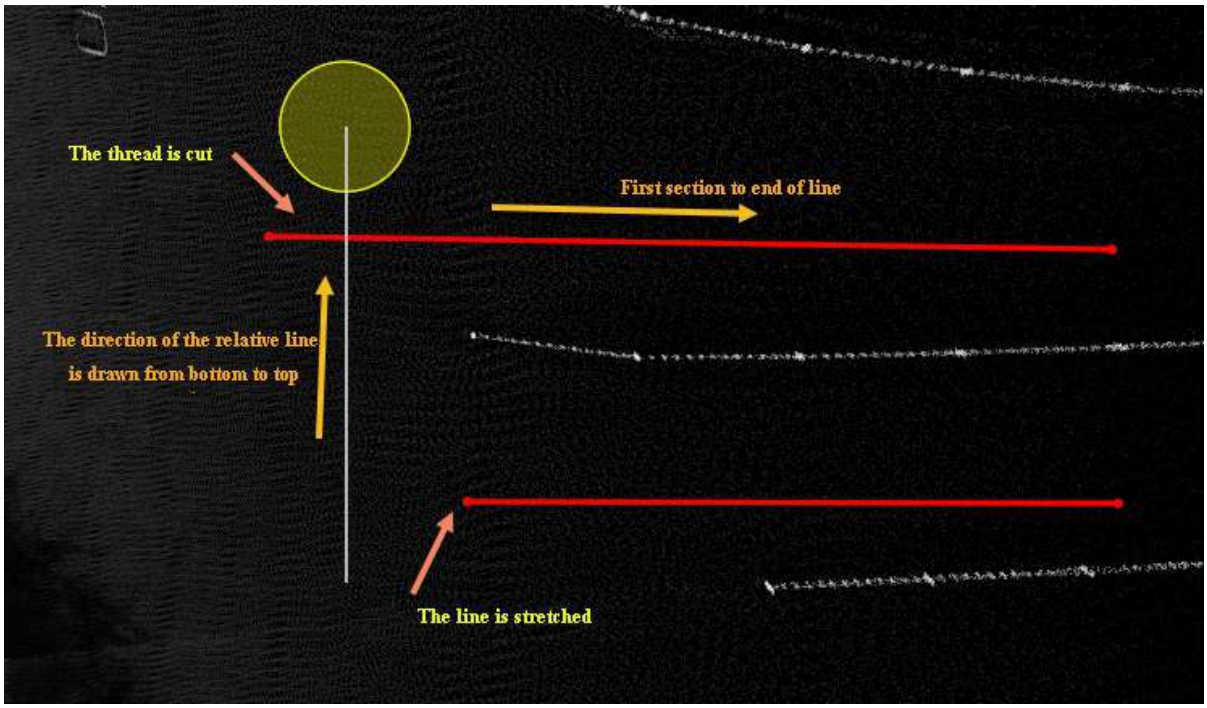
- Draw from left to right relative to the direction of the selected line.



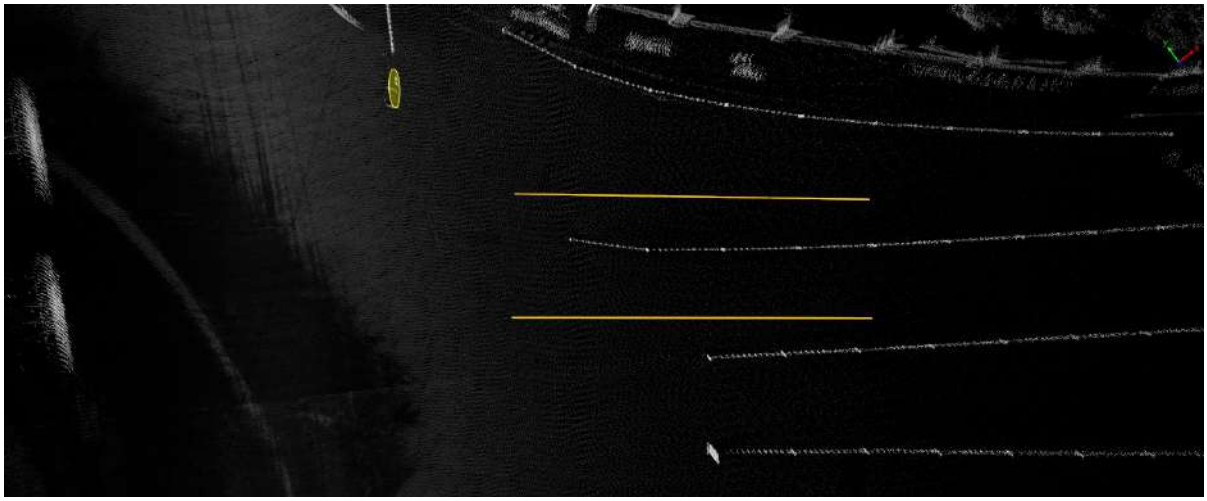
Draw the target line from left to right


Cutting/stretching result

- Draw from right to left relative to the direction of the selected line.



Draw the target line from right to left



The target line is drawn from right to left, the first end of the selected line is cut or stretched as a result

Split and Merge Polygon

Function Description： This function allows the user to split a convex polygon into two convex polygons, or merge two convex polygons into a convex polygon vector object.

> Note: Only the polygon objects under the road surface layer are supported

Steps

1.Click **Split and Merge Polygon**  button in the Vector Editor, the operation mode selection dialog box will pop up.

The pop-up window selects the current operation as cutting or merging

Parameter Description:

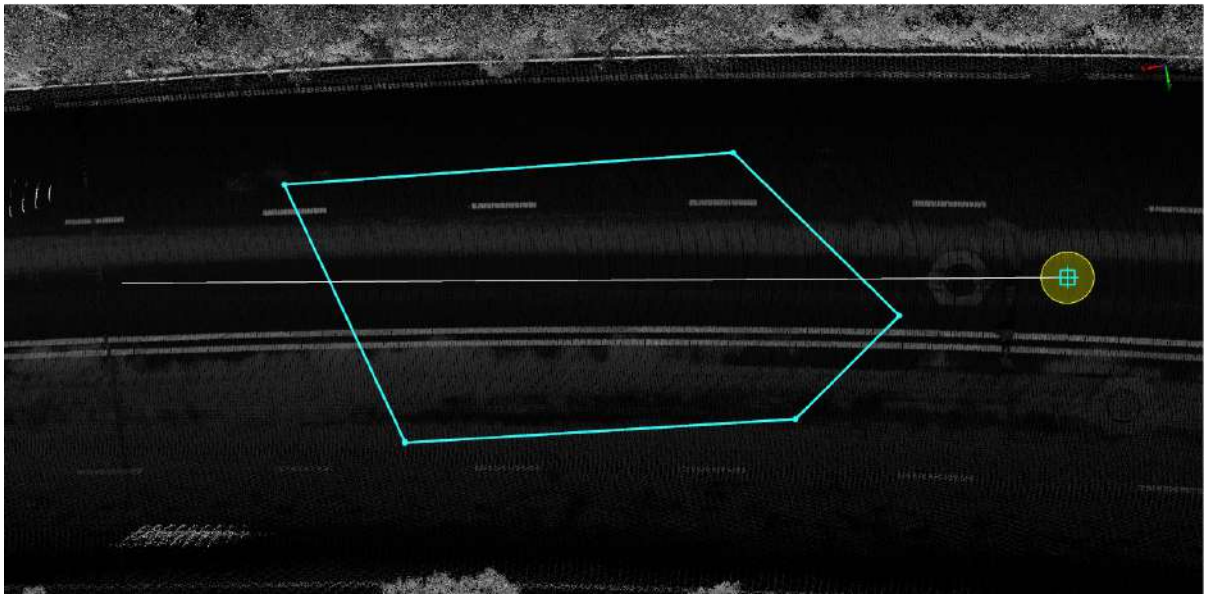
- **SplitPolygon**: Split mode.
- **MergePolygon**: Merge mode.

SplitPolygon

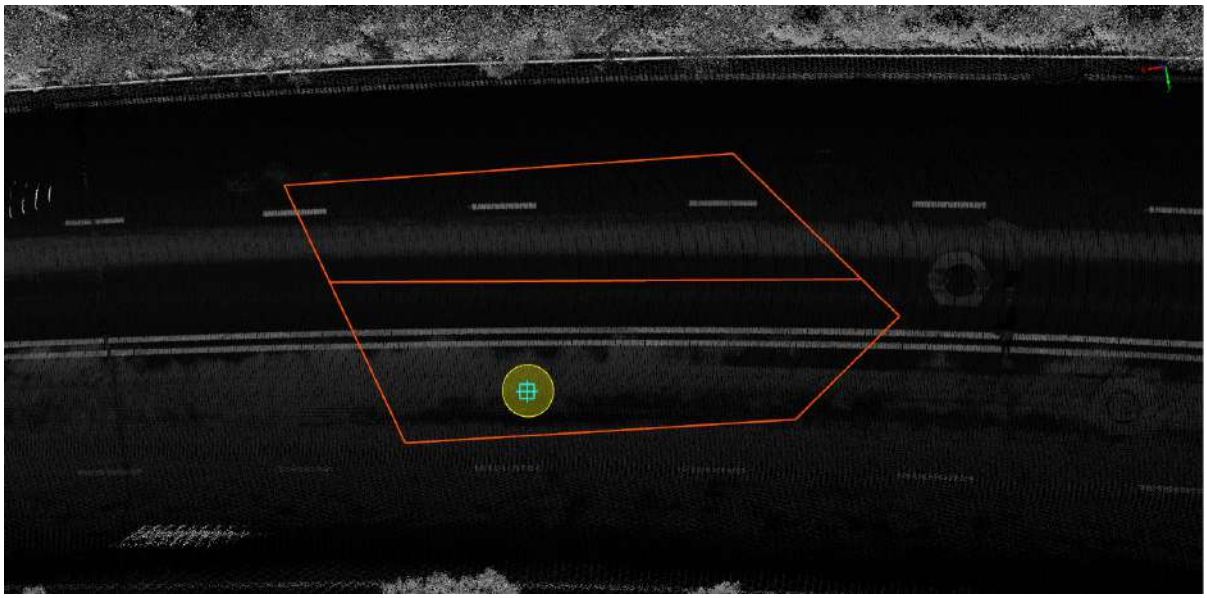
1.Left click to select the polygon to be split, and the polygon will be highlighted in blue.

2.Right click two key points to form a cutting line, as shown by the white line in the figure below, to complete the cutting of the polygon.

> Note: The cutting line needs to intersect the two sides of the polygon.



Draw cutting line

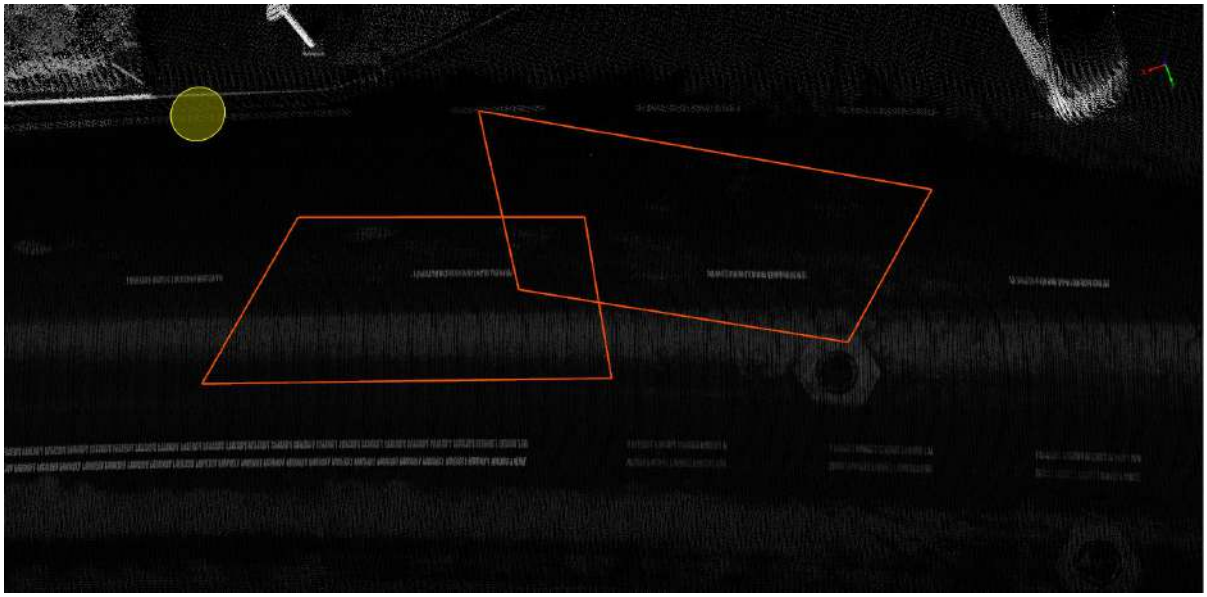


Cutting result

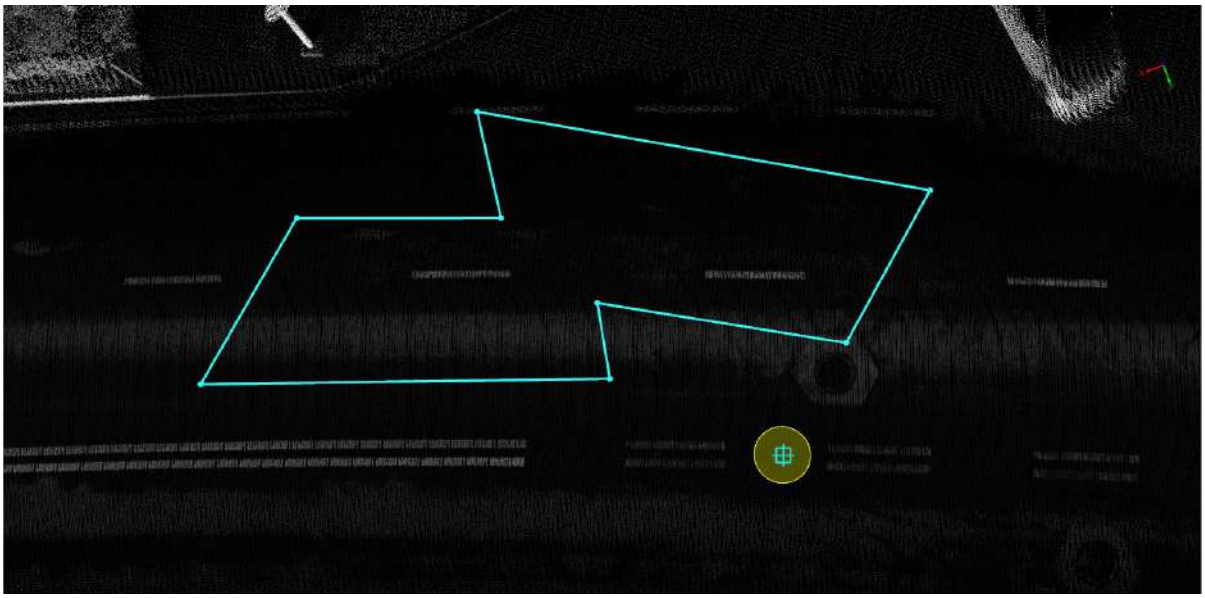
MergePolygon

1.Left click on the two polygons intended to be merged to select, and complete the merging of the polygons.

> Note: The polygons that need to be merged need to have intersections.



The two polygons before the merge



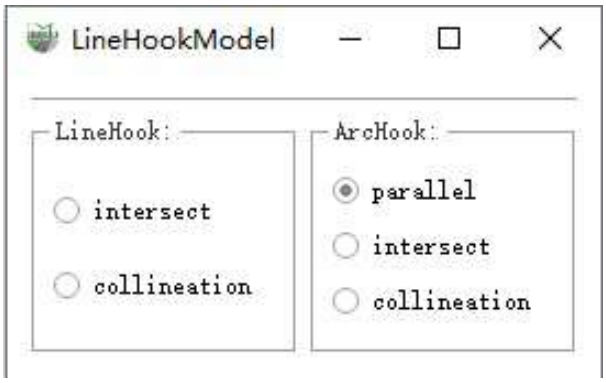
Merge result

Link Line

Function Description： This function allows the user to link lines by way of circular arc or straight line connection, where the relative position relationship of the vectors are collinear, parallel or intersecting.

Steps

1.Click **Link Line**  button in the Vector Editor, LineHookModel pop-up window will appear. Select the corresponding mode to perform the link operation and see the detailed description below for details.



The pop-up window selects the connection mode and the relative relationship of the lines

Parameter Description

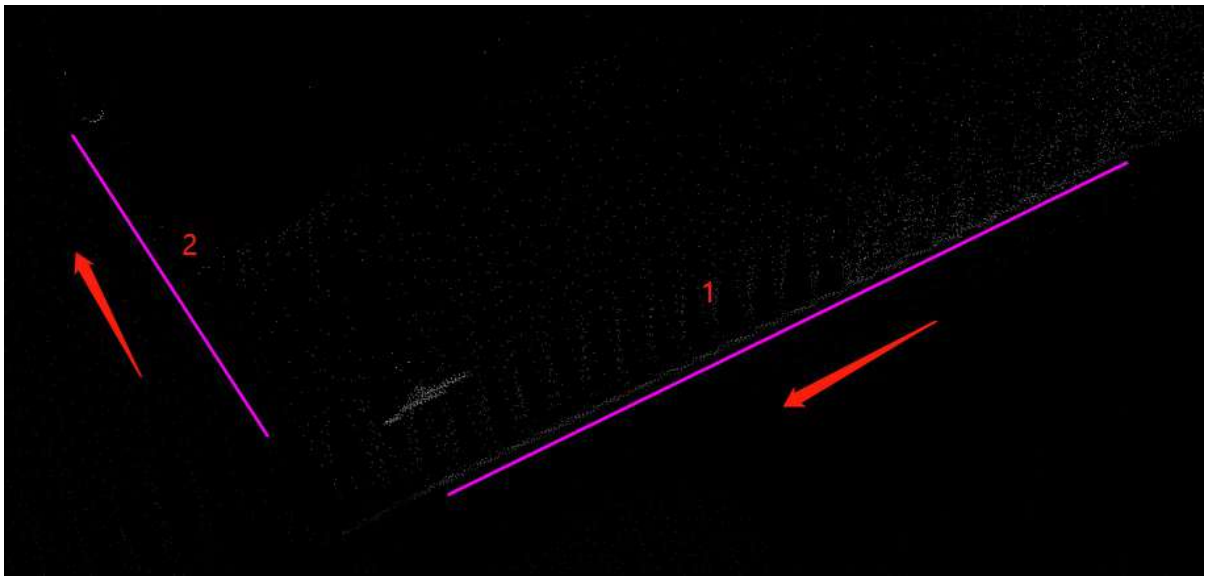
- **LineHook**: According to the spatial geometric relationship of the two lines, they are connected end to end or extended to intersect.
- **ArcHook**: According to the spatial geometric relationship of the two lines, the end points of the two lines plus a user-defined point are selected, a total of three points are used to determine a circular arc; or a point is selected by user-defined and connect the two lines in a circular arc tangent to the two lines.

2.**LineHook**

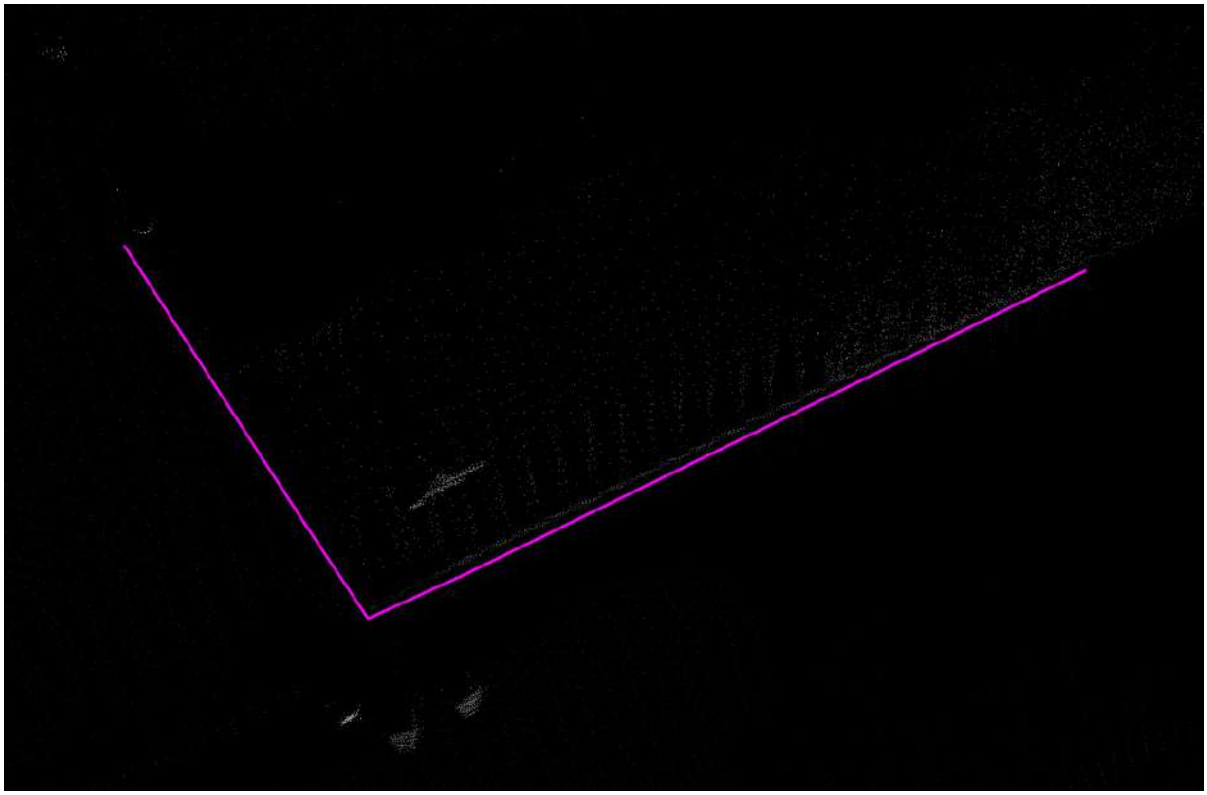
（1）**intersect**

Select **LineHook**, and the relative relationship of the lines is **intersect**, click the left mouse button one by one, and select the two lines that need to be linked to complete the automatic linking. Among them, the order of clicking must be consistent with the direction of the line. As shown in the figure below, you need to **first click** line 1 and then **click** line 2 to achieve the desired effect.

> Note: Intersect means that the straight lines passing through two line segments can intersect at one point.



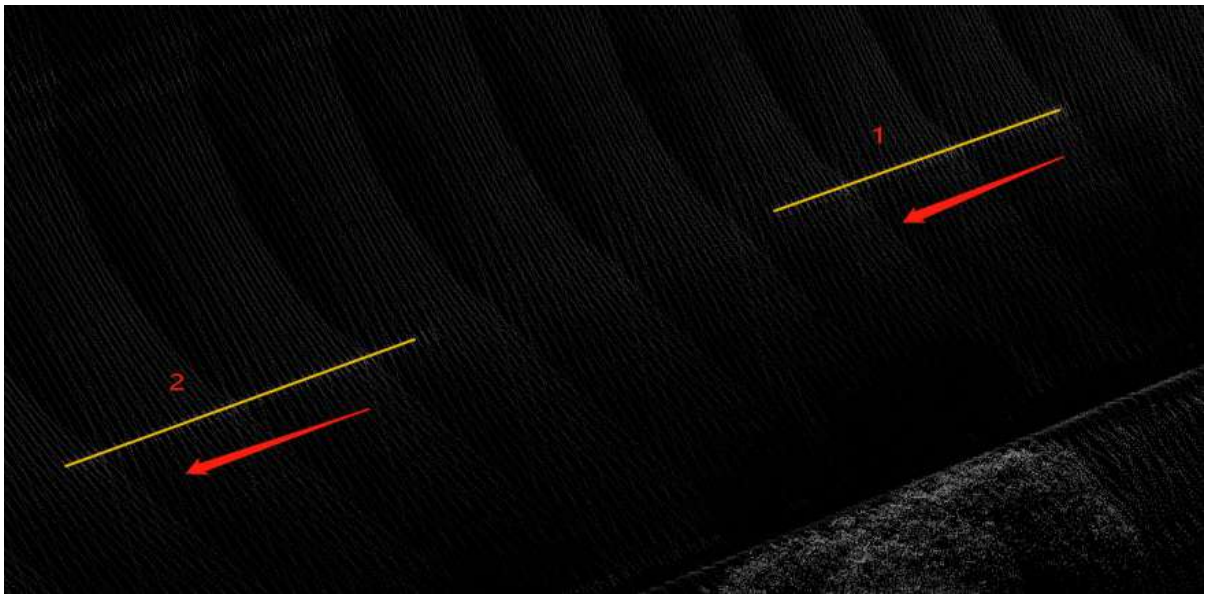
The two lines before linking


Linking result

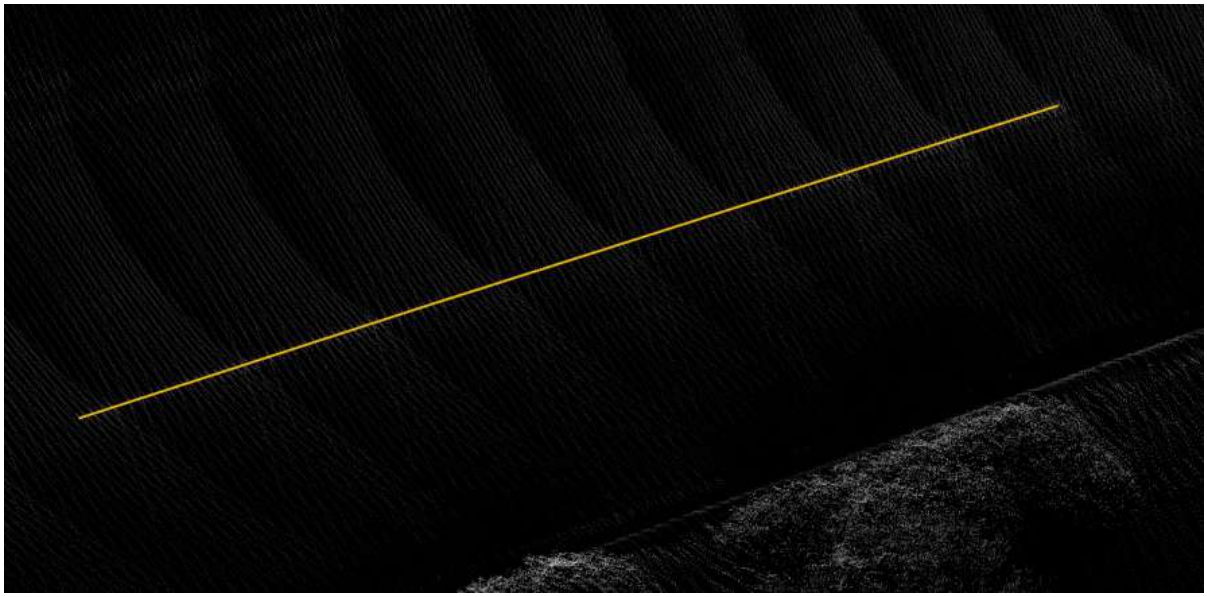
（2） **collineation**

Select **LineHook**, and the relative relationship between the lines is **collineation**, click the left mouse button one by one, and select the two lines that need to be connected to complete the automatic connection. Among them, the order of clicking must be consistent with the direction of the line. As shown in the figure below, you need to **first click** line 1 and then **click** line 2 to achieve the desired effect.

> Note: Collineation means that the straight line passing through the two line segments is approximately on a straight line.



The two line before linking

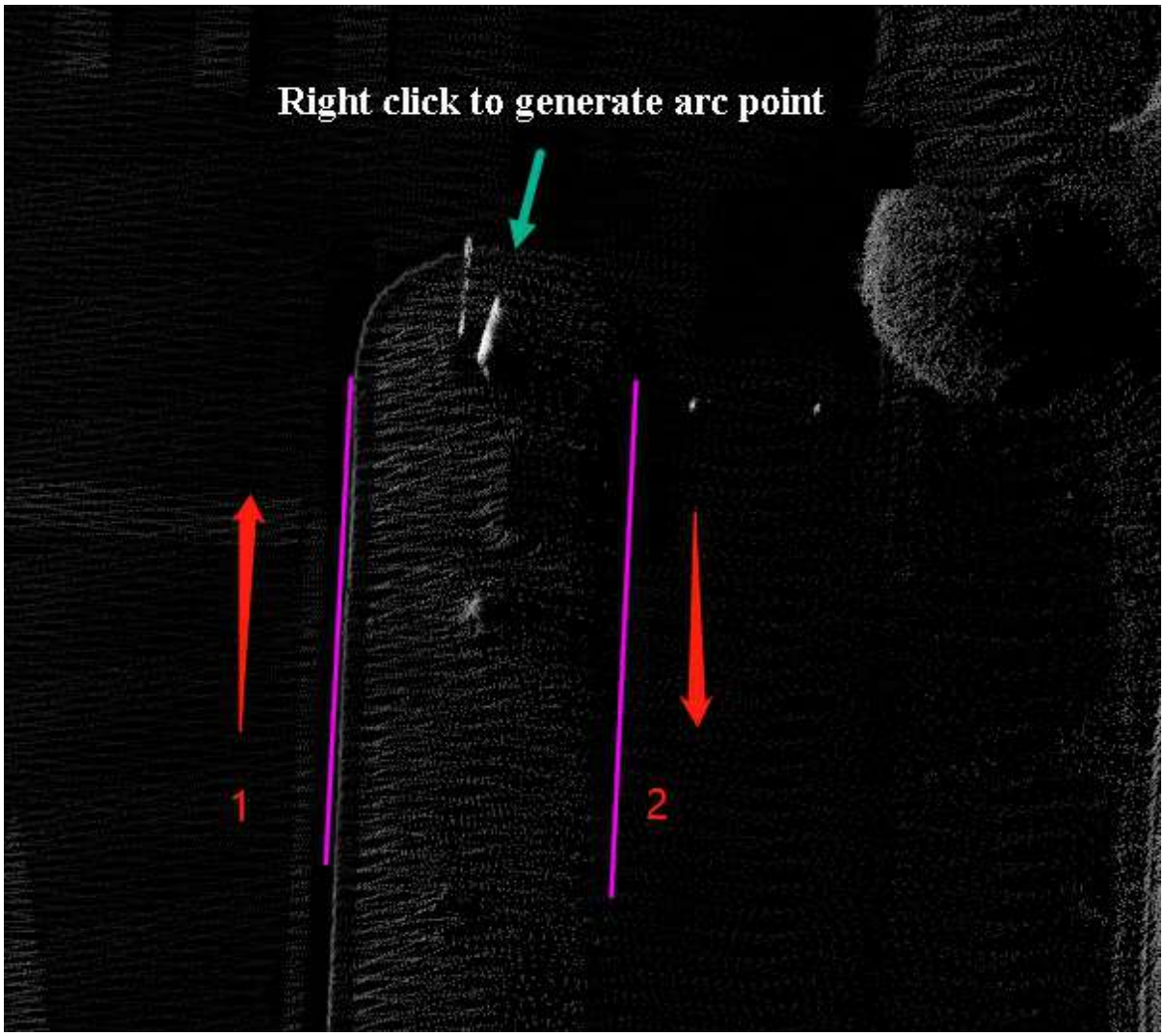

Linking result

3.**ArcHook**

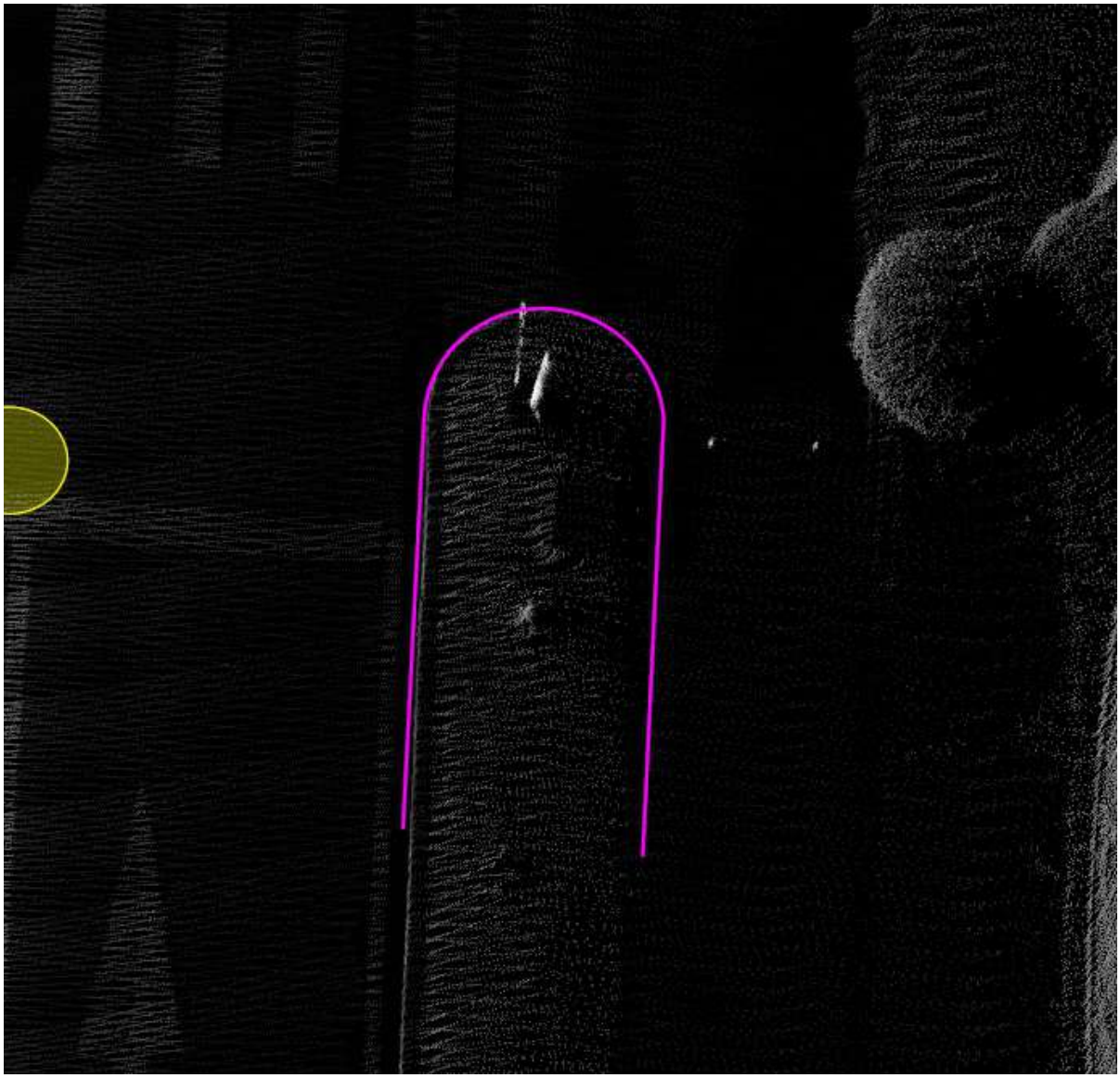
（1）**parallel**

Select **ArcHook**, and the relative relationship of the lines is **parallel**, click the left mouse button in turn to select the two lines to be hooked, and the order of clicking must be consistent with the direction of the line, as shown in the figure below, you need to **first click** line 1 and then line 2 to achieve the desired effect. Finally **right-click** to select the key point of the arc to be generated, and the arc hook in **parallel mode** can be completed.

> Note: Parallel means that two lines are approximately parallel in space.



The two lines before linking

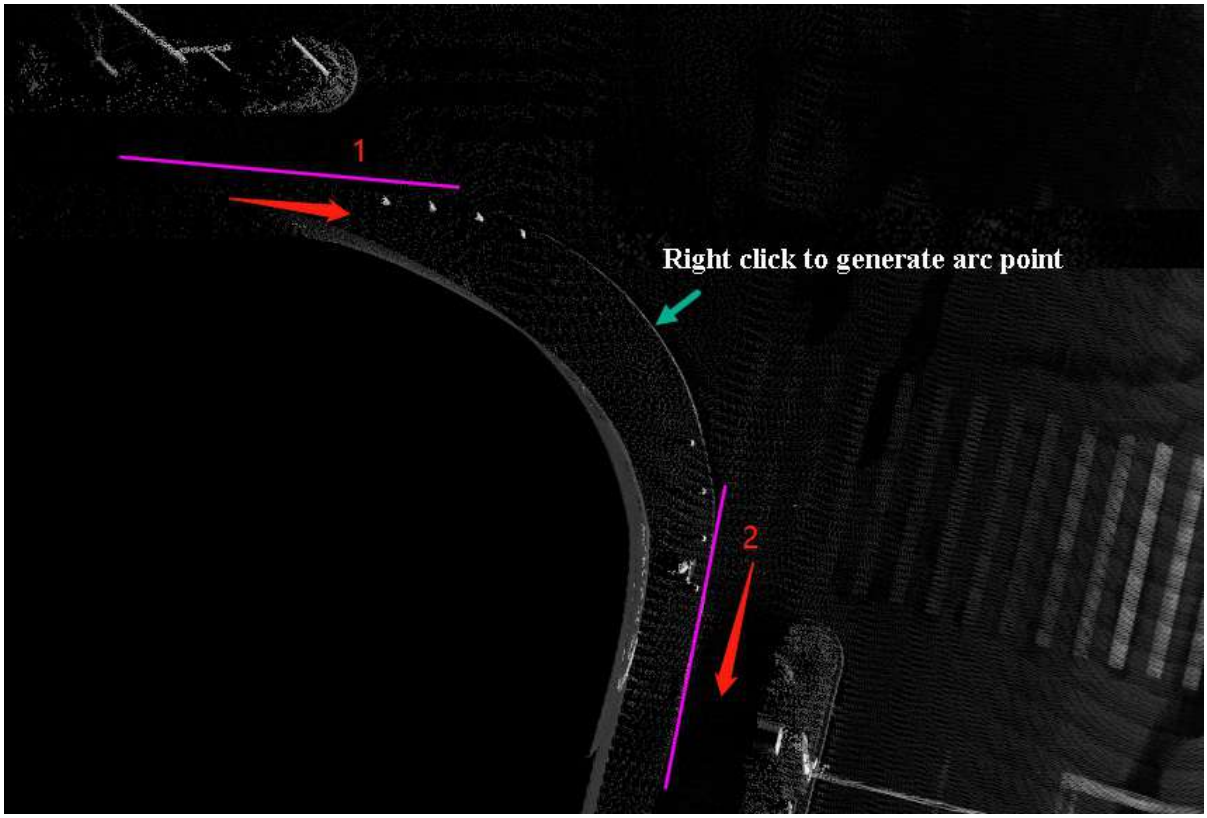


Linking result

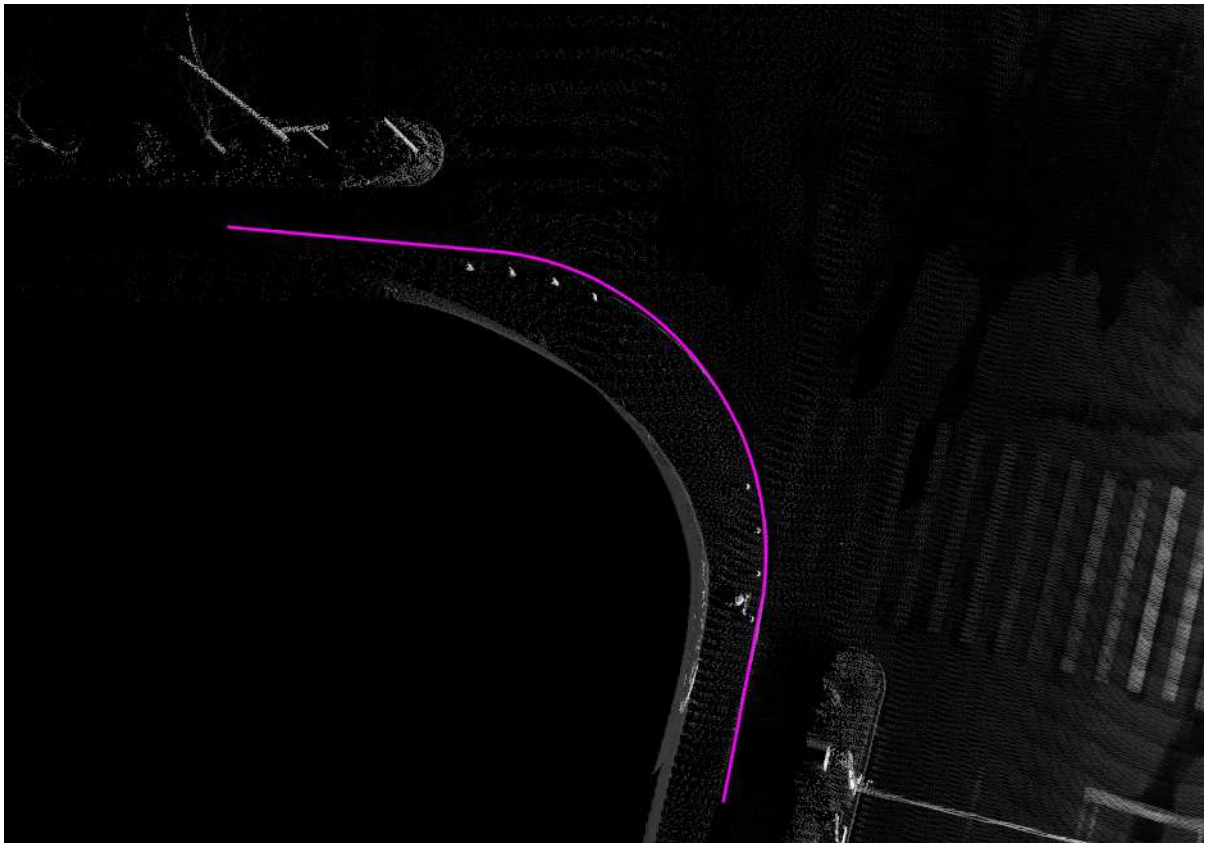
（2）**intersect**

Select **ArcHook**, and the relative relationship of the lines is **intersect**, click the left mouse button in turn to select the two lines to be hooked, and the order of clicking must be consistent with the direction of the line, as shown in the figure below, you need to **first click** line 1 and then line 2 to achieve the desired effect. Finally **right-click** to select the key point of the arc to be generated, and the arc hook in **intersection mode** can be completed.

> Note: Intersect means that the straight lines passing through two line segments can intersect at one point.



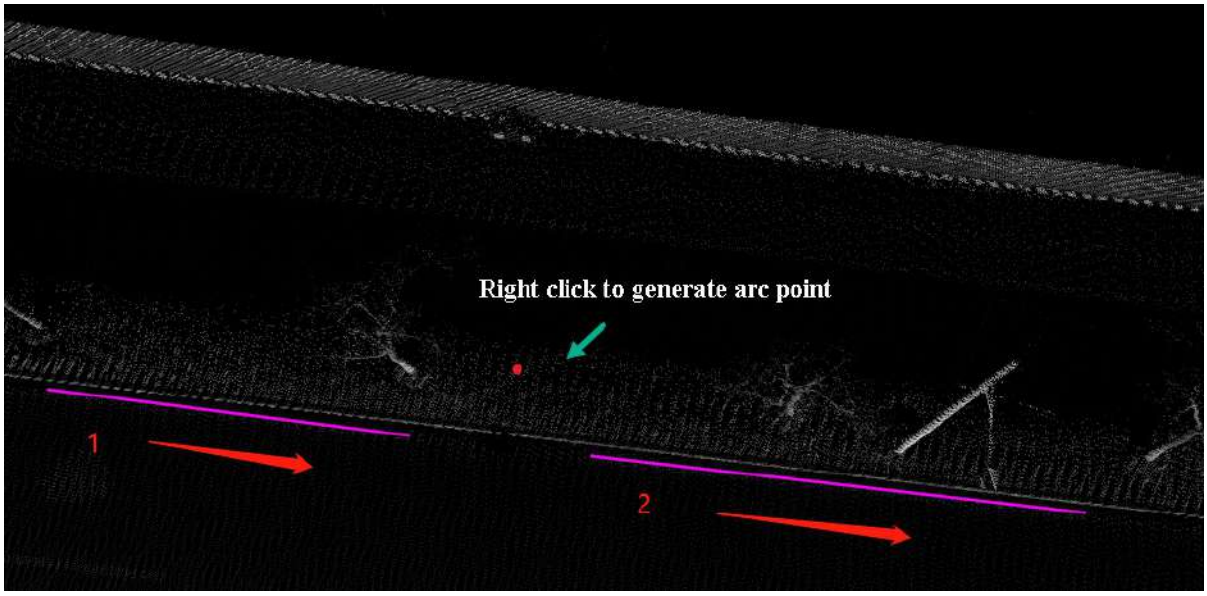
The two lines before linking


Linking result

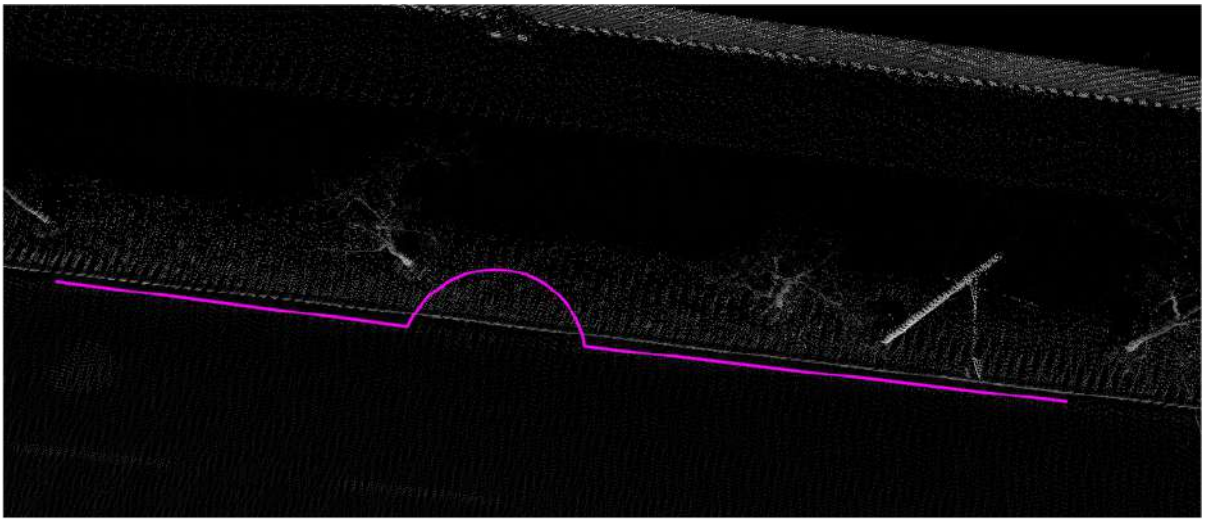
（3）**collineation**

Select **ArcHook**, and the relative relationship of the lines is **collineation**, click the left mouse button in turn, and select the two lines to be hooked. The order of clicking must be consistent with the direction of the line. As shown in the figure, you need to **first click** line 1 and then line 2 to achieve the desired effect. Finally **right-click** to select the key point of the arc to be generated, and the arc hook in **collineation mode** can be completed.

> Note: Collineation means that the straight line passing through the two line segments is approximately on a straight line.



The two lines before linking



Linking result

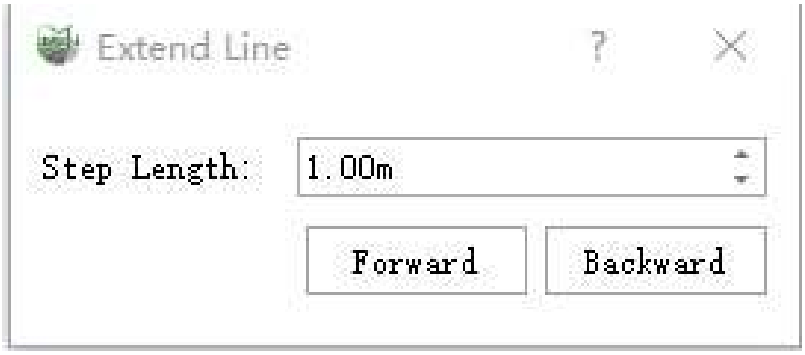
Extend Line

Function Description： The Extend Lines function allows the user to adjust the vector length to make the line object extend in the direction of the head or tail.

Steps

1.Click the **Extend Line**  button in the Vector Editor.

2.**Extend Lines** dialog box pops up.



Extend Lines dialog box

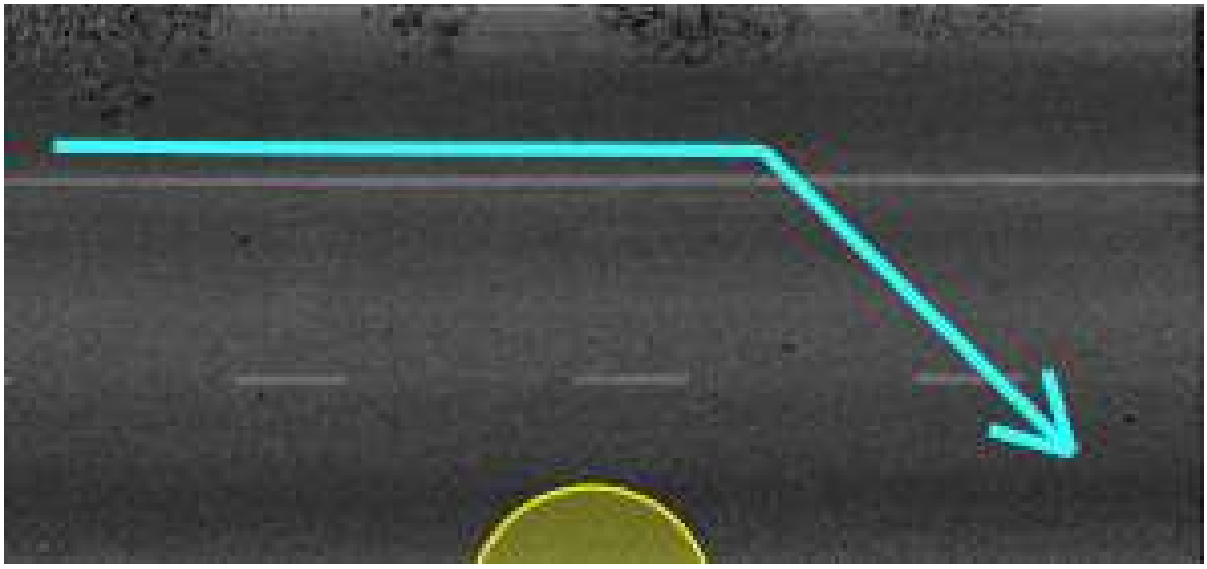
Parameter description:

Step Length: The default is 1.0 meters, which means the length of the line object to extend forward or backward each time.

> Note: When using shortcut keys, make sure that the 3D window is the current active window.

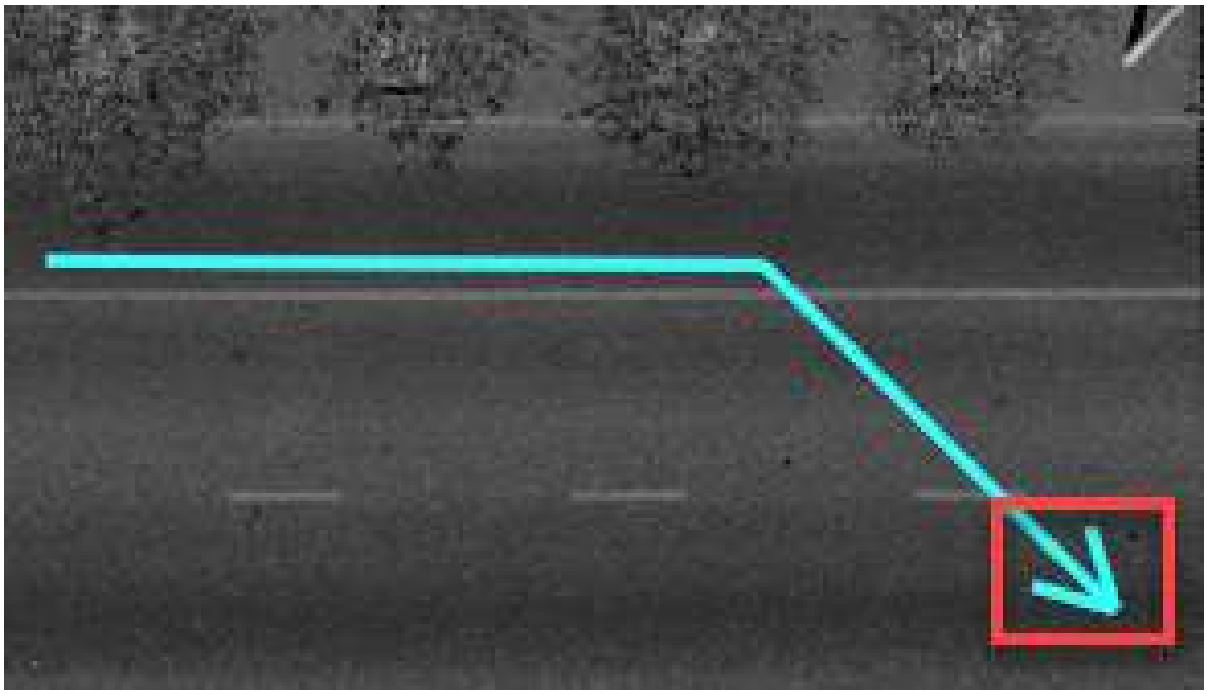
Shortcut keys	Function
F	Forward
B	Backward
Shift+Z	Fallback extension point
H	Hide shortcut key pop-up dialog box

3.Click the mouse to select the line object to be extended.

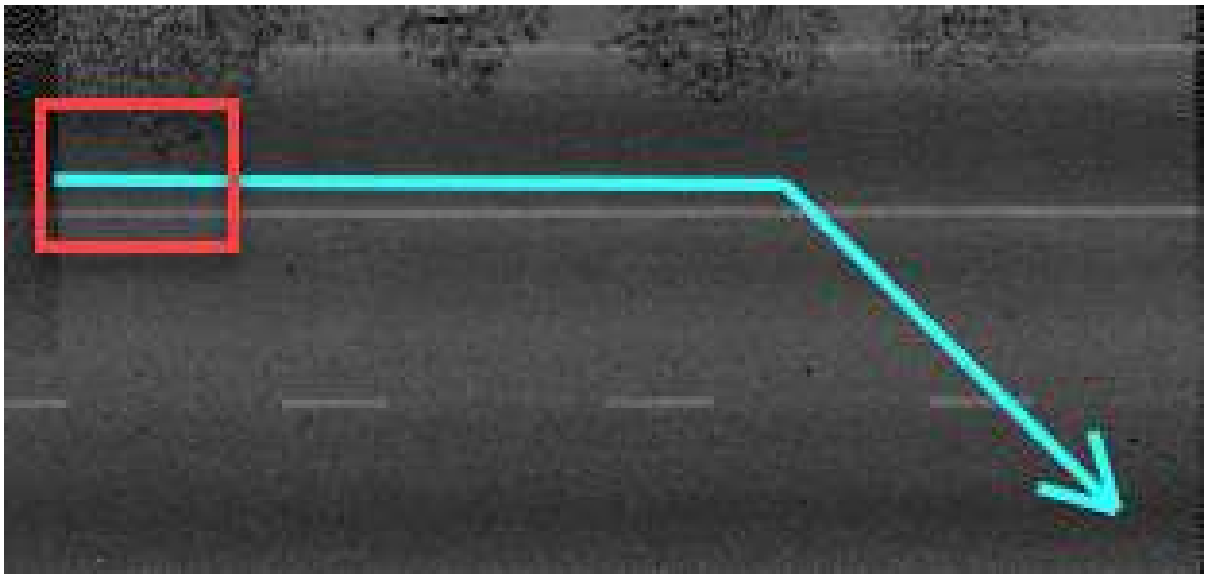


Select the line to be extended

4.Click the Forward or Backward button in the dialog box, and the line object will be extended in the corresponding direction.



Extend forward



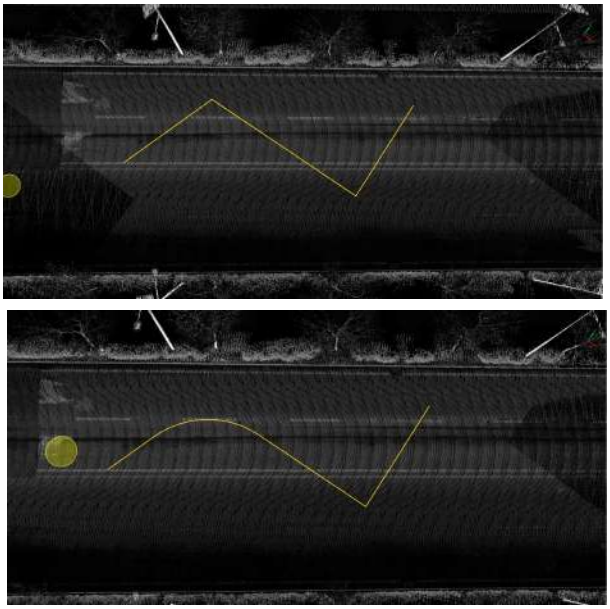
Rendering of extend forward

Fillet

Function Description: The Fillet function allows the user to adjust the size of the angle through the edge of the same node.

Steps

1.Click **Fillet**  button in the Vector Editor, click the left mouse button to click the two intersecting line segments consecutively, move the mouse, determine the needed angle according to the requirements, and double-click the mouse to complete.



Fillet

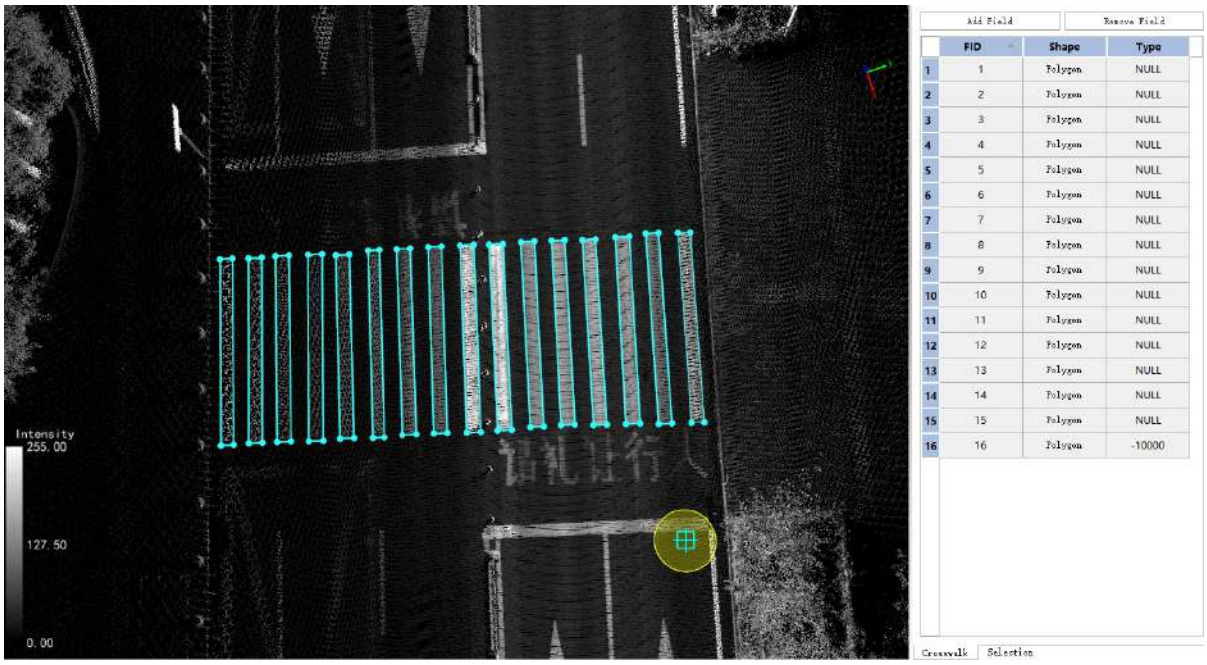
Split Mulitpolygon or Mulitpoint

Function Description: This function allows the user to split the vectors of the same layer into a single vector object.

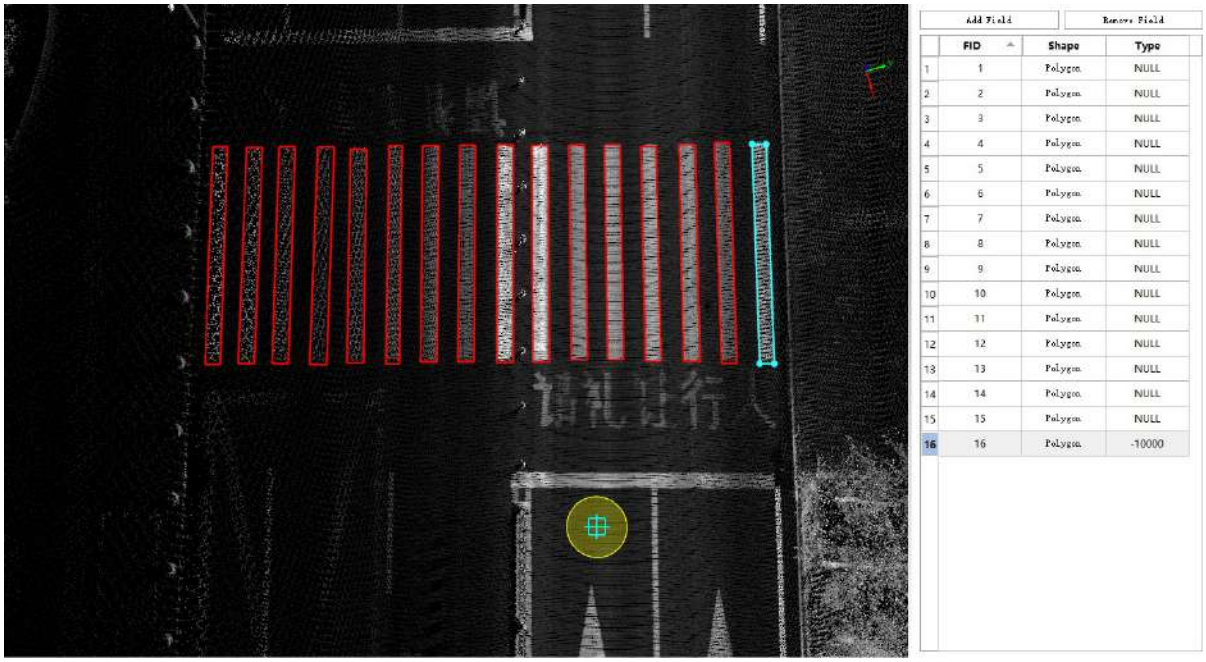
steps

1.Click **Split Mulitpolygon or Mulitpoint**  button in the Vector Editor, click the left mouse button and drag the mouse to select the vector object to be split.

> This function can only process the vectors in the same layer, the details are as follows:



Vectors to be split



Add Field | Remove Field

	FID	Shape	Type
1	1	Polygon	NULL
2	2	Polygon	NULL
3	3	Polygon	NULL
4	4	Polygon	NULL
5	5	Polygon	NULL
6	6	Polygon	NULL
7	7	Polygon	NULL
8	8	Polygon	NULL
9	9	Polygon	NULL
10	10	Polygon	NULL
11	11	Polygon	NULL
12	12	Polygon	NULL
13	13	Polygon	NULL
14	14	Polygon	NULL
15	15	Polygon	NULL
16	16	Polygon	-10000

The split vector

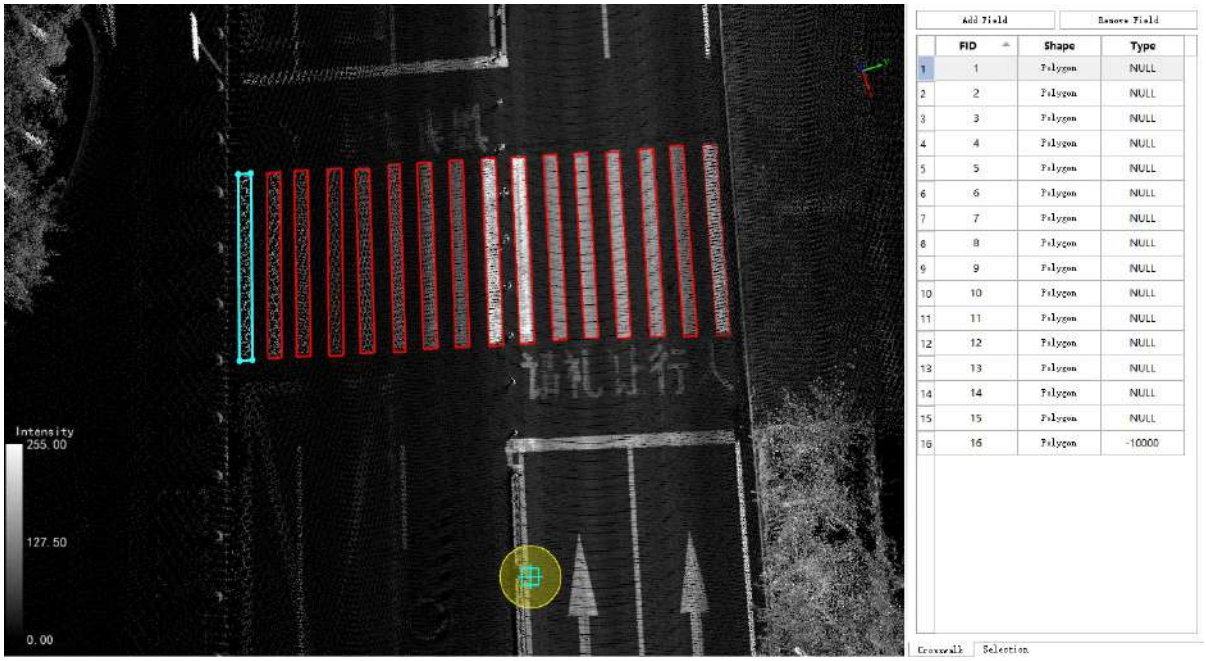
Combine Polygons or Points

Function Description: This function allows the user to combine different vectors in the same layer to form a vector object.

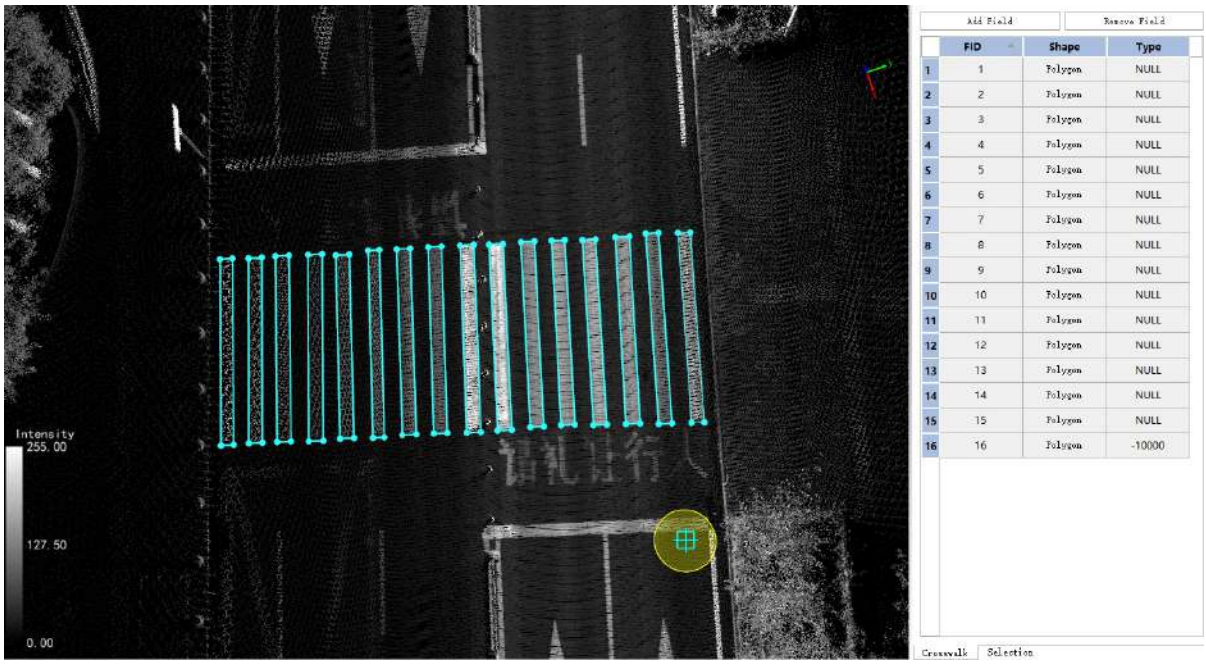
Steps

1.Click **Combine Polygons or Points** button in the Vector Editor, click the left mouse button and drag the mouse to select the vector objects that need to be combined together.

> This function can only process the vectors in the same layer, the details are as follows:



Vectors to be combined



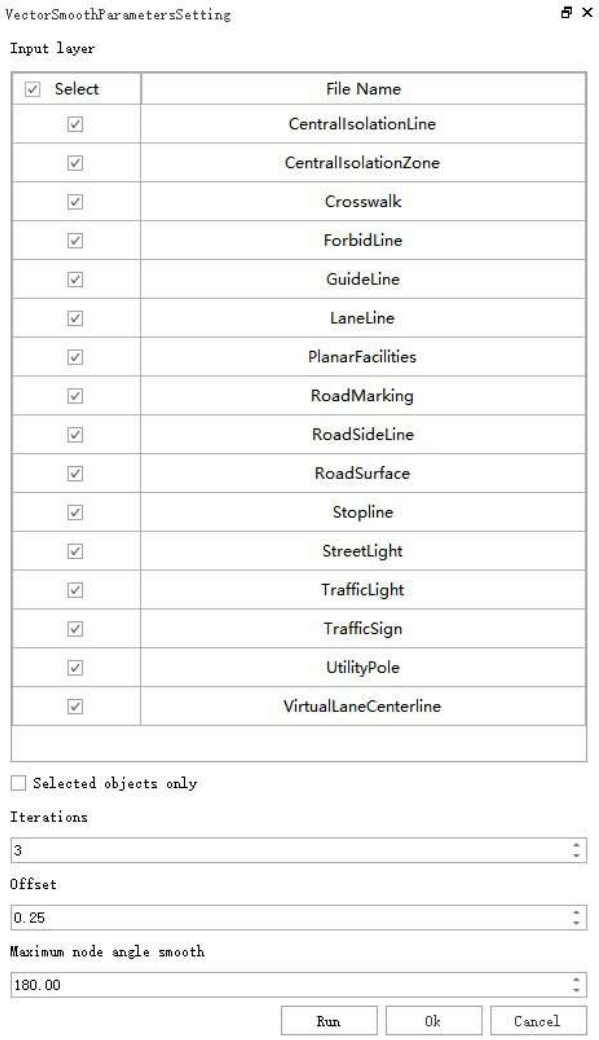
The combined vector

Smooth

Function Description：Make unsmooth vector objects spreading and smooth.

Steps

1.After clicking the **Smooth** button in the vector editor，Pop-up box for setting vector smoothing parameters.

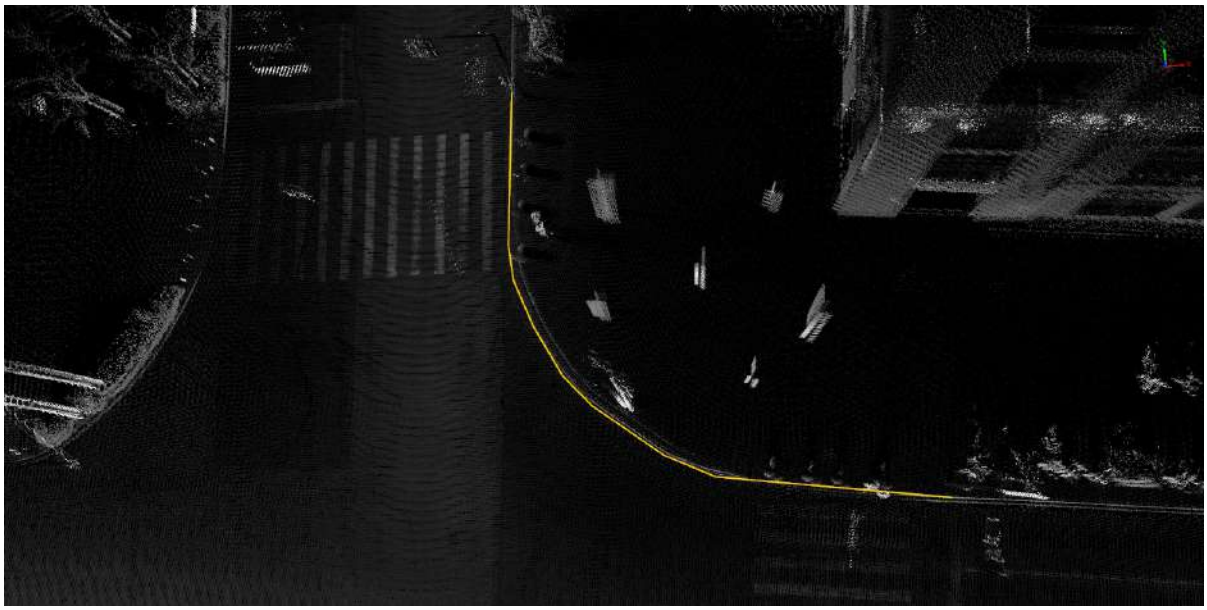


Parameter Description

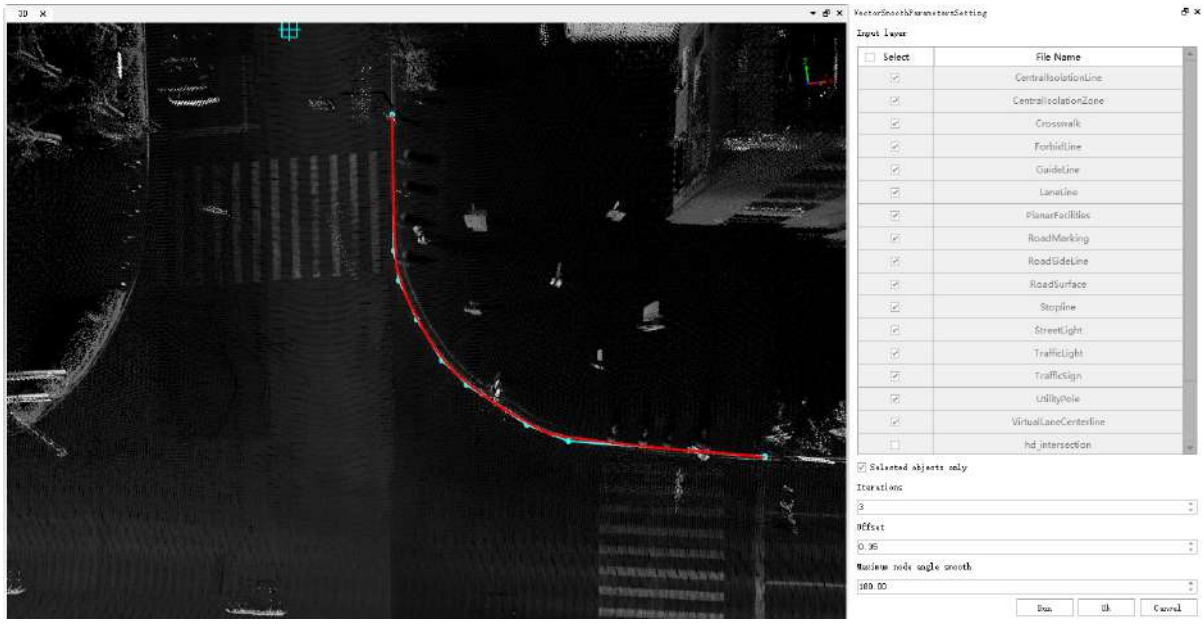
- **Input layer**：Select layers according to requirements
- **Selected objects only**：After checking the box, click on the object you want to change
- **Iterations**：Default iteration 3. Up to 100 iterations. Set the number of iterations as required
- **Offset**：Default offset 0.25. up to 10. the distance between the actual address of the memory cell and the segment address of the segment it is in
- **Maximum node angle smooth**：The default maximum smoothing angle is 180°. The minimum smoothing angle is 1°. Set the smoothing angle according to the requirements
- **Run**：Preview of the effect after setting parameters
- **OK**：After setting the parameters and previewing the effect, click the OK button to save the settings

- **Cancel**：Cancel Settings

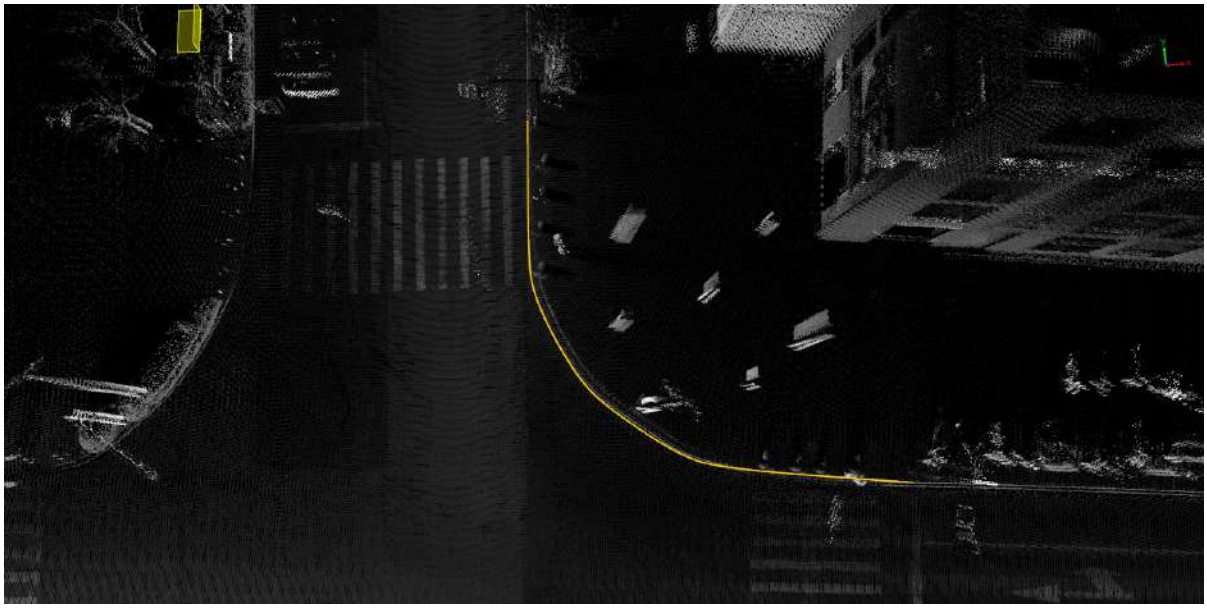
2.The result of smoothing is shown in the following figure.



Vector to be smoothed



Preview effect



Vector after smoothing

Auxiliary

- Drape
- Add Trace Line
- Add Trace Polygon
- Draw By Intersection
- Draw By Perpendicular

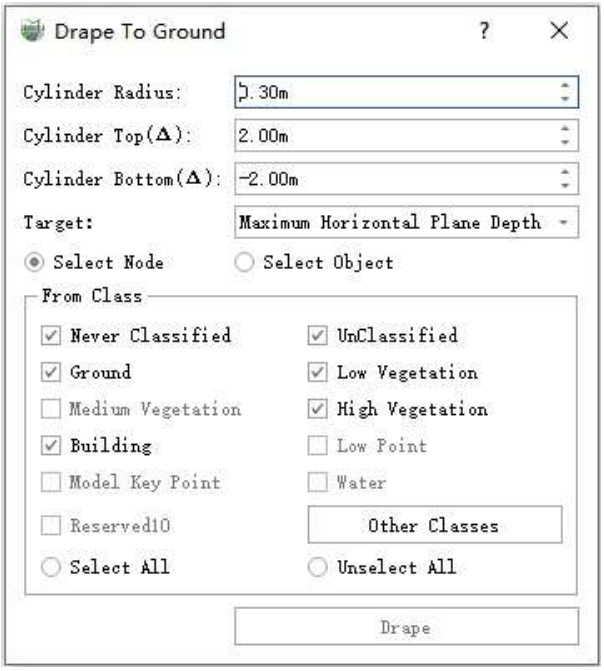
Drape To Ground

Function Description： Fit the vector object or part of the node of the vector object to the ground setting position.

Steps

1.Click the **Drape To Ground**  button in the vector editor.

2.Pop up **Drape To Ground**



Drape To Ground

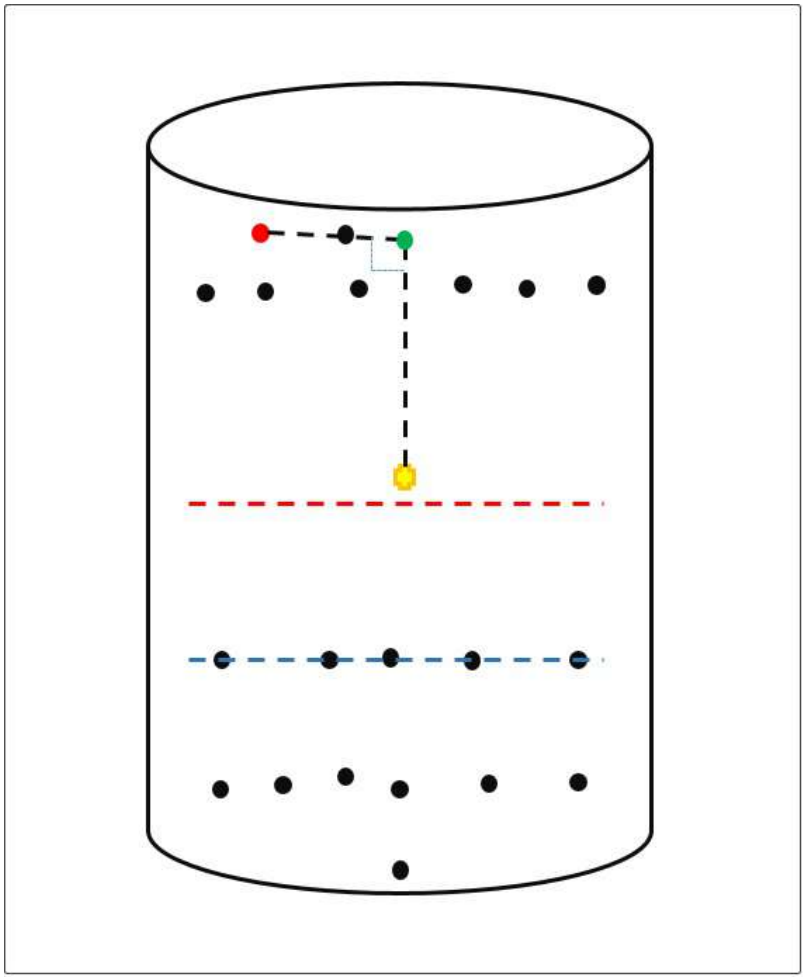
Parameter Description:

The Drape To Ground function, within a cylinder, performs the search and calculation of the point cloud to achieve the ground-fitting. The cylinder, with the principal axis direction parallel to the Z-axis and the center of the principal axis as the vector node in the current calculation, is referred to hereafter by the search cylinder as a surrogate for the cylinder.

- **Cylinder Radius：** Default is 0.3 m. Search for the radius of the column.
- **Cylinder Top(Δ)：** The default is 2.0 m, which means the upper surface is located 2.0 m above the current node, and the relative height of the upper surface of the search cylinder to the current node. Can be negative, which means the upper surface is located below the current node.
- **Cylinder Bottom(Δ)：** The default is -2.0 m, which means the lower surface is located 2.0 m below the current node, and the relative height of the lower surface of the search cylinder to the current node. Can be a positive value, which means the lower surface is located above the current node.

> Note: The upper surface value should be greater than the lower surface value

- **Target：** Default is the lowest depth, the current node needs to fit to the surface location, physical meaning see the search column schematic below
  - Minimum depth: without changing the XY value of the current node, the Z value takes the average Z value of the effective layer where the lowest point is located
  - Maximum depth: without changing the XY value of the current node, the Z value takes the average Z value of the effective layer where the highest point is located
  - Maximum point depth: without changing the XY value of the current node, the Z value is taken as the average Z value of the layer with the highest number of points in the search cylinder
  - Average depth: without changing the XY value of the current node, the Z value takes the average of the lowest depth and the highest depth
  - Median depth: without changing the XY value of the current node, the Z value takes the Z value of the middle valid layer in the search cylinder
  - Lowest point: move the current node to the lowest point within the search cylinder
  - Highest point: move the current node to the highest point of the search column



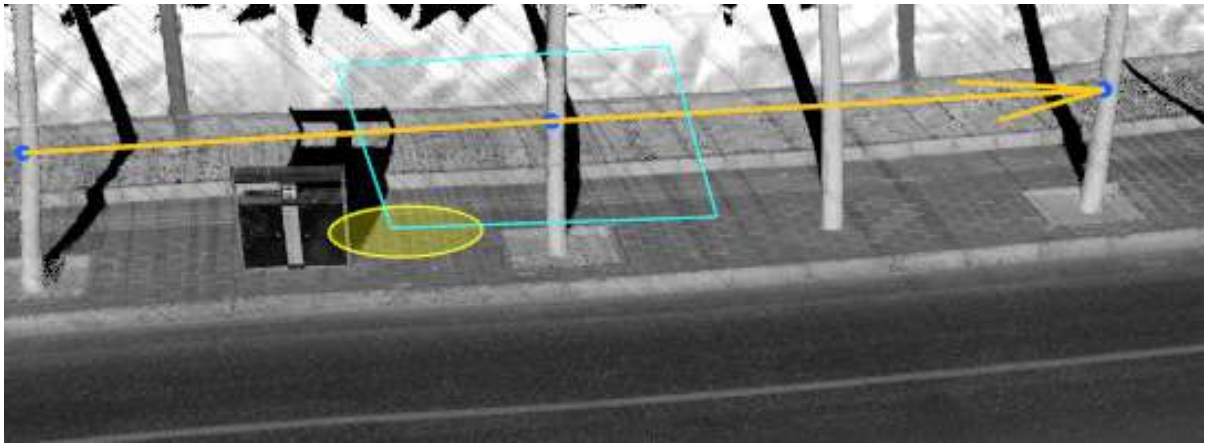
Schematic diagram for searching columns

The middle yellow point is the current node, the red point is the highest point, the green point is the virtual point represented by the highest depth, the red dashed line represents the average depth, and the blue dashed line represents the median depth

- **Select Node**：Polygon box selection method, select the area to be grounded, the nodes of vector objects located in the area will be grounded
- **Select Object**：Tap to select a vector object and fit all the nodes of the object, you can tap multiple objects and calculate at the same time
- **From Class**：Select the category point cloud to be used for the calculation
- **Drape**：The node selection method is not available. After selecting the object, click the button to start the fitting operation.

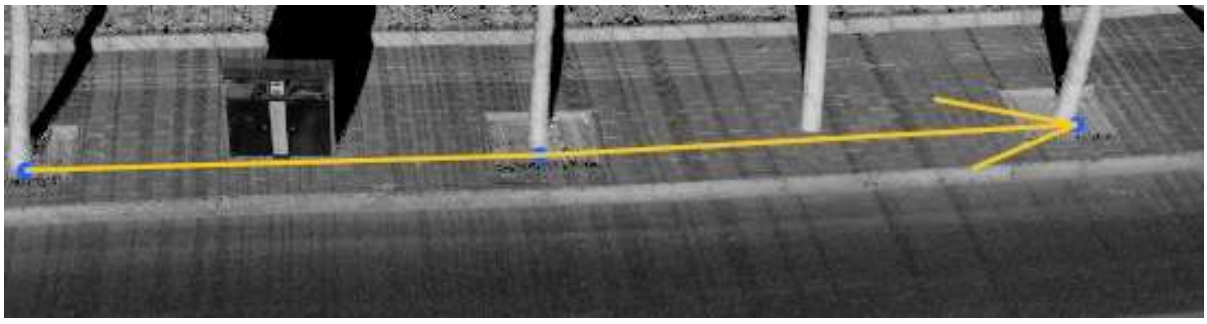
Select node mode

1.Mouse box the nodes that need to be grounded.



Check the nodes that need to be grounded

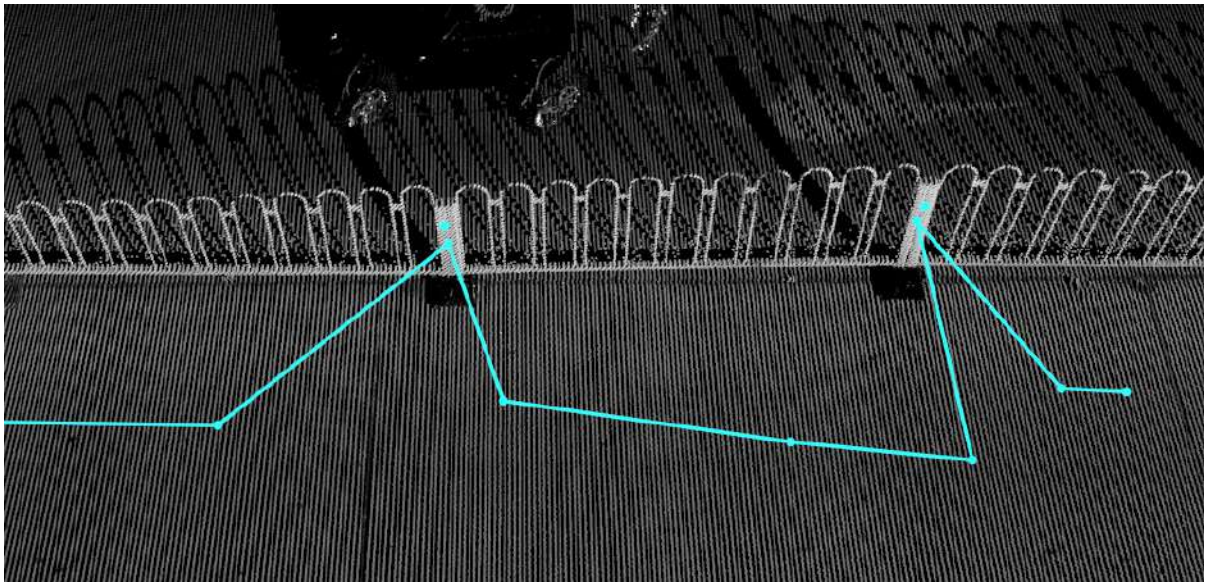
2.After the box is selected, double-click the left mouse button and the program will automatically calculate and ground the vector nodes in the range.



The effect of the node after grounding

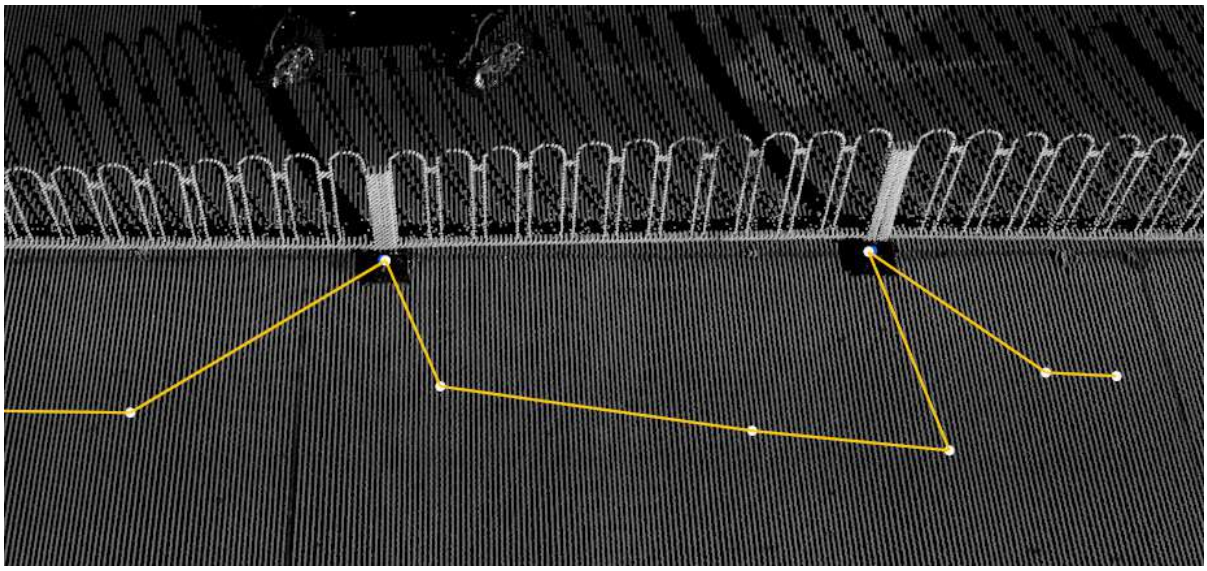
Select object method

1.Mouse click on the object to be grounded, click on the process, you can use the right mouse button, back a selection



Tap on the object to be grounded

2.Click the fit button to ground the selected object



The effect of the object on the ground

Add Trace Line

Function Description： This function allows the user to trace the vector in view to collect line features.

Steps

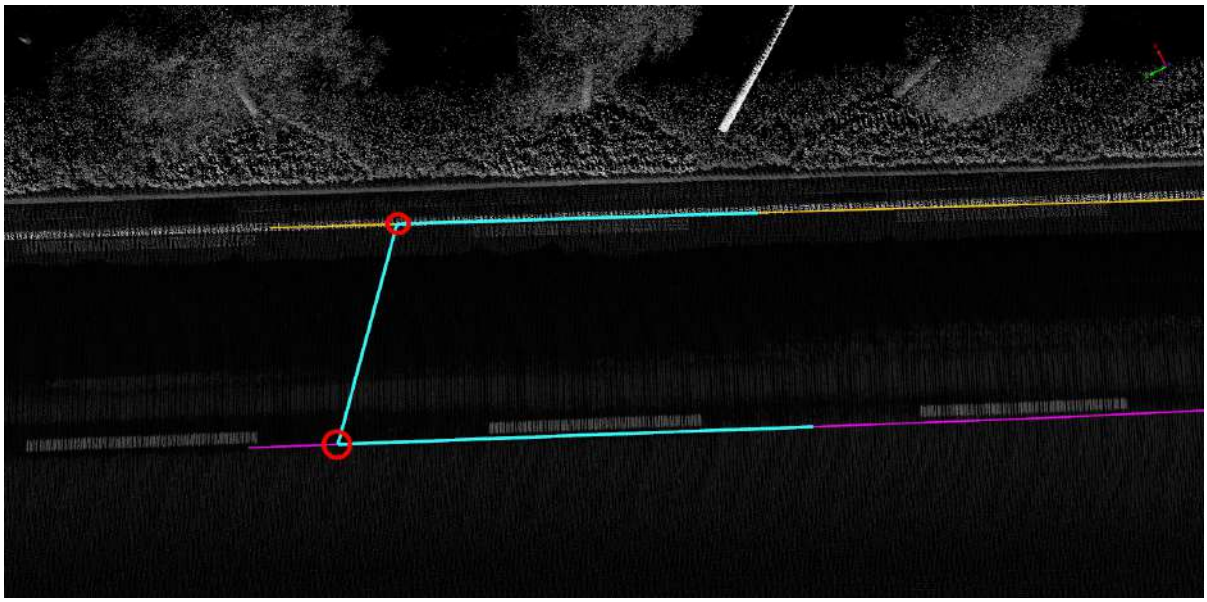
1.First switch the layer to the line layer.

2.After clicking the **Add Trace Line** button in the Vector Editor, point at any line vector in the view, the collection will start at the point closest to the clicked position on the vector.

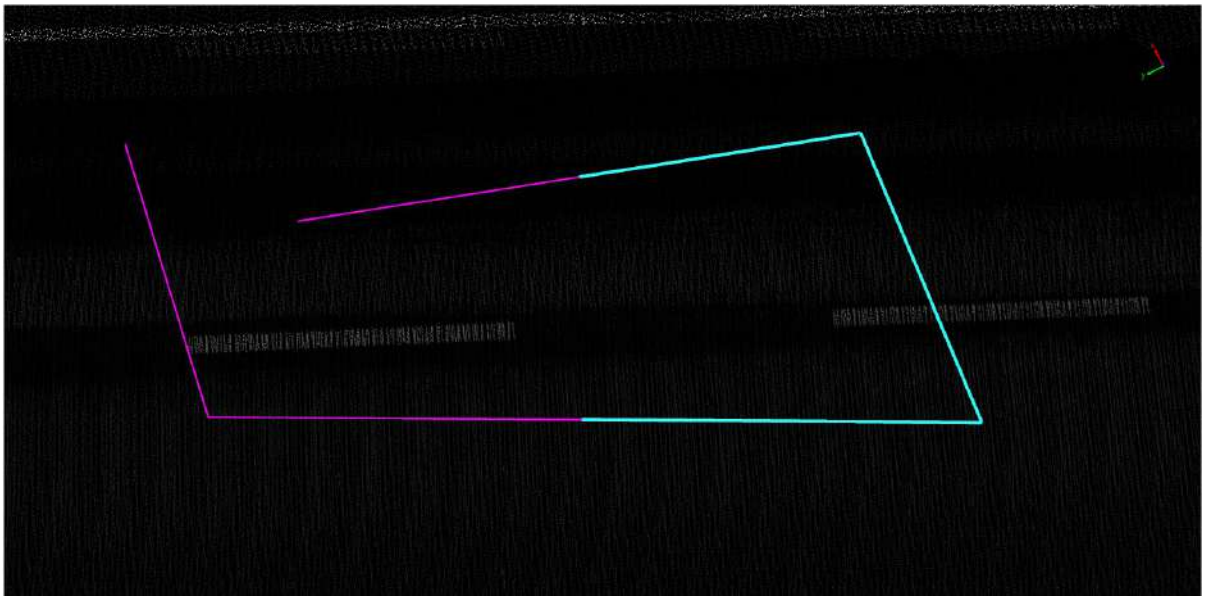
3.If you move the mouse, the currently collected line will track the mouse to add the corresponding key points, and moving in the reverse tracking order will remove the key points.

4.If you need to switch to other lines, there are two ways:

- If the two lines are not connected by a line at the switching place, you need to click the left button at the switching place, and then click the left button again on the adjacent line to be switched, and you can continue to draw on another line.



- If the two lines are connected by a line at the switching place, you only need to move the crosshair to the connected line to continue tracking. The mouse does not need to click during the tracking and switching process.



5.Double-click to end the current track, and a point will be added to the closest point of the double-click.

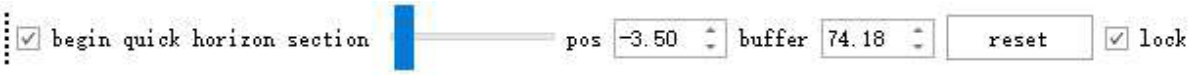
6.Clicking the right mouse button during the tracking process will clear the current tracking process.

Draw By Intersection

Function Description: This function allows the user to draw a polyline or polygon with the intersection of two adjacent lines as the basic logic.

Steps

1.This function requires that all points must be on a plane, so the user will need to determine a reference plane first, refer to the locking function of begin quick horizon section:



2.Select the line layer or polygon layer, activate **Draw By Intersection** function.

3.The collection method takes 4 points as a unit, first collect 2 points to determine the reference line, collect the third point as the anchor point, and obtain the intersection point between the connection line between the fourth point and the anchor point and the reference line, at the same time, the line between the intersection point and the fourth point is taken as the current reference line and circulates in turn.

4.If it is currently a surface layer, the connection between the last 2 points of the current point sequence will intersect with the connection between the first 2 points to form a closure. If the angle is less than 15 degrees or greater than 165 degrees, it will be directly connected.

Draw By Intersection

Draw By Perpendicular

Function Description: This function allows the user to draw vertical polylines or polygons with adjacent sides according to the layer type.

Steps

1.Select the line layer or polygon layer, activate **Draw By Perpendicular**  function.

2.First collect 2 points to determine the current baseline, and then add a point to determine a line vertical to the current baseline, and the newly added line will become the current baseline.

3.If the current layer is a surface layer, the last point will try to find a point on the first line, making the point perpendicular to the line connecting the last point and the first line, and closing based on this point.

Draw By Perpendicular

Edit Vertices

Function Description：Edit the nodes in the vector.

Steps

1.Click the **Edit Vertices** button in the node editor，Then click to select the vector object of the node to be edited. The dialog box function for vector node editing will be activated. As shown in the figure.

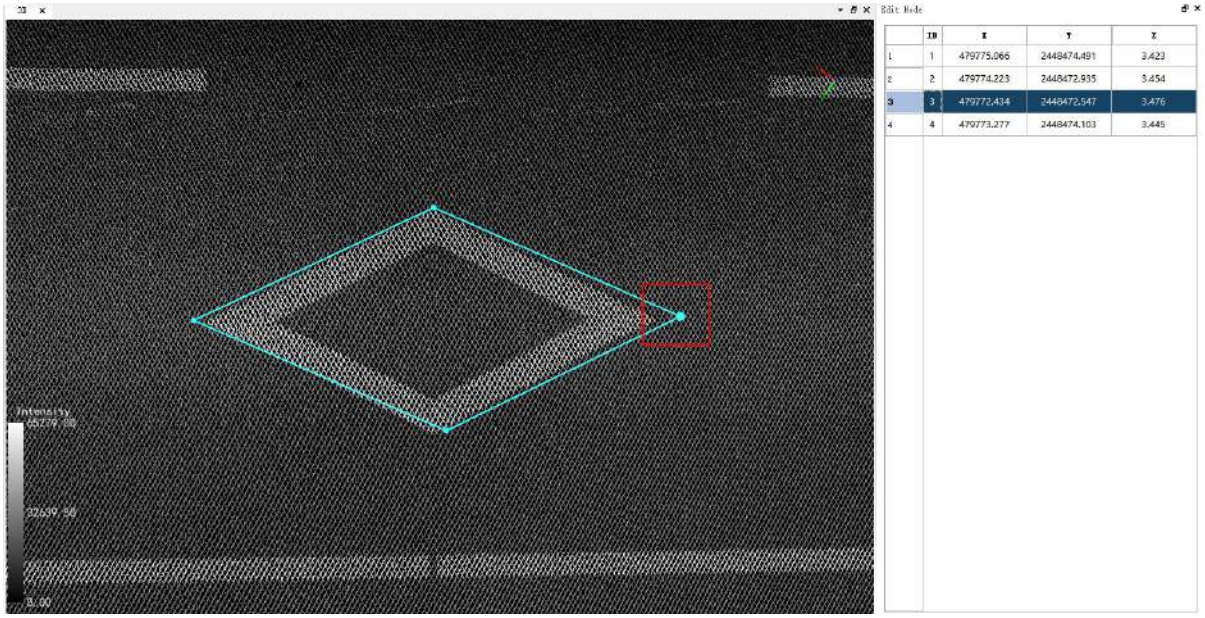
Edit Node

	ID	X	Y	Z
1	1	479775.066	2448474.491	3.423
2	2	479774.223	2448472.935	3.454
3	3	479772.434	2448472.547	3.476
4	4	479773.277	2448474.103	3.445

Node editing interface

2.The main functions of the node editor are as follows.

2.1 **Selected Highlight** : Mouse click on a selected node to edit one or more rows of the table will highlight the selected node of the vector in the 3D scene.



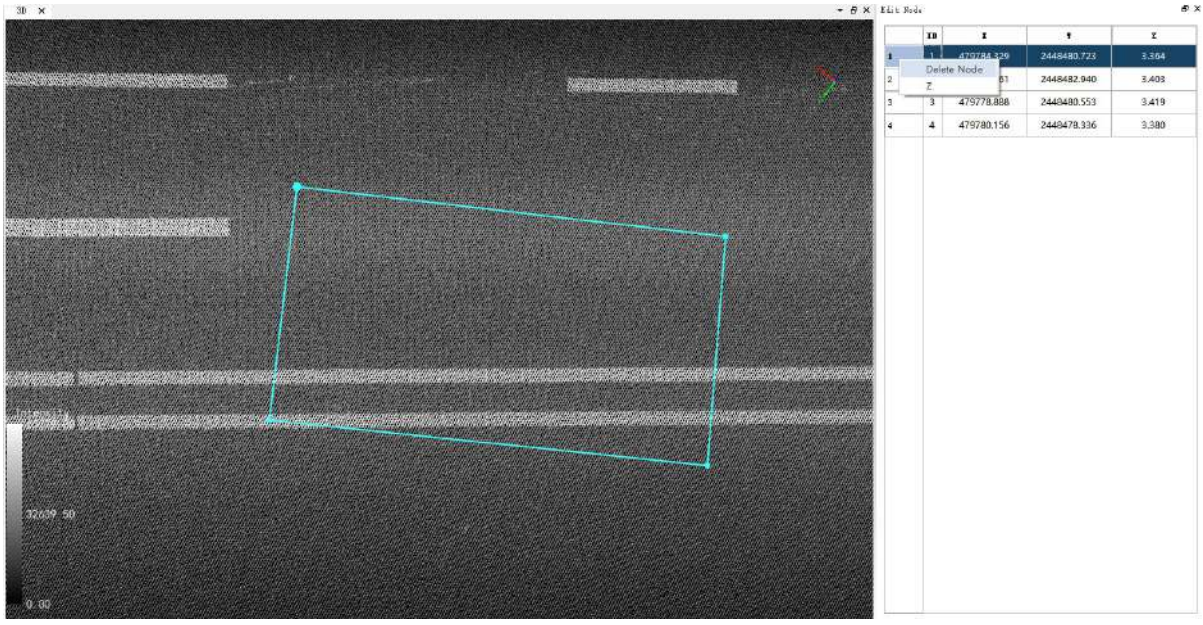
2.2 **Double-click to jump, highlight** : Double-clicking on a row of the node edit form with the mouse will switch the view of the 3D scene to that node and highlight the node.

2.3 **Delete Node**: There are two ways to delete a node.

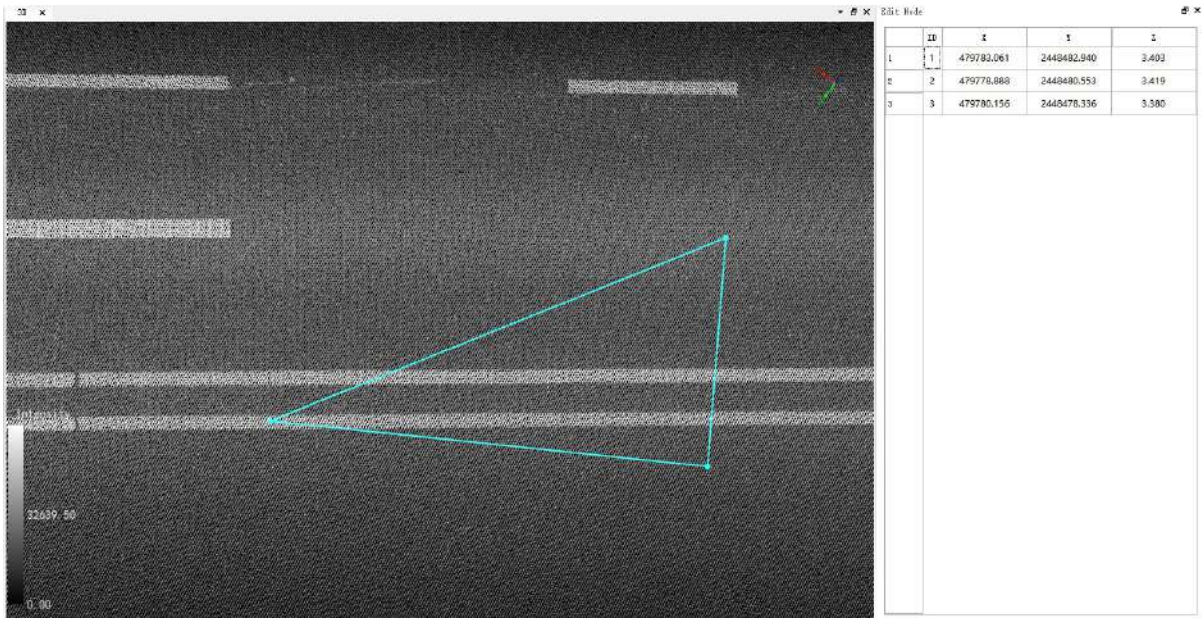
> Note: If the node to be deleted is an internal node, the nodes on both sides of the deleted node will be directly connected while the current node is deleted.

- 1.Select a row in the node edit form to delete a node. Or click on the top left cell in the node editing form, select all nodes, and delete all nodes. Take a row as an example, click the right mouse button to bring up the

drop-down menu, select the function of "Delete Node", which will delete the corresponding row in the table, and at the same time will delete the selected node of the vector in the 3D scene and redraw the vector. The following figure shows.

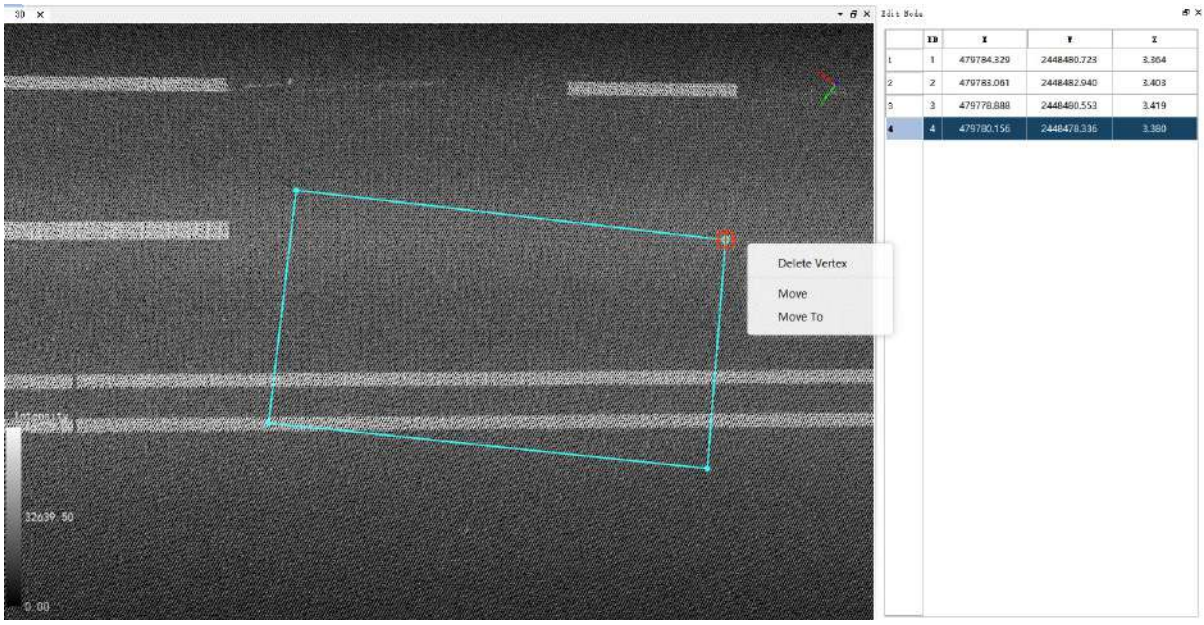


Before node deletion

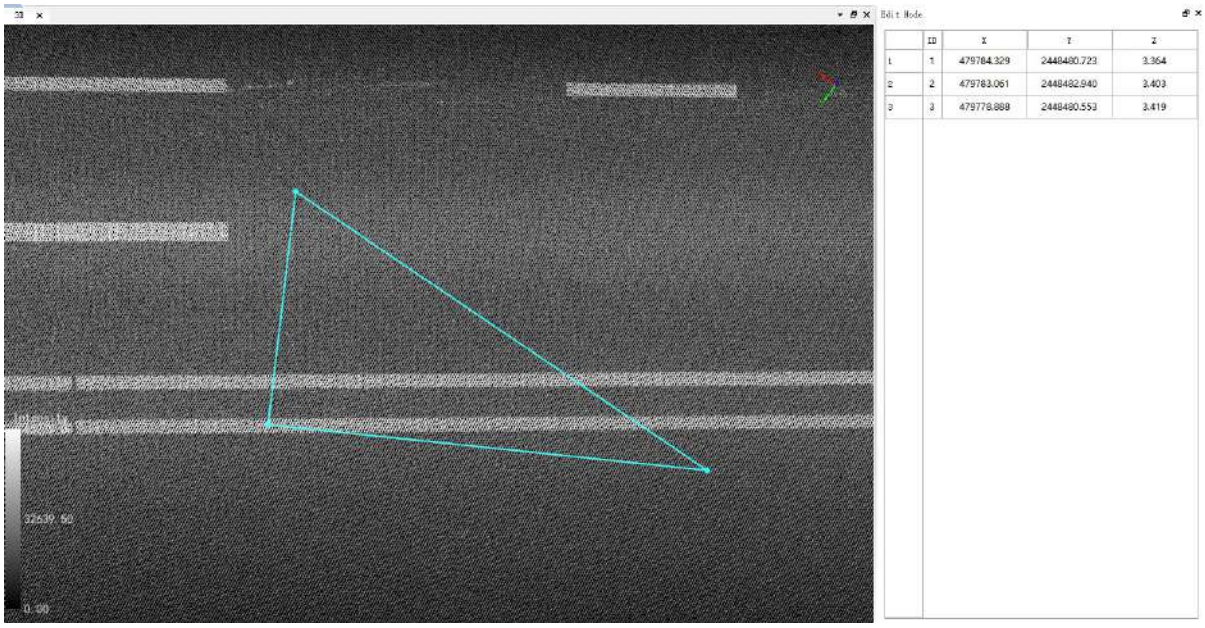


After node deletion

- 2.Move the mouse to the position of the deleted node, the cross wire turns red, indicating that the node is captured. Then click the right mouse button to pop up the drop-down menu, select the function of "Delete Node", which will delete the corresponding row in the table, and will delete the selected node of the vector in the 3D scene, and redraw the vector. The following figure shows.



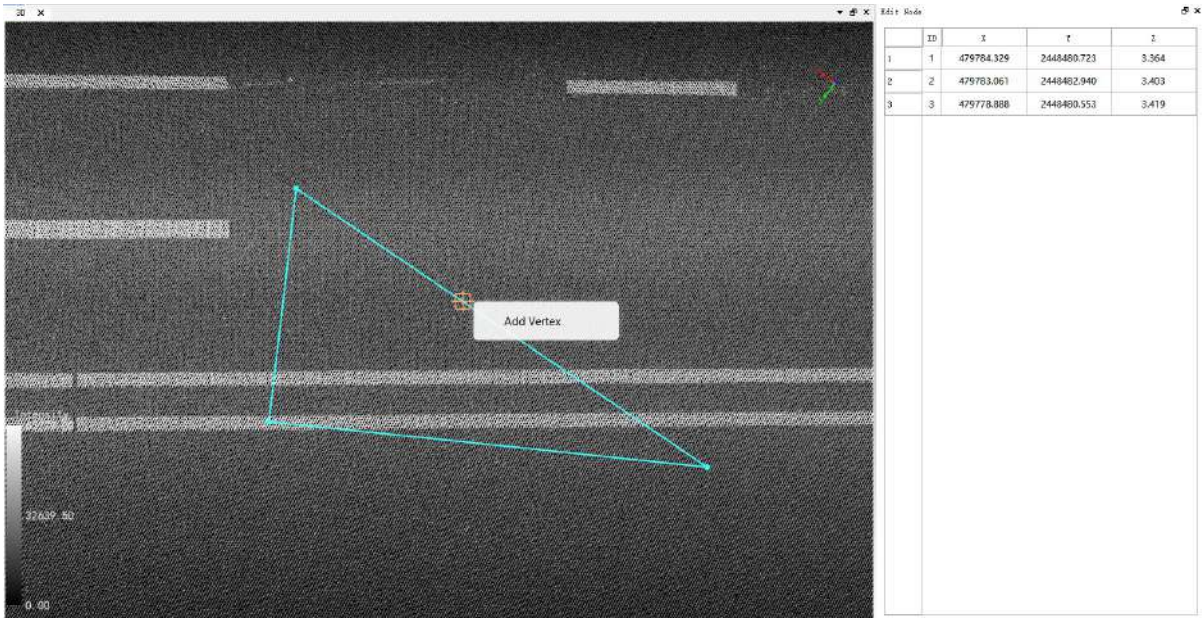
Before node deletion



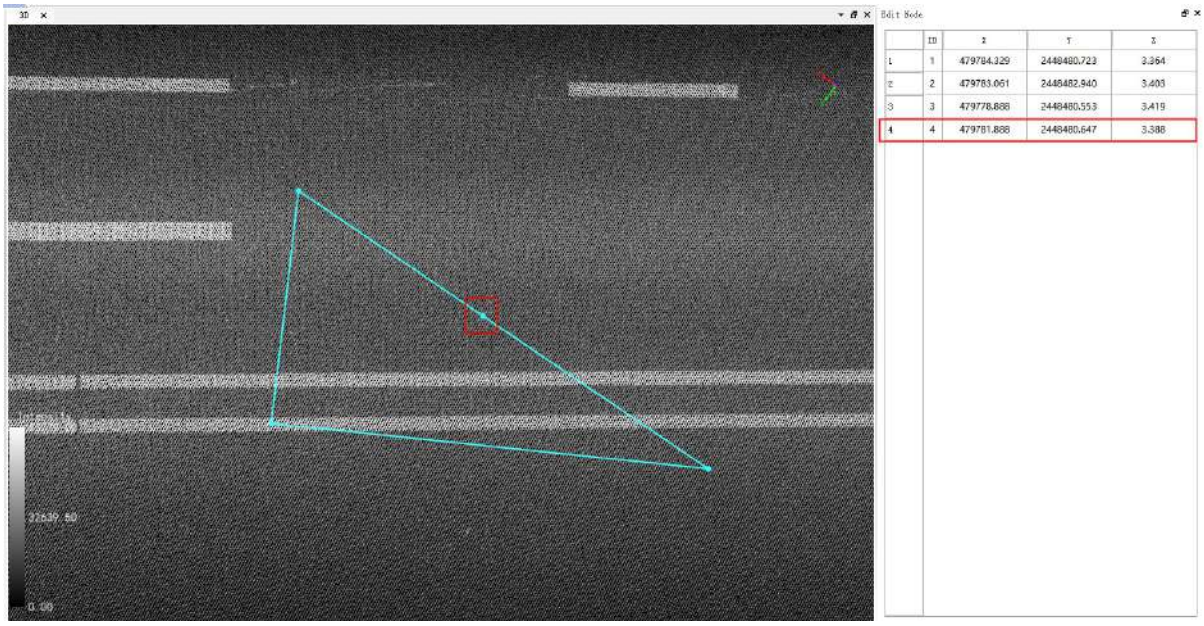
After node deletion

> Note: Keep a minimum of one node for the point layer, two nodes for the line layer, and three nodes for the surface layer.

2.4 **Add Node**: Select the location of the node you want to add in the vector object, the cross turns orange, indicating that the edge of the vector data is captured. Then click the right mouse button, pop-up drop-down menu, select the function of "Add Node", while the vector node will be added in the three-dimensional scene, and the node editing form automatically add the node information. The following figure shows.



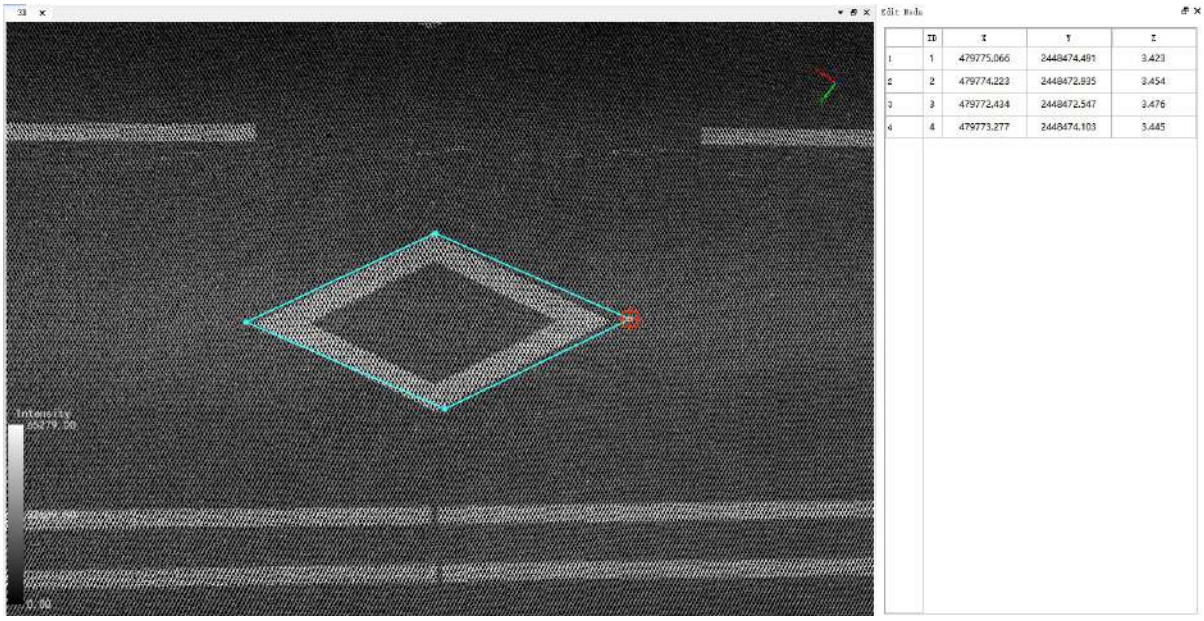
Before node insertion



After node insertion

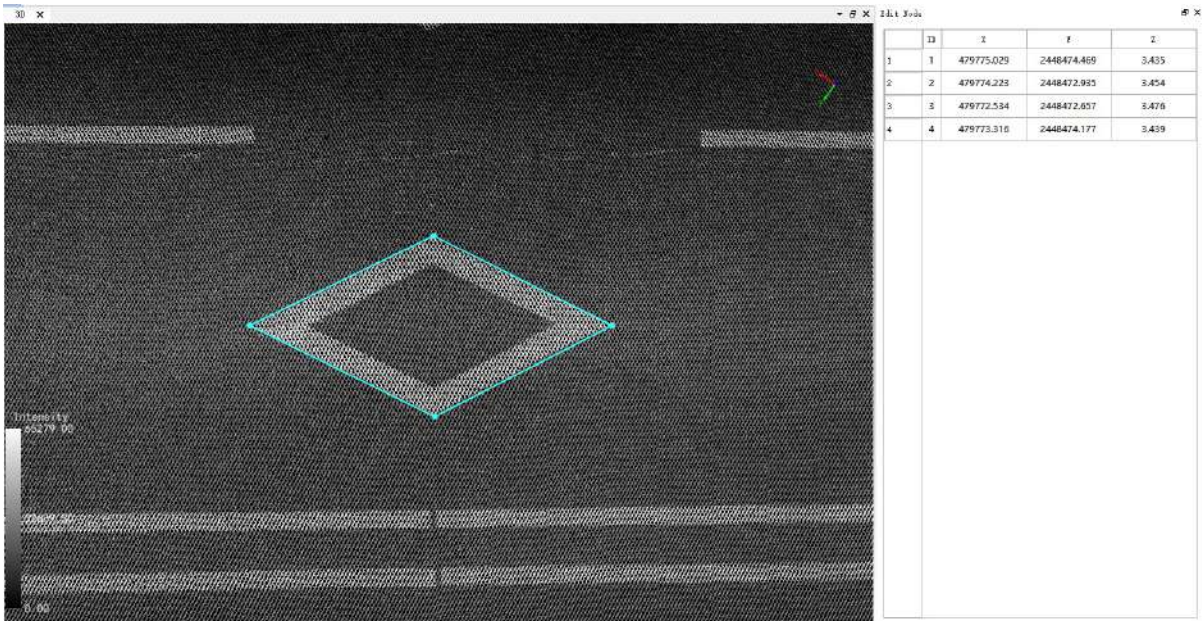
2.5 **Move Node**: There are two methods.

- 1.When the mouse is close to the vector node, the software automatically captures the nearest node and the cross wire turns red, indicating that the node is captured. At this time, you can press the left mouse button to drag the node to the target position. The following figure shows.



	ID	X	Y	Z
1	1	479775.066	2448474.491	3.423
2	2	479774.223	2448472.935	3.454
3	3	479772.434	2448472.547	3.476
4	4	479773.277	2448474.103	3.445

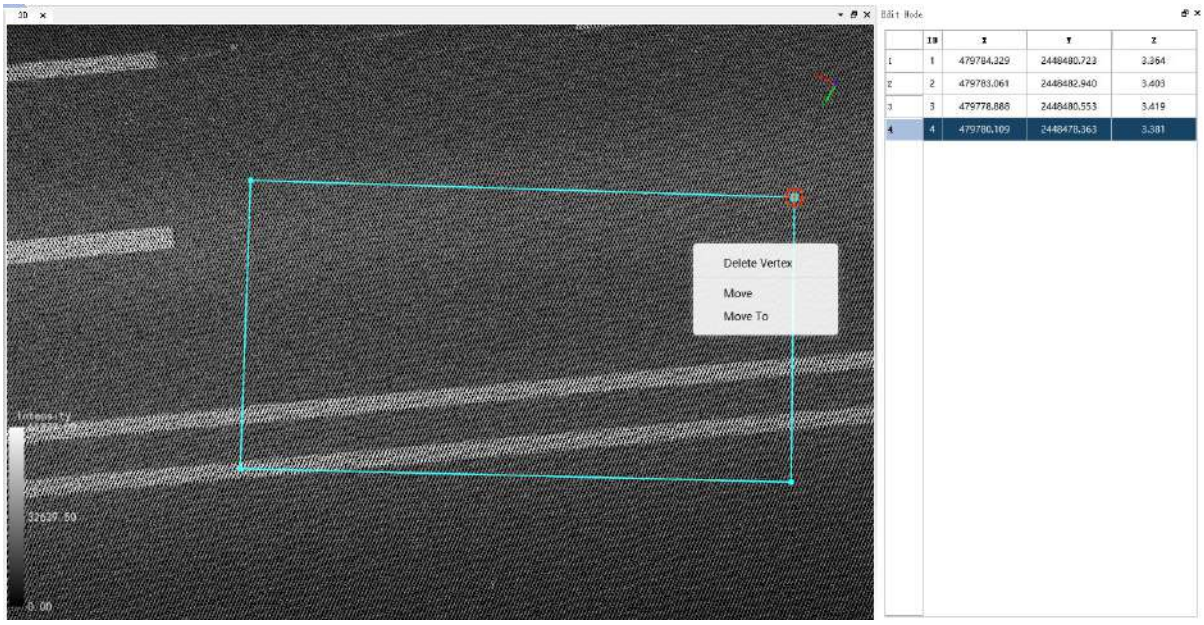
Before moving nodes



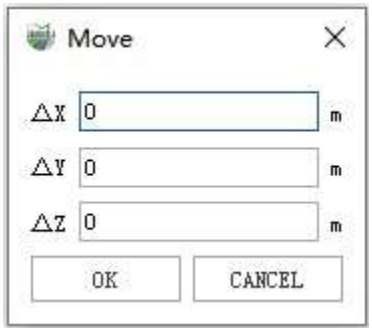
	ID	X	Y	Z
1	1	479775.029	2448474.469	3.435
2	2	479774.223	2448472.935	3.454
3	3	479772.534	2448472.657	3.476
4	4	479773.316	2448474.177	3.439

After moving nodes

- 2.When the mouse is close to the vector node, the software automatically captures the nearest node and the cross wire turns red, indicating that the node is captured, then click the right mouse button to bring up the drop-down box.

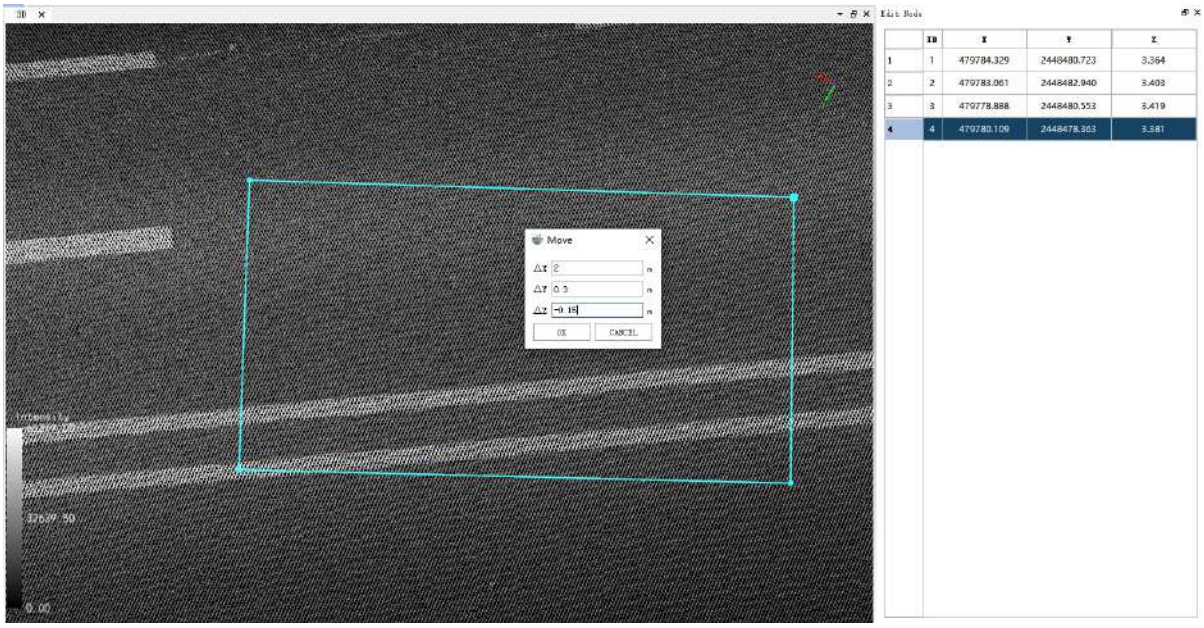


2.1 Check the "Move" function. As shown in the figure below.

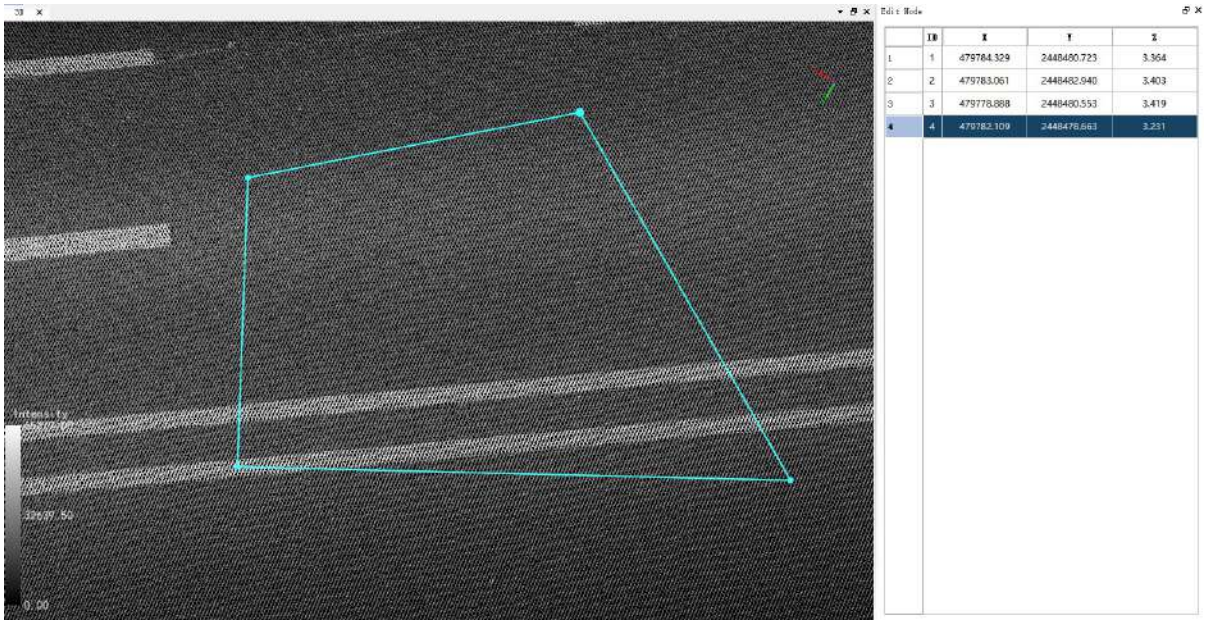


Parameter Setting

- **X**：The input value is the relative displacement of the node on the X-axis
- **Y**：The input value is the relative displacement of the node on the Y-axis
- **Z**：The input value is the relative displacement of the node on the Z-axis



Before moving nodes

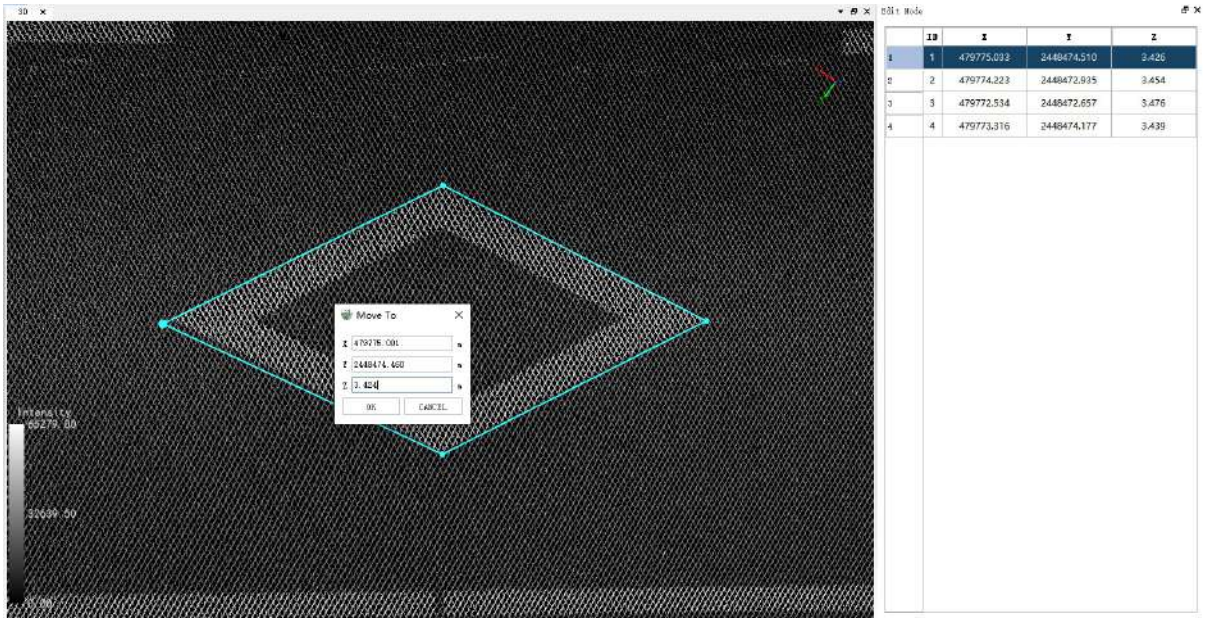


After moving the node

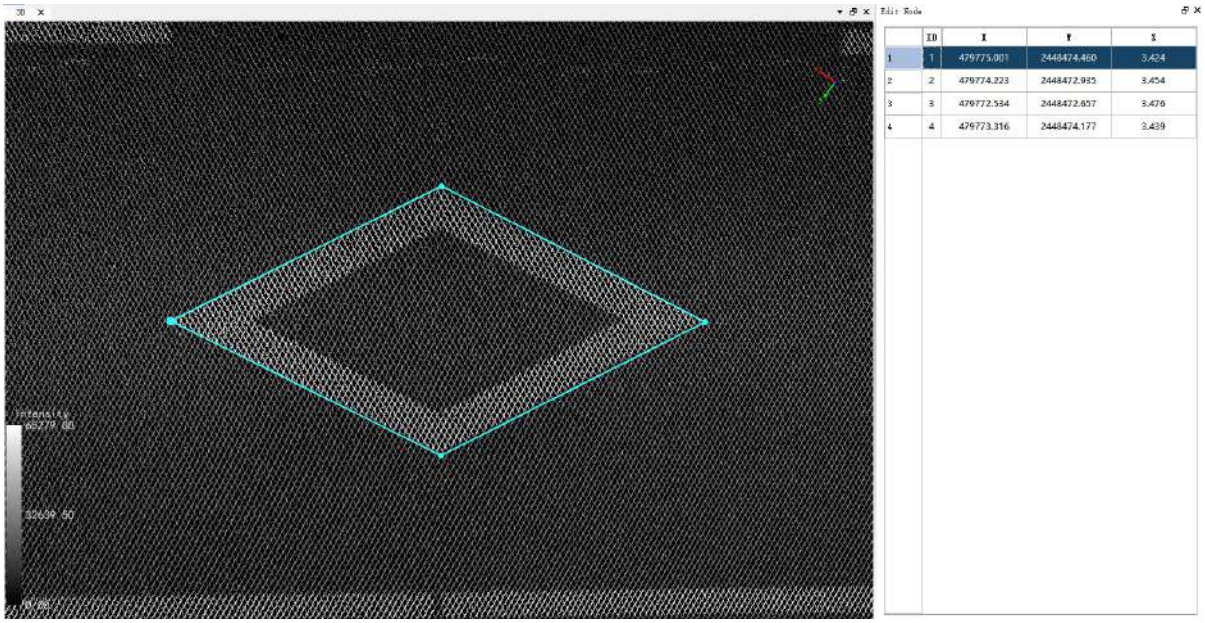
2.2 Check the "Move To" function. The following figure shows.

Parameter Setting

- **X**：The input value is the absolute position of the node on the X-axis
- **Y**：The input value is the absolute position of the node on the Y-axis
- **Z**：The input value is the absolute position of the node on the Z-axis
- **Select Z**：With the checkbox checked, you can pick any position directly on the point cloud as a Z value

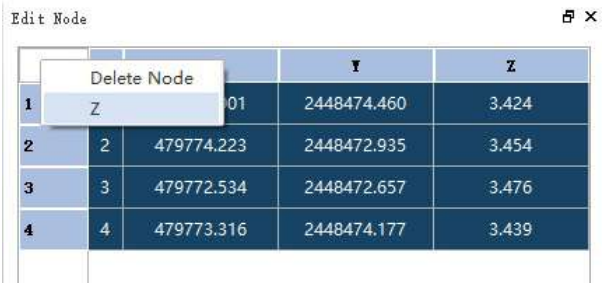


Before moving nodes



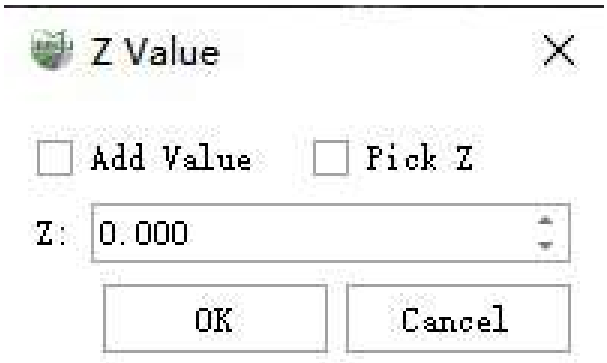
After moving nodes

2.6 **Modify Z value**: Select a row in the node editing table, then click the right mouse button to bring up the drop-down menu, select "Z" to modify a row. Or click the top left cell in the node editing table, select all nodes, and edit all nodes. Then click the right mouse button to bring up the drop-down menu, select "Z" to modify all nodes.



The dialog box of Modify Z Value will pop up:

- **Add Value**:Unchecked "Add Value" will modify the Z value of the selected node to the value in the input box and redraw the vector. If "Add Value" is checked, a new value will be added to the original Z value of the node, and the vector will be redrawn in the 3D scene.
- **Pick Z**:When add Value is unchecked, the checkbox for Select Z can be checked directly, and any position on the point cloud can then be picked as a Z value



Facade Survey

As shown in the following figure, the functions included in the facade survey page are:

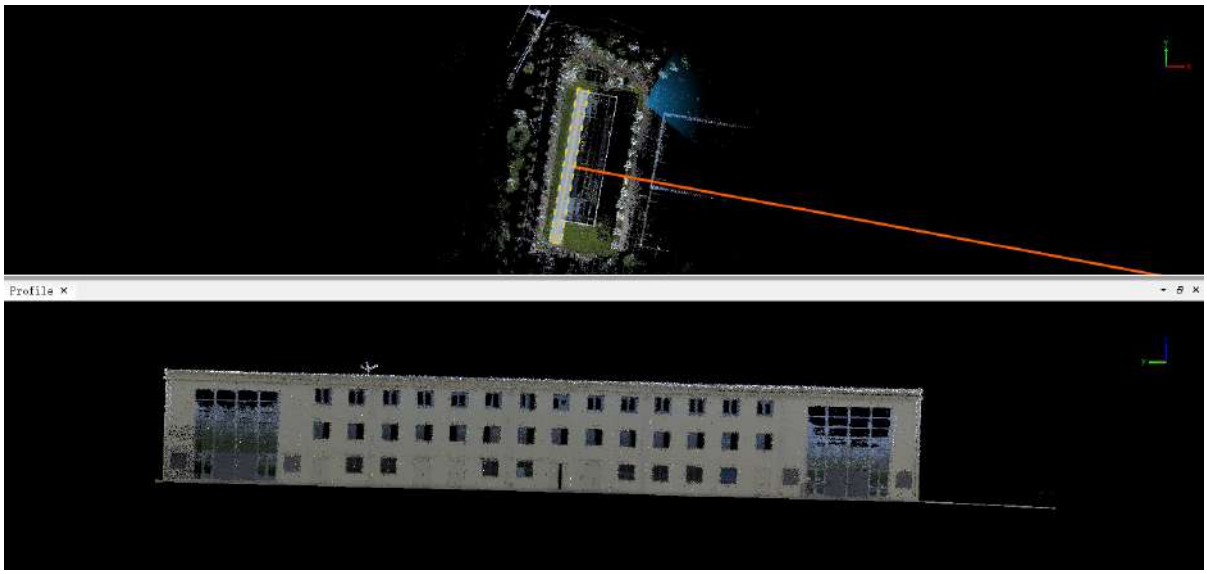
- Set Vertical
- BaseLine
- Boundary Line
- Profile
- Drawing
- Tectonic Collect
- Array
- Edit
- Export

Set Vertical

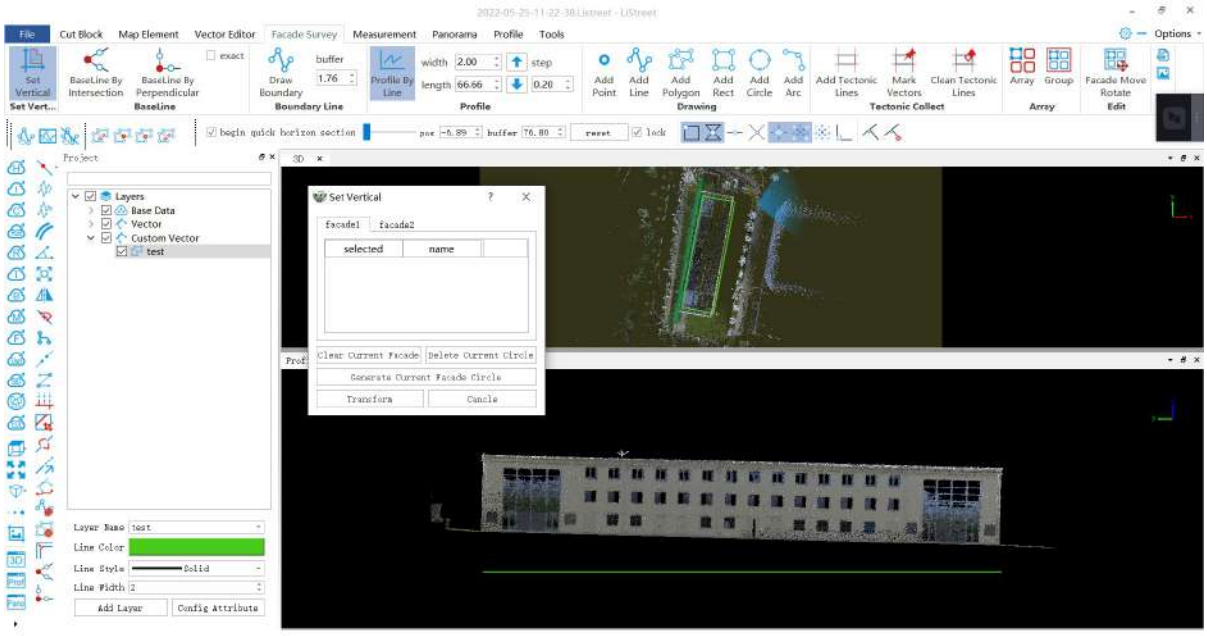
Function Description： This tool allows the user to apply vertical correction of the side facades of a building, and is performed based on two adjacent side facades of a building.

Steps

After the data is loaded, you can take a cross-section to check whether the handheld data is horizontal. If the data is as shown in the figure below, and the side elevation is inclined.

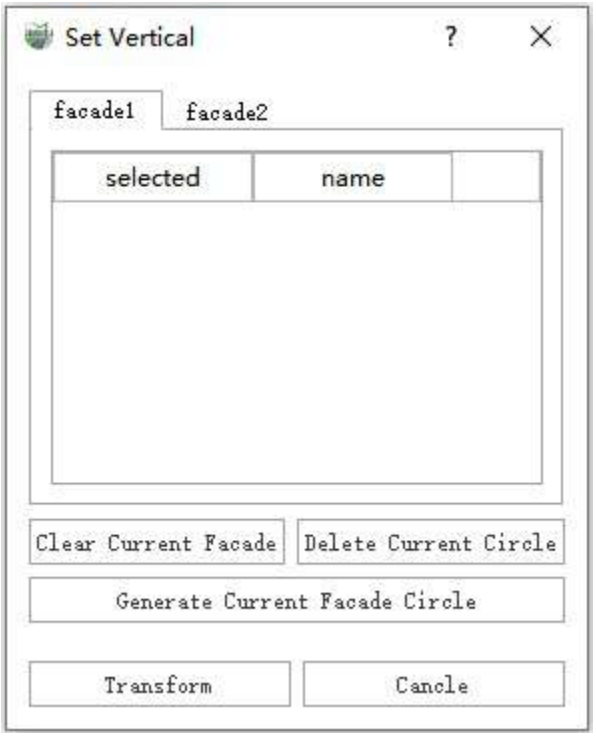


You can activate the "Set Vertical"  function in "Facade Survey" to make the side elevation vertical:



1.Open the point cloud fitting capture tool in the software.

2.Click the Set Vertical button to pop up the set vertical operation pop-up box.



3.Under the "facade1" tab page, on one side elevation of the building, evenly select several (at least three) fitting planes under the same vertical plane from top to bottom (Figure 1 below), and then click "Generate Current Facade Circle" to generate the fitted vertical plane of the current side elevation (Figure 2 below).

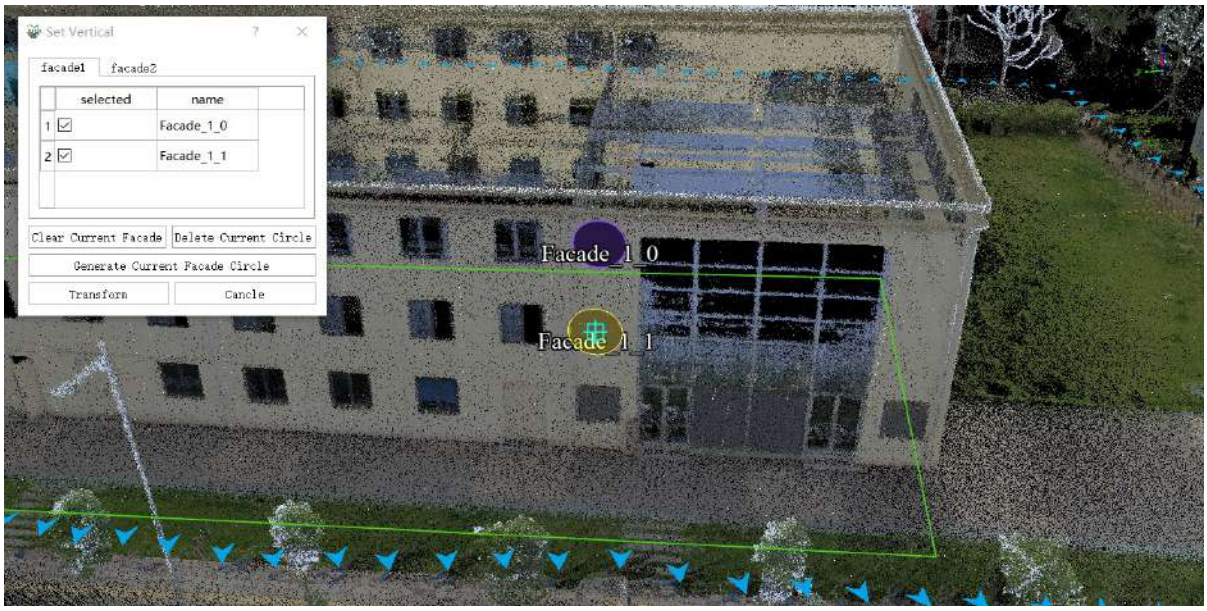


Figure 1 Selecting the Fitting Plane

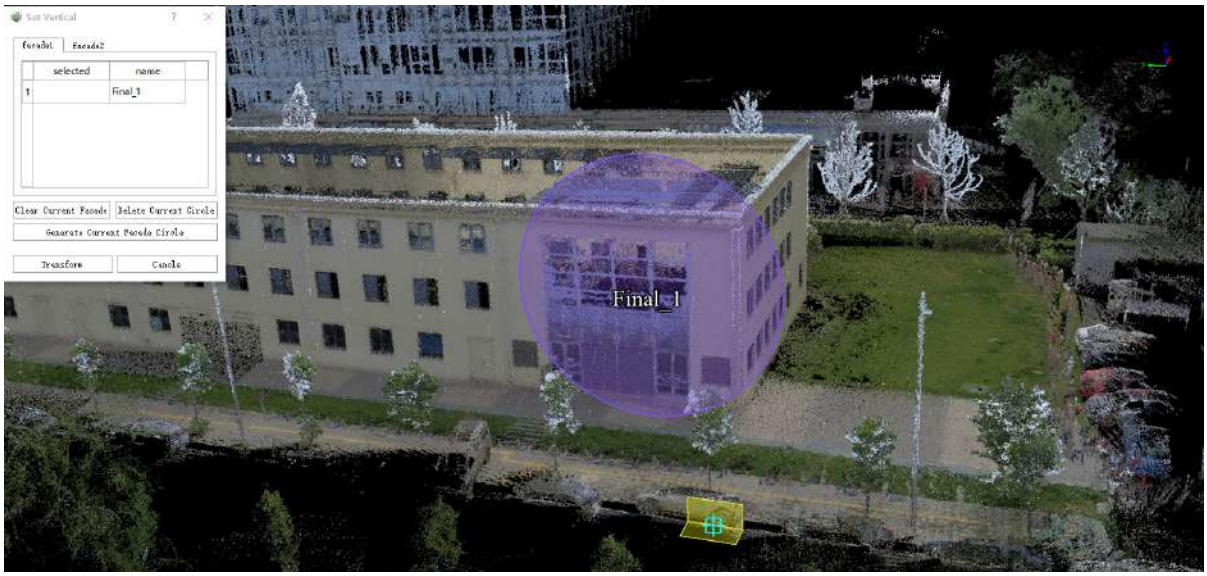
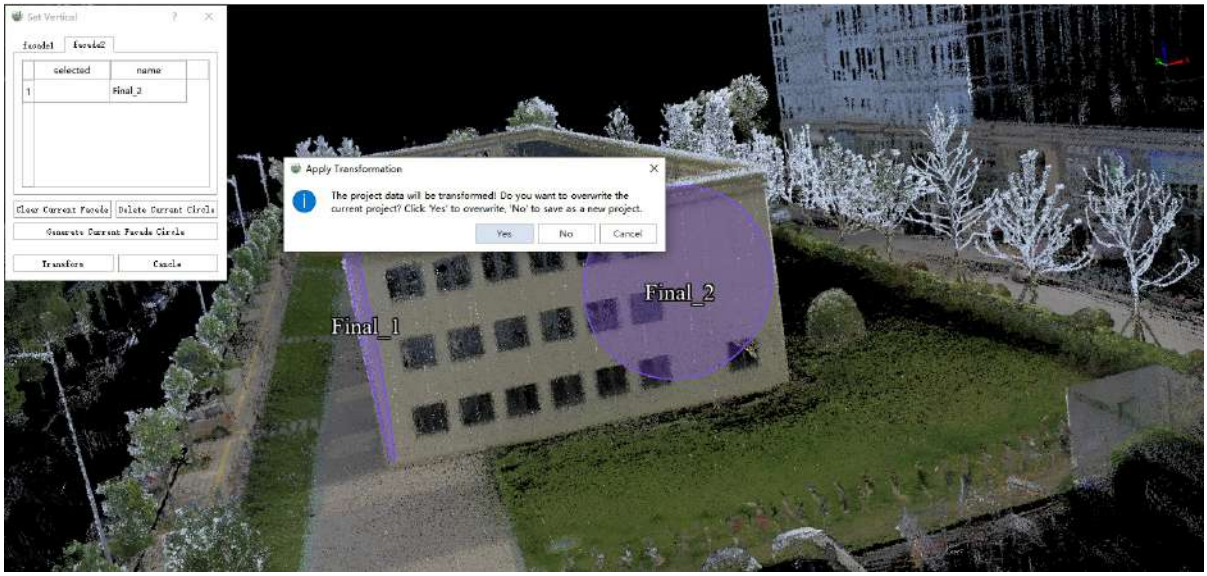


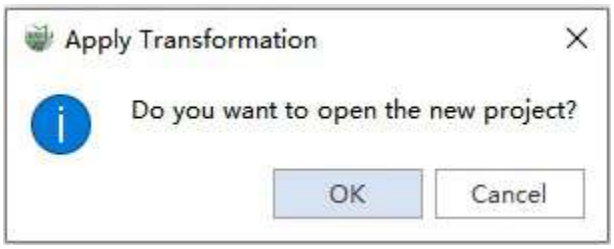
Figure 2 Generate a facade circle

Under the "facade2" tab page, select the side elevation that is connected to the edge of facade 1, and then repeat the operation of facade 1 to generate a second facade circle.

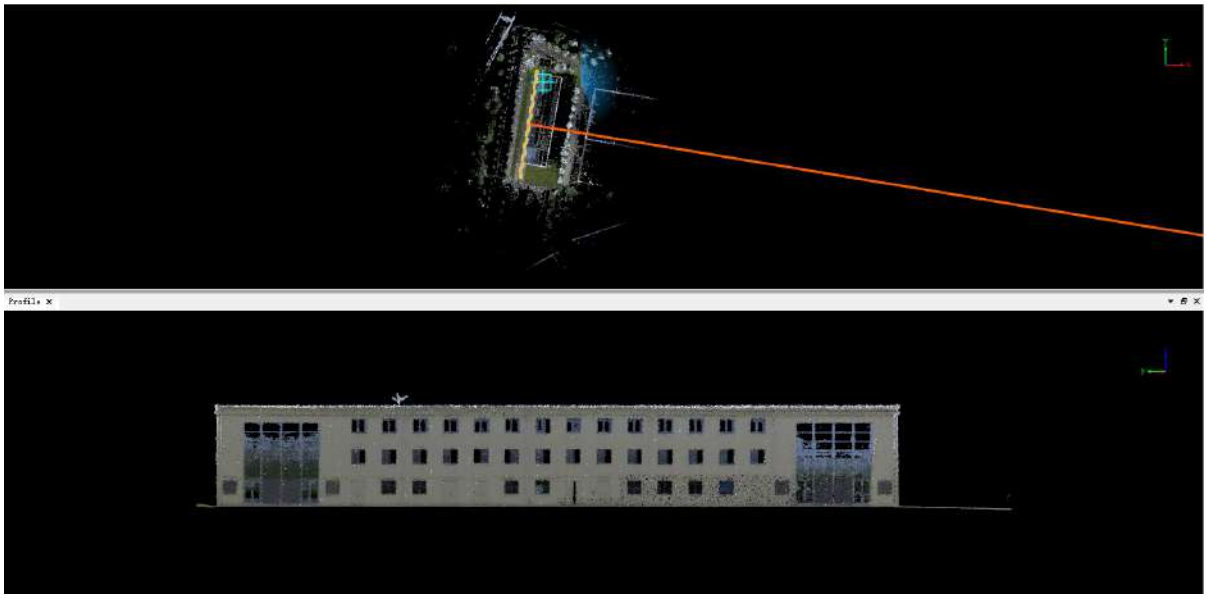
4.After the two side facade circles are generated, confirm that they are correct, and click the "Transform" button to apply the transformation. Select "Yes", the point cloud, panorama and trajectory (if it is set) loaded in the current project will be rewritten, which is equivalent to rewriting the original data.



If you select "No", a new project with a native project name suffix “_Vertical.LiMMP“ will be regenerated according to the converted point cloud, panorama and trajectory (if it is set), and you will be prompted to open a new project after the conversion is completed.



5.The point cloud after resetting the vertical is shown in the figure below.



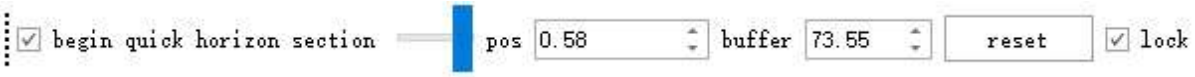
BaseLine

Function Description： This function allows the user to create a baseline from a horizontal structure diagram.

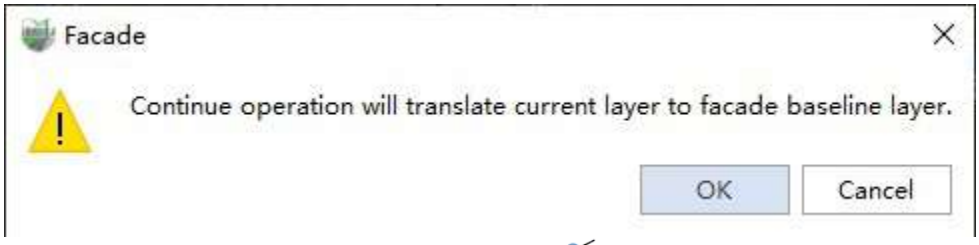
Steps

> Attention: You must have a horizontal structure diagram before you can draw the side facade elements

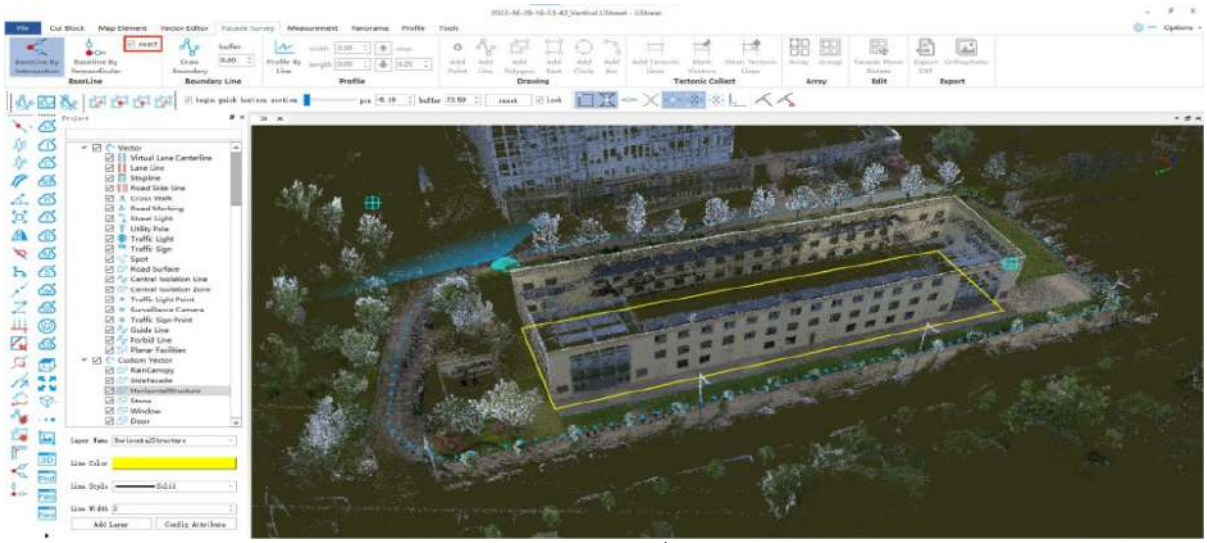
1.Select the layer of horizontal structure diagram, In the "begin quick horizon section" shortcut tool, select a height at which you want to draw the horizontal structure and check "lock".



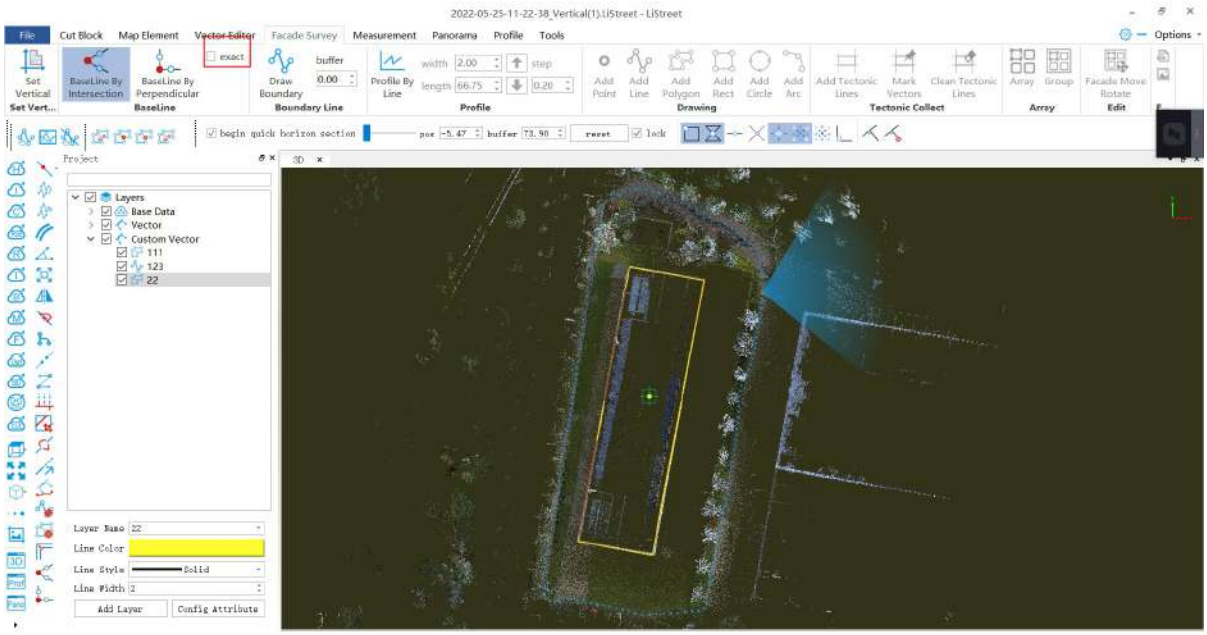
2.Select the tool in "Baseline Collection" under the "Façade Survey" menu bar, and "exact" is checked by default. If you select any tool again, you will be prompted to convert the selected layer into a baseline layer, because the later selection of the edge facade controls the layer selection, only the baseline layer and the range line layer can be selected to intercept the section.



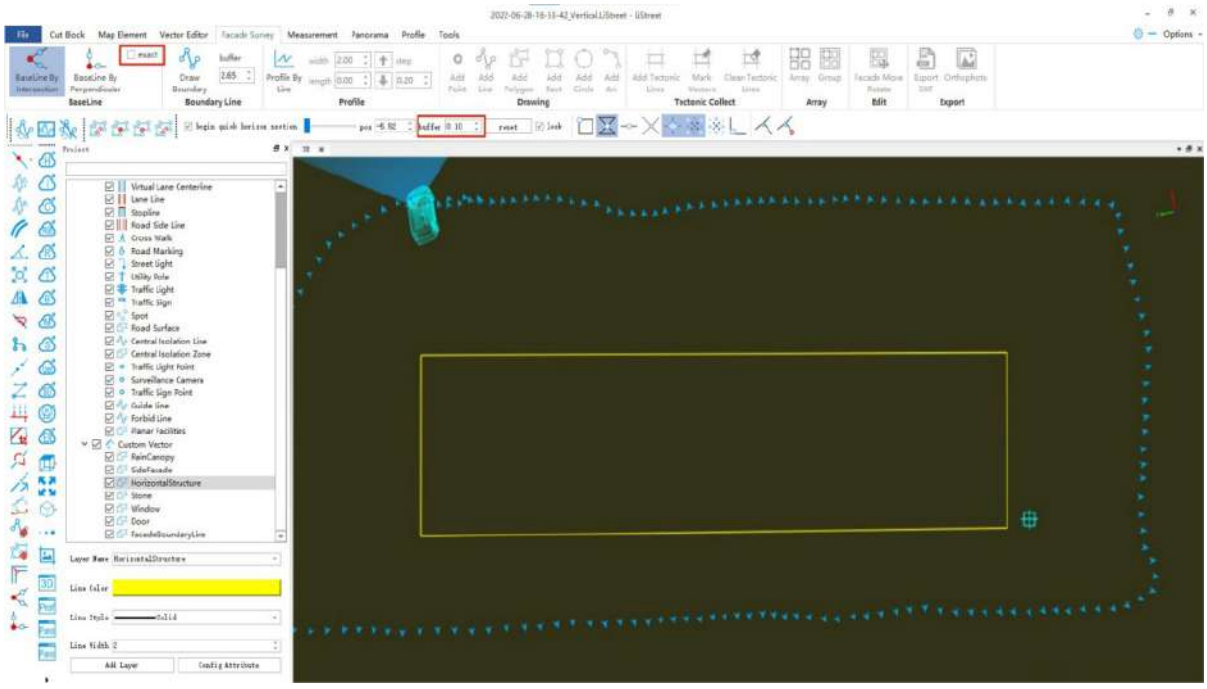
- In precise mode, clicking the **BaseLine By Intersection** button, you need to click on two points of point cloud on each side facade, and double-click on the last point to complete the closure.



- In precise mode, using the **BaseLine By Perpendicular** tool, you need to click two points on the first side elevation, click the point on the previous point cloud on the other side facades, and double-click the last point to complete the closure.



- Uncheck "exact", the two tools will turn the 3D window into top view + orthogonal projection. At this time, it is recommended to change the buffer in "begin quick horizon section" to 0.1 to reduce the thickness of the point cloud, which can increase the precision of the drawing.

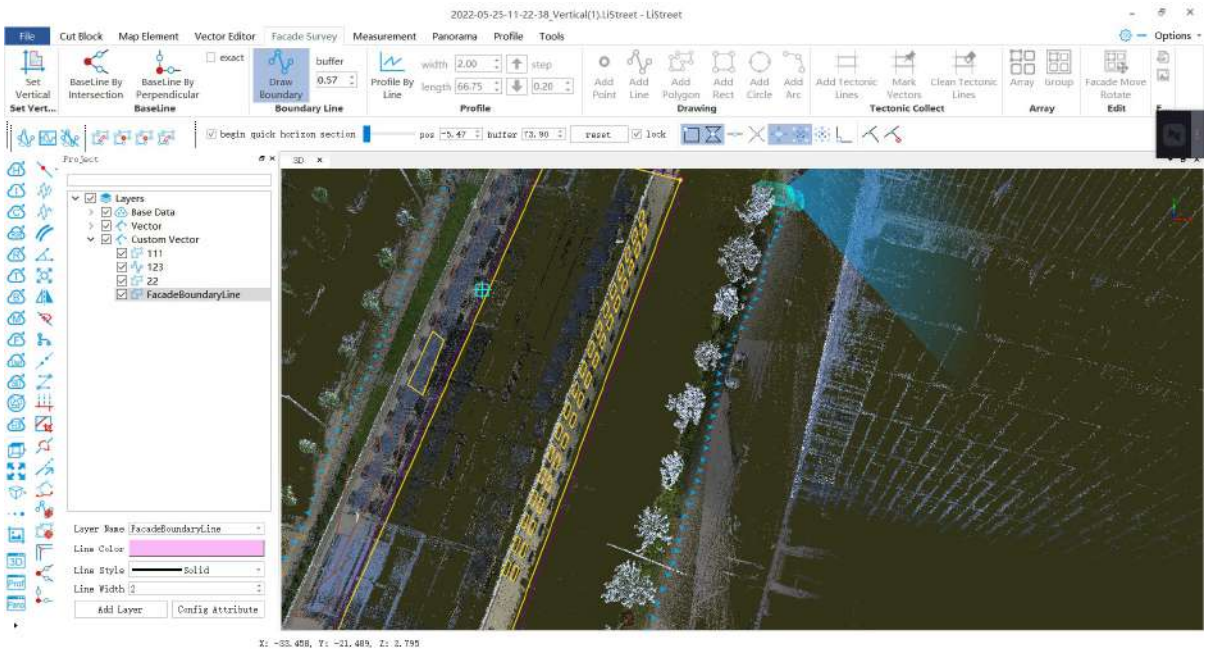


Boundary Line

Function Description： This function allows the user to create a boundary line based on the baseline layer, that can be expanded outward to a certain extent.

Steps

1.According to the facade drawing specifications, if you need to draw a range line, you should activate **Draw Boundary**  tool in **Facade Survey**, select a base line, and you can choose to drag it or draw it out at the length of the boundary line. Enter a fixed value to define the boundary line, which will be drawn in an inline range graph layer.



Profile

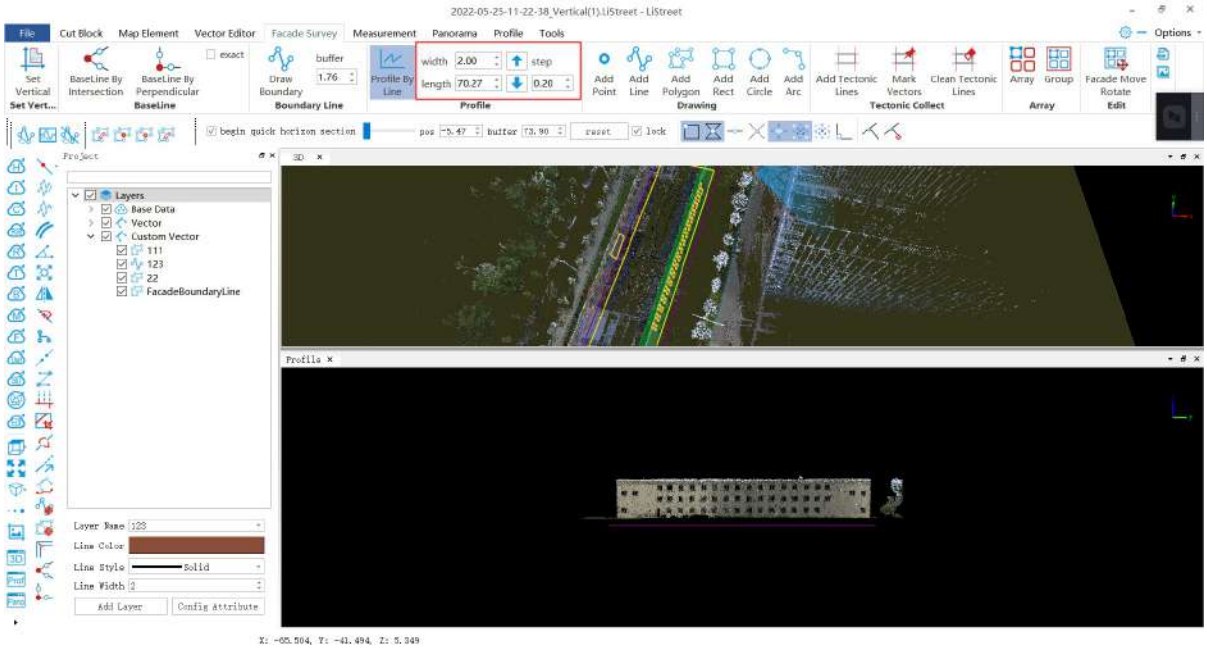
Function Description： This function allows the user to select an edge of the baseline or boundary line, and intercept the point cloud with the default thickness, and generate an profile view.

Steps

1.Click the **Profile By Line**  button to activate the function of generating profile.

2.Click a vector edge of the datum plane drawn in advance, and in the section window, the side profile corresponding to the current edge can be automatically generated.

3.You can adjust the thickness of the point cloud display through "width", and adjust the left and right display range of the point cloud through "length". Adjust the position of the point cloud display "forward" or "backward":



Drawing

Function Description： This function allows the user to draw features within an open dataset and is the same as the drawing function under the vector editing menu page.

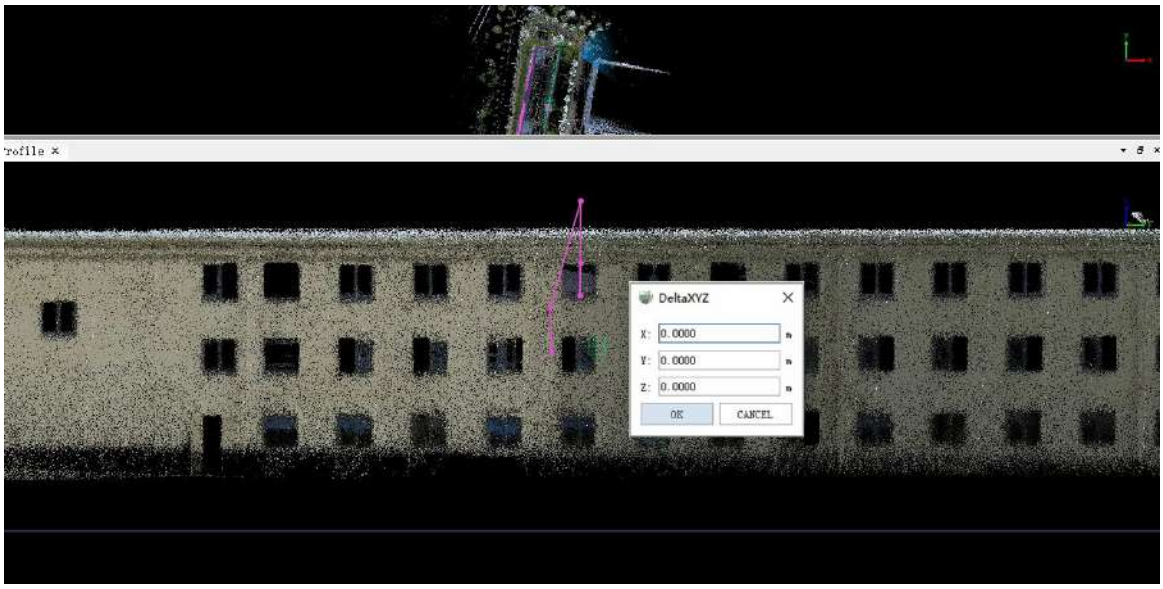
Steps

1.Point element drawing : Select a point layer and select a point in the point cloud under the profile window with the mouse, and a circle will be drawn at this position according to the current layer.

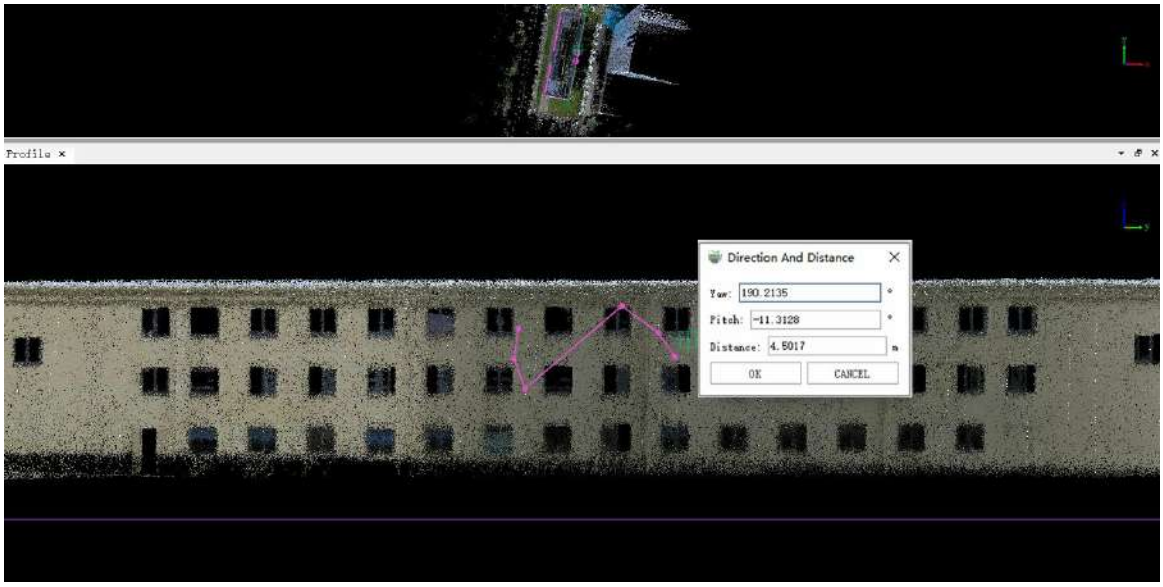
2.Line element drawing : Select a line layer, use the mouse to draw line elements in the point cloud under the profile window, where line features need to be drawn, click the left mouse button continuously to determine multiple nodes of the line. Right click during drawing to activate the right-click menu, as shown in the following figure:

- **SnapToFeature**: You can switch various capture modes at any time in the SnapToFeature menu.
- **BackPoint**: Click BackPoint to fallback nodes in order. (the first node cannot be rolled back). During drawing, the shortcut key of BackPoint is b/B.
- **AbsoluteXYZ**: With the AbsoluteXYZ function, you can specify the coordinates of AbsoluteXYZ during the drawing process. The coordinate value entered in the pop-up box is the position drawn by the node.

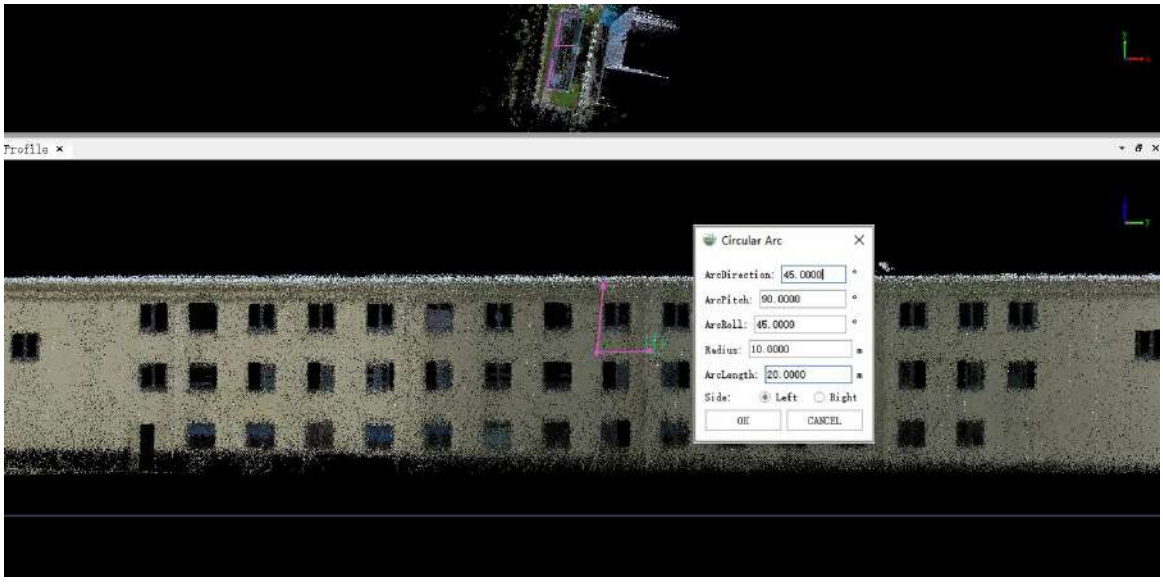
- **DeltaXYZ**: With the DeltaXYZ function, you can specify the coordinate position of the next node relative to the previous node in the drawing process. The coordinate value entered in the pop-up box is the position relationship between the next node and the previous node.



- **Direction/Distance**: With the Direction/Distance function, you can specify the direction and distance of the next node relative to the previous node in the drawing process. The yaw angle is the included angle with the Y axis of the coordinate system, the pitch angle is the included angle with the XY plane of the coordinate system, and the distance is the distance between the next point and the next point in the above angular direction. In the profile window, there is no need to modify the yaw angle.



- **Circular Arc**: With the circular arc function, an arc with a specified direction, size and position can be generated during the drawing process. As shown in the following figure, the ArcDirection refers to the angle of the arc relative to the Y axis of the coordinate system, ArcPitch refers to the direction relative to the XY plane of the coordinate axis, and ArcRoll refers to the rotation angle around the XY plane of the coordinate axis. Radius refers to the length of the two ends of the generated arc, ArcLength refers to the side length of the generated arc, and Side controls the current plane position to the left or right. (Note: the profile window is not applicable to arc calculation, and the generated arc will not be displayed completely in the profile window)



- **ReverseDirection**: During the line drawing process, the direction of the drawn vector line can be reversed with the ReverseDirection function, and the drawing can continue based on the new direction.
- **Finish**: During the drawing process, you can double-click the last node with the left mouse button to complete the drawing, or you can click the Finish button in the pop-up box with the right mouse button to end the drawing of the current object.
- **Cancel**: During the drawing process, you can click the right mouse button to pop up the Cancel button to exit the current drawing.

3.Polygon element drawing  : Select a surface layer, use the mouse in the point cloud under the profile window, where the surface element needs to be drawn, and click the left mouse button continuously to determine multiple nodes of the surface. Click the right mouse button during the drawing process to activate the right-click menu. The function is the same as right click function of line feature drawing.

Function	Shortcut key	Descriptors
Line, Polygon	S/s	Short press the s key to switch the drawing mode from drawing a straight line to drawing an arc. The arc adopts the three-point mode. The first point is the last point before short press s, the second point is the end point of the arc, and the third point controls the arc

4.Rectangular element drawing  : Select a surface layer and draw a rectangular element by drawing.

5.Circle element drawing  : Select a surface layer and draw a circle element by three points.

6.Arc element drawing  : Select a line layer and draw arc line elements by three points.

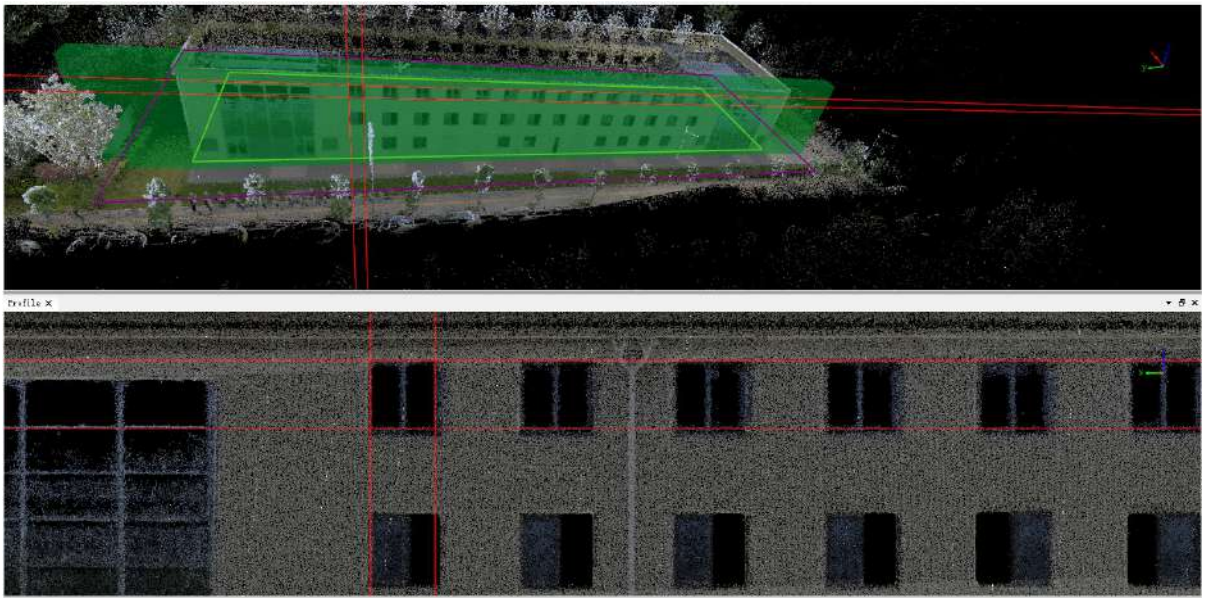
> Note: the use of basic feature drawing tools in facades is the same as that of basic feature drawing tools in vector editing module.

Tectonic Collect

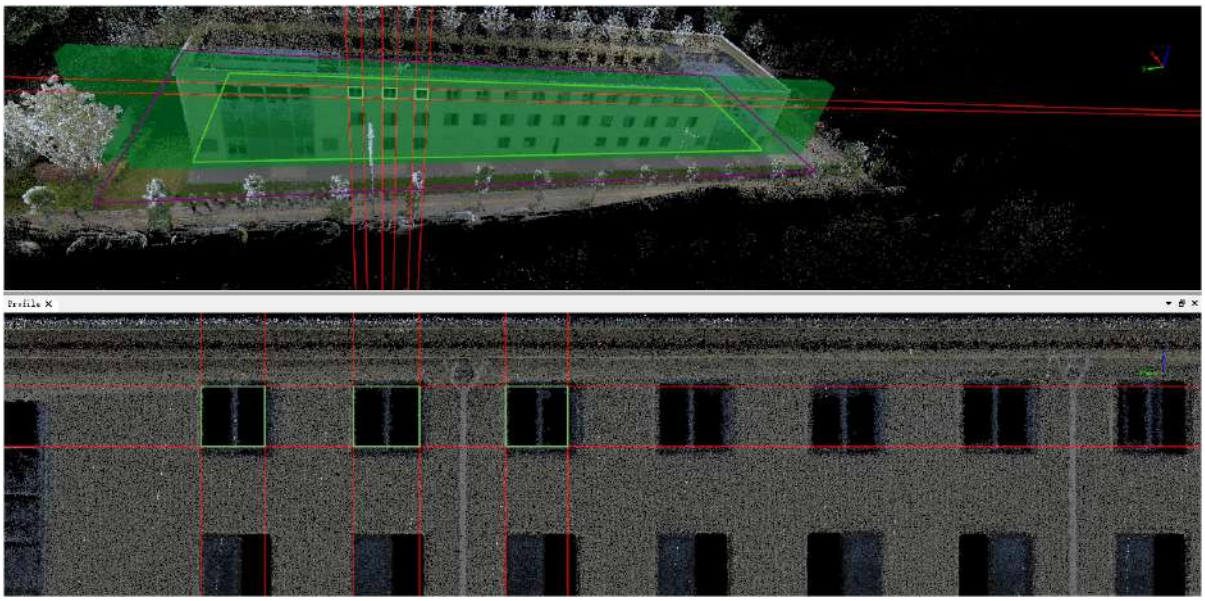
Function Description： This function allows the user to use features to draw side elevations that are regularly parallel.

Steps

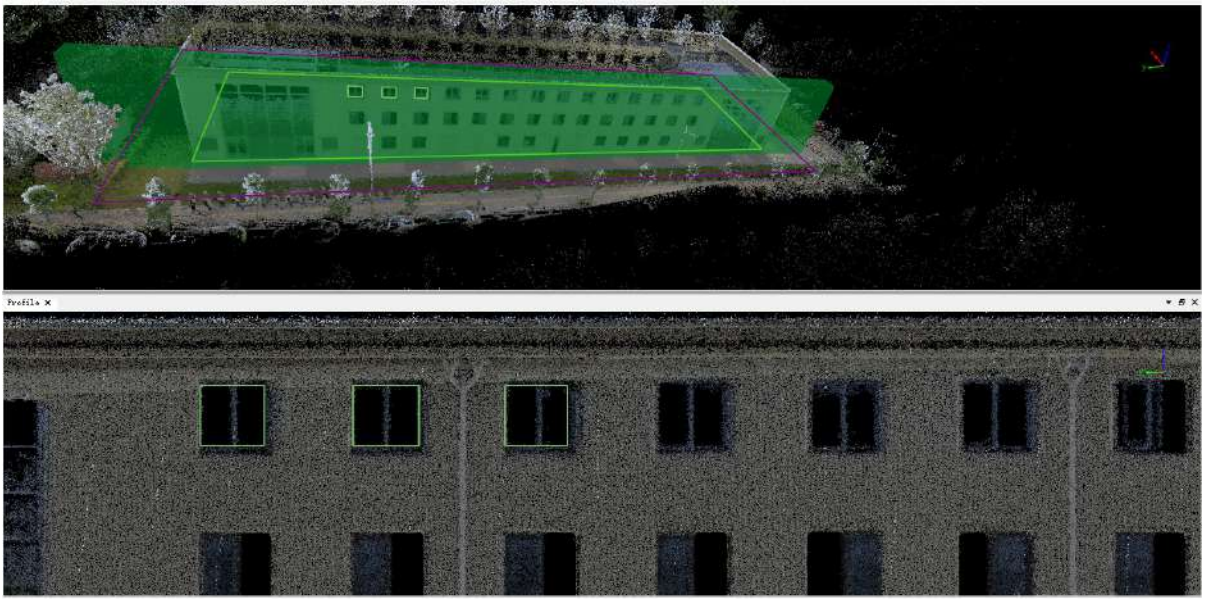
1.Activate **Add Tectonic Lines**  , left click to add horizontal tectonic line, ctrl + left click to add vertical tectonic line, and select the surface layer to be drawn.



2.Click **Mark Vectors**  2.to automatically generate a rectangle in the middle of the four tectonic lines, which can mark multiple vectors continuously. (When adding a tectonic line, the right mouse button is used to roll back the previous tectonic line, and you can roll back multiple times)



3.Click **Clean Tectonic Lines**  to display only the generated vector.

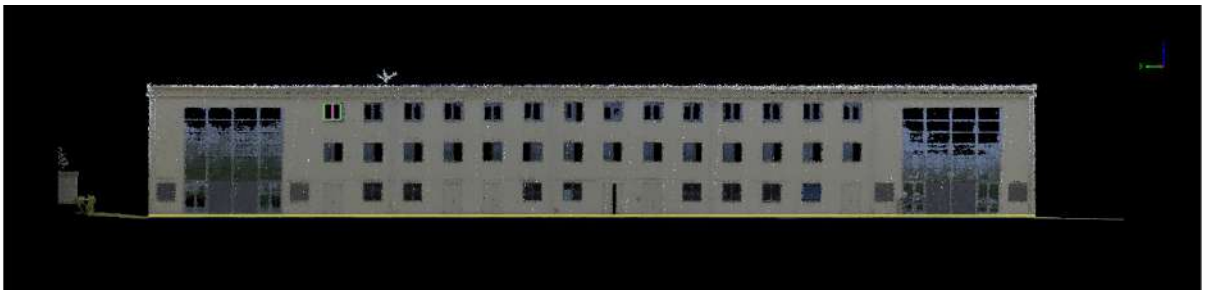


Array

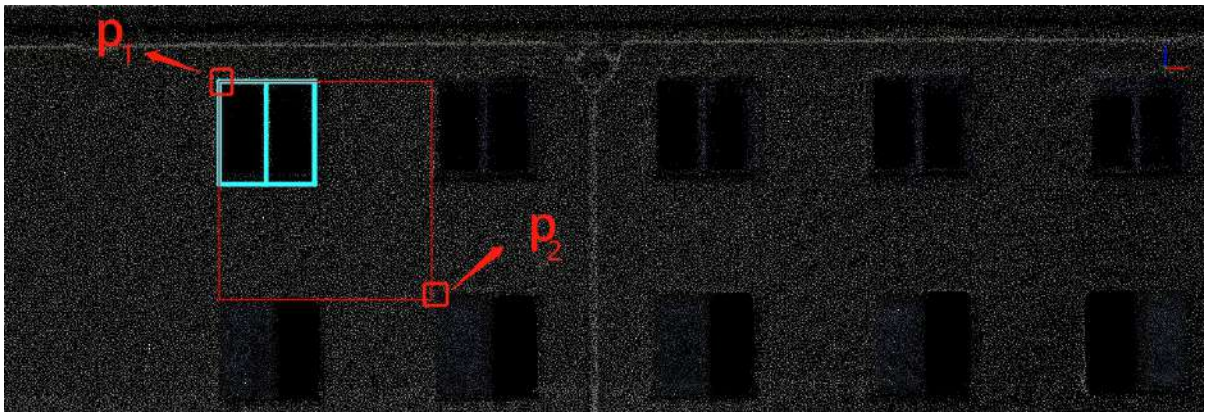
Function Description: This tool allows the user to quickly draw elements with the same structure arranged at equal intervals.

Steps

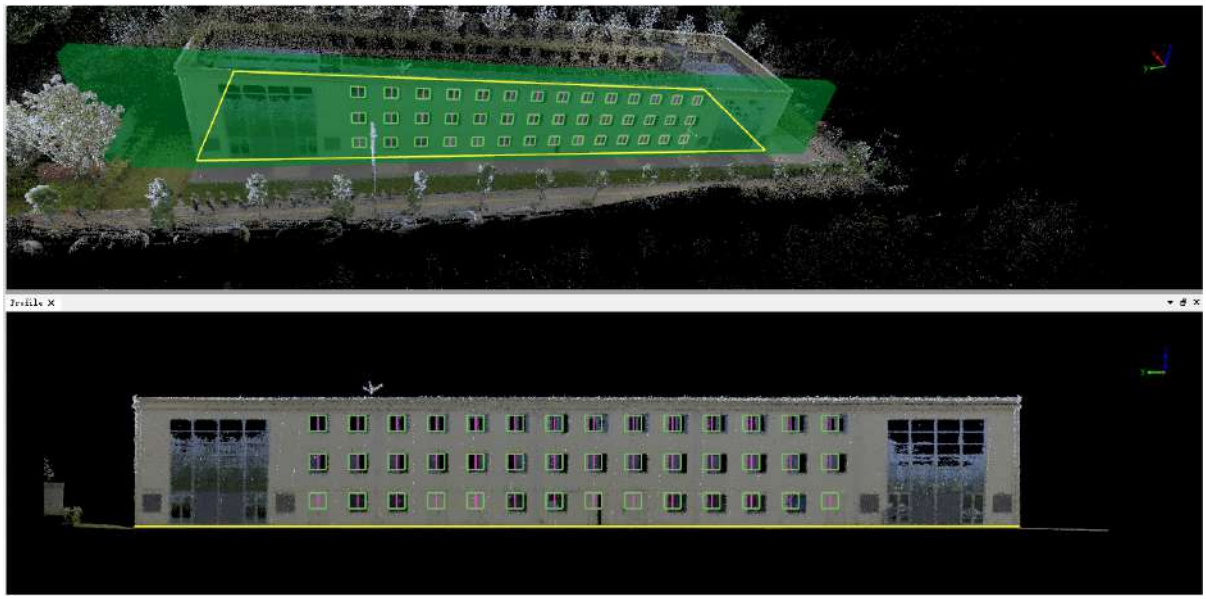
1.As shown in the figure below, the side façade windows are arranged nearly equally spaced and have the same window structure.



2.At this point, you can use the "Array" button, activate the tool, box select the objects that need to be arrayed with the left click, and then click "Enter" to combine them, As the figure shown below, click left mouse button to drag a box. The first point is the starting point, and the second point is the first point that needs to be copied. You can re-pull the box several times to select the position. After determining the length and width of the box, click "Enter" to lock it.



3.And then drag the mouse in the direction of the box or in the opposite direction until all the arrays are completed, and double-click the mouse to complete the array. This enables you to draw a whole wall of windows and their interior elements very quickly.

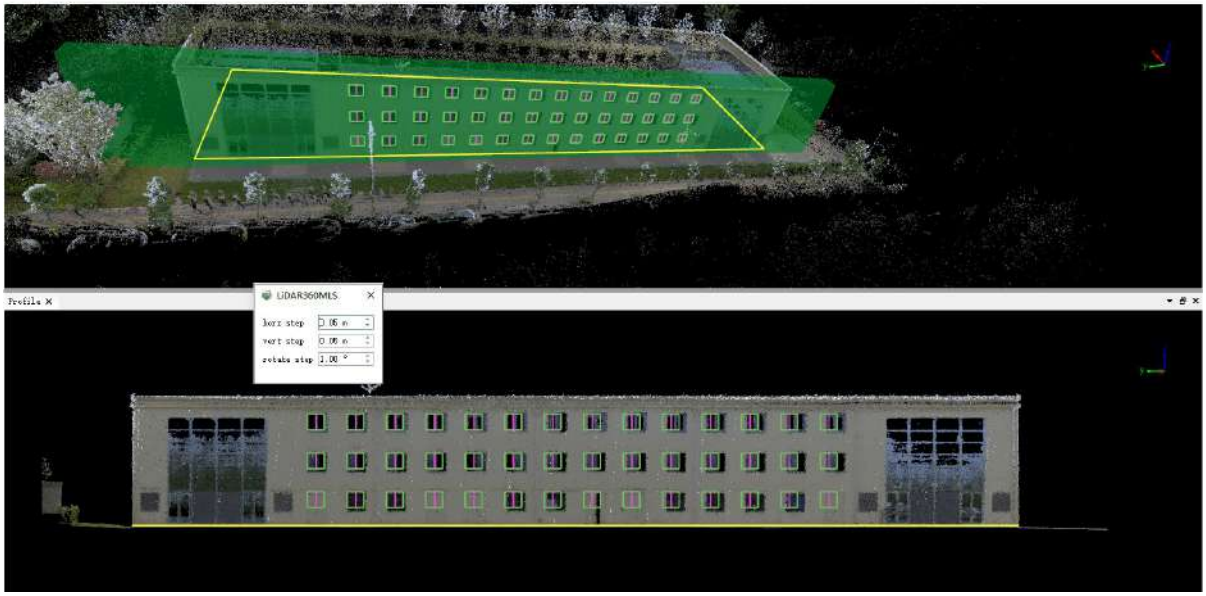


Facade Move Rotate

Function Description: If some elements are not drawn accurately, this allows the user to utilize motion and rotation to compensate and adjust the drawn elements to improve their accuracy.

Steps

1.Activate **Facade Move Rotate**  , box select the elements that need to be fine-tuned with the left click to adjust the movement and perform rotation compensation.



2.Use the up, down, left and right keys on the keyboard to fine tune the position, Q(q) /R(r) to rotate counterclockwise / clockwise, and press "Enter" to lock after adjustment:

3.After adjustment, the changes are shown in the figure below:

Copy Vector

Function Description: This tool allows the user to draw elements to any position of the current profile.

Steps

1.Click "Copy Vector"  button in Editing activate the copy vector function.

2.Press and hold the left mouse button, box select one or several objects to be copied, and then click “Enter”.

3.You will see that the selected objects have been grouped together and moved with the mouse.

4.Then move the mouse to the location where you want to copy, and click the left mouse button to complete the copy process. The left mouse button can always be used to click and place.

5.After all copies are completed, right-click to exit copying.

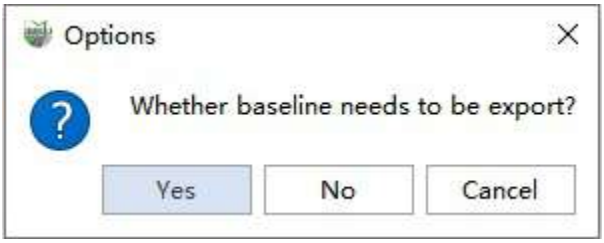
Export

Function Description: The export tool allows the user to export the engineering data into various formats as the input of third-party software for data display or further data processing.

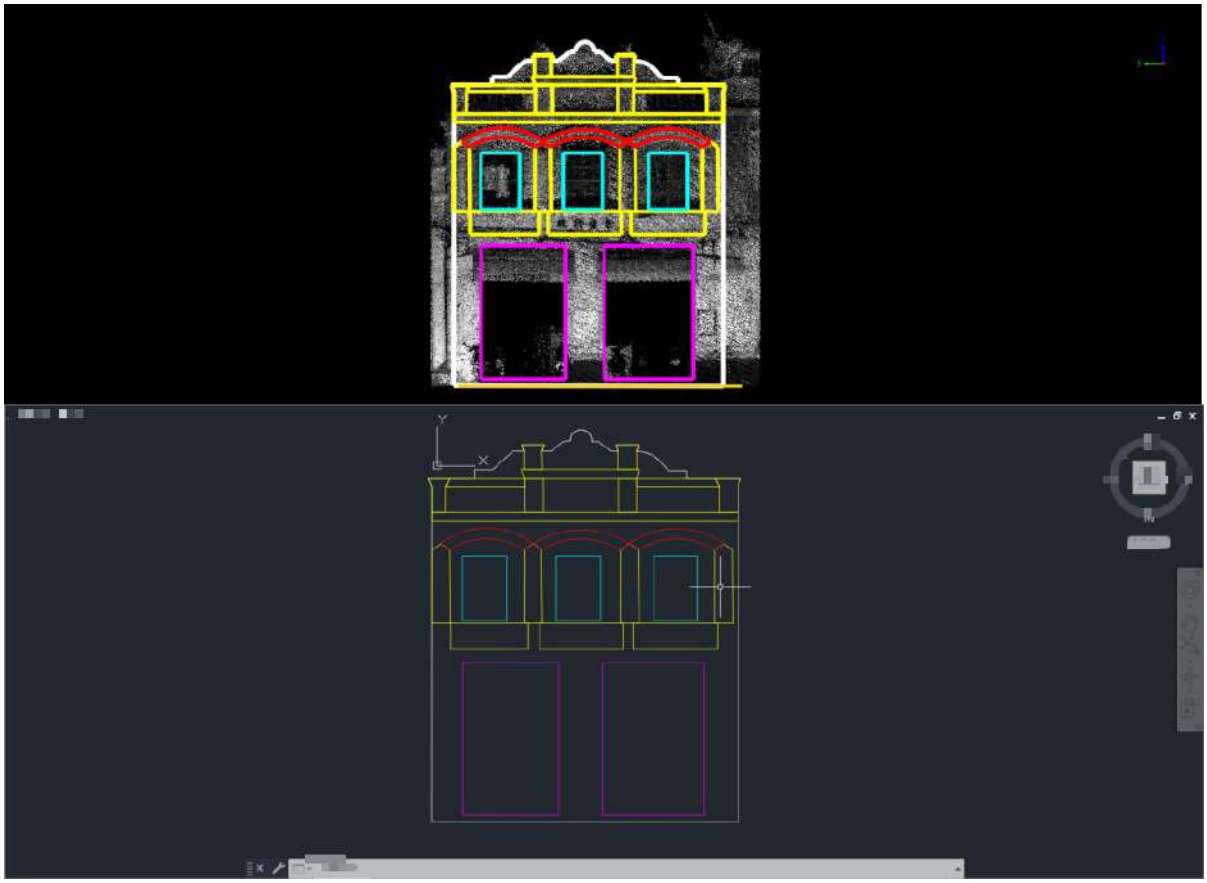
Steps

Export DXF format

1.Click the **Export DXF**  function, and you will be prompted whether to save the baseline.

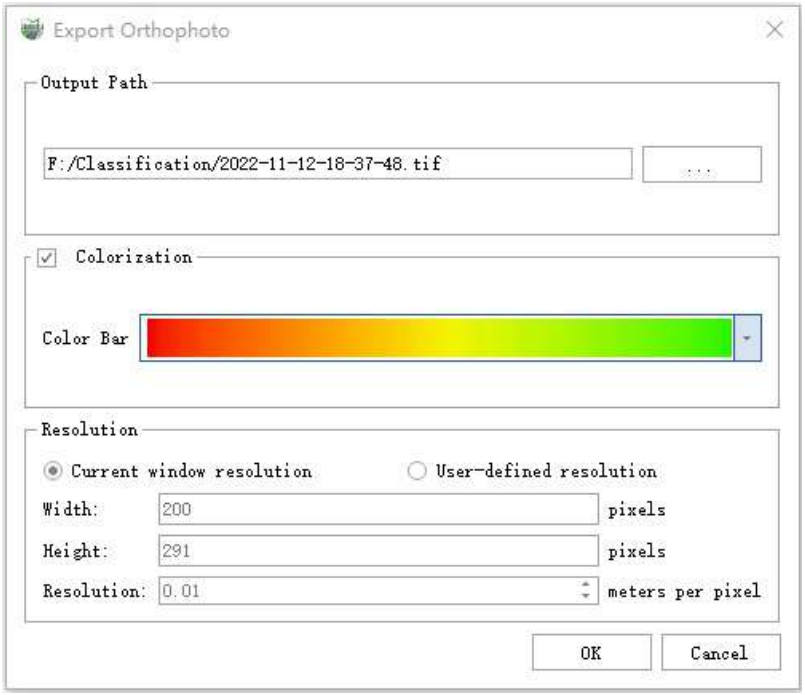


2.If you click Yes, the datum line and boundary line (if any) corresponding to the profile will be saved together, and the exported DXF can be opened with CAD.



Export Orthophoto

1.Click the **Orthophoto** function, and the dialog box for exporting orthophoto will pop up, as shown in the figure. Exporting orthophoto can be combined with horizontal section tools and vertical section tools, and the exported orthophoto map can be opened and measured in other GIS software.



2.Colorization: whether to set the render color of the exported orthophoto image as an option. You can customize the color bar for export rendering according to your needs.

3.Set resolution:

- **Current window resolution**: the orthophoto map is exported according to the resolution of the current 3D window, and the resolution will be updated and changed in real time when the mouse zooms the scene in real time.
- **User-defined resolution**：The orthophoto map is exported according to the user's input resolution.

4.After setting the resolution, click “OK” to export the orthophoto map.

5.Export the orthophoto of the current side elevation after setting colorization, as shown in the figure below.

Appearance

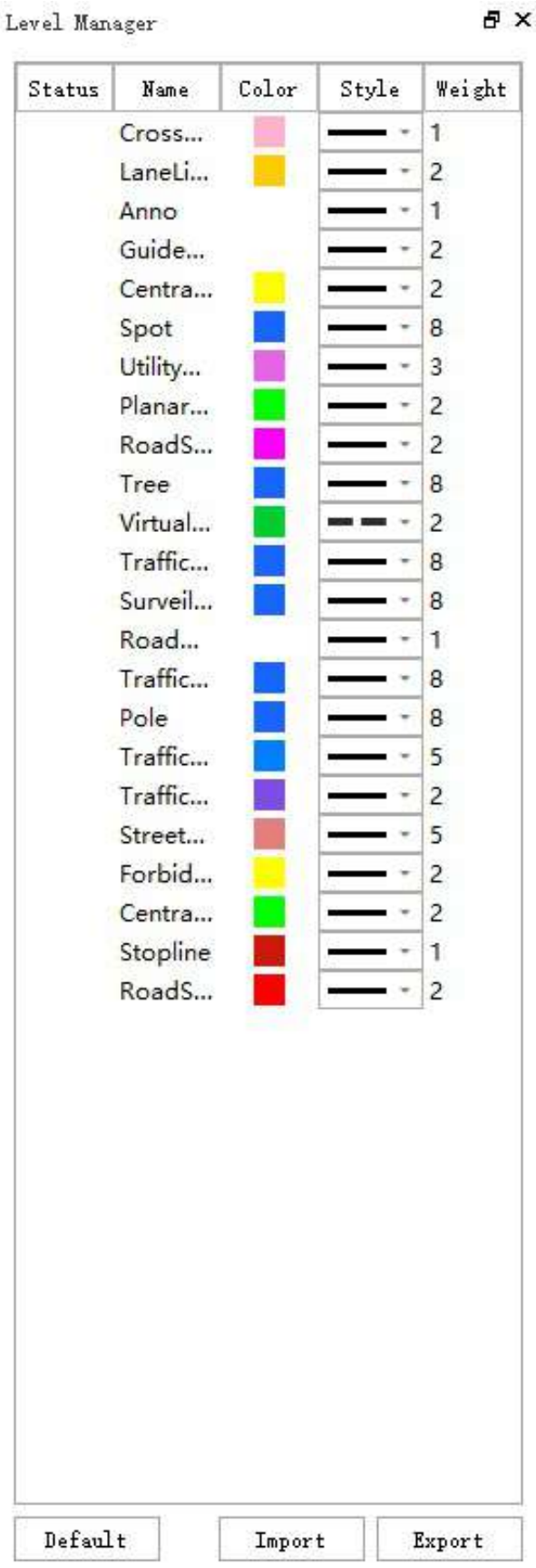
- Layer Manager
- Symbol
- Annotation
- Label Layers

Layer Manager

Feature Description： This tool allows the user to modify the color and line type of the layer and support operations such as importing and exporting layers that have already been set.

Steps

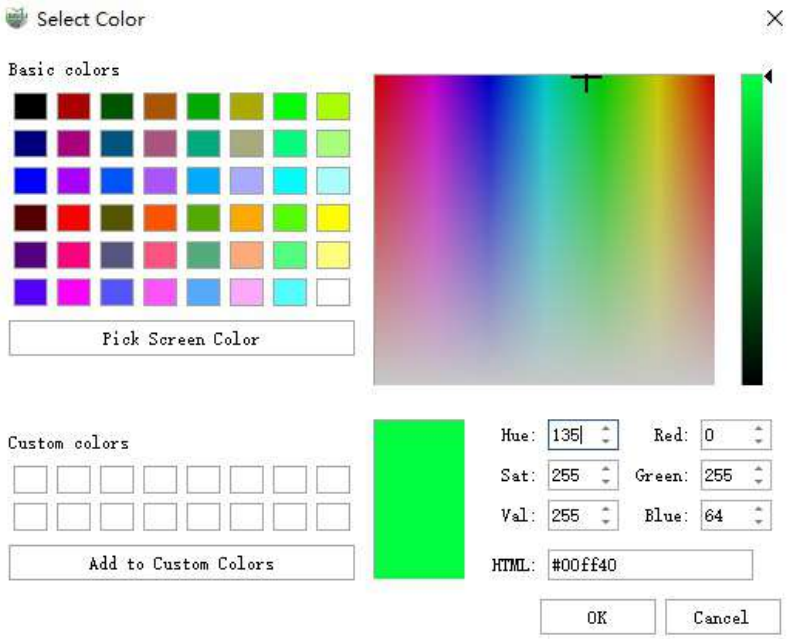
1.click **Layer Manager**  button and the **Layer Manager dialog box** will pop up.



Parameter description：

- **Name**：Displays the name of each vector.

- **Color**：Modifies the color of the layer.
  - **Double-click** to open the settings dialog box.



- **Style**：Modifies the line type of a layer.
- **Weight**：Sets the width of the line.
- **Import**：Imports**.ini**configuration files.
- **Export**：Exports the set layer and save it as a configuration file ending with *.ini* suffix.
- **Default**：Restores the default values of all parameters.

Symbol

Show Symbols

Feature Description： This tool allows the user to control the display and visibility of symbols, if point features in the project symbols have been set.

Code Table

Feature Description： This tool allows the user to open the existing built-in symbol library of the software.

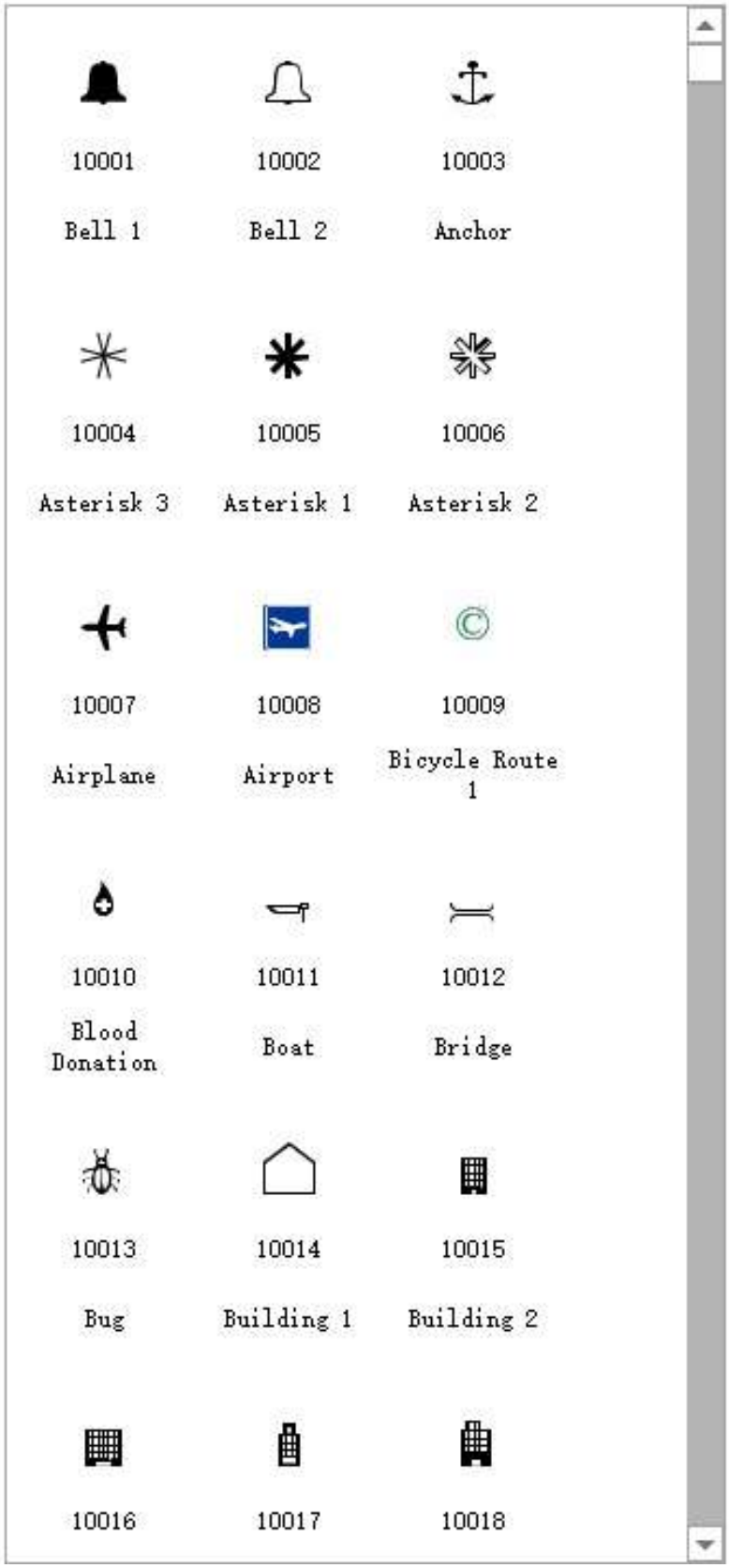
Steps

1.Click **Code Table**  button and the **Symbol dialog box** will pop up.

Symbol

Symbol Library: Universal

Symbol Query:



Parameter description：

- **Symbol Library**：Select Standard or Universal Mode
- **Symbol Query**：Enter via the text box to search for the symbol you want.

Add Feature

Feature Description：This tool allows the user to collect point features directly with symbols.

Steps

1.Click **Add Feature** button, **Code table** pops up.

2.Once you've determined which point feature layer to draw, select the appropriate symbol for your needs and collect it at the corresponding location on the point cloud.

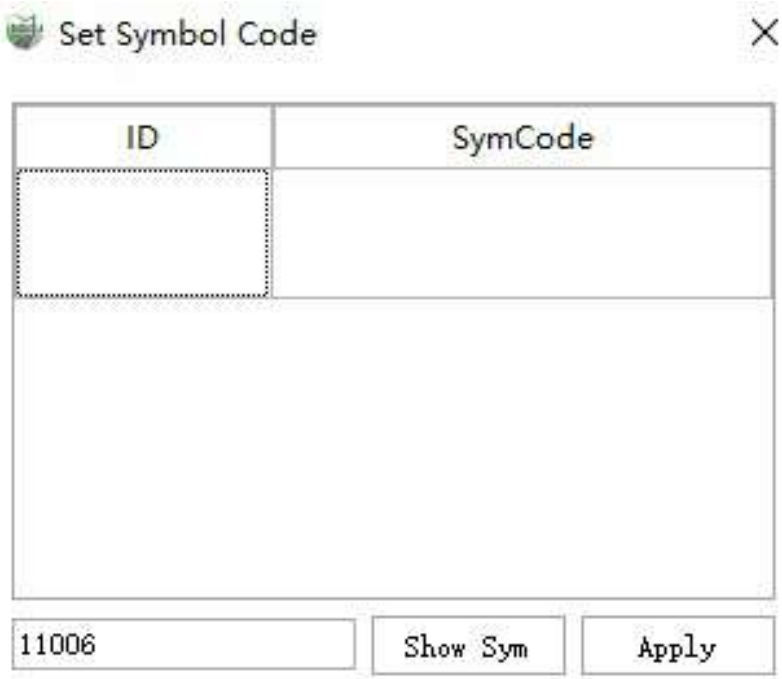
> Click **Show Symbol** button to display the icon of the corresponding symbol.

Set Code

Feature Description：This tool allows the user to modify existing symbols or set corresponding symbols for point features without symbols.

Steps

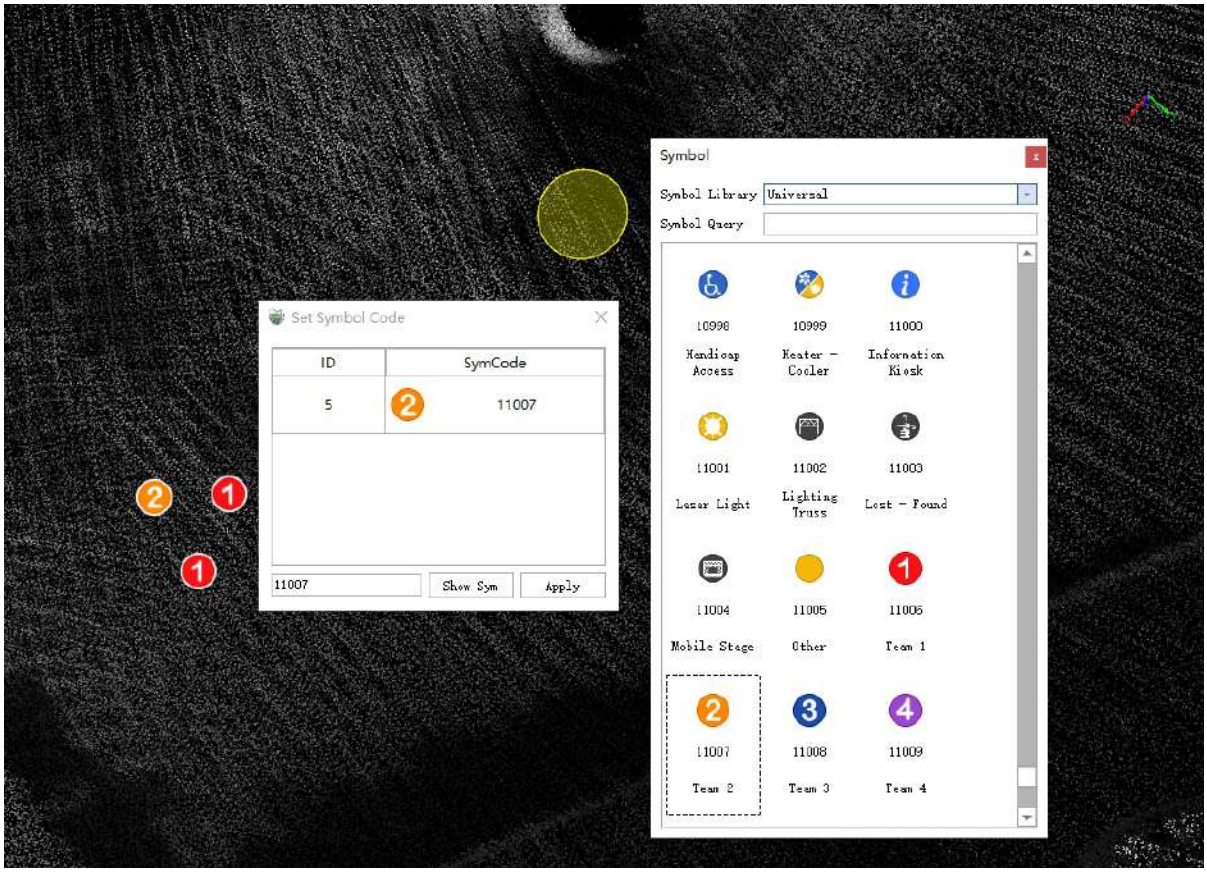
1.Click **Set Code** button and the **Set Symbol Code dialog box** will pop up.



Parameter description：

- **Show Sym**：Displays the symbol table and modifies the current symbol.
- **Apply**：Modifies the symbol.

2.Click the point feature on the point cloud to modify/set the symbol, the Set Symbol Code dialog box displays the feature ID and symbol code. You can re-select a new symbol in the symbol library or enter the code of a new symbol in the text box below the Set Symbol Code dialog box. Click Apply to complete the symbol modification or add new settings.


Set Symbol Code
ID
SymCode
5
11007
11007
Show Sym
Apply
Symbol
Symbol Library
Universal
Symbol Query
10998
Handicap Access
10999
Heater - Cooler
11000
Information Kiosk
11001
Laser Light
11002
Lighting Truss
11003
Lost - Found
11004
Mobile Stage
11005
Other
11006
Team 1
11007
Team 2
11008
Team 3
11009
Team 4

Annotation

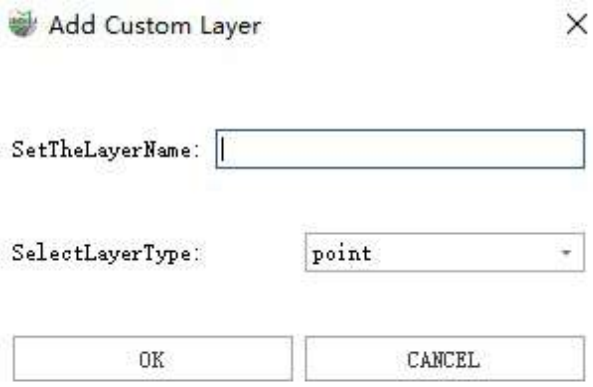
Add text annotation

Function Description： This tool allows the user to annotate the nodes/edges of point clouds, vectors, or vector objects.

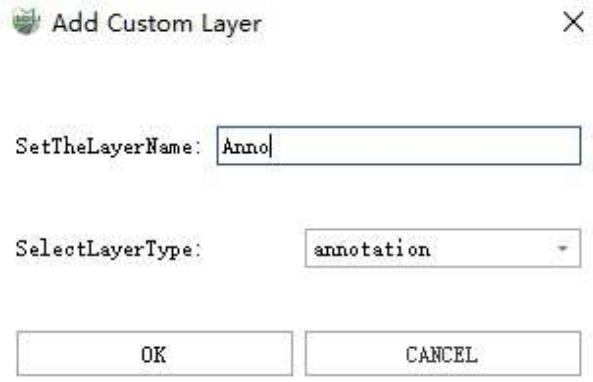
> This function will create a new annotation layer.

Steps

1.Custom vector > Right-click to add a layer, **Add Custom Vector** pops up.



2.Set the layer name as required, select the layer geometry type **Anno**, and click OK.



3.click **Add text annotation** button and the **Anno Text dialog box** will pop up.

Parameter description：

- **Font**：Sets current font.
- **Style**：Current font available styles.
- **Color**：Current font color.
- **Opacity**：Sets text transparency.
- **Attach Feature**：The current text is annotation for a feature or feature edge.
- **Apply**：Click to save the current settings.

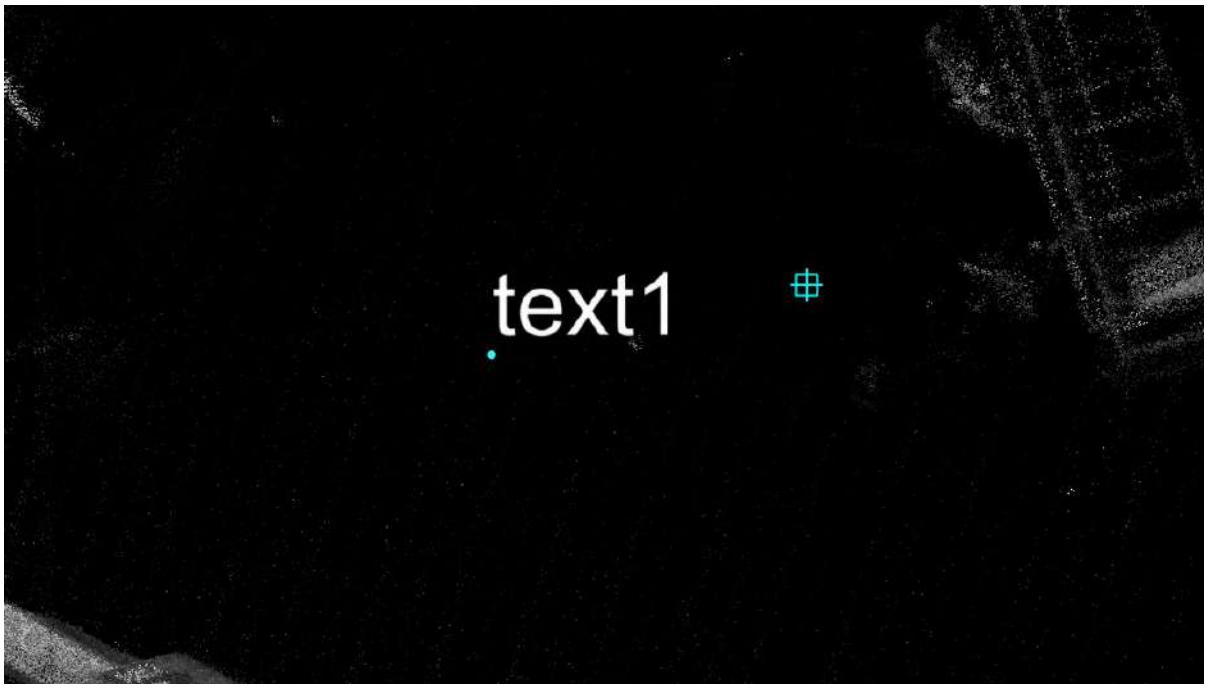
Modify Annotation

Function Description：This tool allows the user to modify existing annotations.

Steps

1.Click **Modify Annotation**  button, click the annotation you want to modify, and the **Anno Text** dialog box will pop up.

2.Re-edit in the text editing box, set the font, color, Opacity, etc, and click Apply, you to "save" the current settings. Additional element information cannot be modified.


text1

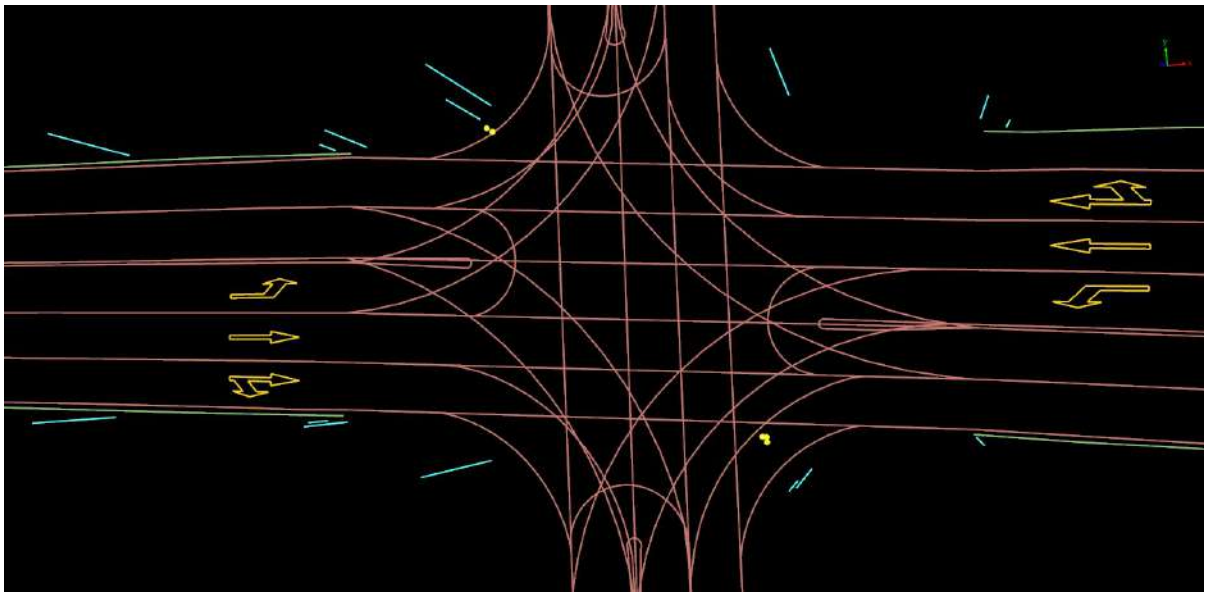
Label Layers

Function Description： This tool allows the user to set the font, size, color and other styles of the layer label, as well as the direction of the label and other operations.

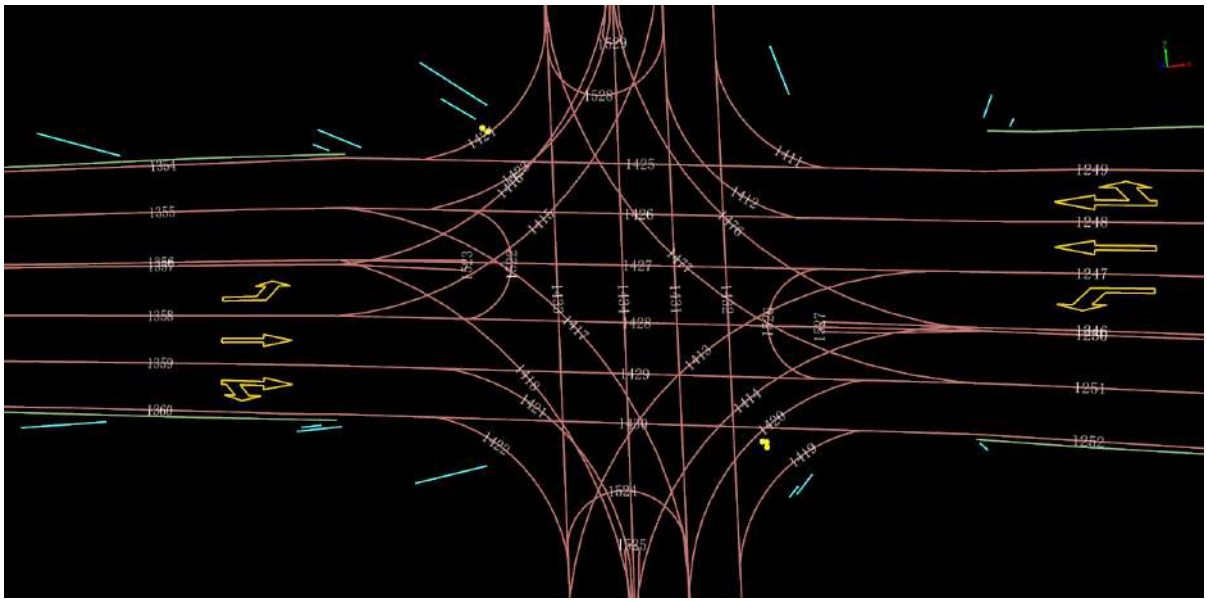
> Note: This function is used to ensure that there is vector result data in the project and that the attribute table field already has a value.

Steps

1.Right-click on the layer where the vector has been drawn in the directory tree and check the "Label" option to activate the display of the layer label.



Before the layer label is checked



After the layer label is checked

2.In the layer label menu bar, you can modify the layers and fields of the display label, and you can modify the style of the display label.

- **Layer**：Select the layer on which you want to display labels.
- **Field**：Select the corresponding field based on the layer.
- **Label Feature**：Shows or hides labels.
- **Label Direction**：Choose a view or scene.
- **Font**：Choose the style of the font.
- **Size**：Sets the size of the font.
- **Color**：Sets the color of the font.
- **Bold**：Sets the font to be bold.

> Note: Currently, label styles cannot be set by layer

Road Analysis

It is mainly used for road-related analysis work.

Mainly includes：

- Road condition analysis
  - Pavement Damage Detection
  - Pavement Damage Report Export
  - Clearance Analysis
  - Clearance Report Export
  - Observation Observation Analysis
  - Target Observation Analysis
- Road cross-section analysis
  - Road Reference Line
  - Generate Section Views
  - Show/Hide
  - Clear Sections
  - Section Analysis Window Control

Road Condition Analysis

- Pavement Damge Detection
- Pavement Damage Report Export
- Clearance Analysis
- Clearance Report Export
- Observation Observation Analysis
- Target Observation Analysis

Pavement Damge Detection

Function Description: This tool allows the user to extract the point cloud of road surface cracks, damages, etc., so as to realize damage detection.

> To ensure the correctness and accuracy of the results, it is strongly recommended to perform point cloud classification and denoising in advance.
>
> The pavement damage detection function is only applicable to point cloud data scanned by high-precision lasers such as Riegl.

Steps

1.Click **Pavement Damge Detection** button, the parameter dialog box will pop up.

Parameter dialog

Parameter Description:

- **Select File**: Check/uncheck the LiData file to be processed.
- **From Class**: Choose the ground category, incorrect selection can have a big impact on the result.
- **To Class**: Write the detected damage point to the target category. If you do not want to change the category, you can set the target category to the same category as the source category.
- **Rectangle Mode**:
  - **Roughness**: The roughness of the proposed road surface, points less than the set roughness parameter value, will not be calculated as damage. When the quality of the road point cloud is poorer and thicker, it is recommended to set the roughness to be larger, and to set the roughness to be smaller when it is better and thinner. It is not recommended to set values below 0.01 meters.
  - **Thickness**: According to the Z value of the manual frame, select point cloud data with a certain thickness to participate in the calculation, which can prevent some noise points from participating in the calculation.
  - **Fitting Length**: Set the length of the point cloud used by the reference line for calculating the depth of damage, the default is 2.5 meters.
  - **FittingDepth**: Set the width of the point cloud used by the reference line for calculating the depth of

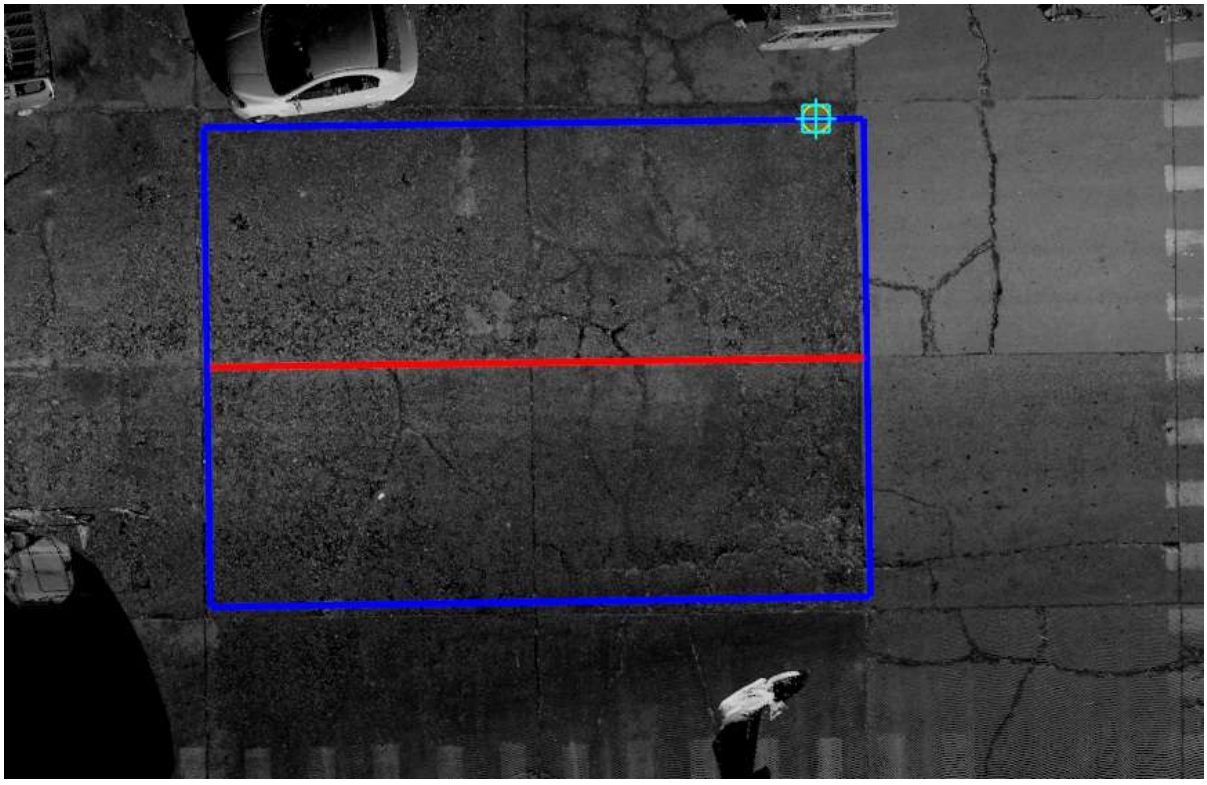
damage, the default is 0.1 meters.

> **Length is along the advance direction**; width is perpendicular to the advance direction. In the detection process, the method of block detection is adopted, and the length and width of the block generally use the default parameters, unless the damaged position is larger than the default fitting length along the direction of the frame, and needs to be adjusted.

- **Centerline Mode**:
  - **Left Width**: The distance of the left border from the selected centerline.
  - **Right Width**: The distance of the right border from the selected centerline.
  - **Thickness**: Same as rectangle mode.
  - **Fitting length**: Set the length of the point cloud used by the reference line for calculating the depth of damage, the default is 2.5 meters.
  - **Fitting width**: Set the width of the point cloud used by the reference line for calculating the depth of damage, the default is 0.1 meters.

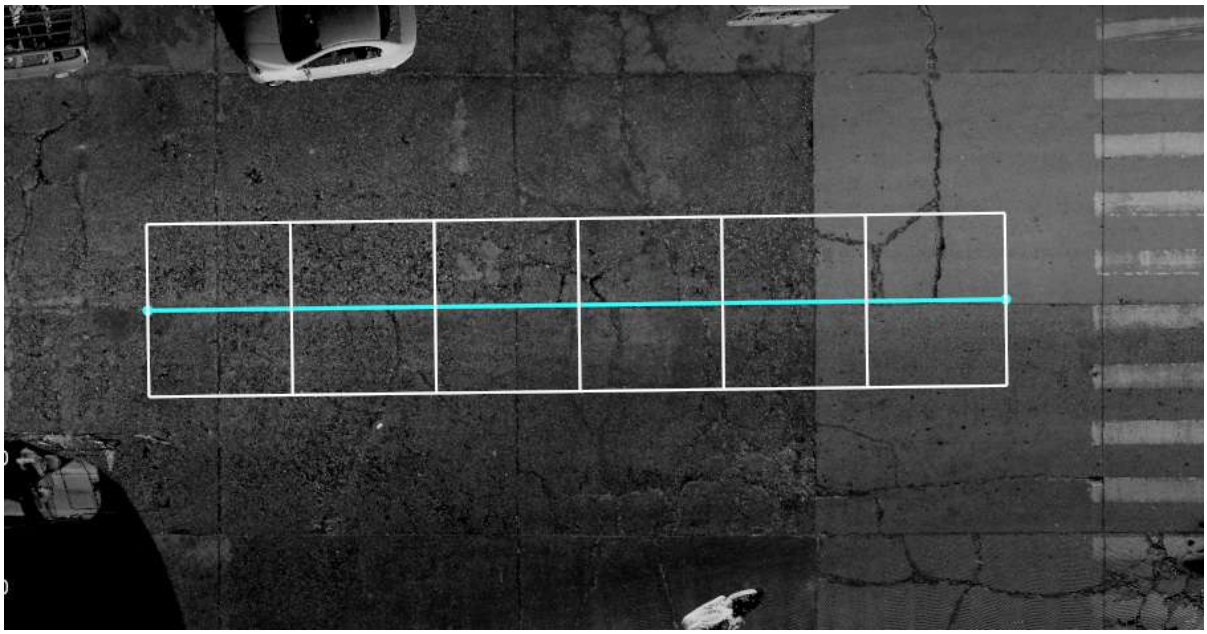
> Length is along the advance direction; width is perpendicular to the advance direction. In the detection process, the method of block detection is adopted, and the length and width of the block generally use the default parameters, unless the damaged position is larger than the default fitting length along the direction of the frame, and needs to be adjusted.

2.**Rectangle Mode**: Set the parameter three key point method, pull the rectangle frame, and calculate.



Rectangular Mode

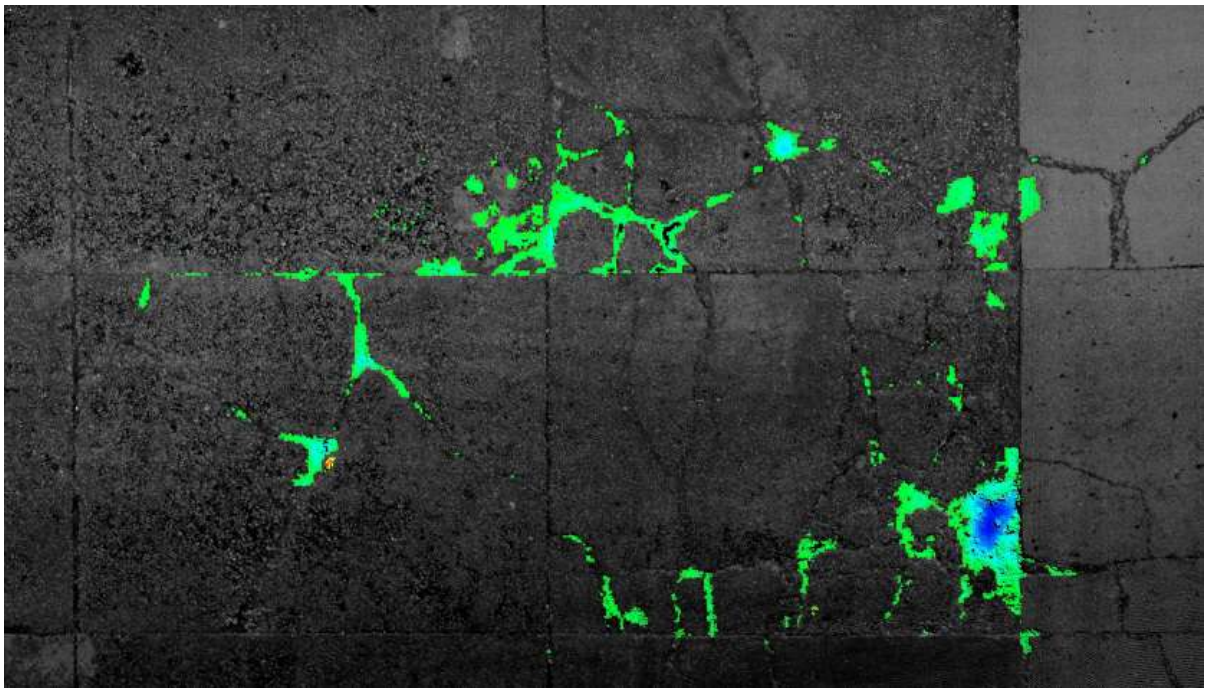
3.**Centerline Mode**: Click on an existing vector line as the centerline, adjust the left and right widths to cover the area you want to calculate, and click OK to calculate.



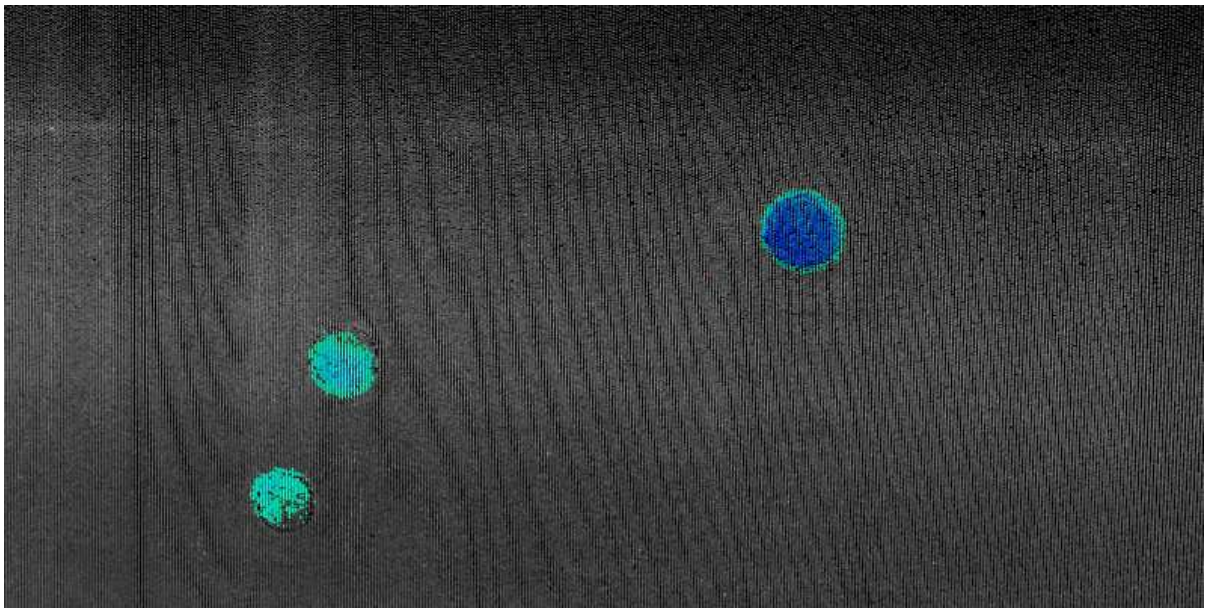
Centerline Mode

4.When the probe is finished, the view will automatically jump to the overhead view of the boxed area and a **Report folder** will be generated automatically. The report folder is located in the same directory as the current project file and the naming convention is: Project name + Report.

5.In the combined display interface, turn on the intensity display to integrate road damage, and you can clearly see the point cloud of the damaged location.

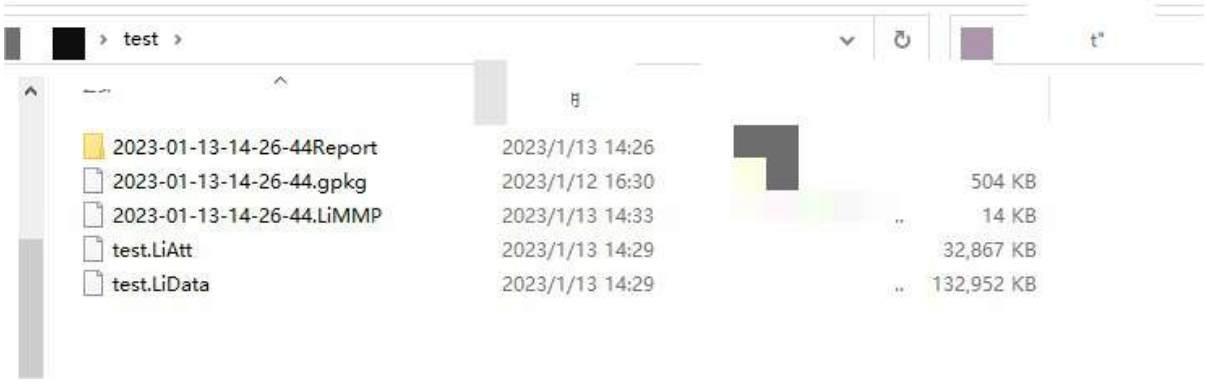


Results Display 1



Results Display 2

Example of data erasure for result display 2, where the roughness is set to 10m



Report Folder

> It is strongly recommended that no changes are made to the report folder as this may cause the report file generation to fail

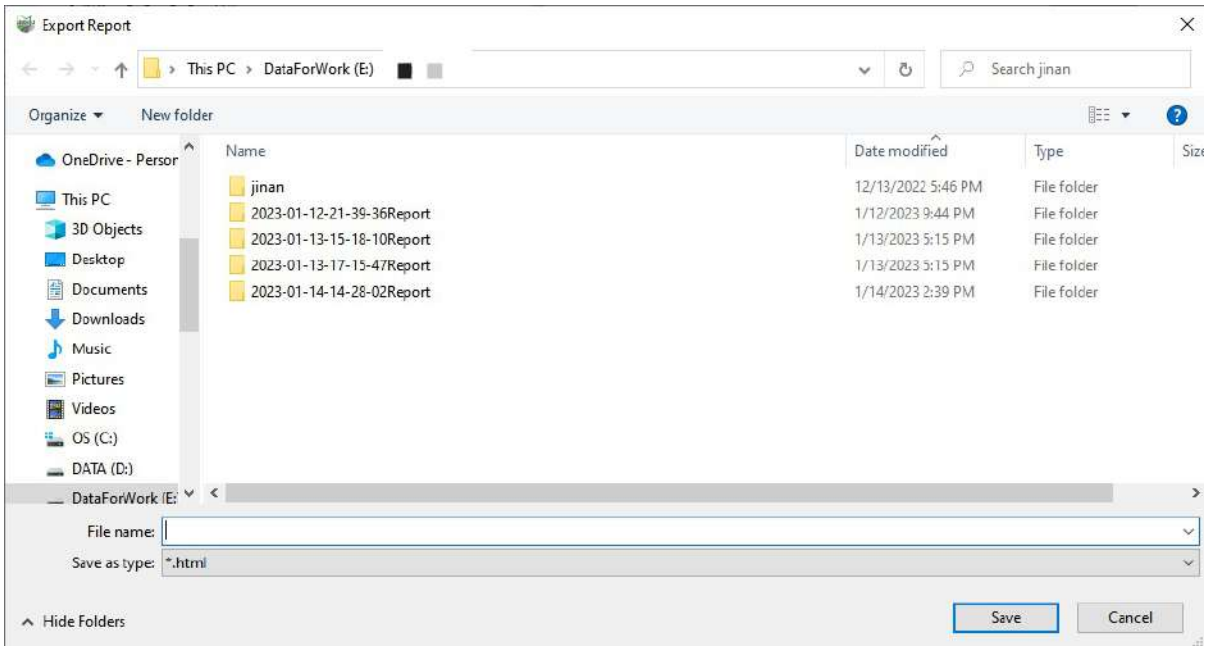
Pavement Damage Report Export

Function Description：Generates an overall report file based on the results of each previous road damage detection.

> Prerequisite for this function：there is a report folder in the current project file directory and the folder is not empty, otherwise this function is not available

Steps

1.In the Road Analysis panel click on the drop down button next to **Road Damage Detection** and select **Export Report** button.The Save File dialog box pops up, select the name of the report file and the location where you want to save it.



Save File Dialog

Report content

The content of the report is divided into three main parts: table headers, table of contents and specific content

Header and Table of Contents

Pavement Damge Report

Generated in2023.01.13

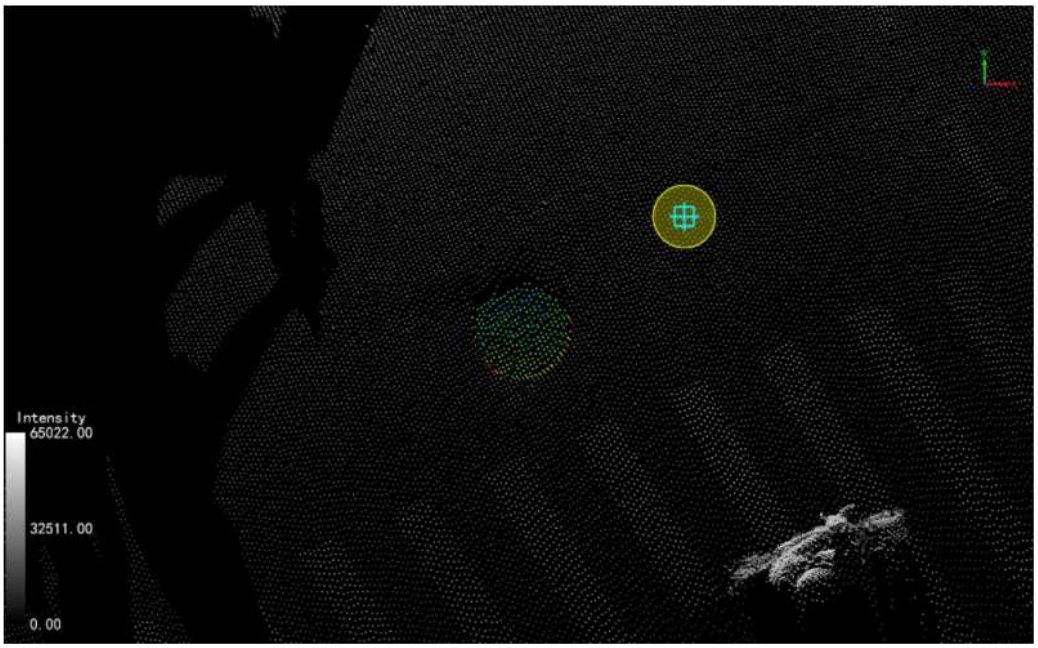
Catalogue:

Click on Region under the table of contents to jump to the corresponding location

Specific content

Input Parameters:

Generated Time	Generated in2023.01.13 17:03:02
Files	test
From Class	0、1、
Pattern	Rect
Roughness	0.010000
Thickness	0.200000
Fit Length	2.500000
Fit Depth	0.100000



Output:

Total Points	4819
Damage Points	319
Damage Points Proportion	6.619631%
Max Damage(Below)	-0.042940
Max Damage(Above)	0.000000
Danger Cluster Center	512181.051390 , 512182.241581 , 109.432273

Explanation of contents.

- **Input Parameters**: The parameters used for this probe
  - Generated Time: the time of this probe
  - Files: the name of the LiData file used for this probe

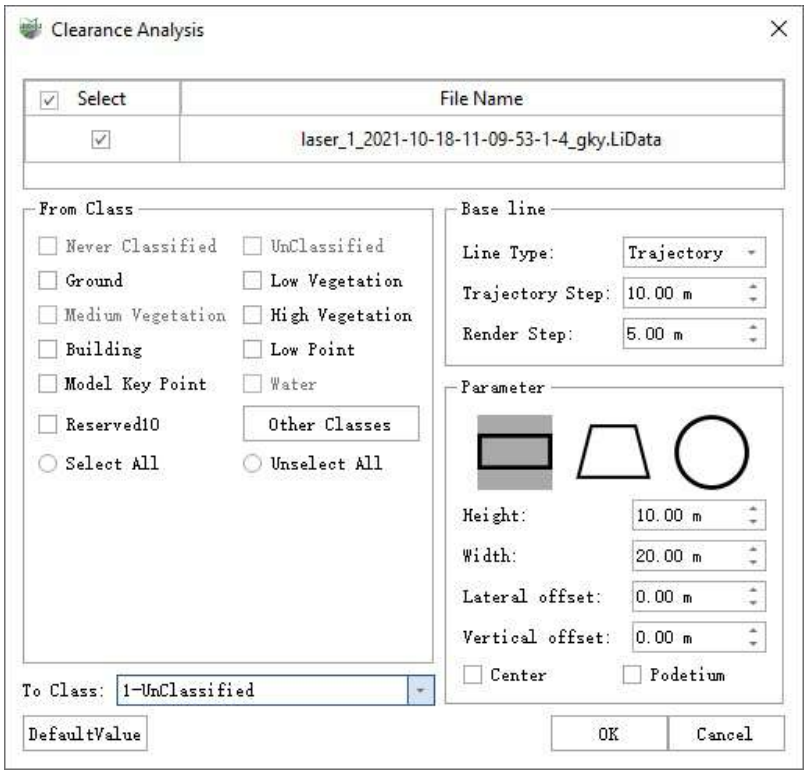
    - From Class: the source class used for this probe
    - Pattern: the detection using the box selection method, Rect for rectangular mode, Center Line for centre line mode
    - Roughness, Thickness, Fit Length, Fit Depth: represent the roughness, thickness, fit length, fit width, specific reference road damage detection
- **Output**: the statistics of this detection
    - Total Points: the number of all the points selected in this detection box
    - Damage Points: the set of damage points at this detection
    - Damage Points Proportion: the proportion of the set of damage points in this probe to the total number of points in the box
    - Max Damage (Below): The maximum damage value below the pavement for this probe
    - Max Damage (Above): the maximum damage value above the road surface
    - Danger Cluster Center: the cluster centre of the set of damaged points

Clearance Analysis

Function Description: This tool can be used to calculate whether there are occlusion points on the selected path by selecting a template and analyzing the path, and analyzing the points in the clearance area as the target category, which will be described in detail below.

Steps

1.Click **Clearance Analysis** button, **Clearance Analysis dialog** box pops up.



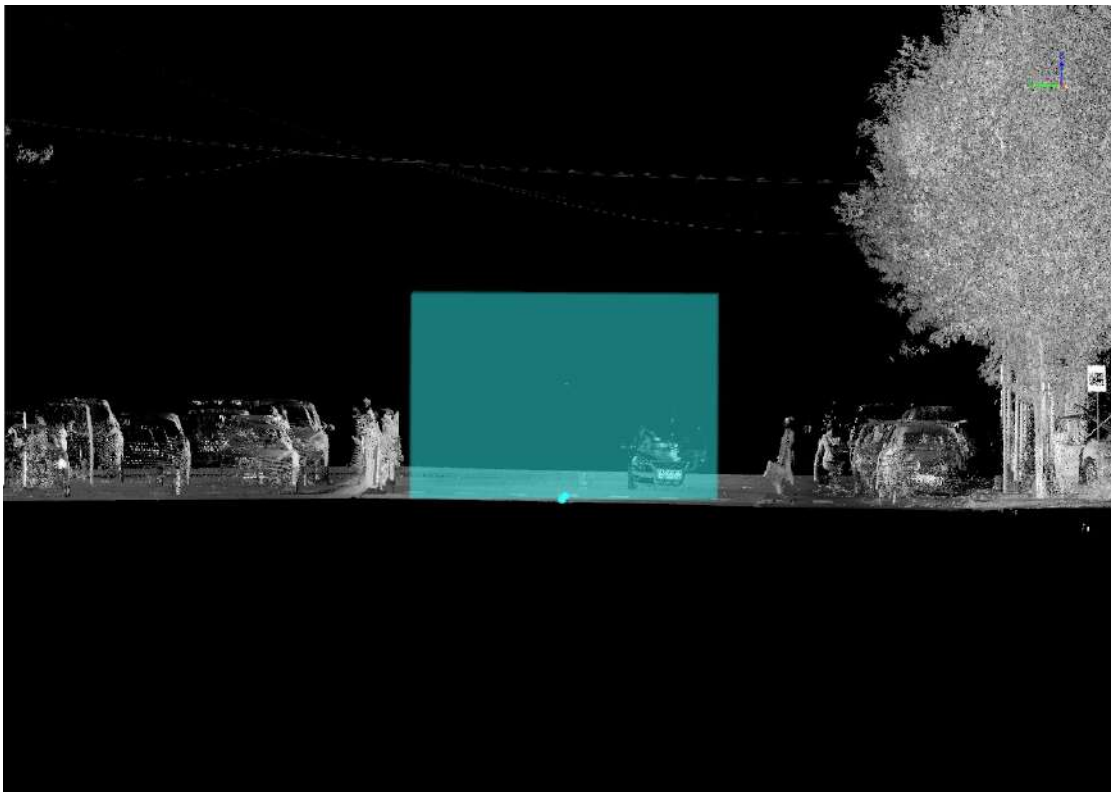
Parameter Description:

- **Point cloud selection**: Select the point cloud file to participate in the calculation.
- **From Class**: The category that participates in the occlusion calculation.
- **To Class**: If the point is occluded, the class is set to the target class.
- **Line Type**: There are three modes: Trajectory, Vector Line, Custom Line.
- **Render Step**: The distance between two display slices.

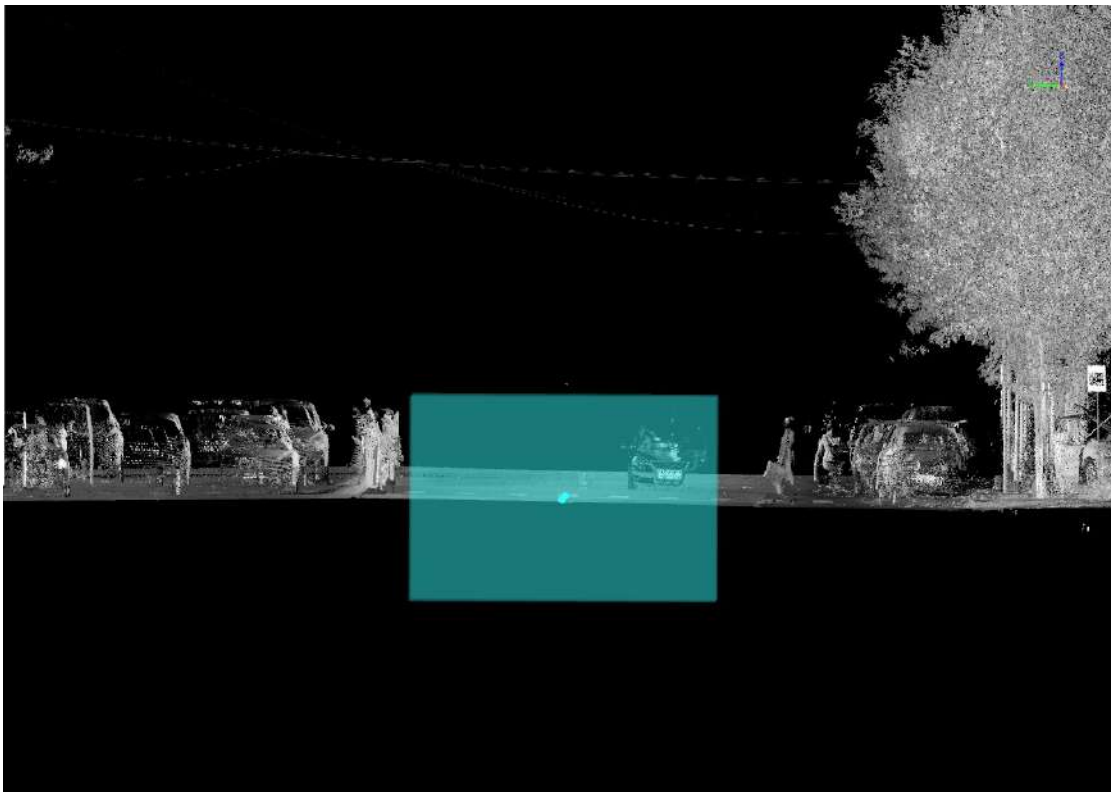
> Note: The slice position is not the actual vector node position, it is only for display.

Template selection: Three types of templates are supported: Rectangle, Trapezoid, and Circle.

- **Rectangle Mode**:
  - Uncheck the center: the position of the middle point of the lower bottom edge of the rectangle is the designated center position.



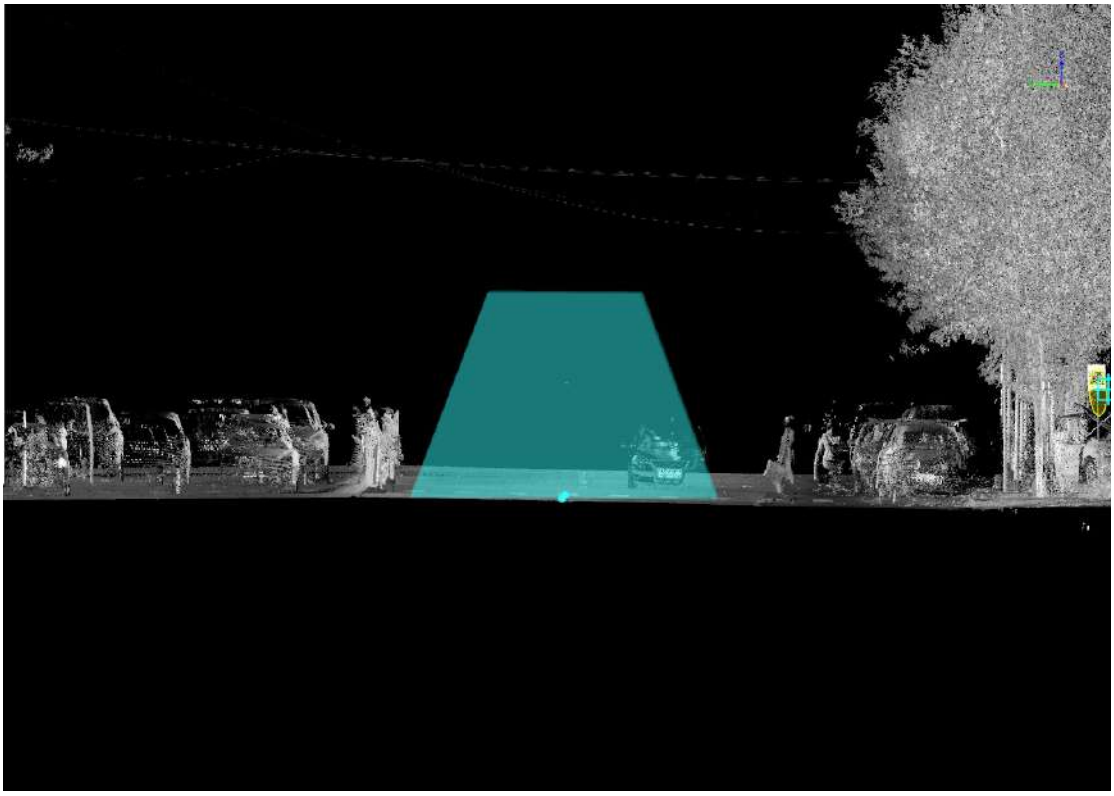
- Check the center: the geometric center position of the rectangle, which is the designated center position.



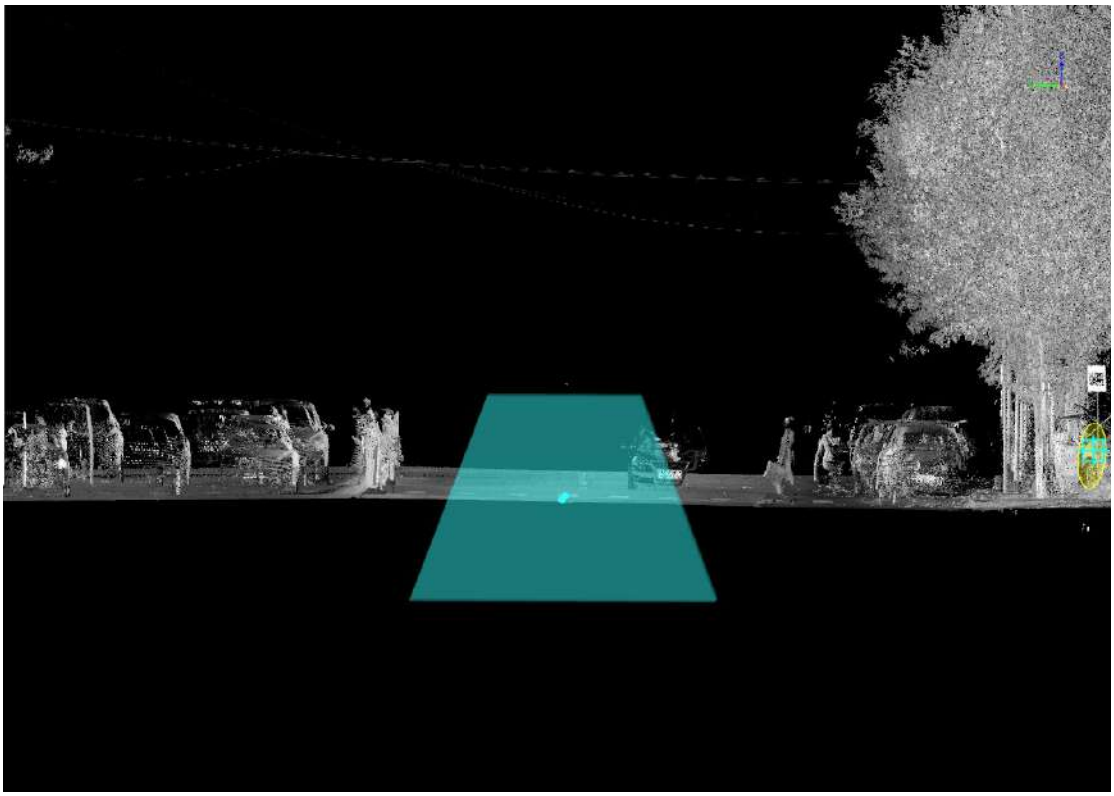
- **Height**: Used to set the height of the analysis area rectangle.
- **Width**: Used to set the width of the analysis area rectangle.
- **Lateral offset**: Using the selected baseline as the reference line, the horizontal distance between the specified center position of the rectangle and the reference line, negative values will be on the left side of the reference line advancing direction and positive values will be on the right side.

- **Vertical offset**: Using the selected baseline as the reference line, the vertical distance between the specified center position of the rectangle and the reference line, the upper portion of the reference line (Z axis) is a positive value, and the lower portion is a negative value.

- **Trapezoid Mode**:
  - Uncheck Center: The middle point position of the lower bottom edge of the trapezoid is the designated center position.

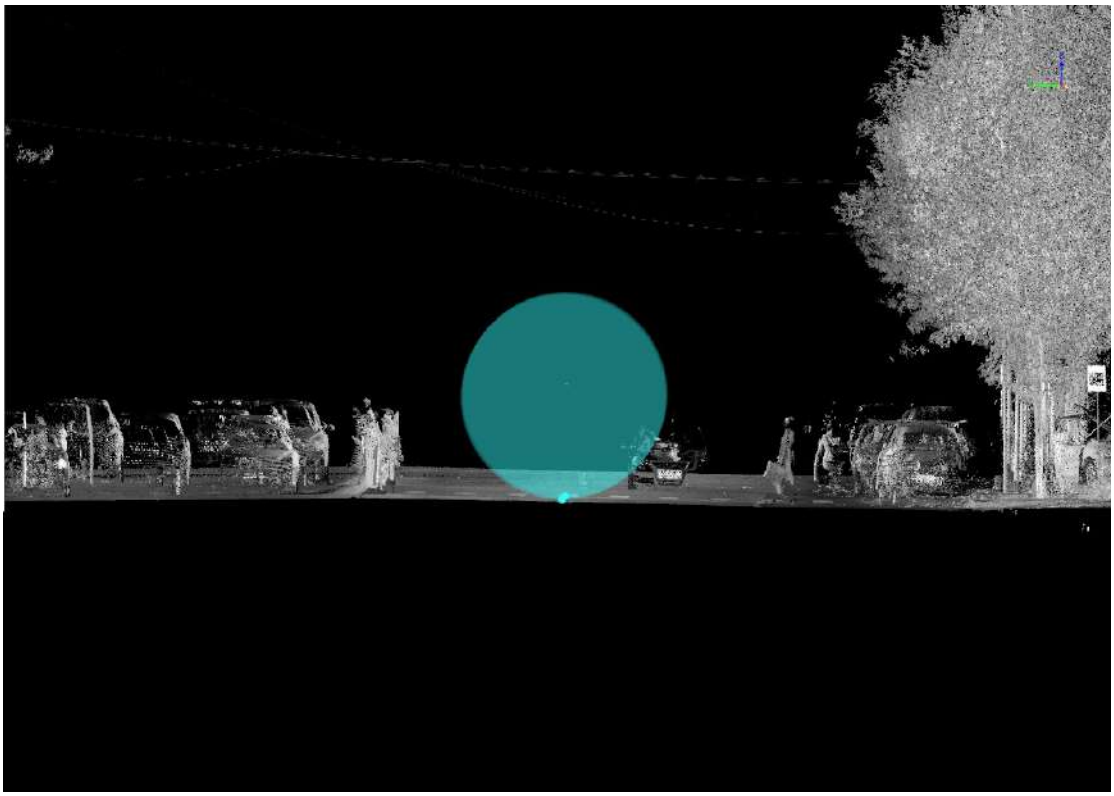


  - Check the center: the geometric center position of the trapezoid is the designated center position.

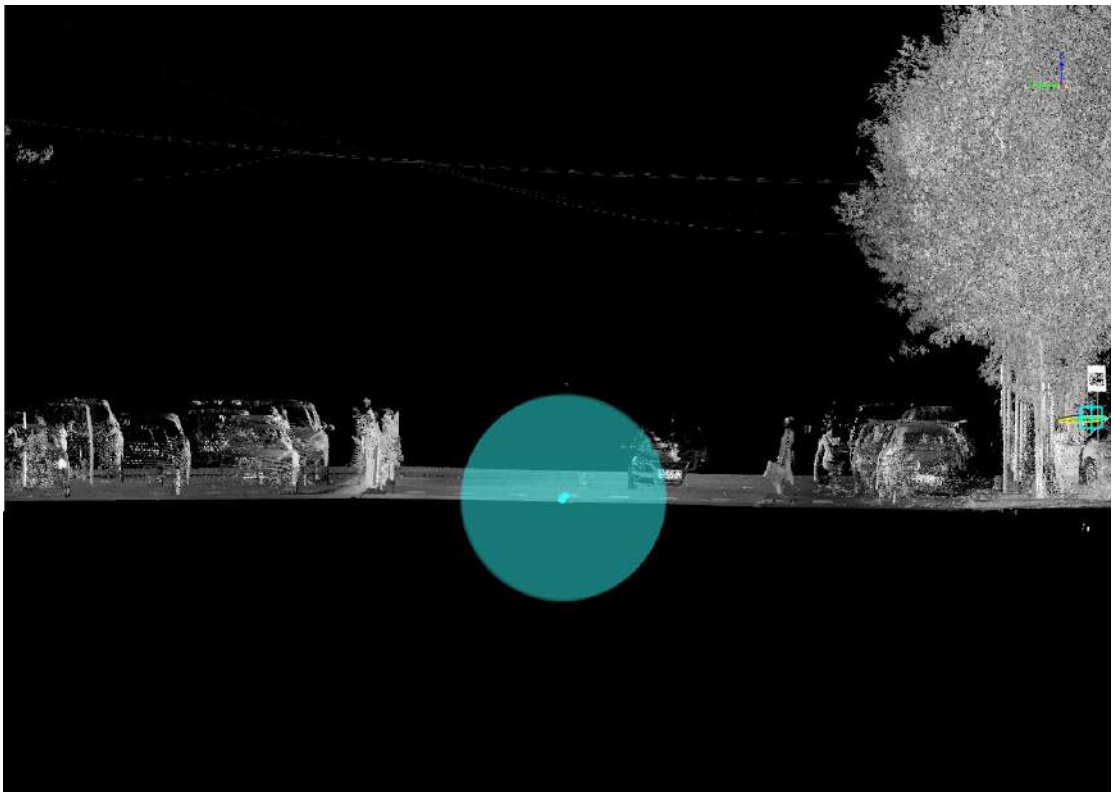


- **Height**: Same as rectangle
- **Upper base length**: Used to set the width of the analysis area rectangle.
- **Bottom base length**: Used to set the width of the analysis area rectangle.
- **Lateral offset**: Same as rectangle.
- **Vertical offset**: Same as rectangle.

- **Circle Mode**:
  - Uncheck the center: the bottom position of the circle is the designated center position.



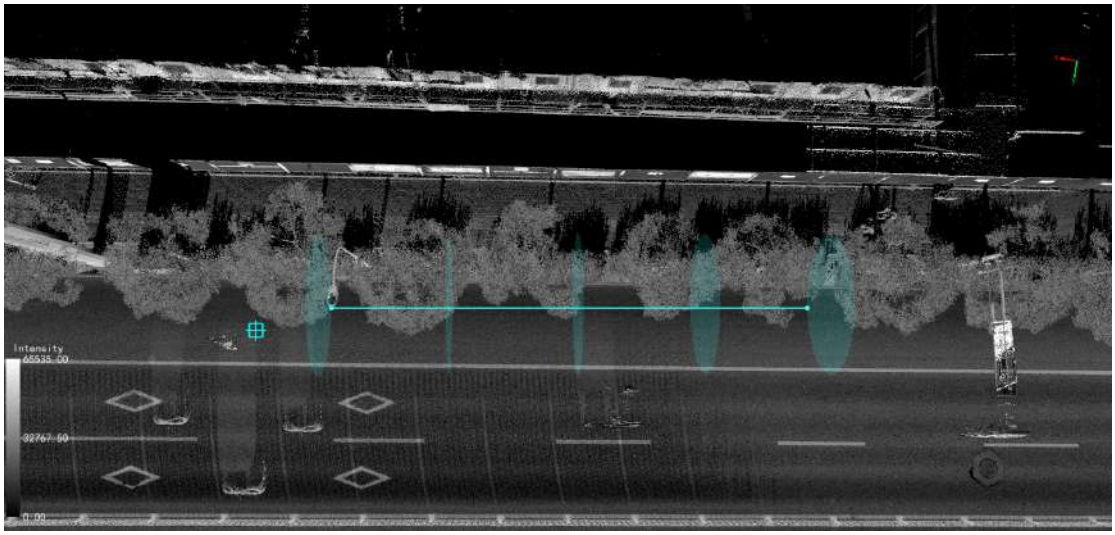
- Check the center: the center position of the circle is the designated center position.



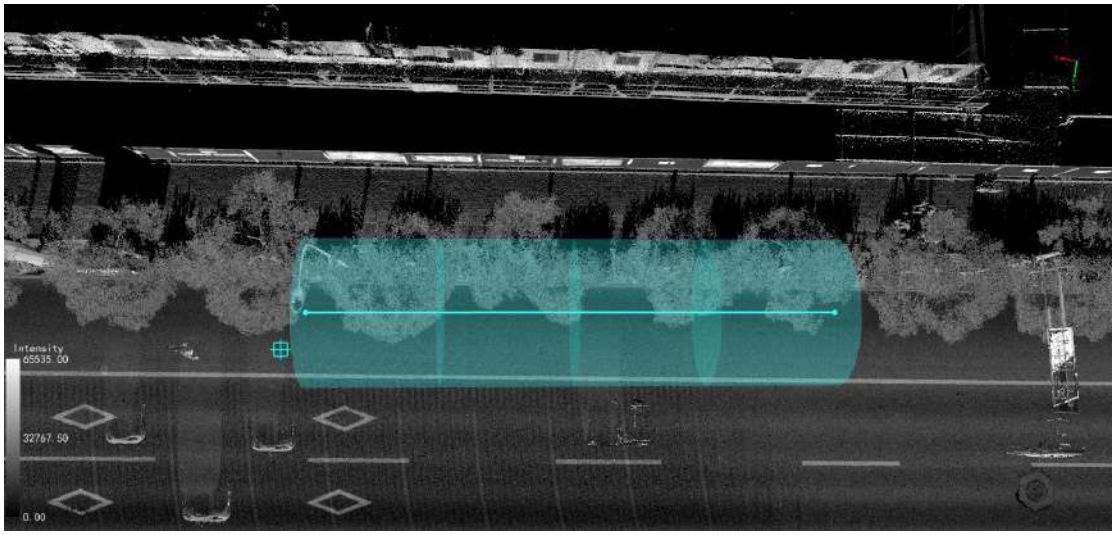
- **Radius**: The size of the radius of the circular parting.
- **Lateral offset**: Same as rectangle.
- **Vertical offset**: Same as rectangle.

- **Center**: See the three modes for details.

- **Podetium**: If unchecked, the clearance area will be displayed as a slice, if selected, it will be displayed as a cylinder.
    - Unchecked



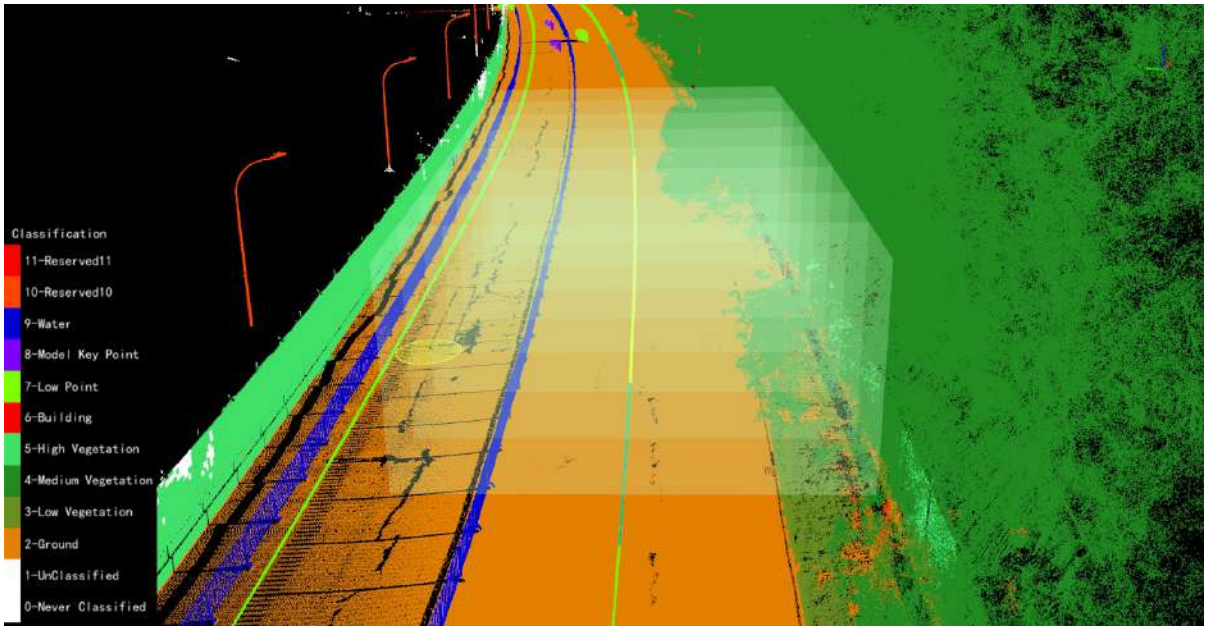
    - Checked



- **Default Values**: Click this button to restore all parameter default values.

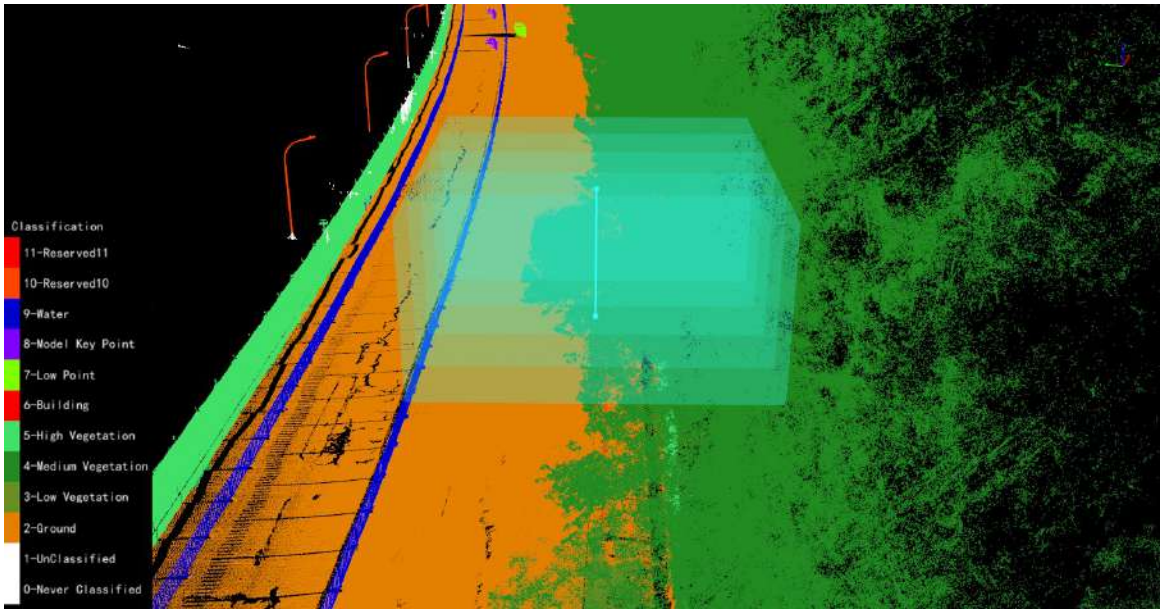
2.Select the line type (trajectory, vector line, custom line).

- **Trajectory**: The track file must be loaded when creating a new project, and then click on the starting position of the track segment to be selected, push the mouse, the track will be selected in real time, and double-click at the position to end the selection in order to select a certain segment of the track as a reference line.



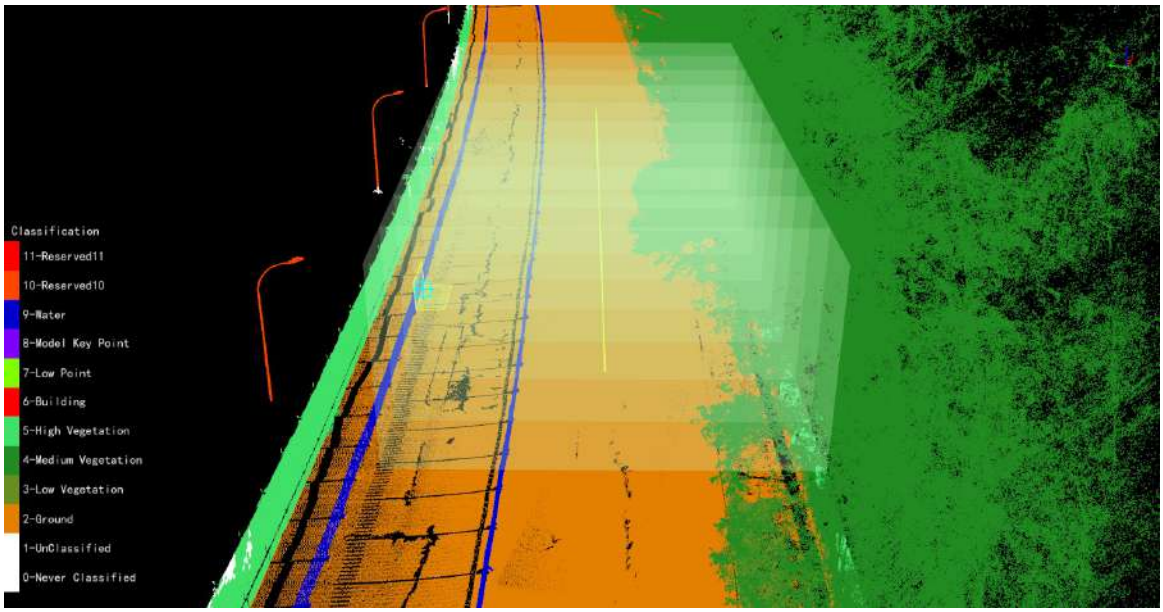
Observation point selection track

- **Vector line**: Select an existing vector line as a reference line.



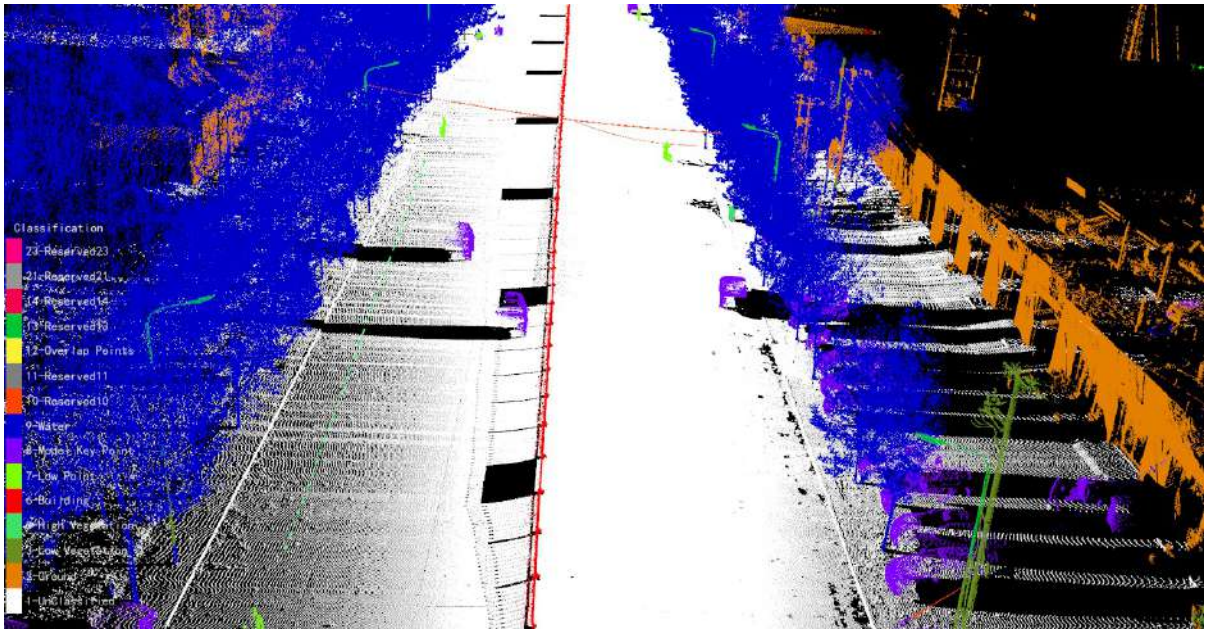
Observation point selection vector line

- **Custom Line**: After selecting a custom line, you can draw a temporary vector line at the position to be analyzed as a reference line.

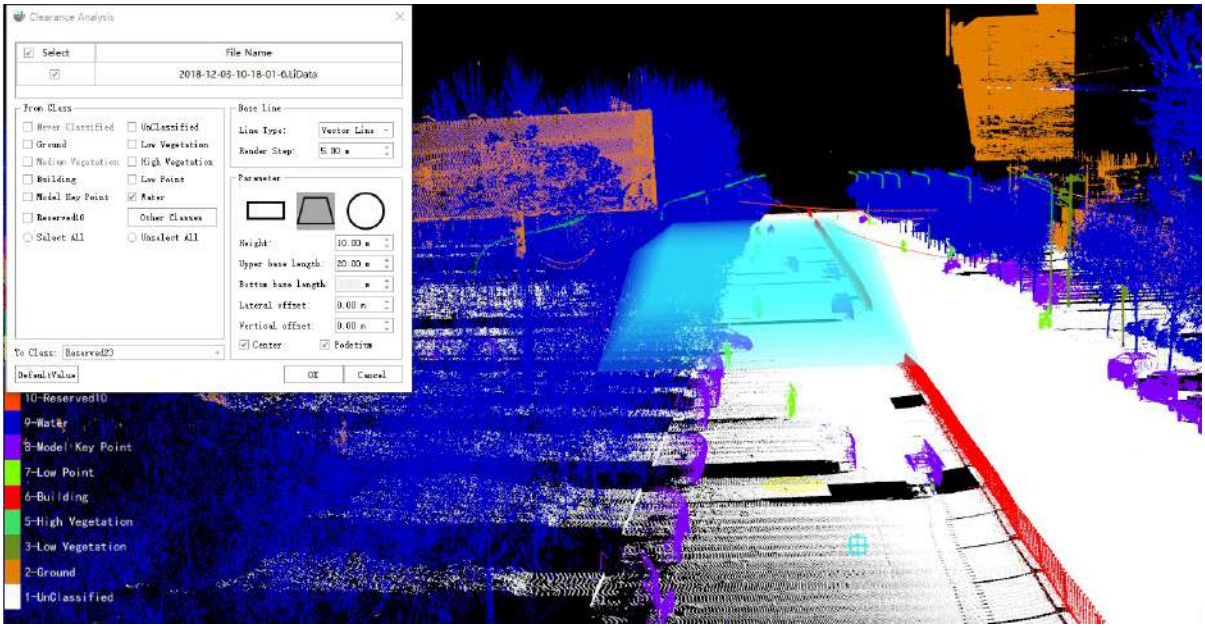


Observation point selection custom line

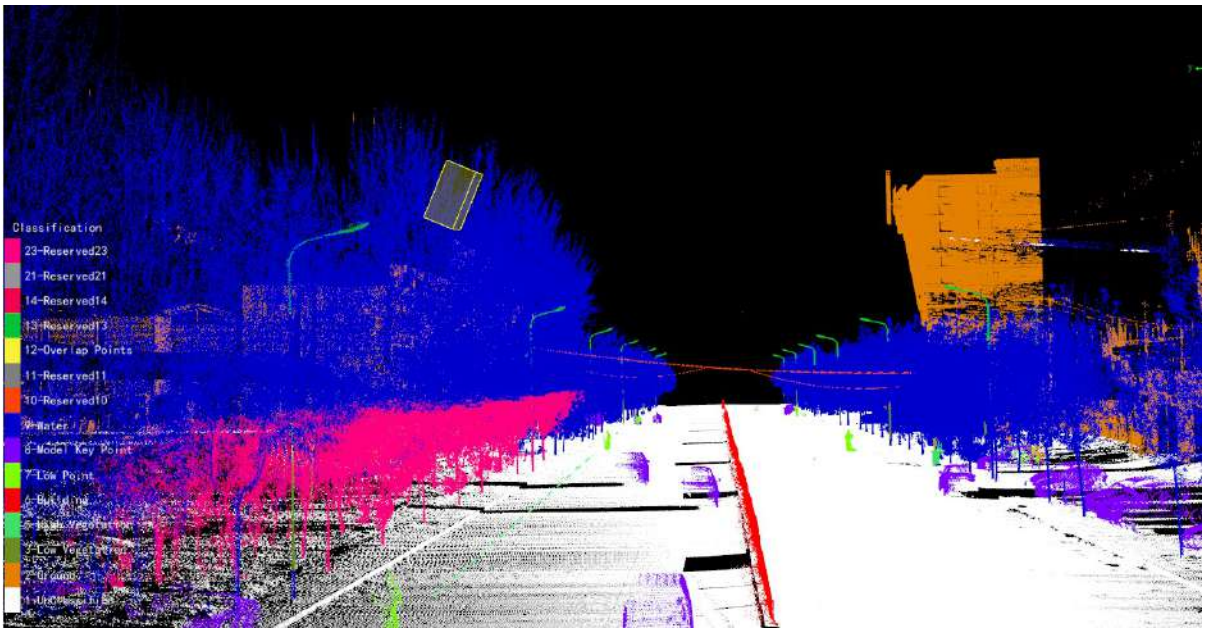
3.Select the source category, target category, and select the template (rectangle, trapezoid, circle) according to the actual needs, and set the parameters, then click OK.



Before clearance analysis

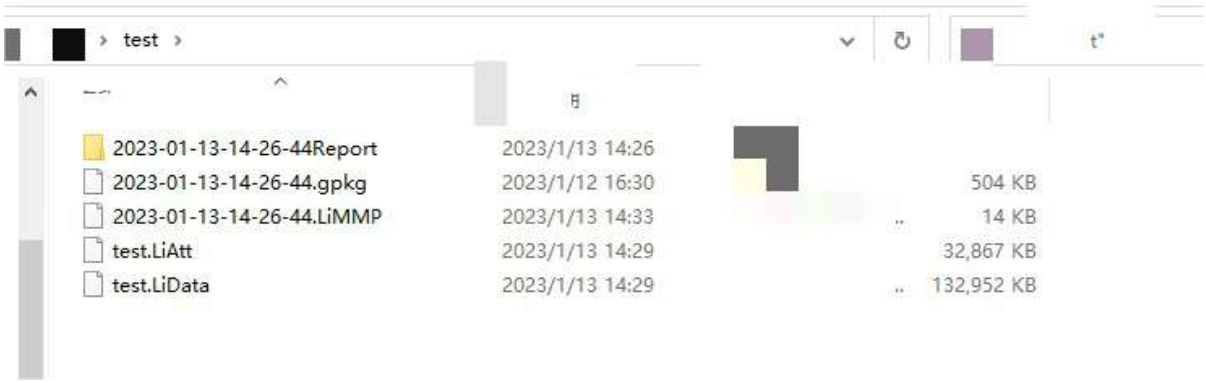


Select the template, after setting the parameters



Results after analysis

4.At the end of the analysis, the viewpoint will automatically jump to the overhead view of the boxed area and a **Report folder** will be automatically generated. The report folder is located in the same directory as the current project file and the naming convention is: Project name + Report.



Report Folder

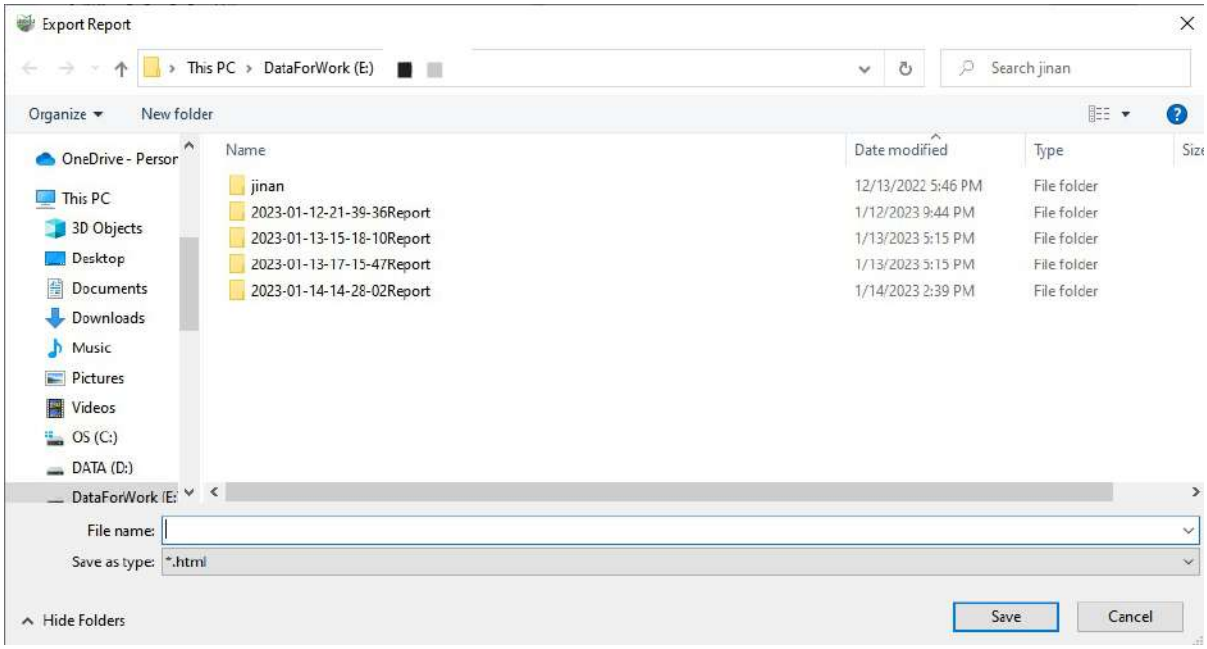
Clearance Report Export

Function Description：Generate an overall report file based on the results of each previous analysis.

> Prerequisite for this function：there is a report folder in the current project file directory and the folder is not empty, otherwise this function is not available

步骤

1.In the Road Analysis panel click on the drop down box next to **Clearance Analysis** and select **Export Report**  button to bring up the Save File dialog box, select the name of the report file and the location where you want to save it.



Report content

The content of the report is divided into three main parts: table headers, table of contents and specific content

Header and Table of Contents

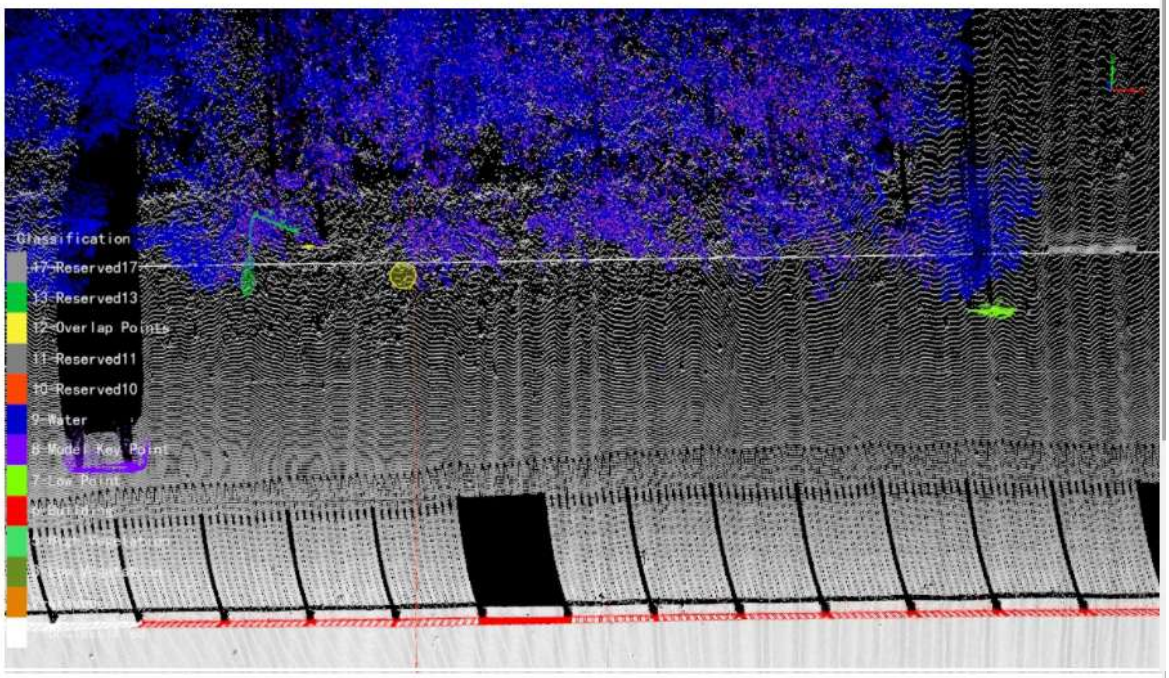
Click on the Region under the table of contents to jump to the corresponding location

Specific content

Region 1

Input Parameters:

Generated Time	Generated in2023.01.13 18:36:47
Files	2018-12-03-10-18-01-6.LiData
From Class	9、
Base Line Type	Custom Line
Render Step	5.000000
Base Line Position	Bottom Middle
Pattern	Rect
Height	10.000000
Width	20.000000
Lateral Offset	0.000000
Vertical Offset	0.000000



Output:

Total Points	2522025
Danger Points	119407
Danger Points Proportion	4.734568%
Danger Cluster Center	593605.875000 , 4211417.000000 , 32.273766 593603.562500 , 4211417.500000 , 27.642365 593597.187500 , 4211419.500000 , 27.797459 593601.312500 , 4211414.000000 , 34.121826 593608.562500 , 4211410.500000 , 29.096058 593613.812500 , 4211421.500000 , 35.059727 593604.437500 , 4211416.500000 , 27.483723 593608.125000 , 4211416.500000 , 27.471399 593601.312500 , 4211412.000000 , 33.613480 593611.562500 , 4211419.500000 , 37.170547

Explanation of content.

- **Input Parameters**: The parameters used for this probe

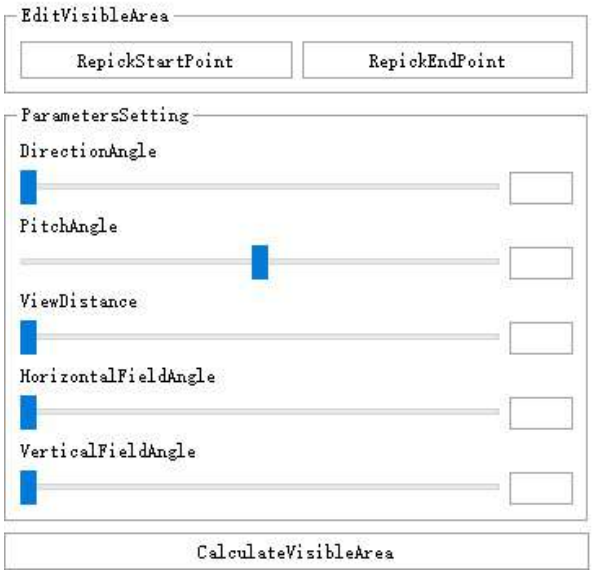
    - Generated Time: the time of this probe
    - Files: the name of the LiData file used for this probe
    - From Class: The source class used for this probe
    - Base Line Type: the type of base line used for this probe
    - Base Line Position: the position of the base line used for this probe
    - Pattern: The type of template used for this probe
    - Height, Width, Lateral Offset, Vertical Offset: represent the height, width, horizontal offset, vertical offset respectively, refer to Clearance Analysis
- **Live screenshot of this probe**
- **Output**: Statistics of this survey
    - Total Points: the number of points used in this survey
    - Danger Points: the number of danger points at this detection
    - Danger Points Proportion: the proportion of the set of danger points in this detection to the total number of points used
    - Danger Cluster Center: Cluster centre of the set of danger points

Observation point visual field analysis

Function Description: This tool can be used to establish an observation point and analyze the occlusion of the target point from the observation point.

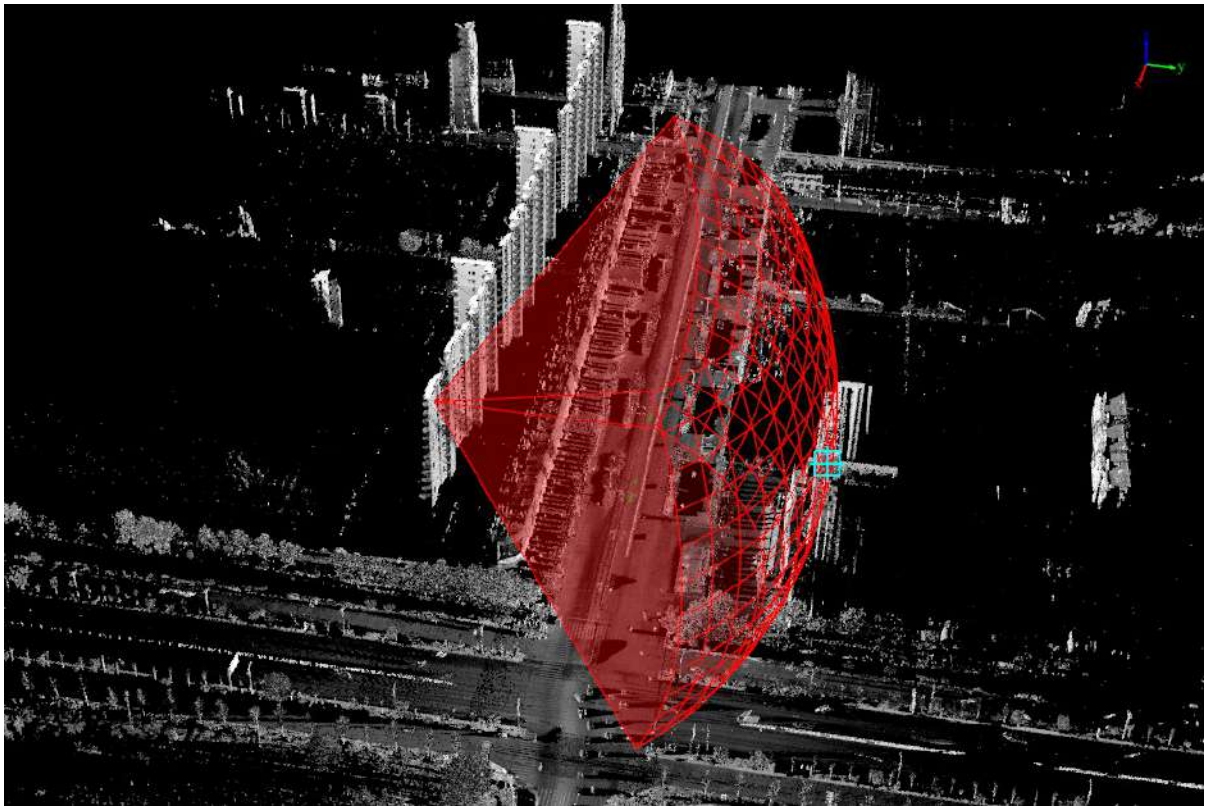
Steps

1.In the road analysis menu bar, click **Observation point visual area analysis**  button, the visual field editing settings page pops up.



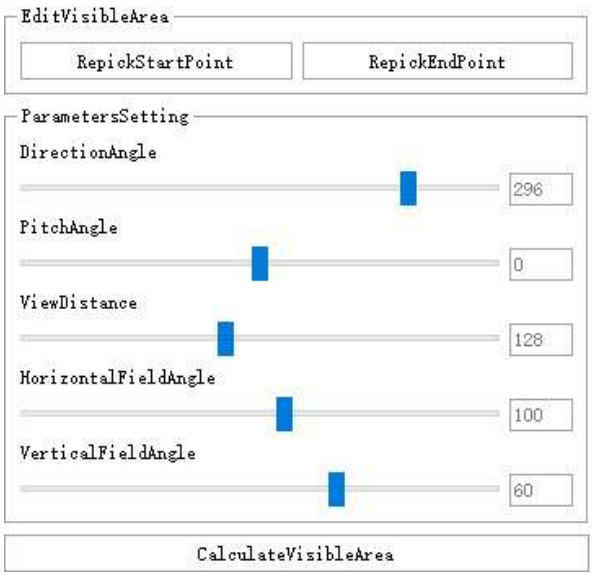
Visual field editing

2.Use the left mouse button to select the position of the observation point in the 3D viewport, and then slide the mouse. The 3D viewport will render the observation range from the selected point to the position of the mouse movement in real time. The vertebral body：



Observation range Visual vertebral body

3.Double-click the left mouse button at the target point to determine the end point of the vertebral body. At this time, the optic vertebrae will no longer change with the mouse.



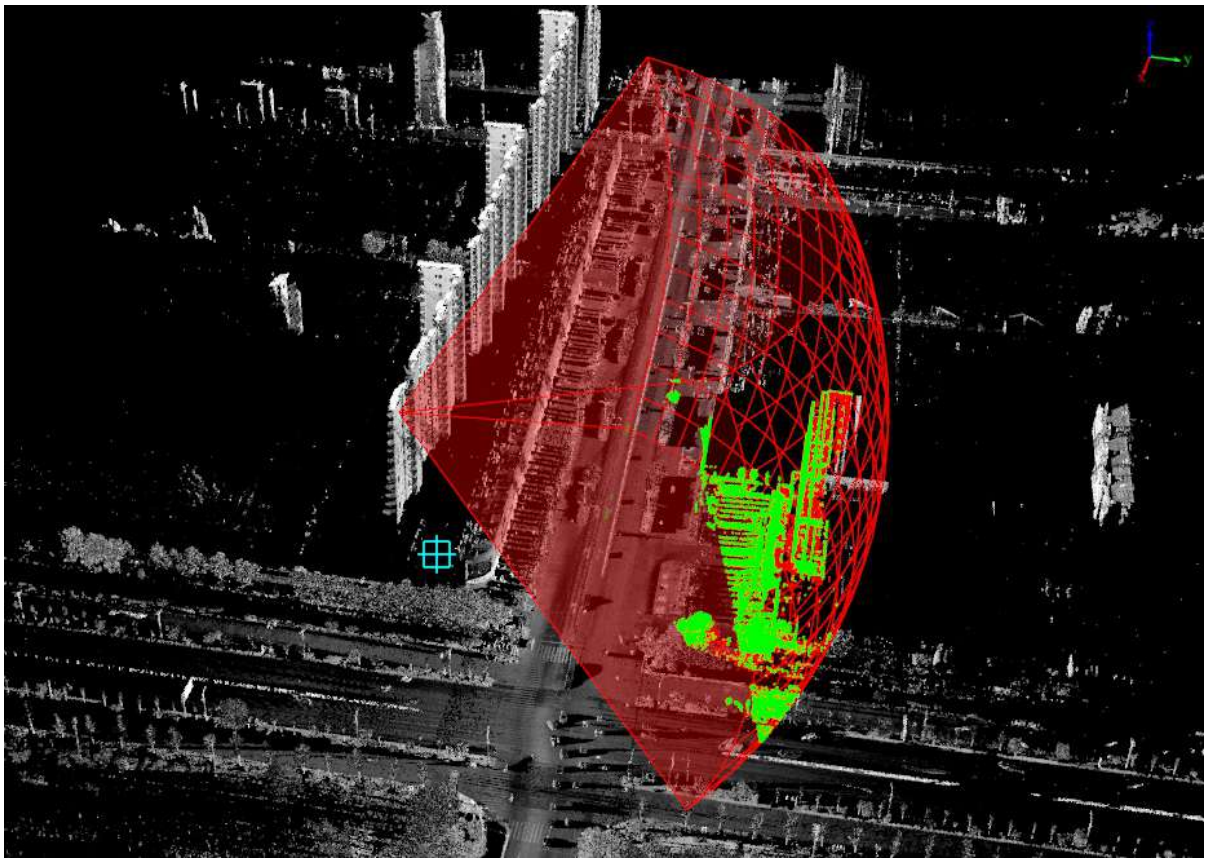
After the starting point is selected, the angle is displayed

4.In the pop-up settings window, the various angles of the current visual vertebral body will be displayed. If you want to adjust the starting position of the observation and the viewing angle range of the observation, you can set it in the parameter adjustment pop-up window, click the **RePick start point** button, you can reselect the starting point of the observation in the 3D viewport. Click the **RePick end point** button to reselect the end point position of the optic vertebrae.

5.If you need to adjust the viewing angle, you can set it by dragging and dropping the slider in the parameter settings.

- **Parameter setting：**
  - Slide the **Direction Angle** slider to adjust the horizontal orientation of the visual vertebral body.
  - Slide the **Pitch Angle** slider to adjust the vertical orientation of the visual vertebral body.
  - Slide the **Visual Distance** slider to adjust the distance of the observation range.
  - Slide the **Horizontal Field Angle** slider to adjust the observation range in the horizontal direction.
  - Slide the **Vertical Field Angle** slider to adjust the observation range in the vertical direction.

6.When all adjustments are completed, click the **Calculate Visible Area** button, and wait for the calculation progress bar at the bottom of the software to end, you can display the calculation results of the visual field in the 3D viewport.



Observation point visual field analysis and calculation results

Target

Function Description: This tool can be used to calculate the visibility between the target point cloud and the observation point, which can be used to detect whether the location of the traffic facility is reasonable and whether it meets the requirements of traffic safety.

Steps

1.Click on the road analysis page **Target**  button in the road analysis page, the settings page will pop up on the software page at this time:

Parameter adjustment pop-up window

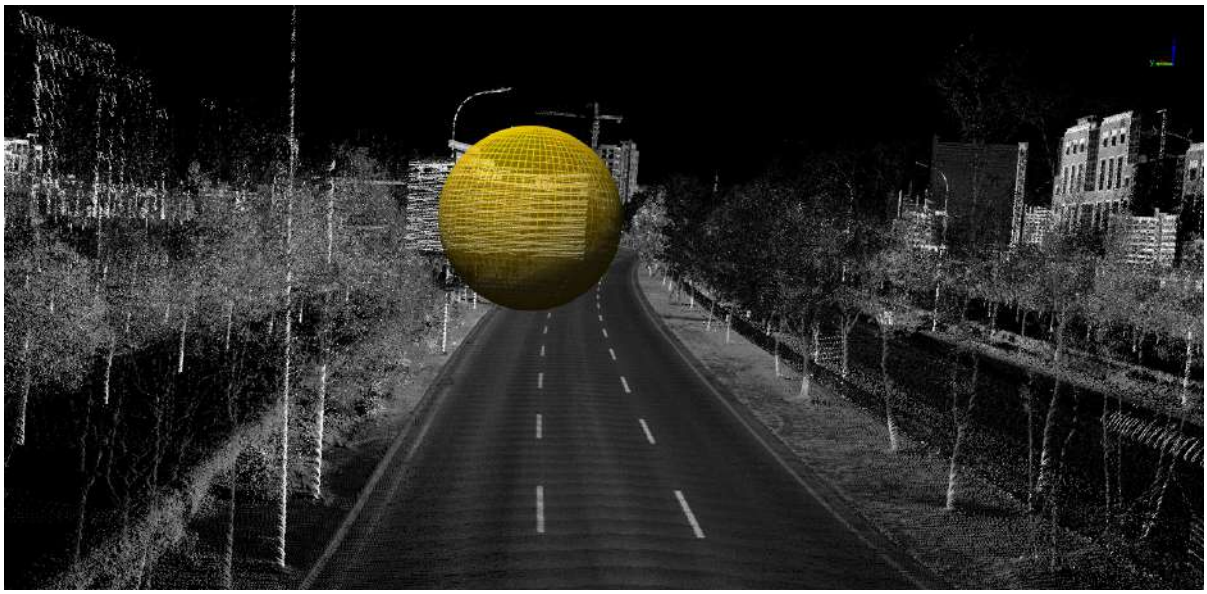
Parameter description：

- **Select the target point**: Select the cluster of points to be observed through the ball selection.
- **Select observation point**: Select observation point data, including three modes: trajectory segment selection, existing vector line selection, and custom line drawing.
- **Visual parameters**: Adjust the visual parameters to obtain different visual effects.
  - **Trajectory Step size**: When the track segmentation mode is selected, set the observation point to take the point step size on the track. since the trajectory points are too dense, the trajectory points are thinned at a step interval, and the thinned trajectory points are used for visual calculation.

- **Voxel Size**: he smaller the voxel, the finer the analysis result and the longer the calculation time. The default is recommended.
- **Occlusion Points Threshold**: When the number of occlusion points between the target point and the observation point is less than the threshold value, it is considered that there is no unobstructed
- **Obscuring ratio range**: The obscuring rate is divided into three levels, which are represented by different colors (customizable).After the analysis is over, by adjusting the level range, the display effect will be refreshed in real time.

- **Save and Load**: Read and write visual domain analysis results.
  - **Save**: Save the visual field analysis results, including the center of the target point, the observation point, and the masking rate.
  - **Load**: Import the saved visual domain analysis results and display them in the view (they need to be in the same coordinate system).

2.Click the **Select target point** button, and in the point cloud, use the ball selection method to select the point cluster of the observed point.You can click Empty to re-select.

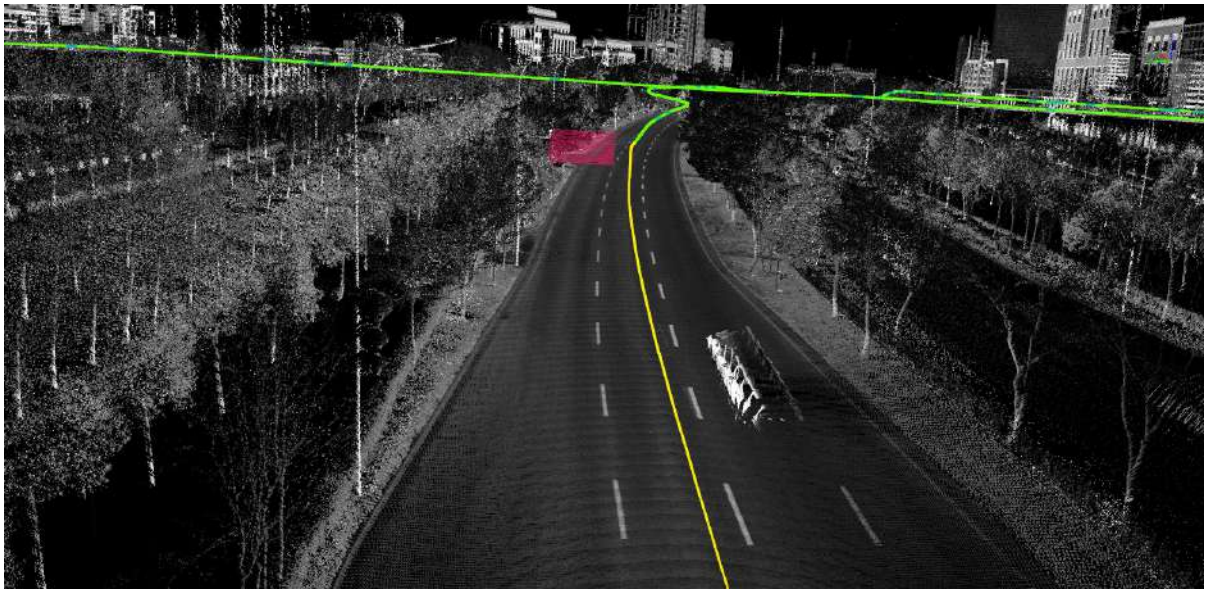


Before the target point is selected

After the target point is selected

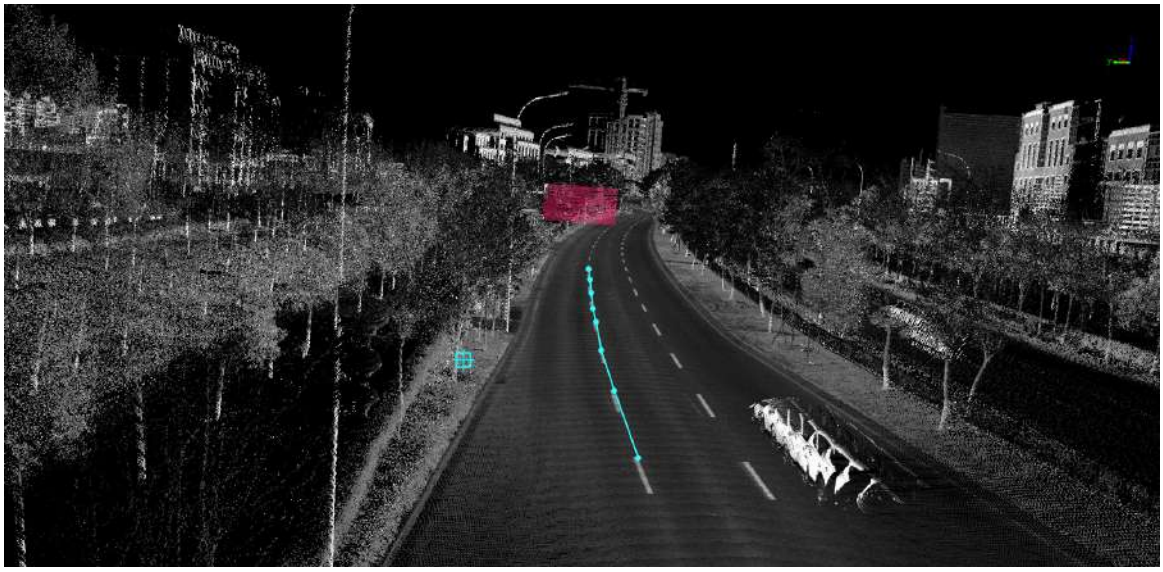
3.Click the **Select observation point** button to activate the observation point type drop-down box.

- **Track**: You must load the track file when you create a new project, then click at the starting position of the track segment you want to select, push the mouse, and the track will be selected in real time. Double-click at the location you want to end the selection and you can select a certain segment of the track as the observation point.



Select trajectory at the observation point

- **Vector line**: You must draw a line element in advance, and use the selection vector line mode to select the vector line as the observation point.

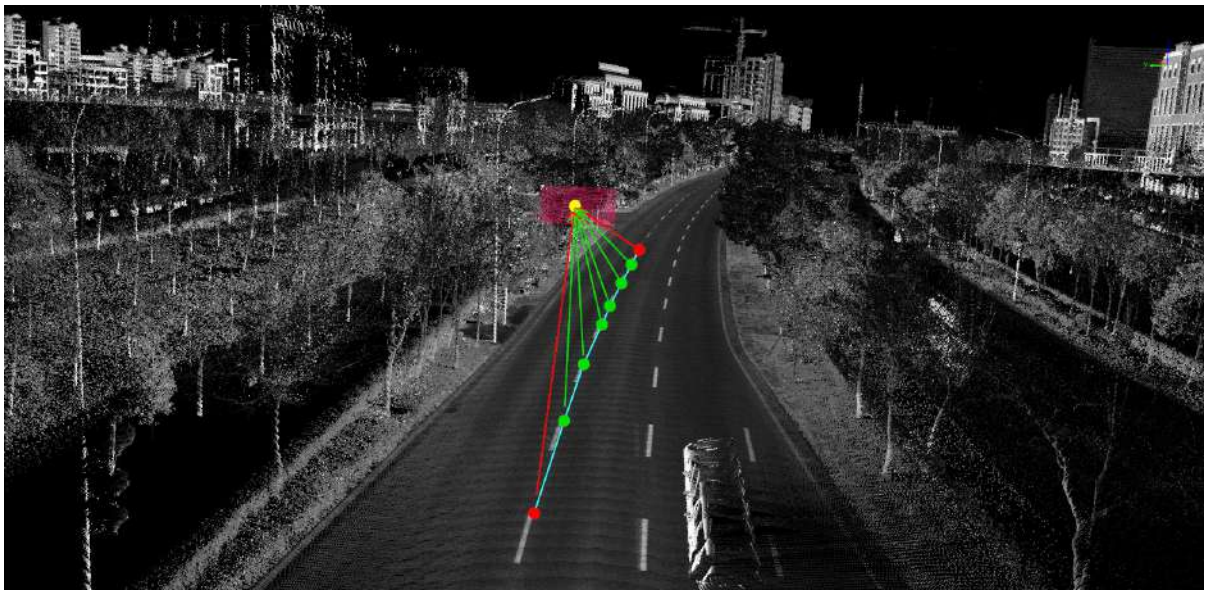


Select vector line at the observation point

- **Custom line**: After selecting the custom line, you can draw a temporary vector line at the location that needs to be analyzed as an observation point

Select custom line at the observation point

4.After clicking OK, perform visual domain analysis.



Analysis results

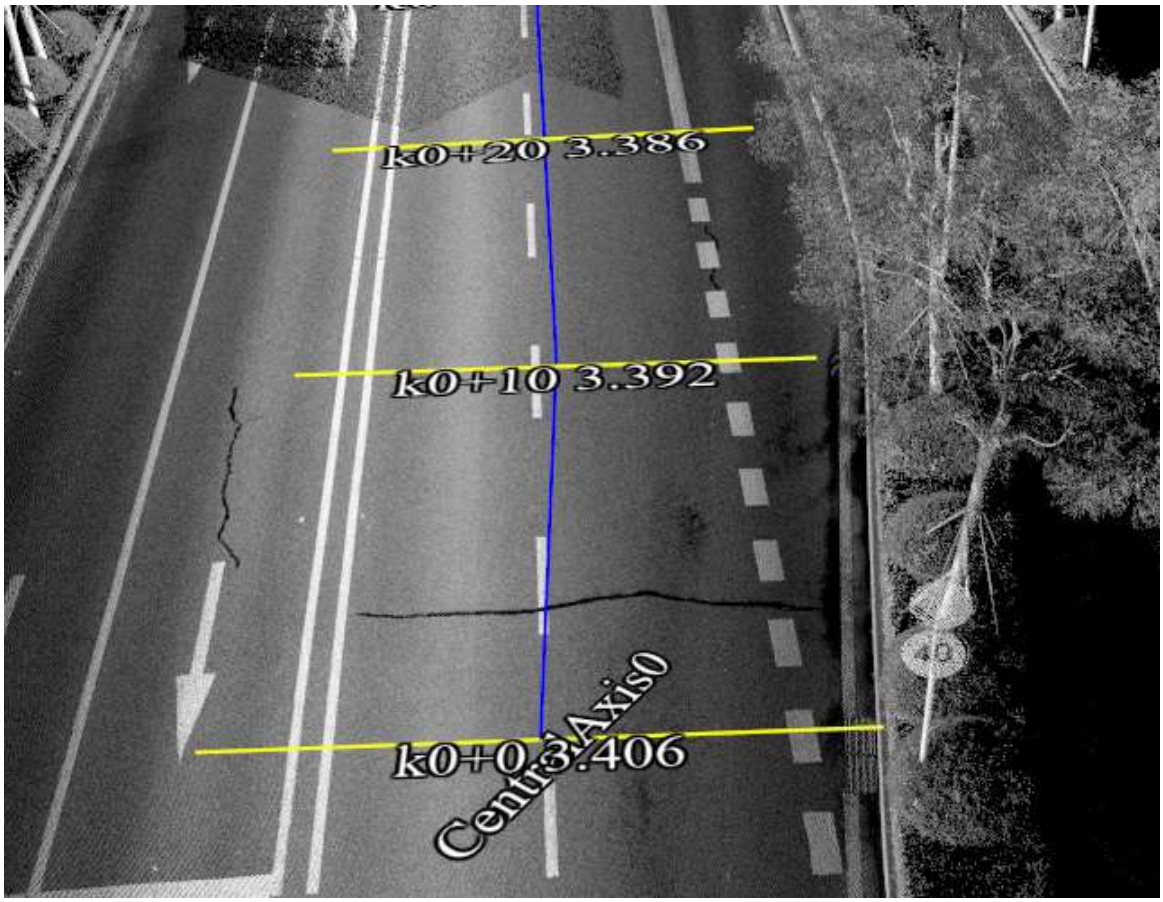
5.The analysis results can be exported a *.vis file by clicking the save button, you can also reload the file to display and view in the future.

Road Section Analysis

Road Reference Line Setting

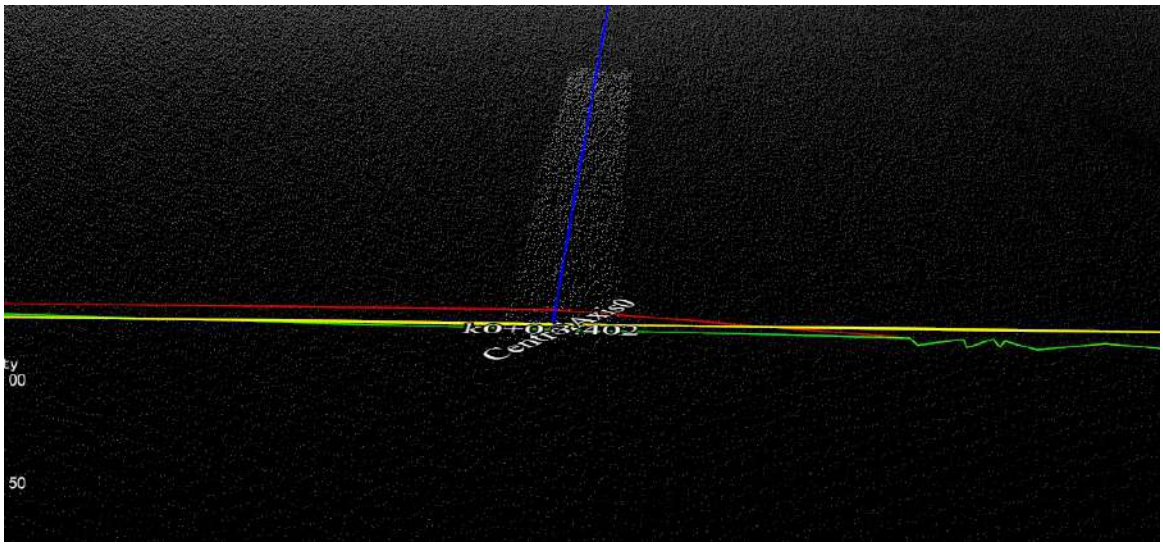
Term Descriptions

- **Vertical reference line**: It represents the direction of the road and is a virtual reference line, as shown in the blue line in the figure below.
- **Cross-section reference line**: Perpendicular to the direction of the road, it is a virtual reference line and the basic reference line for cross-section analysis, as shown in the yellow line in the figure below.



- **Design section**: The section data is generated by the design data. The main performance is that the first phase of the current data, that is, the data of different periods in the same location, is used for fault-breaking ratio analysis and filling-in analysis, as shown in the green line in the figure below.
- **Measurement section**: The section data is generated by the current data, as shown in the red line in the figure below.

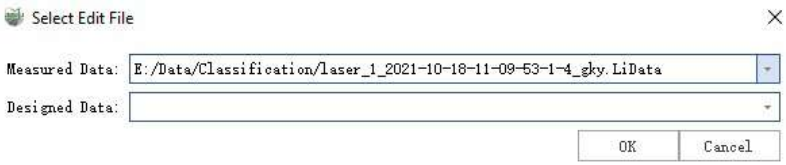
> In order to effectively distinguish, the data is manually exaggerated and displayed



Editor

Function Description: This tool can be used to turn on/off the road cross-section analysis function.

Start editing



Parameter description：

- **Measurement Data**: Used to generate the real cross-section, the default is red, required items, the drop-down items are the point cloud data opened by the current project.
- **Design Data**: Used to generate design sections, the default is green, and it can be left unchecked. If you need to use subsequent fill-in analysis, the design data is required.

> The measurement data and design data are all point cloud data opened by the current project, and the two must have overlapping areas of physical range, so that the generated design section and the actual measurement section have the practical significance of comparative analysis, that is, the measurement data is the actual cross-section data used in this measurement, and the design data is point cloud data from different periods in the same area.

End editing

Exit the overall function of road cross-section analysis and eliminate various vector data generated in the process of the function.

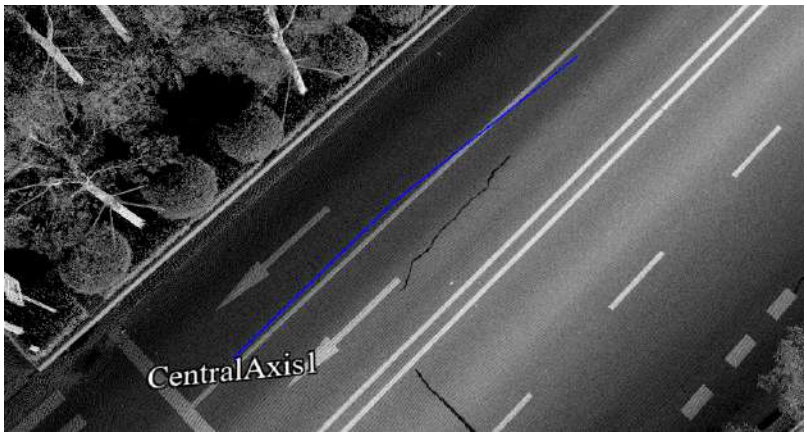
> Before clicking to end editing, please be sure to ensure that the data that needs to be saved has been saved. After exiting the function, all data will be emptied.

Create Polyline

Function Description: Manually draw a direction line (longitudinal reference line) for road cross-section analysis.

Steps

1.Click with the left mouse button.



Dotted line dialog box

2.Double-click the left button to end the creation.

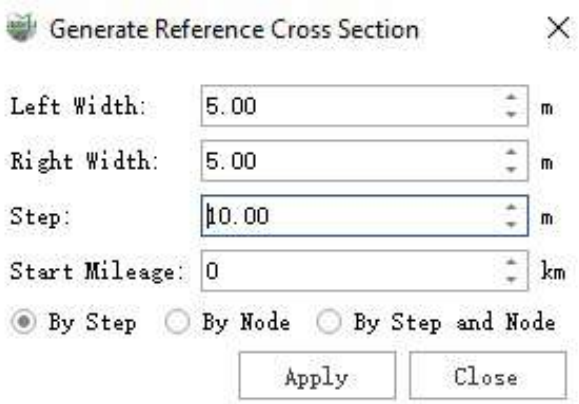
Select Reference Line

Function Description: With this tool, click with the left mouse button to select an existing line object as the vertical reference line.

Generate Reference Cross Section Automatically

Function Description: This tool allows the user to create an is automatically generated reference cross-section according to set parameters.

Parameter dialog box



Parameter dialog box

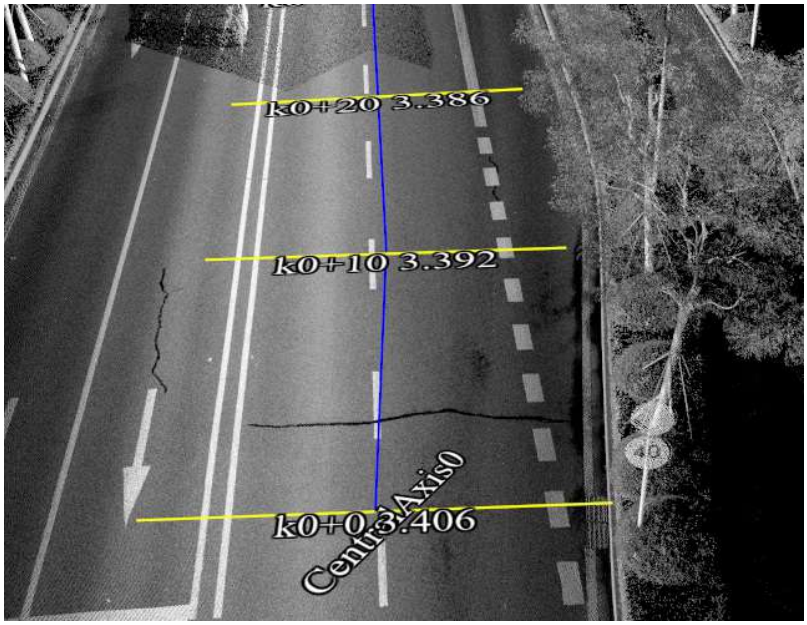
Parameter description

- **Left Width**: The distance from the leftmost side of the cross-section reference line to the vertical reference line, the default is “5”, to be calculated using the road properties, it needs to be set to half the width of the

lane.

- **Right Width**: The distance from the rightmost side of the cross-section reference line to the longitudinal reference line, the default is “5”, to be calculated using the road properties, it needs to be set to half the width of the lane.
- **Step**: The step size to generate the reference cross-section
- **Starting Mileage (default is “0”)**: Generate the starting mileage of the reference cross-section, and the mileage of the cross-section is accumulated on this basis. The starting mileage pile of each section of the road may be different. The cross-section is named in the form of mileage-pile km + m by default. Users can modify the name of the cross-section by themselves.
- **By Step**: Generate a reference cross-section by a specific step size.
- **By Node**: Only the reference cross-section is generated at the node, and the node refers to the node of the longitudinal reference line itself.
- **By Step and Node**: Generate a reference cross-section by a specific step size, and process the nodes at the same time.

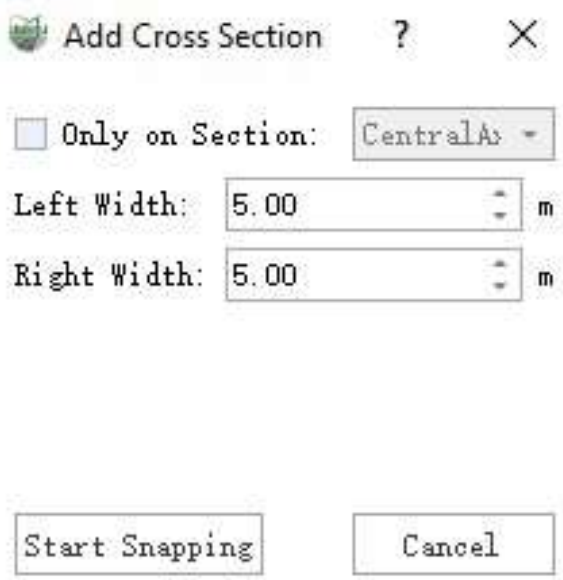
The blue vector line is a manually drawn longitudinal reference line, the name is CentroAxis0, where 0 is the line number, and the yellow line is a cross-section reference line automatically generated according to the step size, the name is K0 +0 3.406, where K0 is the pile number, + 0, +10, etc. Are the mileage from the starting point, 3.406, 3.392, etc. Are the elevation values at the intersection of the cross-section reference line and the longitudinal reference line.



Parameter dialog box

Generate Orthogonal Section Manually

Function Description: This tool allows the user to create an automatically generated reference cross-section according to set parameters.



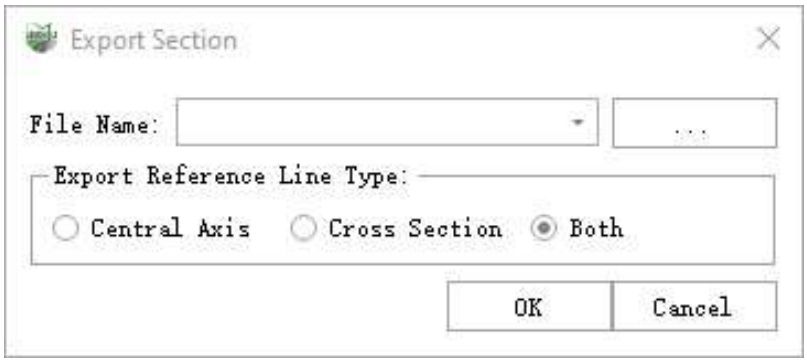
Parameter dialog box

Parameter description

- **Only addon a certain centerline**: Only add on a specific centerline (vertical reference line), which is invalid when other centerlines (vertical reference line) are captured.
- **Left Width**: The default is 5.0 meters, the distance from the leftmost side of the cross-section reference line to the longitudinal reference line.
- **Right Width**: The default is 5.0 meters, the distance from the rightmost side of the cross-section reference line to the longitudinal reference line.
- **Start Snapping**: Start adding a cross-section, move the mouse to any centerline, **When the centerline is highlighted, click the left mouse button**, the new cross-section will be added to the center axis.

Save Reference Line

Function Description: This tool saves the reference line and cross-section line in the form of a file.



Parameter dialog box

Parameter description

- **File name**: Combine the buttons at the back of the edit box to select the location and file name of the file to be saved. The saved results support three formats:.dxf,.shp,.and txt.
- **Export reference line type**:

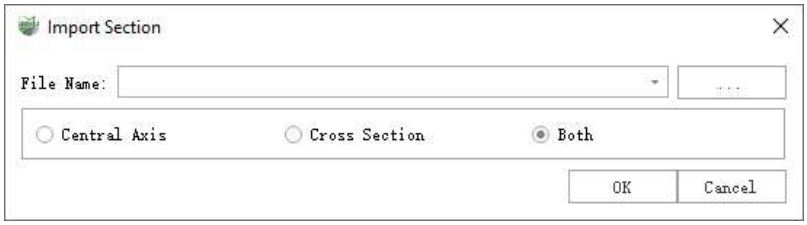
- **Central Axis**: Only the longitudinal reference line is saved.
- **Cross Section**: Only the Cross Section reference line is saved.
- **Axis and Cross-section**: Save the longitudinal reference line and the cross-section reference line at the same time.

Import

Function Description: These tool allows for the external reading of existing section data, including reference lines, section analysis files, and design sections.

Import reference line

Function Description: These tool allows for the external read-in of reference line data saved by the save reference line function.
Parameter dialog box



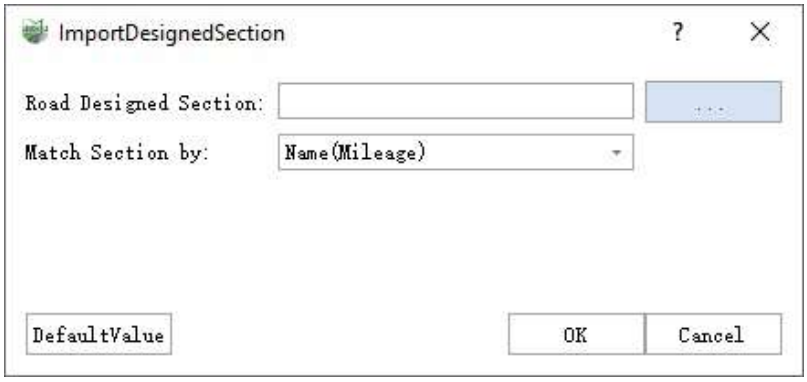
Parameter dialog box

Parameter description

- **File Name**: Combine the buttons at the back of the edit box to select the file to be imported. It supports.dxf,.shp, txt three formats.
- **Import reference line type**:
  - **Central axis**: Only longitudinal reference lines are imported.
  - **Cross section**: Only the cross-section reference line is imported.
  - **Axis and cross-section**: import longitudinal reference lines and cross-section reference lines at the same time.

Import Design Section

Function Description: These tool allows the user to import design section data and read it in format.csv, this file is generated by the export section function. There is no need to import normal measurements. If you need to calculate the filling and digging party or analyze the cross-section ratio, you need to import the design cross-section.



Parameter dialog box

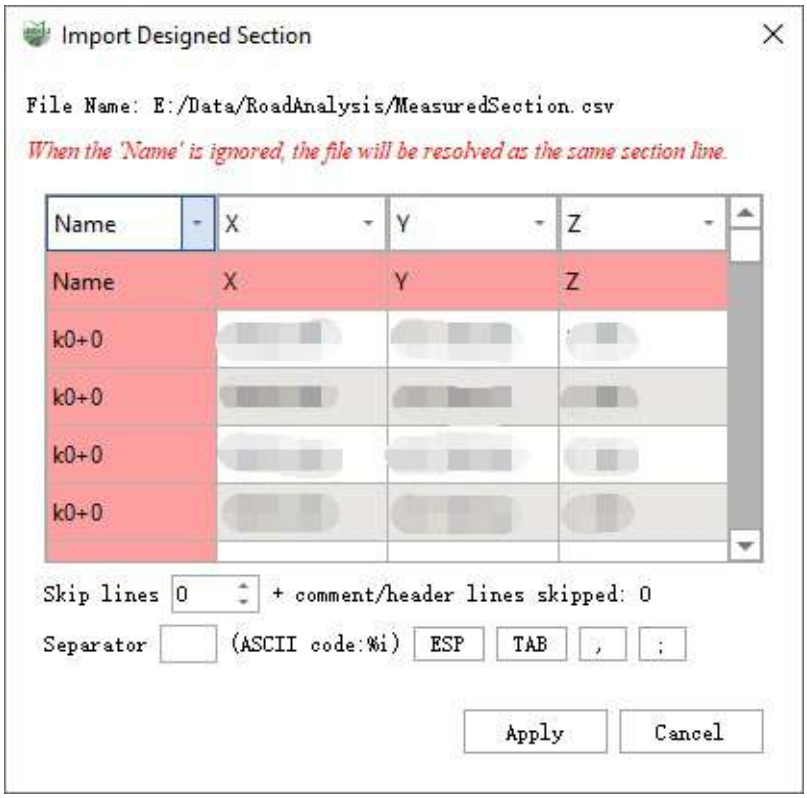
Parameter description

- **Road Designed Section**: Combine the buttons at the back of the edit box to select the design section to be imported.csv file.
- **Matching Section by**: The matching mechanism between the design section and the current measurement section, line to line, currently only supports name matching, that is, only when the measurement section line and the design section line have the same name, they can be matched.

Steps

1.Select the designed section you want to import.

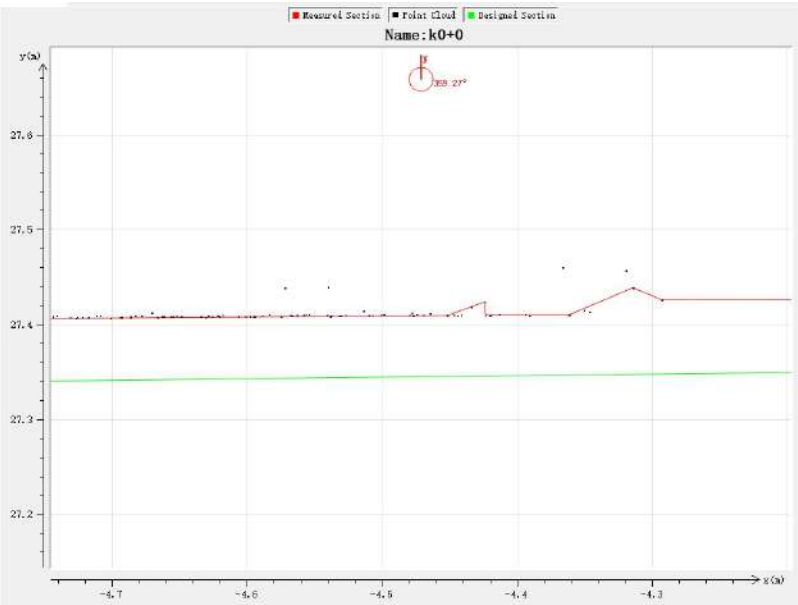
2.Confirm and adjust the format of the read data.



Parameter dialog box

3.Result view.

The green line is the designed section, and the red line is the current measurement section.



Parameter dialog box

Generate Cross-Sectional View

- **Generate Road Section Views**

Generate Road Section Views of the road surface, with the generation method, refer to Section Analysis Window Control.

- **Generate Tunnel Section Views**

Generate Tunnel Section Views of the tunnel, with the generation method refer to Section Analysis Window Control.

Hidden

Function Description: This option controls the concealment of cross-section correlated vector lines.

Show spindle

Function Description: This option controls the concealment of the vertical reference line.

Show horizontal line

Function Description: This option controls the concealment of the cross-section reference line.

Display measurement section

Function Description: This option controls the concealment of the measurement section.

Display designed section

Function Description: This option controls the concealment of the designed section.

Show all

Function Description: This option controls the display and concealment of all reference lines.

Clear Section

Function Description: These options allow the user to clear cross-section related information.

> Note: The clear function will eliminate the existing actual data, and it is **unrecoverable**, please choose carefully

Clear Central Axis and its Measured Sections

Function Description: This option clears the data related to the vertical reference line.

Clear Cross Sections and its Measured Section

Function Description: This option clears the horizontal reference line and the corresponding measurement section.

Clear Measured Sections

Function Description: This option clears measured sections.

Clear All Sections

Function Description: This option clears all data, including vertical reference lines, horizontal reference lines, measurement sections, and designed sections.

Section Analysis Window Control

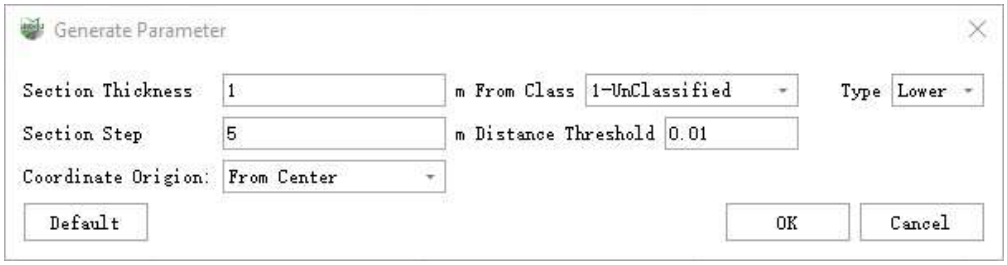
Function Description: This tool calculates, modifies, saves and exports cross-section related operations.

> Note: This function needs to be turned on **Generate RoadSection Views** or **Generate Tunnel Section Views**

Calculate

According to the existing horizontal reference line and related setting parameters, the measurement section is generated.

Click  button, the parameter dialog box pops up.

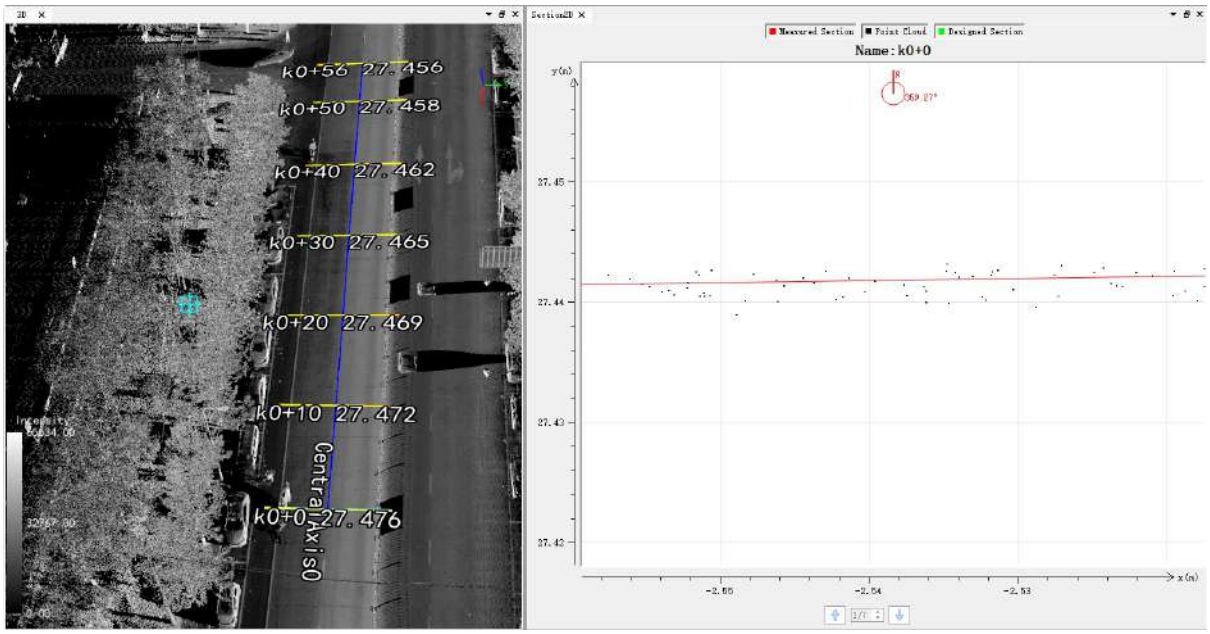


Parameter dialog box

Parameter description

- **Section Thickness**: Generate a section diagram using point cloud data along the width of the horizontal reference line
- **From Class**: Participate in the generation of cross-section point cloud categories, select the source category correctly, the source category is selected incorrectly, and the generated results may be incorrect.
- **Type**: Take the cross-section data according to the elevation type to generate a cross-section diagram.
  - Lower edge (default): Take the lowest point of the cross-section data to generate a cross-section diagram.
  - Upper edge: Take the highest point of the cross-section data to generate a cross-section diagram.
- **Section Step**: The measured cross-section will be segmented and streamlined according to the distance threshold according to the set step size, and the cross-section will use linear interpolation to interpolate points at an integer multiple of the cross-section step size.If this value is set to 0, the section will be streamlined as a whole.
- **Distance Threshold**: The section will be streamlined according to the Douglas algorithm used for this parameter. The larger the value, the fewer points will be retained and the more streamlined, and vice versa. The more points and details will be retained.
- **Coordinate Origin**: The center position of the section, it is recommended to choose the center. If you need to calculate the road parameters, you must choose the center mode.

The red line is the generated measurement section

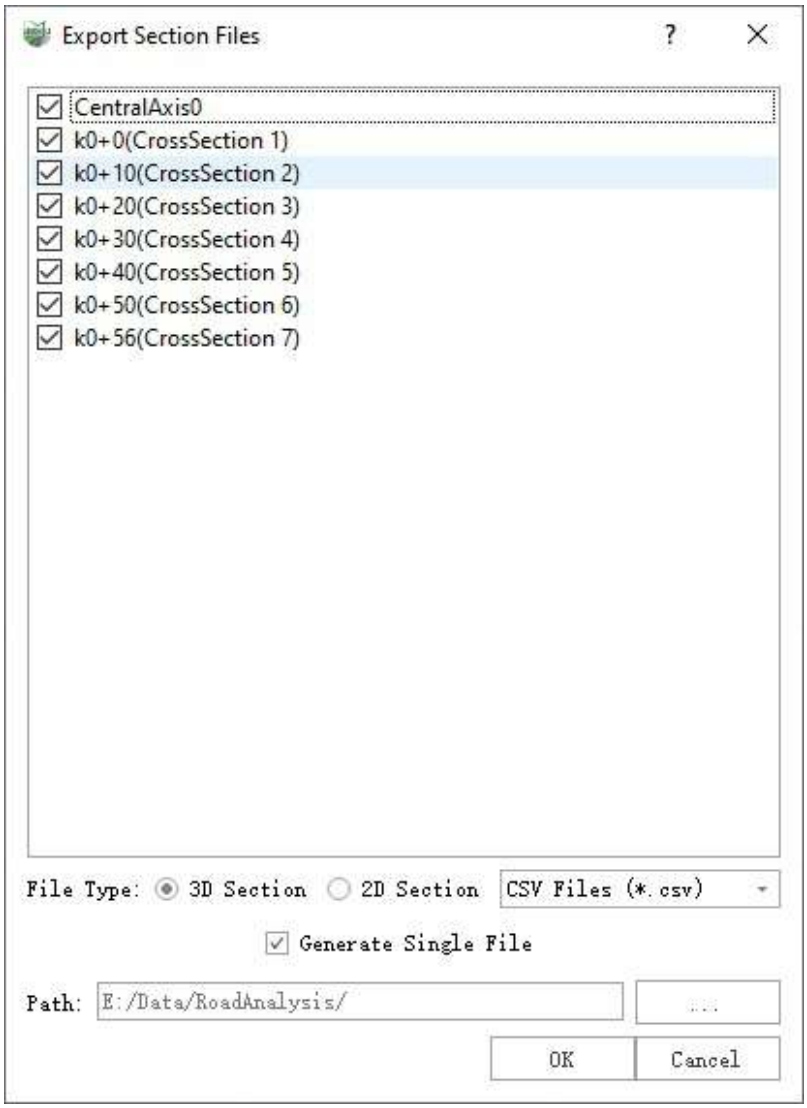


Parameter dialog box

Export section

Function Description: This tool allows users to export 2D or 3D cross-section lines into a variety of ways, and save them into one or more files. It currently supports **.csv**, **.hdm**, **.shp**, **.dxf** and other formats.

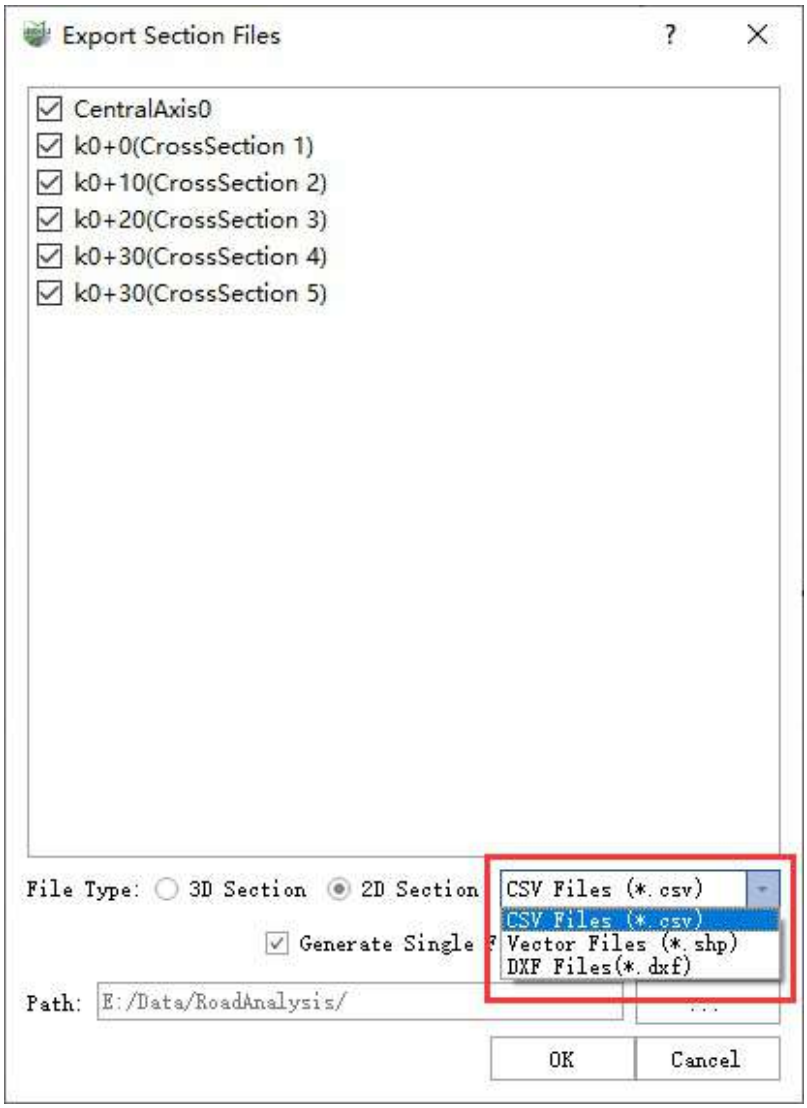
1.Click **Export section** button, the parameter dialog box pops up.



Parameter dialog box

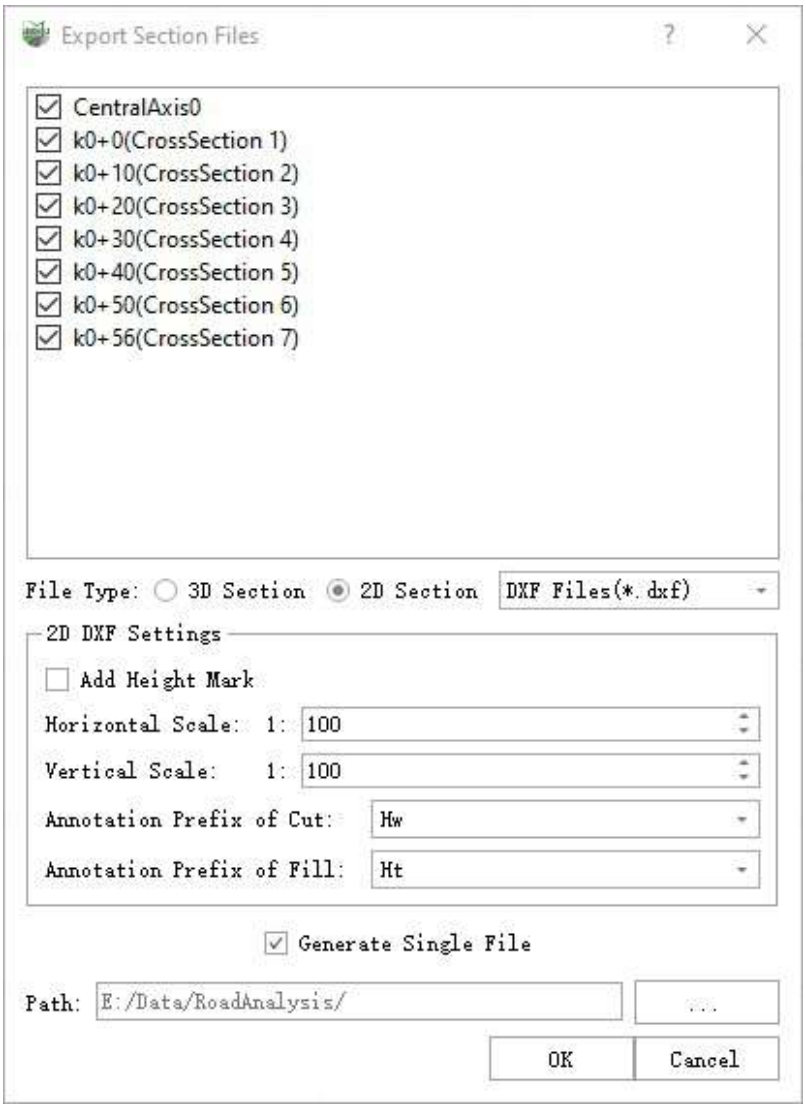
Parameter description

- CSV is a common format used internally by the software, and it is read in csv format when importing the design Cross-section data.
- Files in hdm format supports the data formats of two different software, Weidi and Southern CASS, and can be set according to user needs. The format of the hdm file and the parameter setting interface are shown in the following figure；



Parameter dialog box

- DXF vector files, support the import of third-party software, such as AutoCAD to view the cross-section results and related data. Take the generated file in dxf format in the example below. The parameter settings are shown in the figure below. Set the output path and click OK.

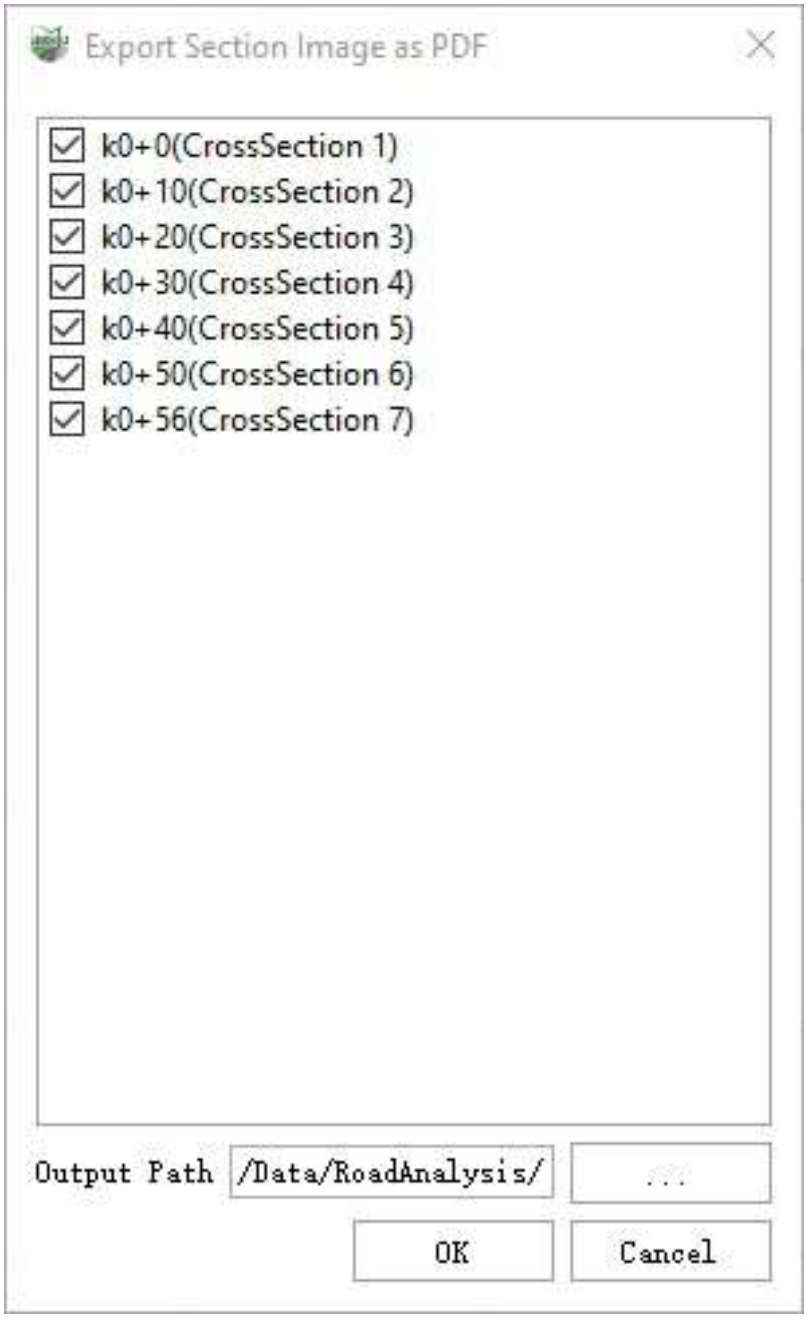


Parameter dialog box

Export Section Image as PDF

1.Click **Export pdf** button, the parameter dialog box pops up.

2.Select the export path to complete the export of section information.



Parameter dialog box

Edit Section Node

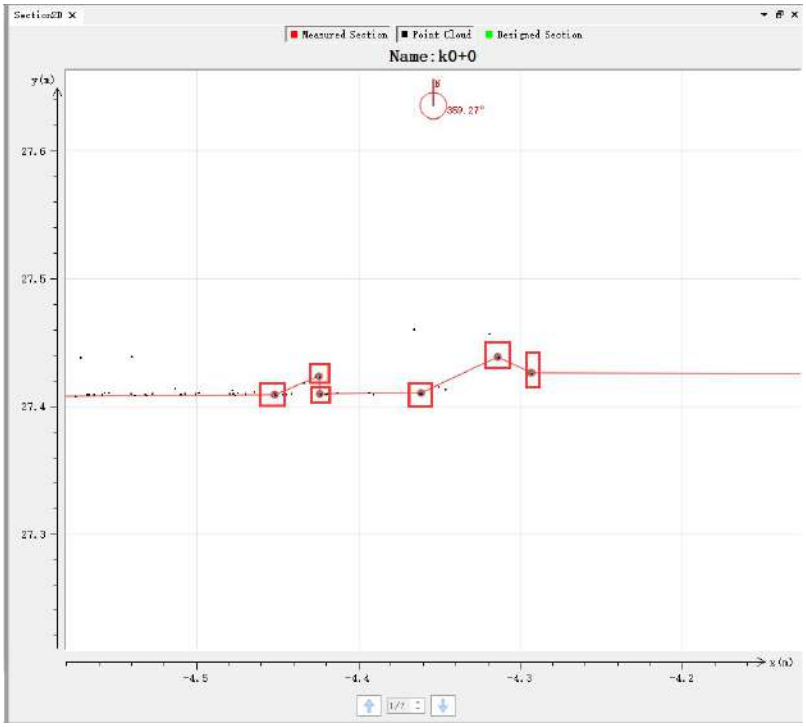
Function Description: This tool can be used to edit cross-section nodes by dragging and dropping nodes, as well as adding nodes. It can be used to edit incorrect or noisy tunnel boundaries.

- **Edit section line nodes**

Click **Edit section line nodes** button, the cross-section line node will be highlighted and become editable. Drag and drop the node to the appropriate location with the mouse to complete the editing of the node. You can see that the editing results will be synchronized to the point cloud window in real time, and the editing results can be viewed intuitively.

- **Add section line nodes**

Click **Add section line nodes** button, the cross-section node will be highlighted and become editable. Click the mouse where you need to add a node, and the node will be automatically added and the node will be added and edited. Similarly, the editing results will be displayed in real time in the point cloud window, and the editing results can be viewed intuitively.

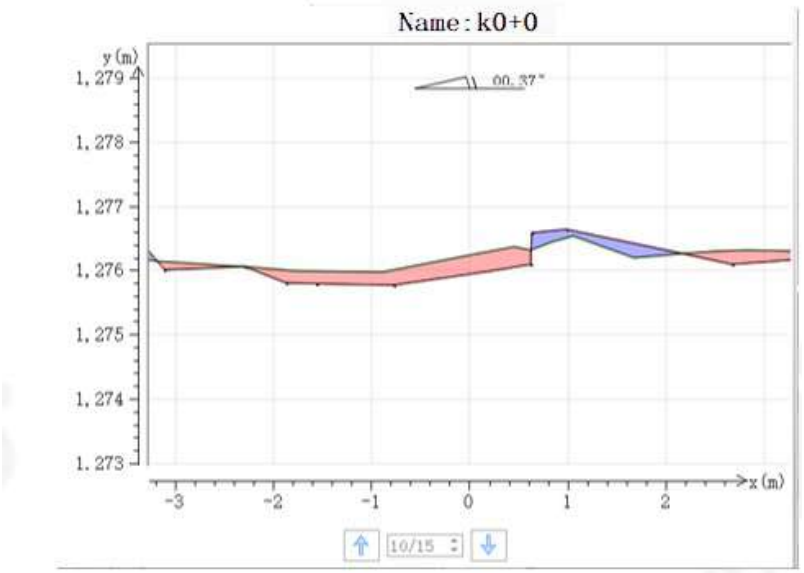


Edit node

Section Compare

Function Description: This tool compares the generated road reference section with the road design section, and calculates the amount of filling and digging. The filling square and the digging square are displayed in different colors (the color can be set).

> Note: This function is only available after the design data or design section has been imported



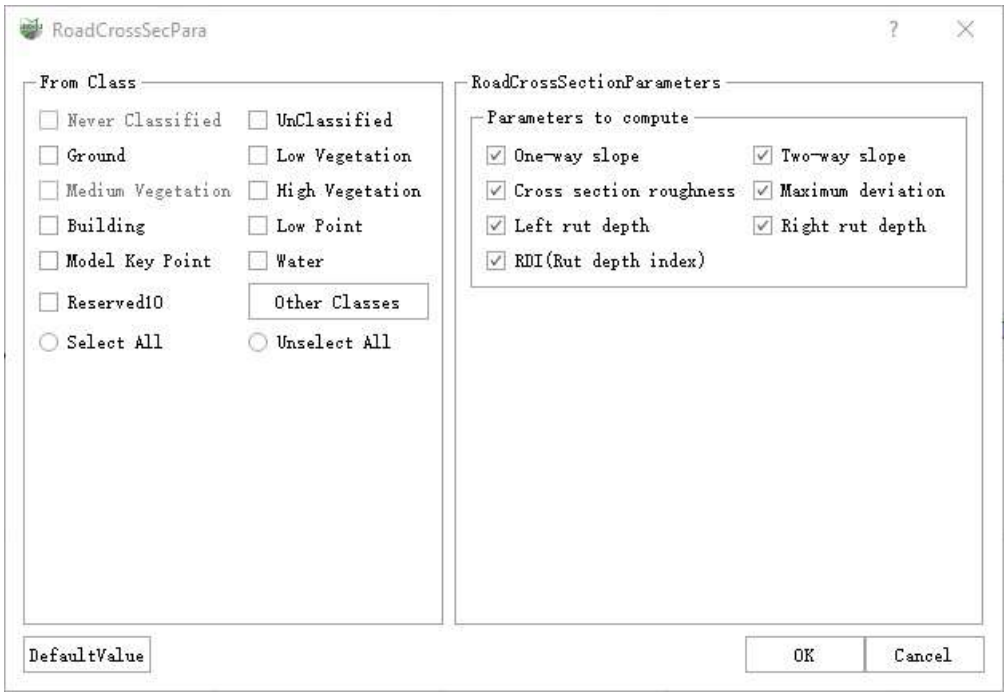
Section Compare

Export compare report

Click  button to export the cross-sectional view and cross-sectional information on the current canvas into a report and output it to the specified folder.

Calculate road parameters

Calculate the relevant road parameters based on the horizontal reference line.



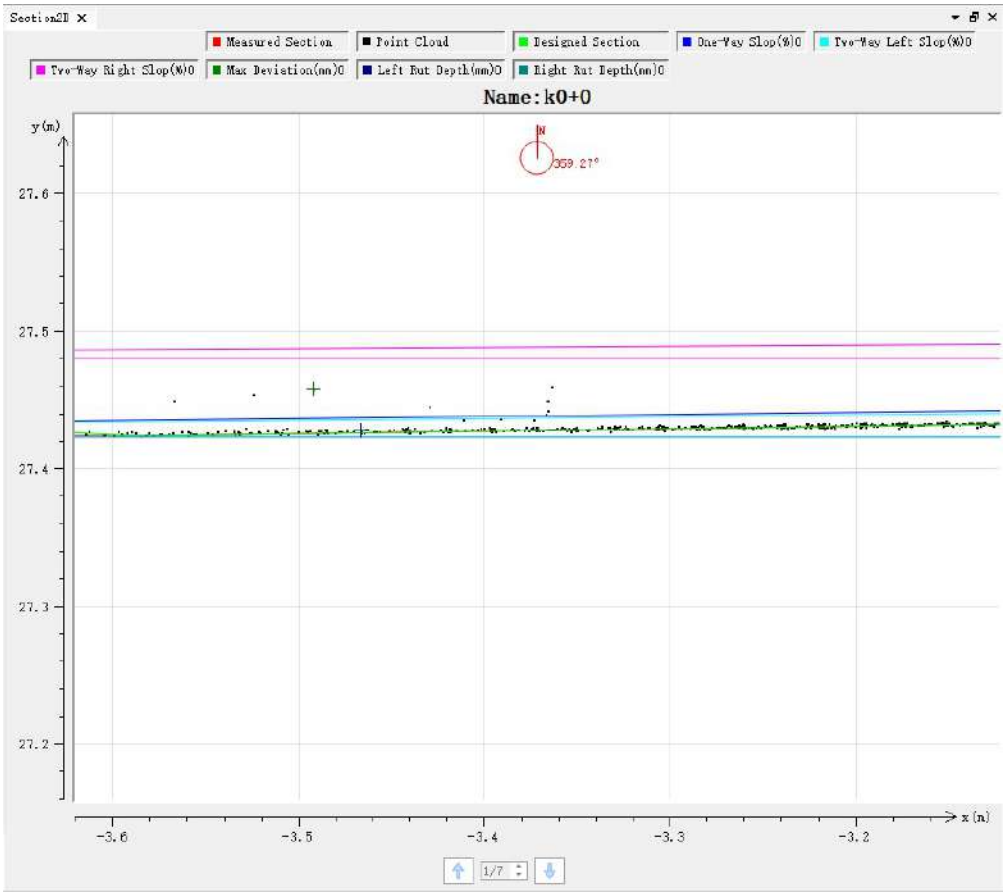
Parameter dialog box

Parameter description

- **From Class**: The point cloud category involved in the calculation of road parameters, the source category is selected correctly, the source category is selected incorrectly, and the generated results may be incorrect.
- **Parameters to be calculated**:

  > They can be displayed on the canvas, click the corresponding button to control the display and concealment of the result parameters.

  - **One-way slope**: The ratio of the height difference between the leftmost point and the rightmost point of the current cross-section to the width of the road surface.
  - **Two-way slope**: Take the center point as the dividing point, and calculate the one-way slope on the left and right sides separately.
  - **Cross-sectional roughness**: The leftmost point and the rightmost point are connected, and the average value of the distance from all other points to the line, in mm (mm).
  - **Maximum deviation**: Tthe leftmost point and the rightmost point are connected, and the maximum value of the distance from all other points to the line, in mm (mm).
  - **Left and right rut depth**: According to the 3m ruler method, calculate the left and right rut depth in mm (mm).
  - **Rut depth index**: RDI.



Section measurement

Measures the length and area of the generated section.

- **Distance measurement**：

Click on the length measurement. After turning on the distance measurement, you can use the left mouse button to click on the canvas to pick up the points that need to be measured. When picking up, you can use the mouse to zoom and pan the canvas at the same time. Double-click the left mouse button to complete the measurement; click the **Distance measurement** button again to end the measurement.

- **Area measurement**:

Click area measurement. After turning on area measurement, you can use the left mouse button to click on the canvas to draw a rectangle of the area to be measured, and double-click the left mouse button to complete the measurement; click the “Area measurement” button again to end the measurement.

Settings

According to user habits, this sets the color of each attribute that needs to be displayed.


Setting
Display
Section Attribute
Show Attribute
Measure Section Color:
Designed Section Color:
Point Cloud Color:
Background Color:
Cut Color:
Fill Color:
OneWaySlop Color:
TwoWayLeftSlop Color:
TwoWayRightSlop Color:
SectionRoughness Color:
MaxDeviation Color:
LeftRut Color:
RightRut Color:
RutDepthIndex Color:
OK
Cancel

View tool

Controls the display of the currently active view.

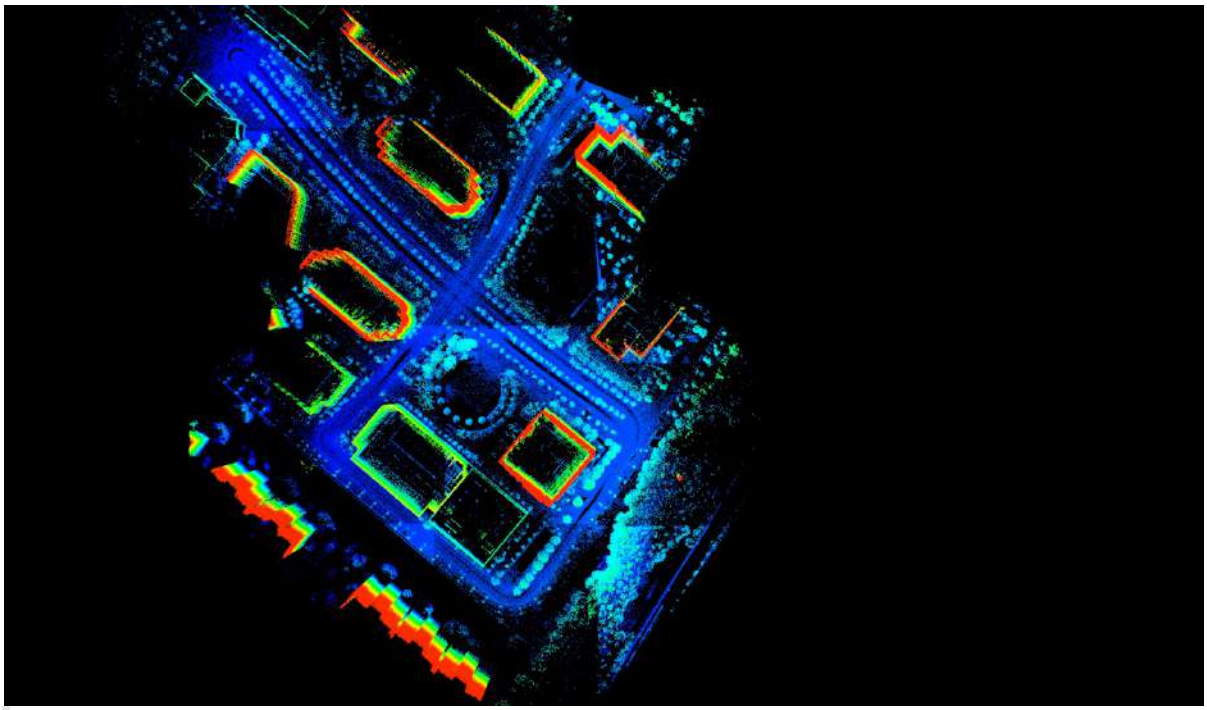
- Top View
- Bottom View
- Left View
- Right View
- Front View
- Back View
- Front Isometric View
- Back Isometric View
- Full Extent
- Set Projection Mode
- Configure Point Size
- Capture Image

Top View

Function Description: This tool allows the user to set the camera position to view from the top, that is, from +z to-z to view the three- dimensional data, the plane is the x-y plane.

Steps

1.Click **Top**  button, the current activation form is displayed as shown in the figure：



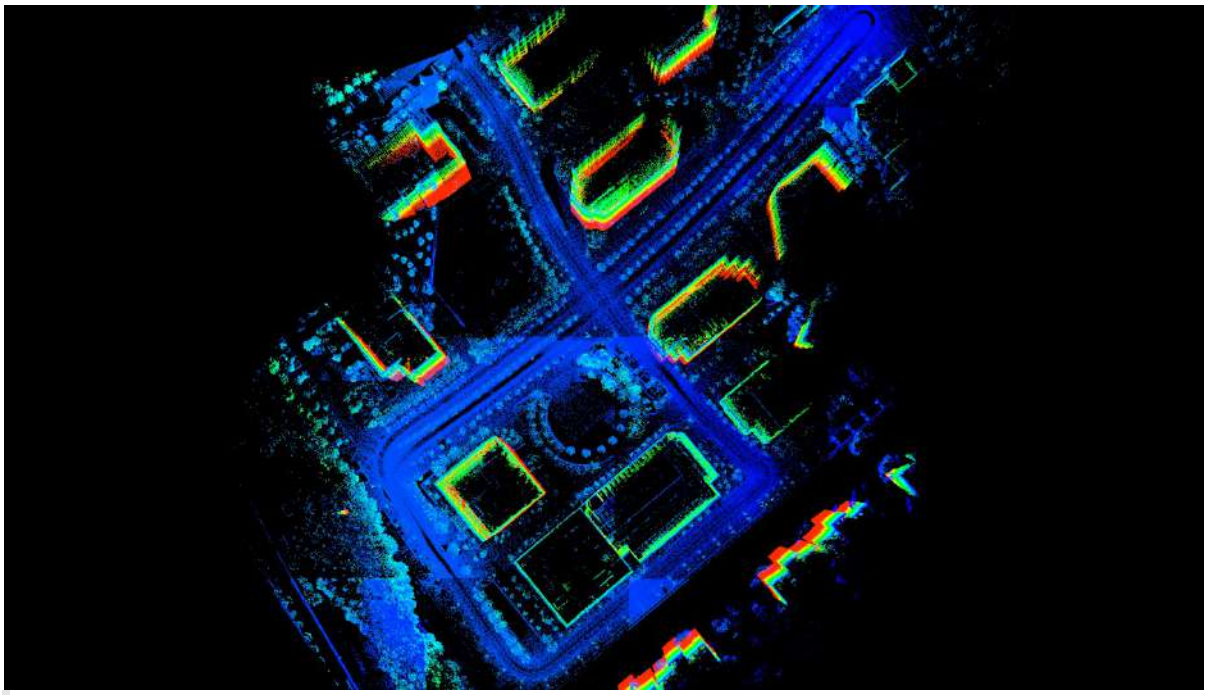
> Note: This function is only for views displayed in 3D; this function does not reset the center position of the viewpoint. If you need to reset to the default viewpoint, please click Global Display.

Bottom View

Function Description: This tool allows the user to set the camera position to view from the bottom, that is, from-z to +z to view the three- dimensional data, the plane is the x-y plane.

Steps

1.Click **Bottom**  button, the current activation form is displayed as shown in the figure：



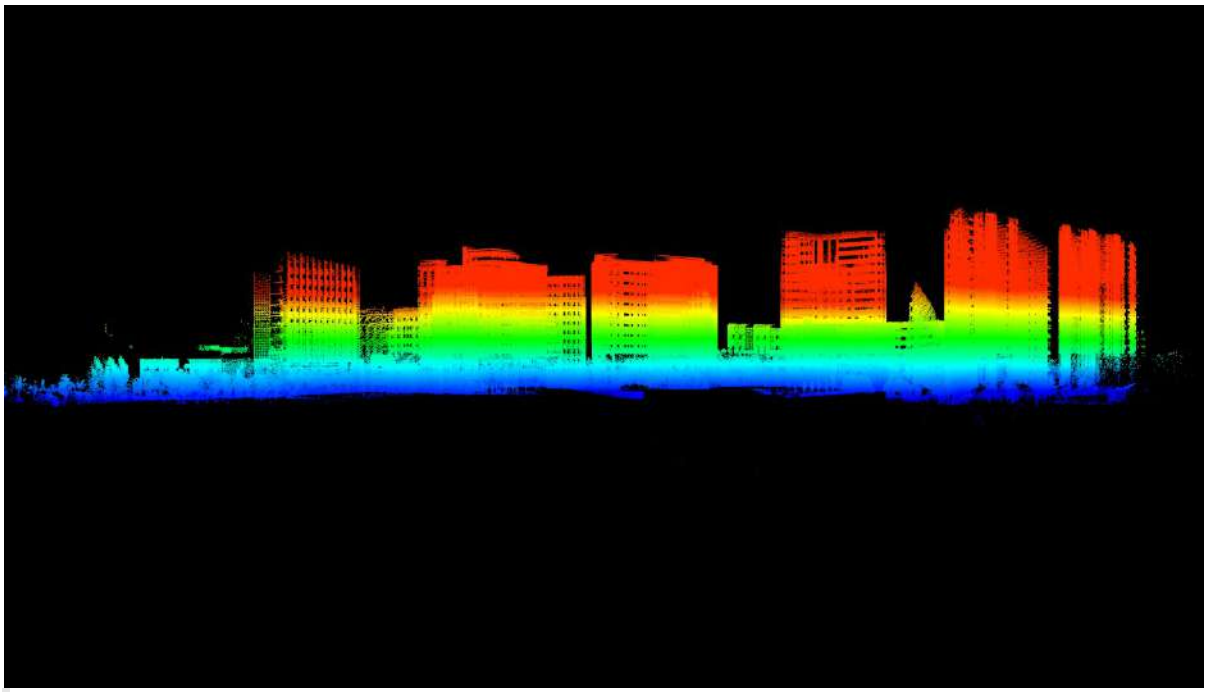
> Note: This function is only for views displayed in 3D; this function does not reset the center position of the viewpoint. If you need to reset to the default viewpoint, please click Global Display.

Left View

Function Description: This tool allows the user to set the camera position to view from the left, that is, from -x to +x to view the three- dimensional data, the plane is the y-z plane.

Steps

1.Click **Left**  button, the current activation form is displayed as shown in the figure：



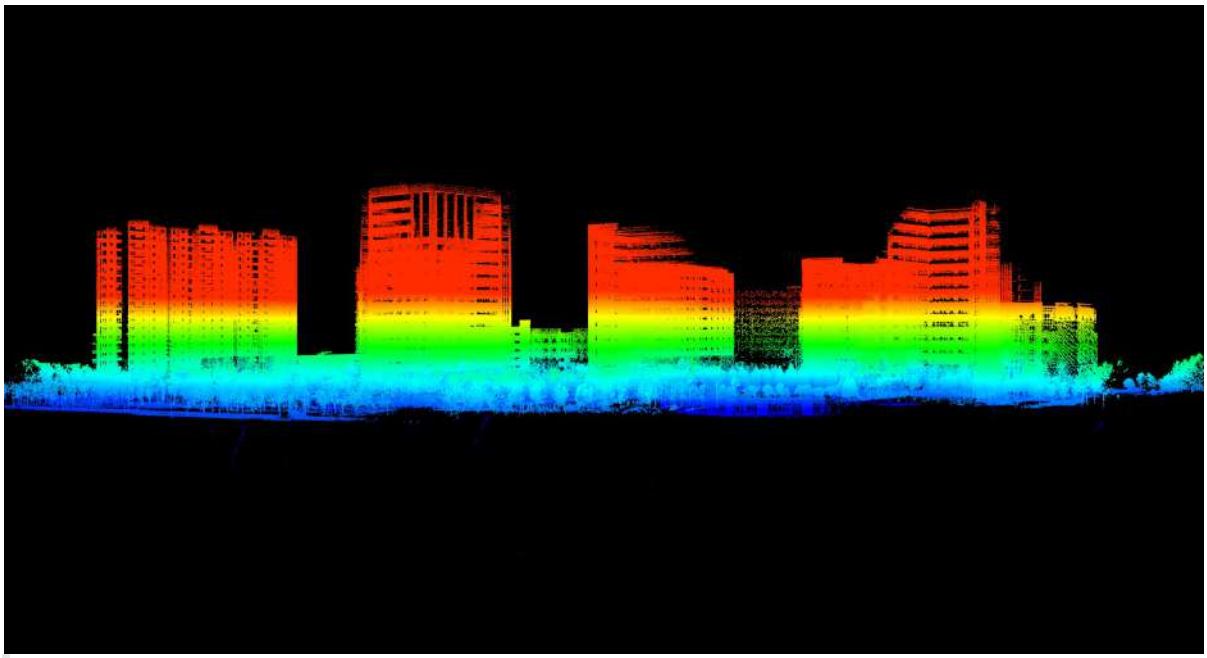
> Note: This function is only for views displayed in 3D; this function does not reset the center position of the viewpoint. If you need to reset to the default viewpoint, please click Global Display.

Right View

Function Description: This tool allows the user to set the camera position to view from the right, that is, view the three-dimensional data from +x to-x direction, and the plane is the y-z plane.

Steps

1.Click **Right**  button, the current activation form is displayed as shown in the figure：



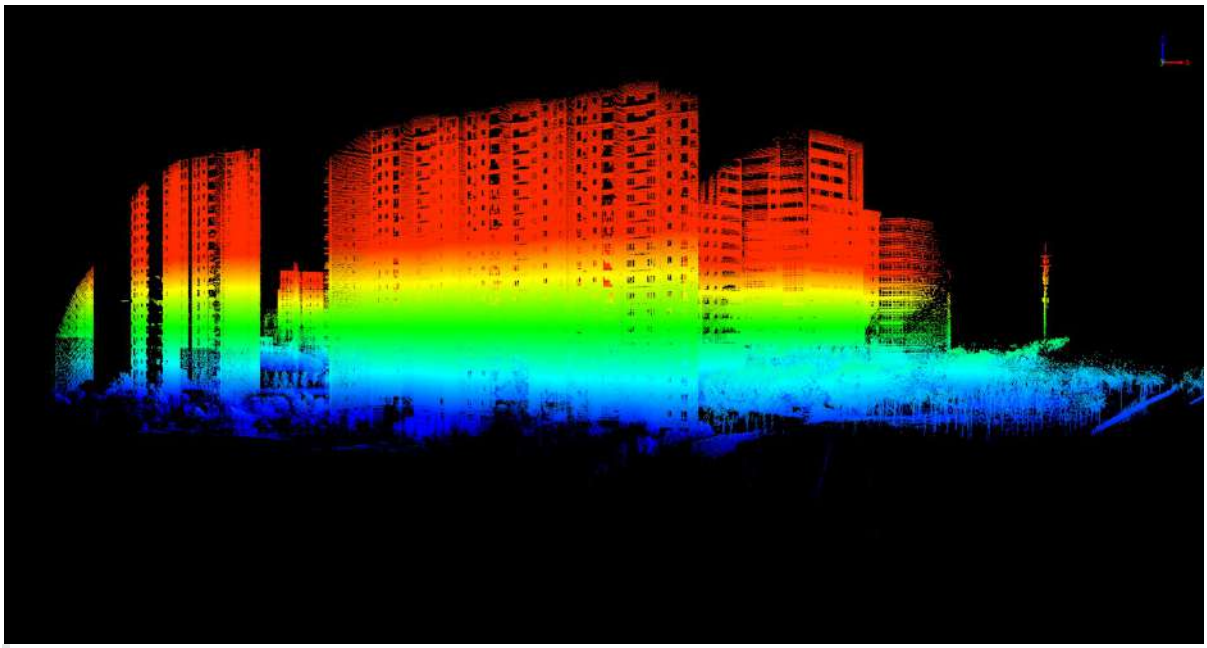
> Note: This function is only for views displayed in 3D; this function does not reset the center position of the viewpoint. If you need to reset to the default viewpoint, please click Global Display.

Front View

Function Description: This tool allows the user to set the camera position to view from the front, that is, view the three-dimensional data from the-y to +y direction, and the plane is the x-z plane.

Steps

1.Click **Front**  button, the current activation form is displayed as shown in the figure：



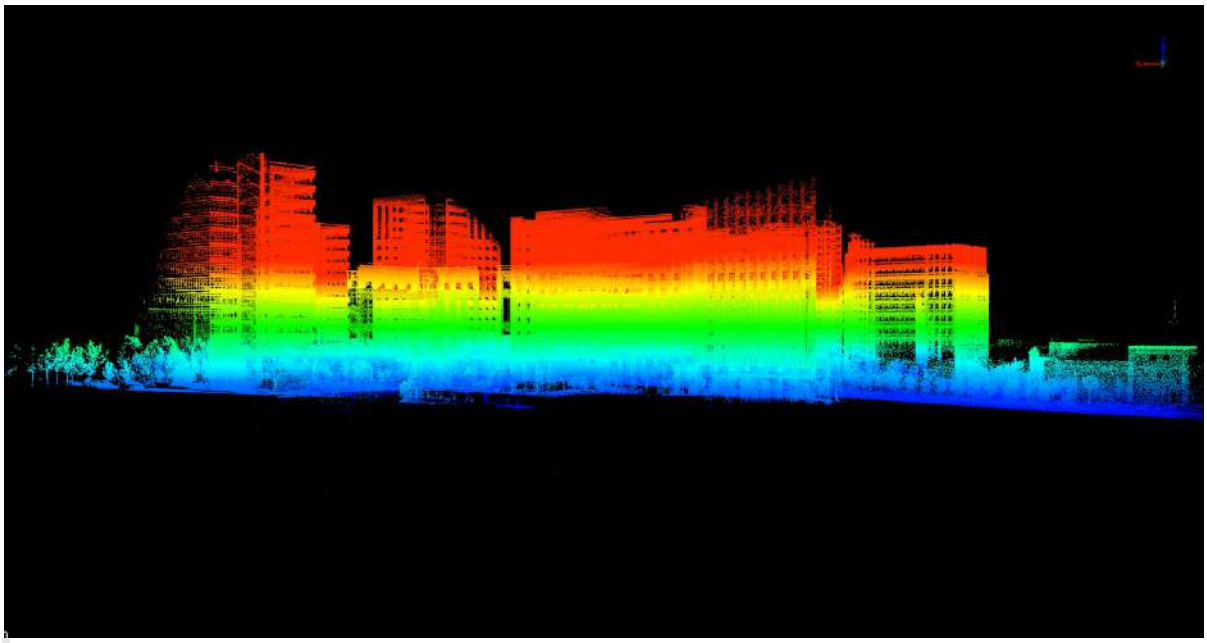
> Note: This function is only for views displayed in 3D; this function does not reset the center position of the viewpoint. If you need to reset to the default viewpoint, please click Global Display.

Back View

Function Description: This tool allows the user to set the camera position to view from the back, that is, view the three-dimensional data from +y to-y direction, and the plane is the x-z plane.

Steps

1.Click **Back**  button, the current activation form is displayed as shown in the figure：



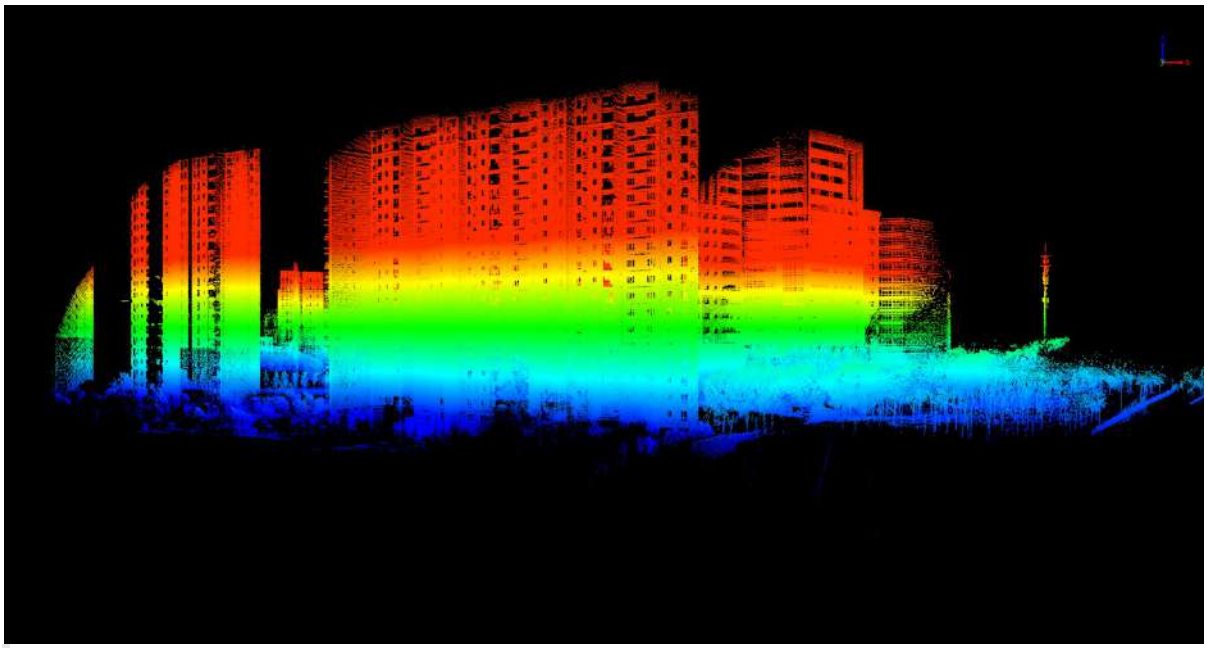
> Note: This function is only for views displayed in 3D; this function does not reset the center position of the viewpoint. If you need to reset to the default viewpoint, please click Global Display.

Isometric Front View

Function Description: This tool allows the user to set the camera position to tilt 45° in front of x-y.

Steps

1.Click **Front Isometric**  button, the current activation form is displayed as shown in the figure：



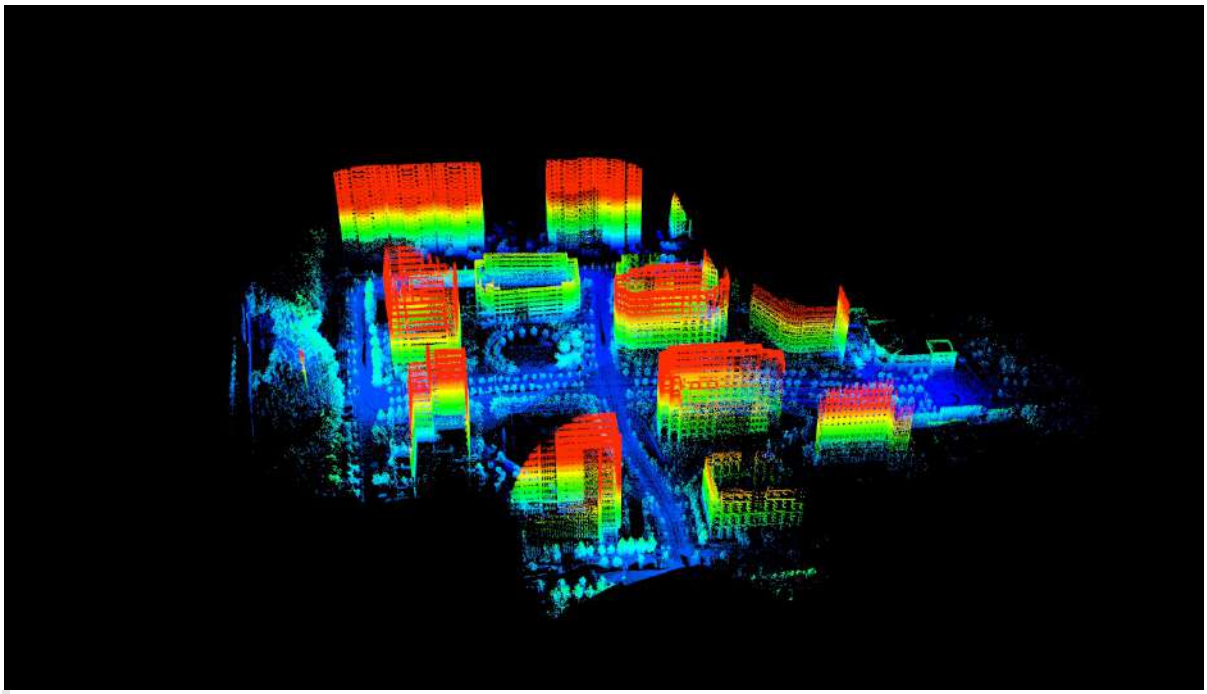
> Note: This function is only for views displayed in 3D; this function does not reset the center position of the viewpoint. If you need to reset to the default viewpoint, please click Global Display.

Isometric Back View

Function Description: This tool allows the user to set the camera position to tilt 45° backward on the x-y surface.

Steps

1.Click **Back Isometric**  button, the current activation form is displayed as shown in the figure：



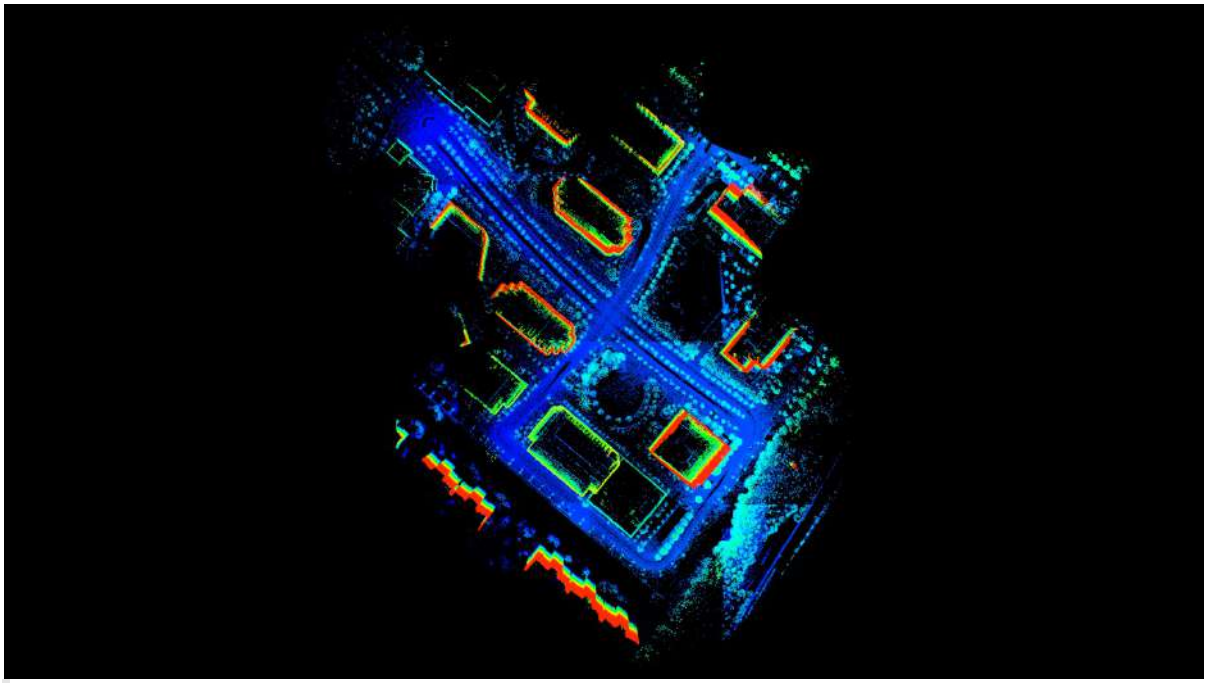
> Note: This function is only for views displayed in 3D; this function does not reset the center position of the viewpoint. If you need to reset to the default viewpoint, please click Global Display.

Full Extent

Function Description: The global display function is suitable for the 3D window of the LiDAR360 MLS software, which is used to make all the data in the 3D window cover the entire window in the form of a top view, in order to achieve the purpose of global browsing of the data.

Steps

1.Click on the toolbar **Full Extent** button, the data in the 3D window will be automatically scaled to cover the entire window, as shown in the figure：



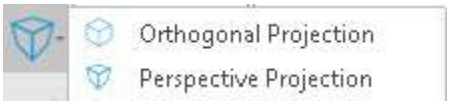
> Note: This function is only for views displayed in 3D; this function does not reset the center position of the viewpoint. If you need to reset to the default viewpoint, please click Global Display.

Set Projection Mode

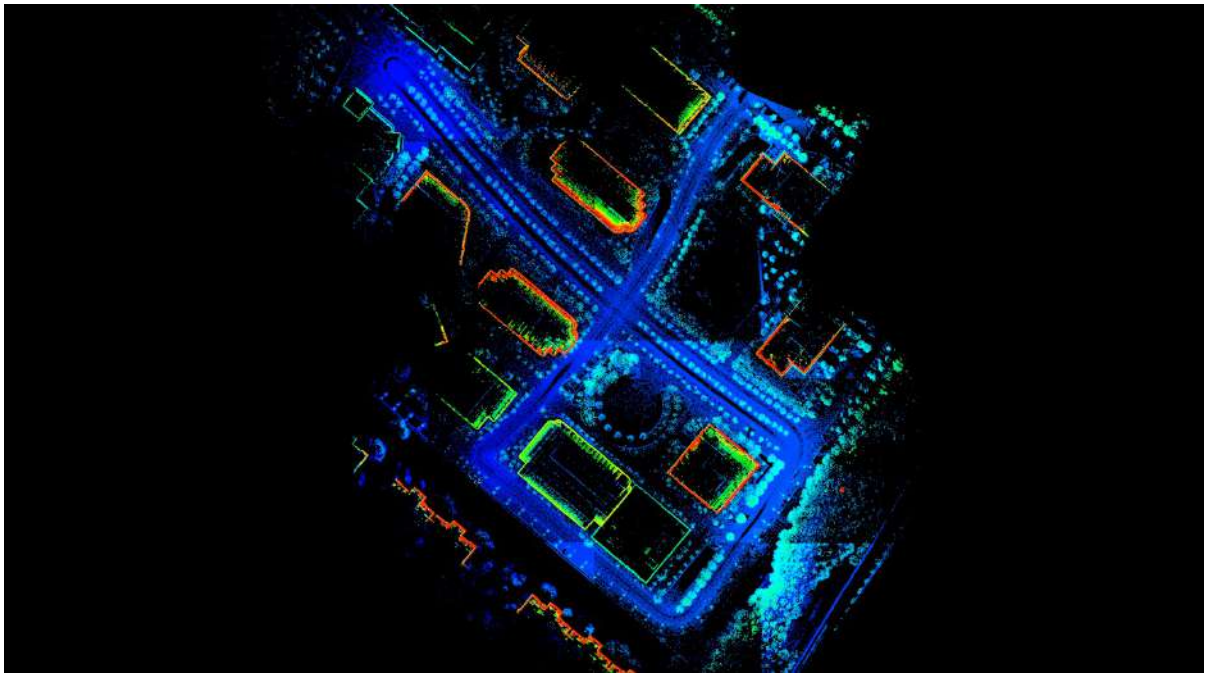
Function Description: This tool allows the user to change the projection method of the view and supports two projection methods, orthogonal and perspective.

Steps

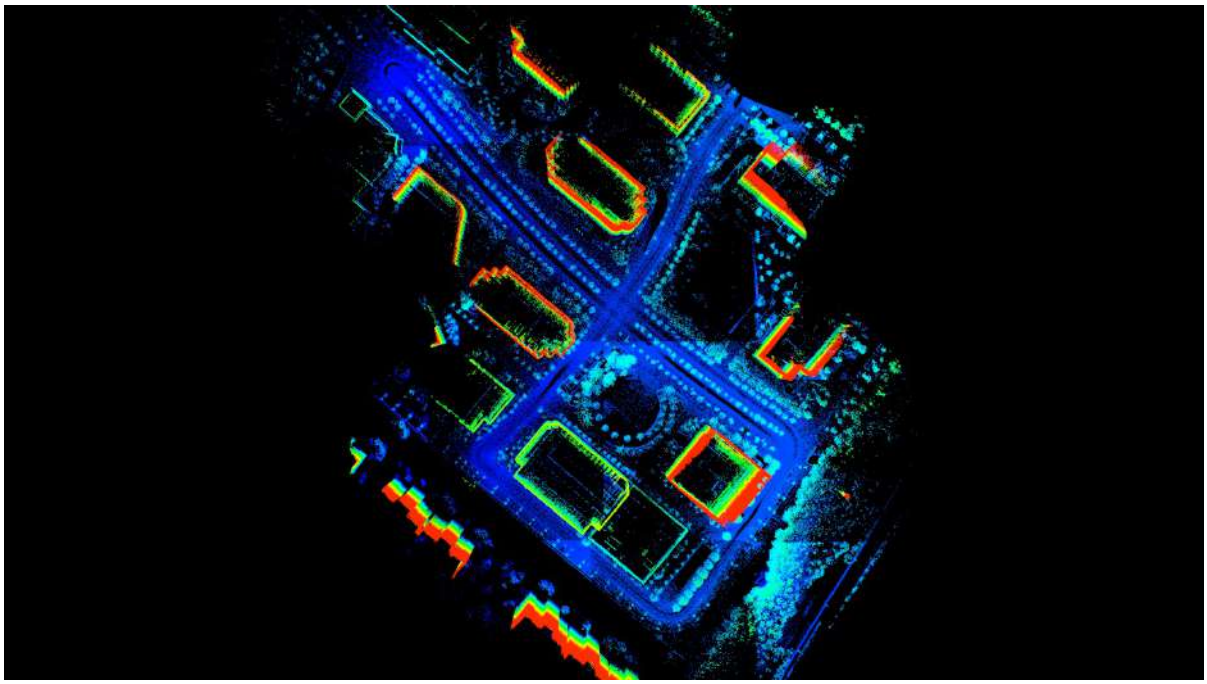
1.Click **Set Projection Mode**  button, pop up the option to select the projection method.



2.If you select orthogonal projection, the point cloud window is orthogonal projection.



3.If perspective projection is selected, the point cloud window is perspective projection.

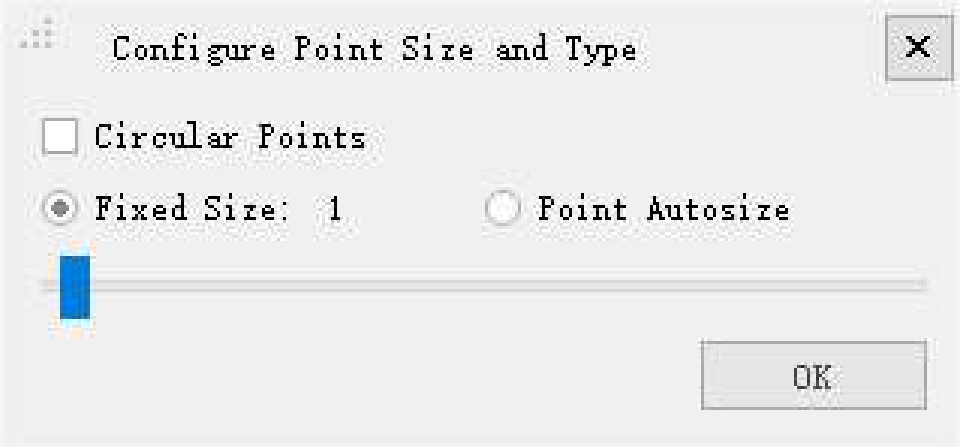


Configure Point Size

Function Description: Set the size and type of points in the three-dimensional point cloud in the entire software system.

Steps

1.Click **Configure Point Size** button, the interface shown in the figure below pops up：



Parameter setting

- **Circular Points (Circle)**: This parameter defines the display type of point cloud. If checked, click to display according to the circle, if not checked, click to display according to the rectangle.
- **Fixed Size (Fixed Size)**: This parameter defines the point of the software system to be displayed in a fixed size. You can slide the slider below to set the point to be displayed between 0-50 pixels.
- **Point Autosize (Auto Size)**: This parameter defines the depth display of the visual vertebral body in the midpoint adaptive form of the software system.

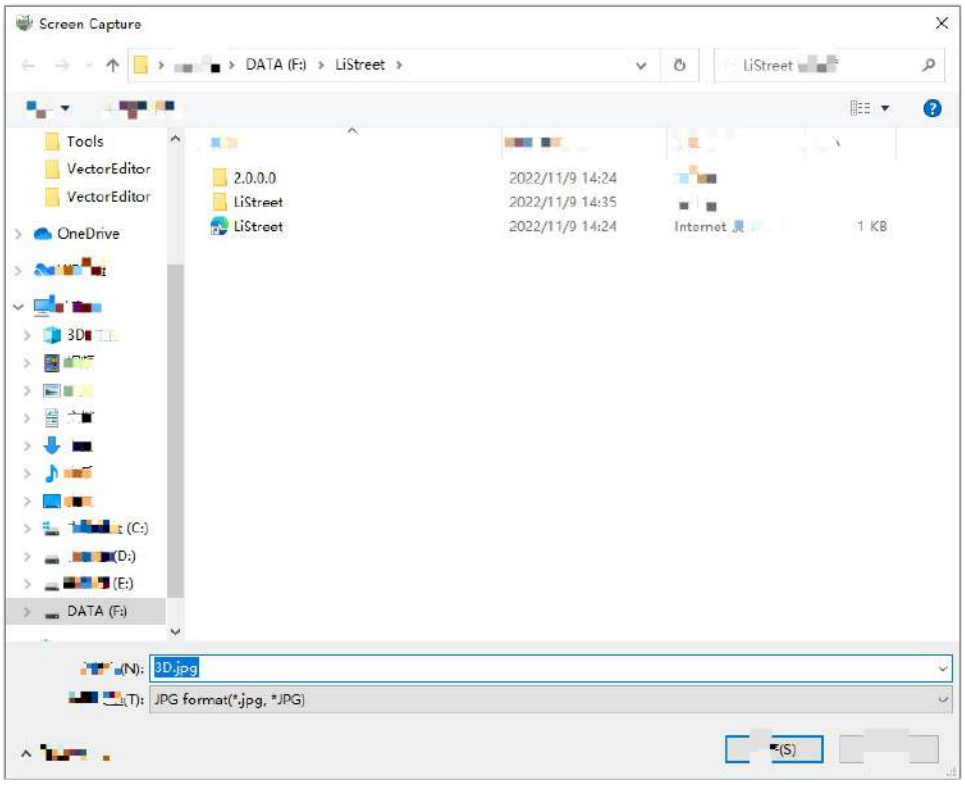
2.Set the size and display type of the point, click OK.

Capture Image

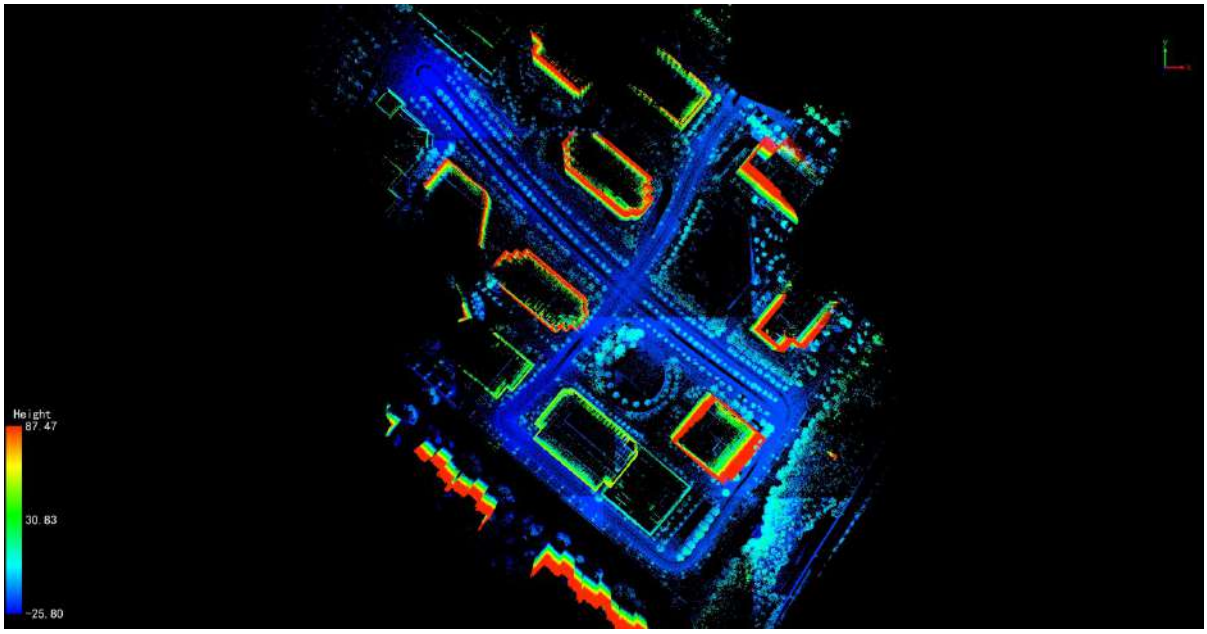
Function Description: This tool allows the user to directly take a screenshot of the point cloud that the user has customized or set a certain viewing angle in the 3D window.

Steps

1.Click **Capture Image**  button, pop up the save path：



2.Click save, you can see the scene of the 3D window screenshot in the corresponding folder.



Color bar tool

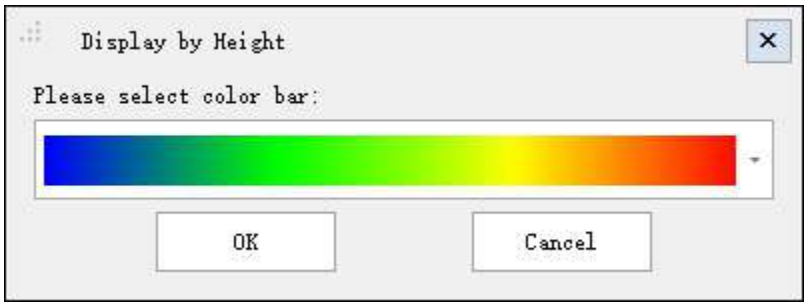
The color toolbar provides several color display modes for the visualization for large point cloud datasets, and the most suitable display method can be selected for different analysis functions (such as display by intensity, display by GPS time, display by echo frequency, etc.). In addition, LiDAR360 MLS provides EDL tools to enhance the display effect, more intuitively reflect the characteristics of the data, and also help to check the quality of the data.

Display by Height

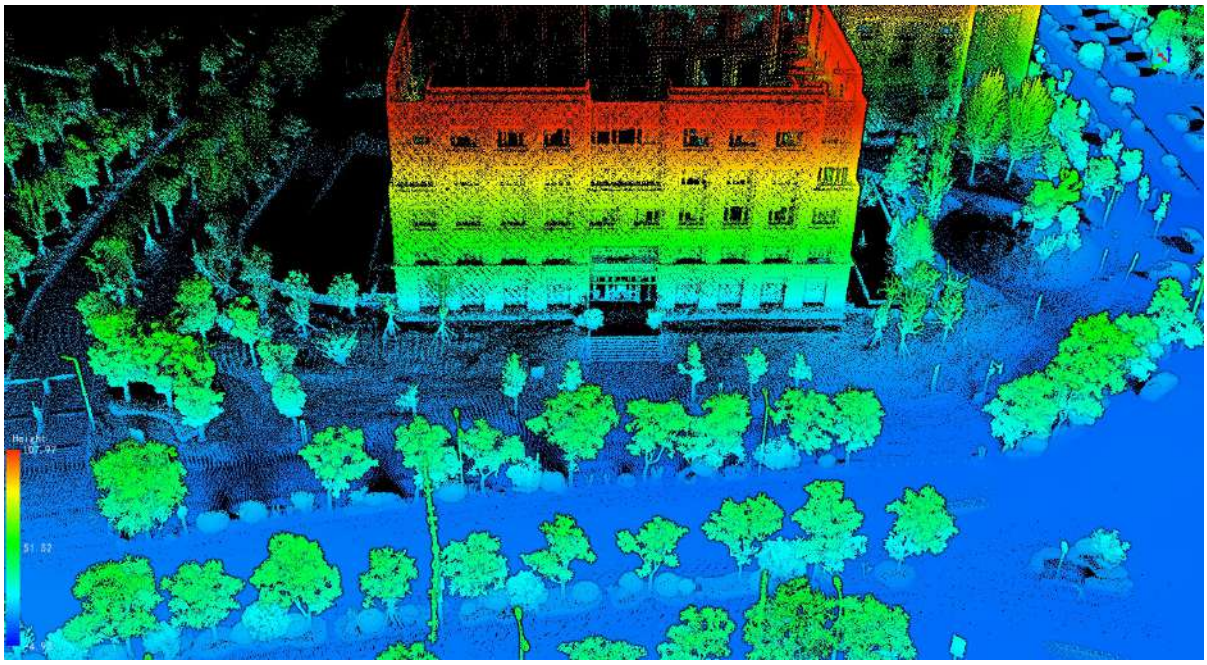
Function Description: It can be used for the display of point cloud data, mapping the elevation attributes of point cloud data to several evenly varying color intervals, and more intuitively showing the changes in the elevation value of point cloud data.

Steps

1.Click  button and the Display by Height dialog box should pop up, as shown in the figure.



2.Select the appropriate color bar in the drop-down box and click the OK button. The color indicator in the lower left corner of the window automatically maps the elevation change range of the point cloud data to the selected color bar. At the same time, the point cloud data in the scene is displayed by elevation, which is enhanced with EDL display.



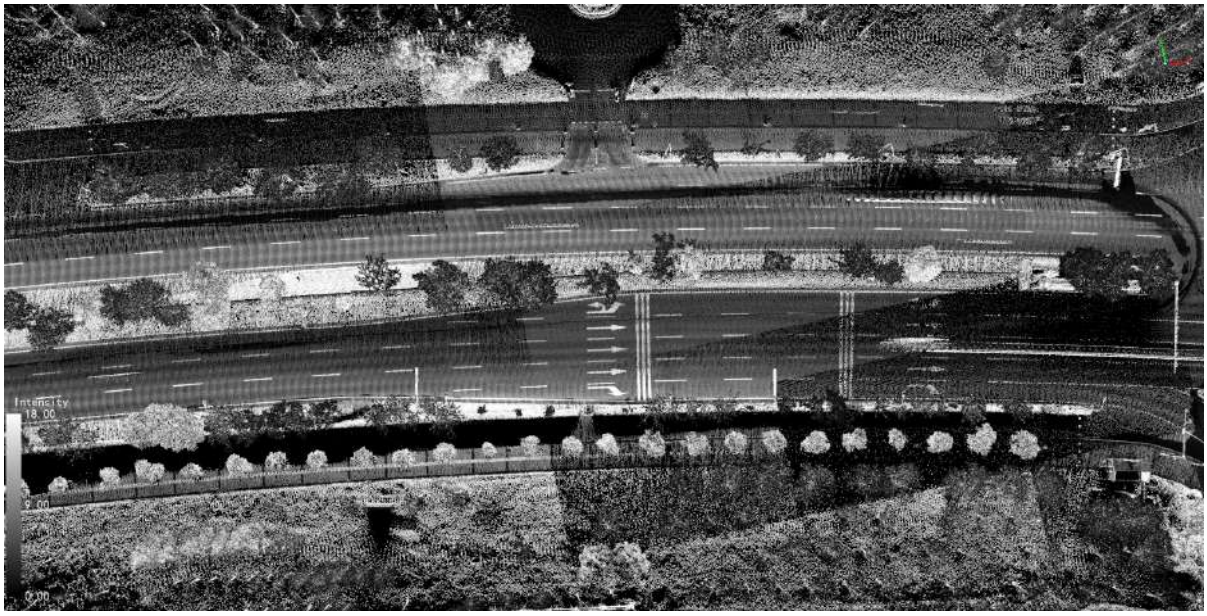
> Note: This function only works on point cloud data.

Display by Intensity

Function Description: This tool can be used for the display of point cloud data, mapping the intensity attributes of point cloud data to evenly varying color intervals, and more intuitively showing the changes in the intensity value of point cloud data.

Step

1.Click  Button, the color indicator in the lower left corner of the window automatically maps the range of changes in the intensity of the point cloud data to the color bar, and the user can choose the color of the color bar. The point cloud data in the scene will be displayed by intensity according to the corresponding color bar.



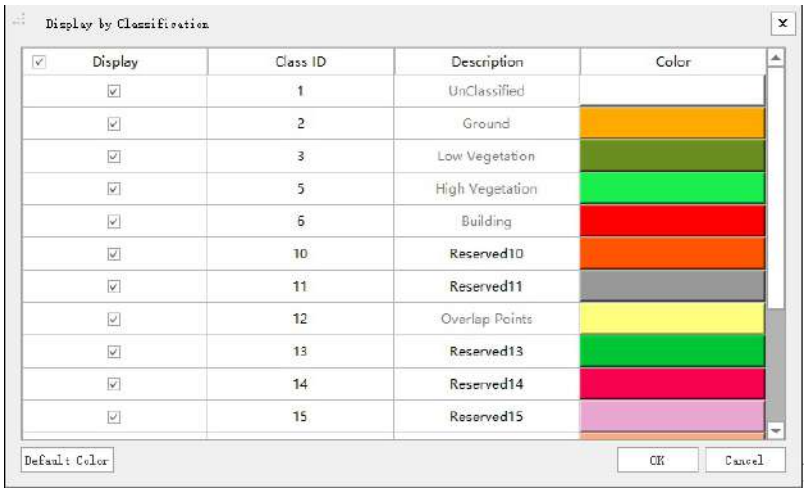
> Note: This function only works on point cloud data.

Display by Classification

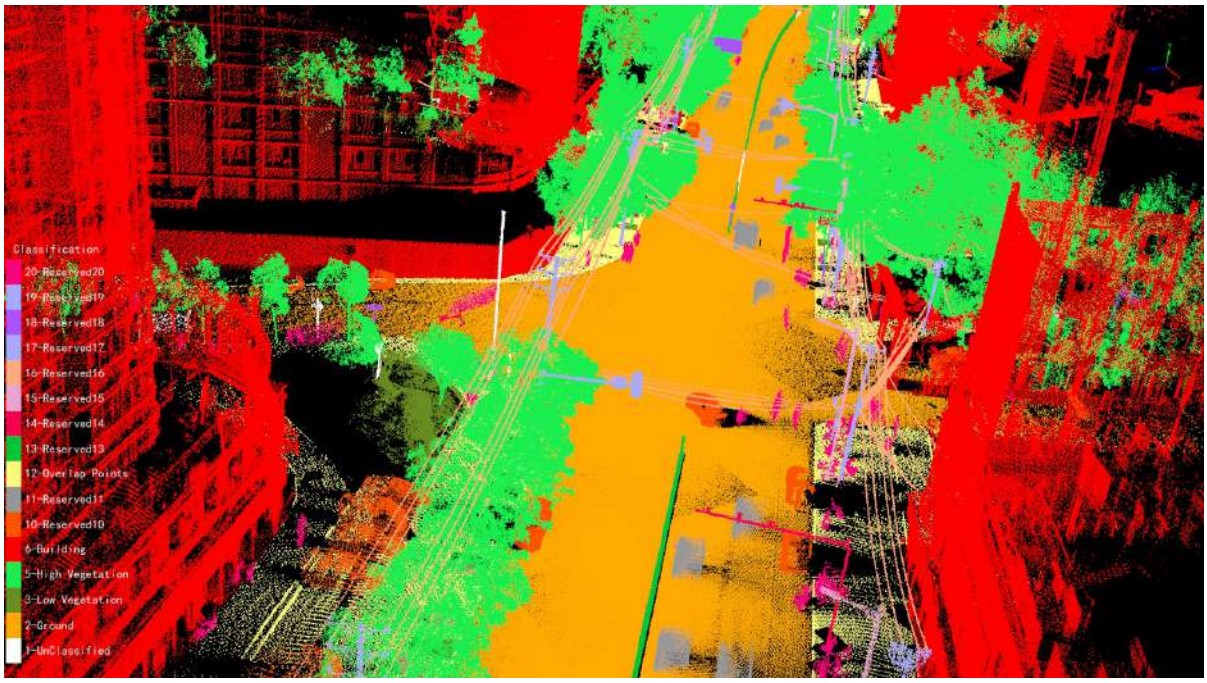
Function Description: This tool can be used for the display of point cloud data, mapping the category attributes of point cloud data to different color values, and more intuitively distinguishing various categories of point cloud data.

Steps

1.Click  button and the Display by Classification dialog box should pop up, as shown in the figure.



2.Select different colors for different categories, click the OK button, and the color indicator in the lower left corner of the window automatically maps different categories of point cloud data to the corresponding colors. At the same time, the point cloud data in the scene is displayed by category, which is enhanced with EDL display.



> Note: This function only works on point cloud data.

Display by RGB

Function Description: This tool can be used for the display of point cloud data as well as the point cloud data drawn with the RGB color attributes of existing point cloud data.

Steps

1.Click  button, the point cloud data in the scene is displayed according to its own RGB value, the effect is shown in the figure.

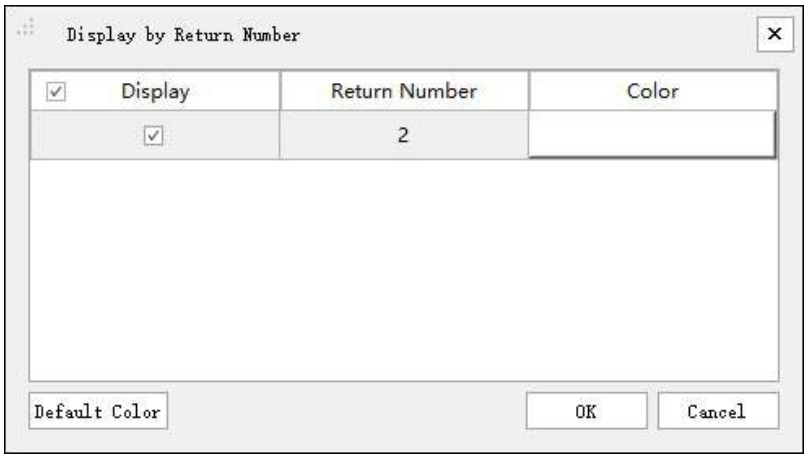
> Note: This function only works on point cloud data that contains RGB attributes.

Display by Return

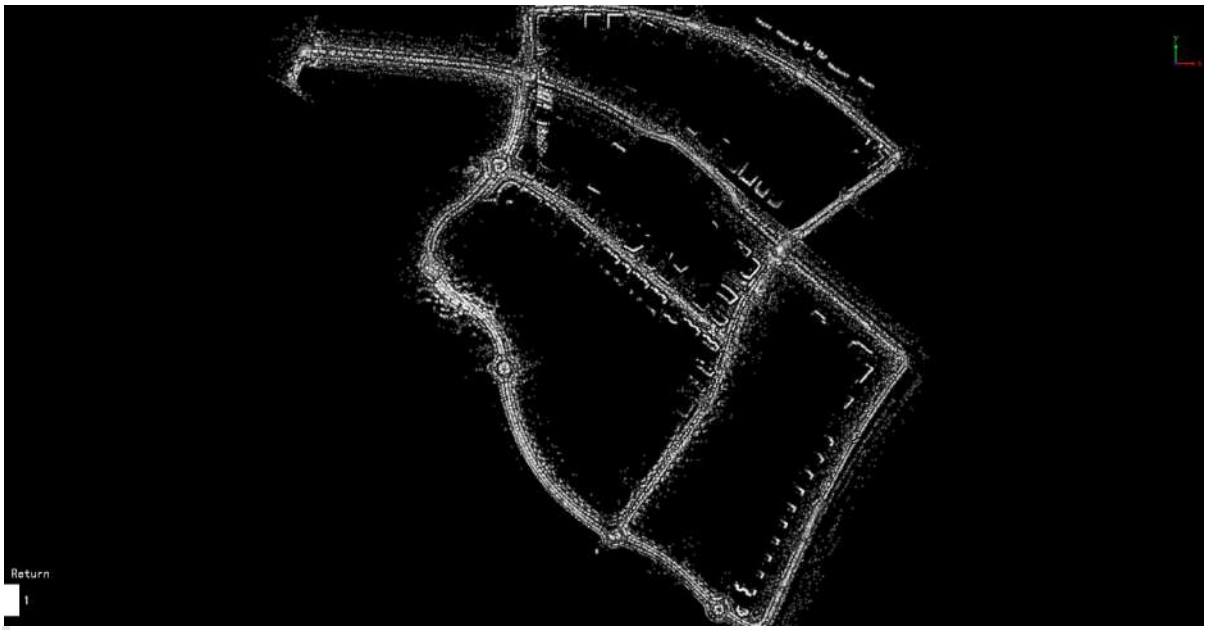
Function Description: This tool can be used for the display of point cloud data, mapping the return number attributes of point cloud data to different color values, and more intuitively distinguishing the echo number of point cloud data.

Steps

1.Click  button and the Display by Return Number dialog box should pop up as shown in the figure.



2.Select different colors for different return number, click the OK button, and the color indicator in the lower left corner of the window automatically maps the different return number of the point cloud data to the corresponding color. At the same time, the point cloud data in the scene is displayed according to the return number, which is enhanced with EDL display, the effect is as shown in the figure.



> Note: This function only works on point cloud data.

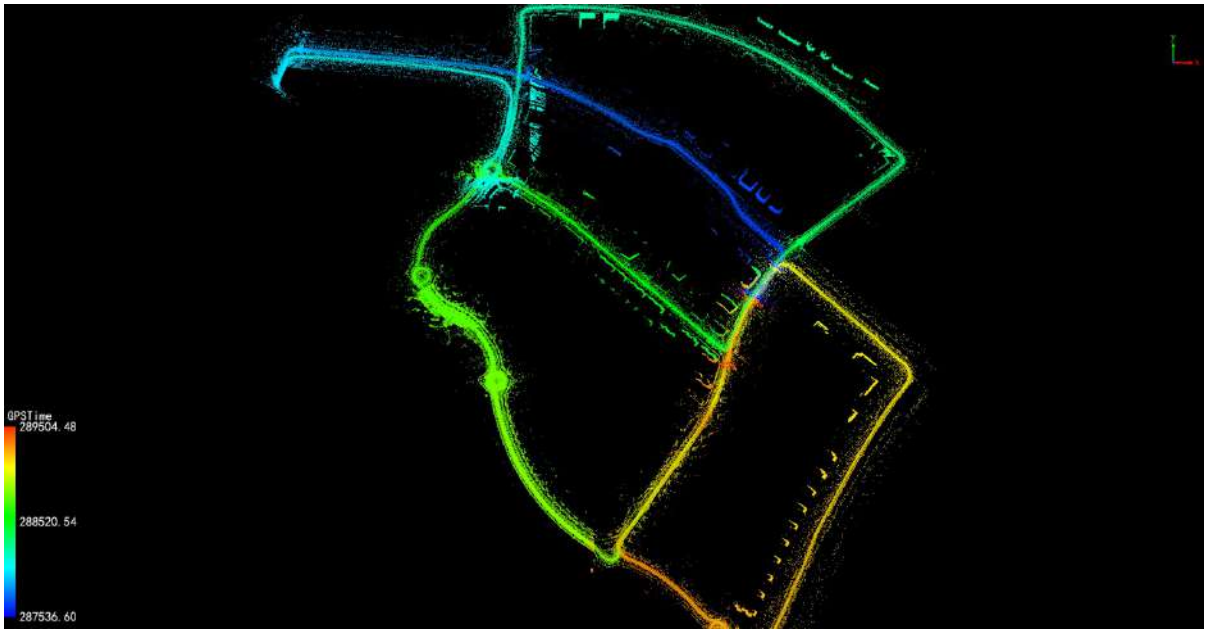
Display by Time

Function Description: This tool can be used for the display of point cloud data, mapping the GPS time attributes of point cloud data to evenly varying color values, and more intuitively showing the changes in the GPS time attributes of point cloud data.

Steps

1.Click  button and the Display by GPS Time should pop up.

2.Select the appropriate color bar in the drop-down box, click the OK button, the color indicator in the lower left corner of the window automatically maps the GPS time change range of point cloud data to the selected color bar, and the scene point cloud data is displayed according to GPS time.



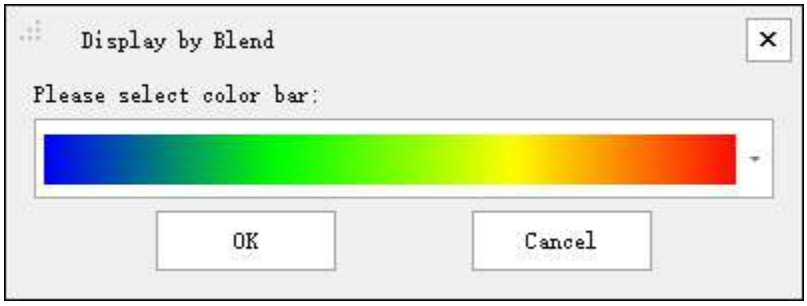
> Note: This function only works on point cloud data.

Display by Blend

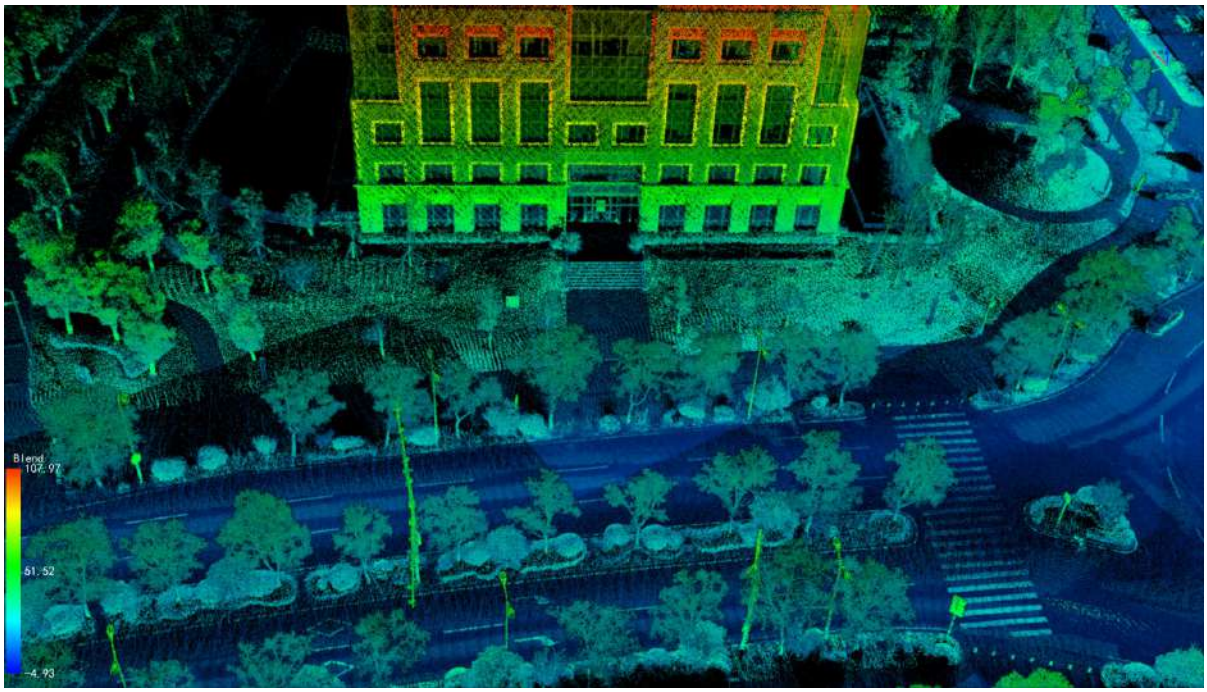
Function Description: This tool can be used for the display of point cloud data, synthesize the elevation attributes and intensity attributes of point cloud data, map to evenly varying color intervals, and more intuitively show the comprehensive changes in the elevation and intensity of point cloud data, while more clearly showing the category and boundary of objects.

Steps

1.Click  button and the Display by Blend display dialog box should pop up, as shown in the figure.



2.Select the appropriate color bar in the drop-down box and click the OK button. The color indicator in the lower left corner of the window automatically maps the elevation value of the point cloud data to the selected color bar. At the same time, the point cloud data in the scene is displayed according to the mixed elevation and intensity, which is enhanced with EDL display.



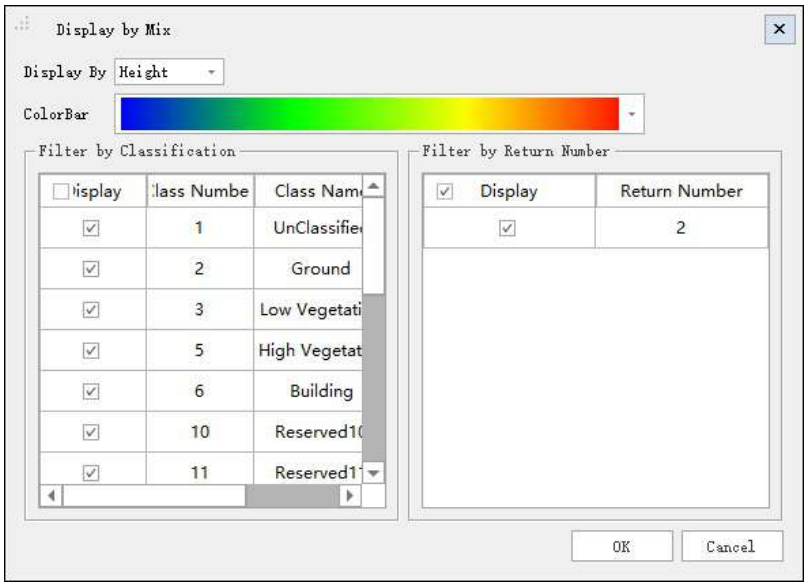
> Note: This function only works on point cloud data, and the mixed display effect is better after PCV processing of point cloud data.

Display by Mix

Function Description: This tool can be used for the display of point cloud data, mapping different attributes of point cloud data to evenly varying color intervals, and providing a way to filter by attribute value to more intuitively show the changes in a certain attribute value of the filtered point cloud data.

Steps

1.Click  1.button and the Display by Mix dialog box should pop up, as shown in the figure.



Parameter Setting

- **Display**: The system uses the attribute value corresponding to this parameter to map to the selected color range.
  - **Height (default)**: The height attribute of point cloud data.
  - **Intensity**: The intensity attribute of point cloud data.
  - **GPS Time**: The GPS time attribute of point cloud data.
- **Color Bar**: A color bar used to map uniform changes in the display properties of the point cloud.
- **Filter by Classification**: Lists the category values used to select the filtered point cloud data.
- **Filter by Return Number**: Lists the values of return number used to select the filtered point cloud data.

2.Select the attributes to display.

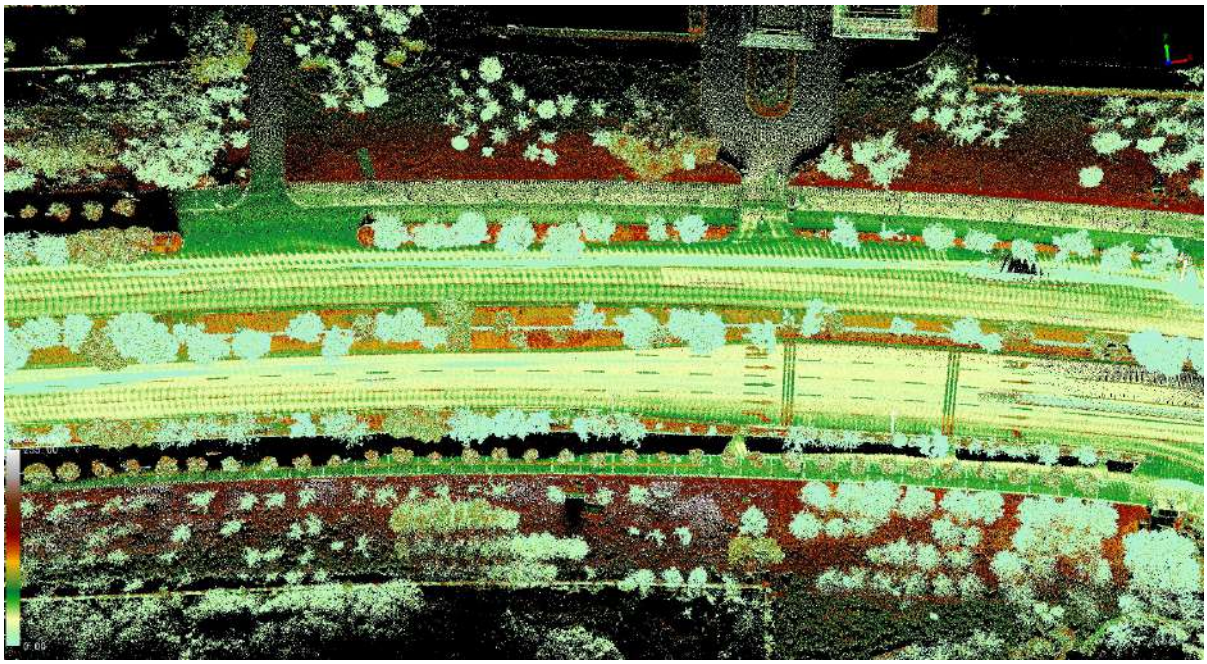
3.Select the appropriate color bar in the drop-down box.

4.Check the category and number of echoes that need to be filtered.

5.Click the OK button, and the color indicator in the lower left corner of the window maps the selected attributes to the selected color bar. At the same time, the cloud data in the scene will be filtered according to the specified attributes first, and then displayed according to the display attributes. The display effect is enhanced with EDL, and the comparison effect before and after the display is combined as shown in the figure.

Before the combination is displayed:

After the combination is displayed:



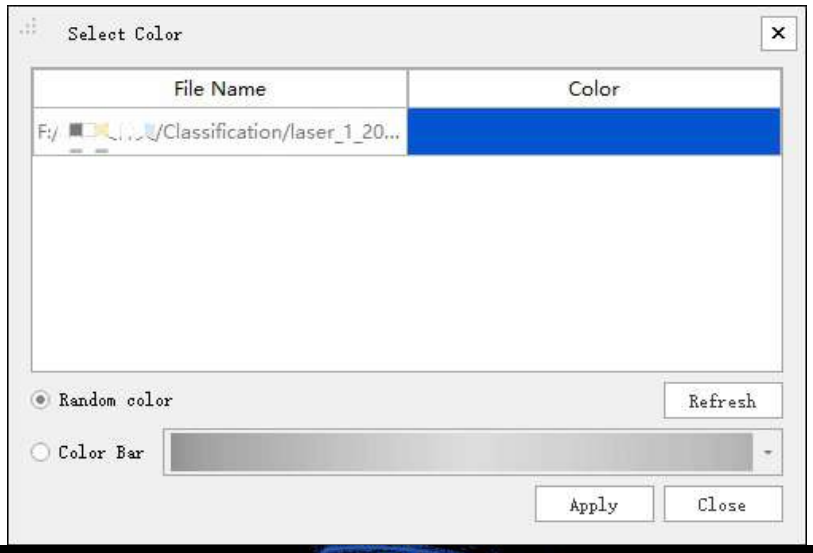
> Note: This function only works on point cloud data.

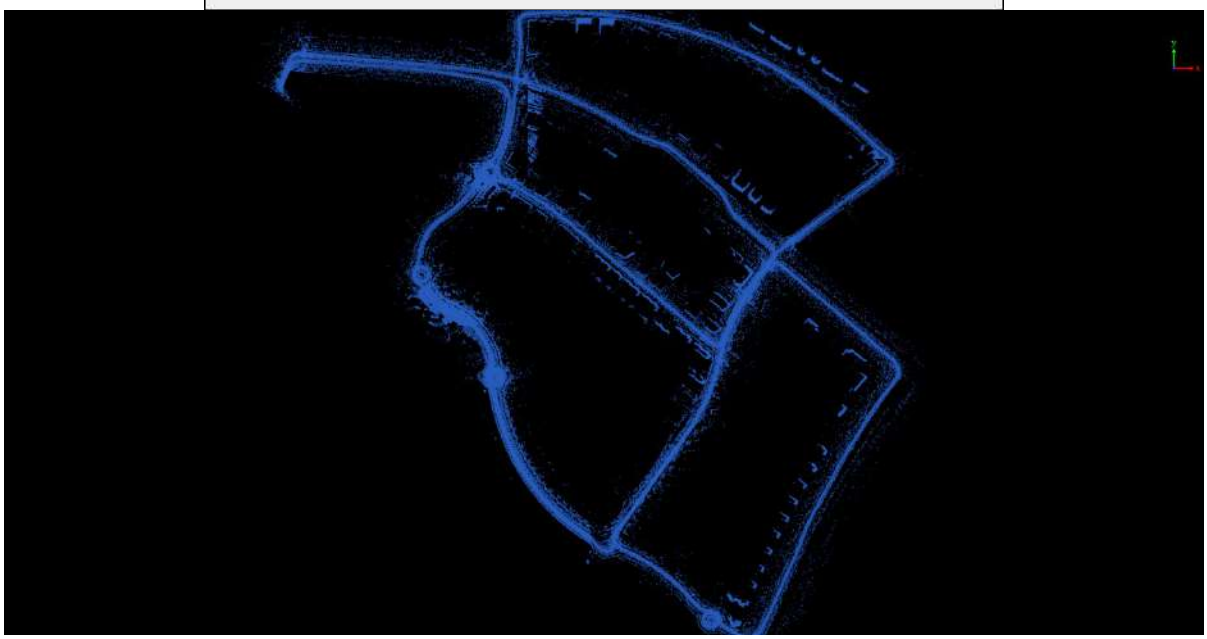
Display by Select Color

Function Description: This tool can be used to display point cloud data by a specified color.

Steps

1.Click  button, click the color button corresponding to each point cloud to display the point cloud as the specified color.



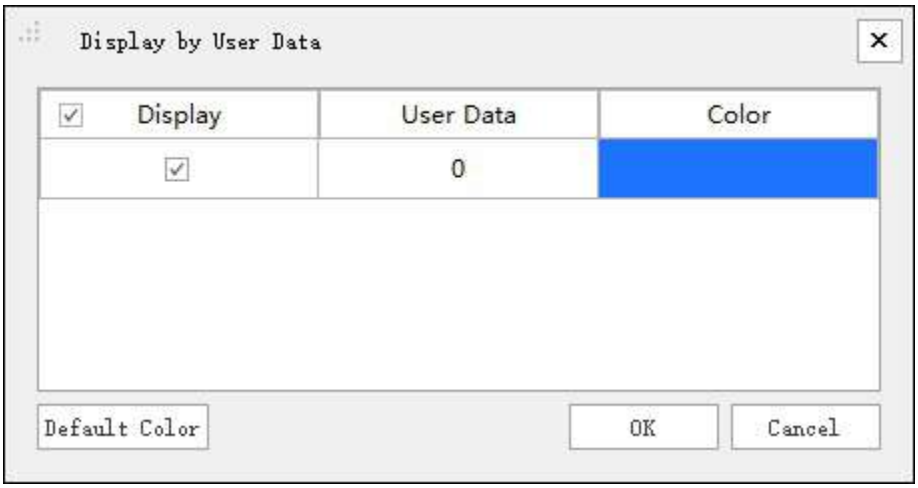


Display by User Data

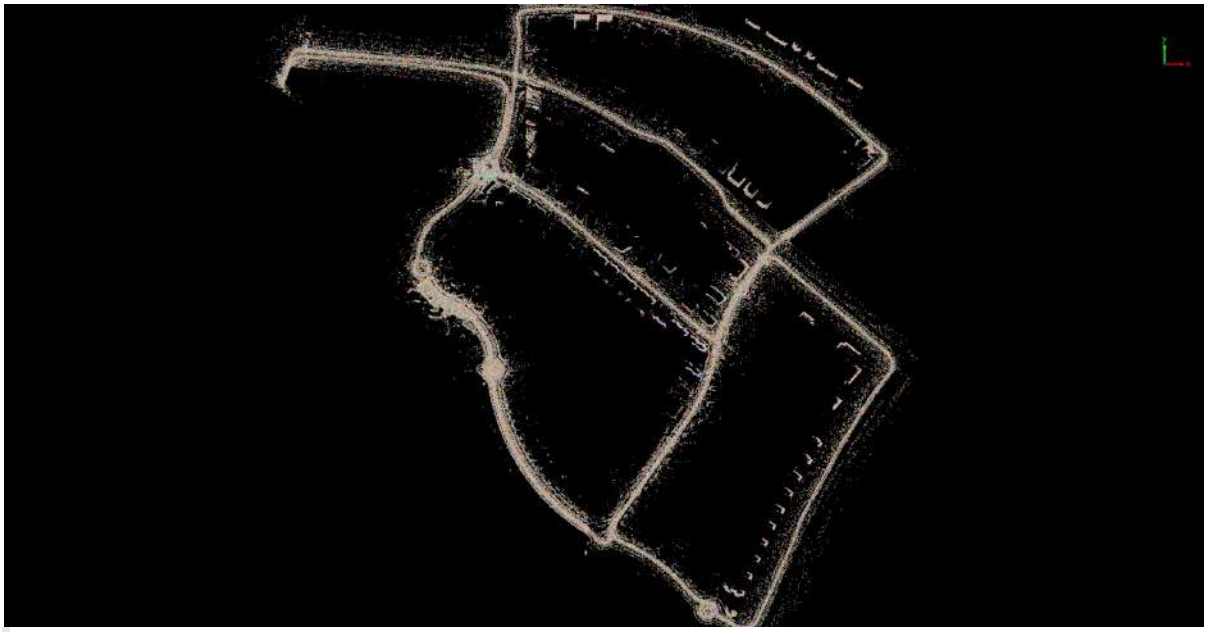
Function Description: This tool can be used for the display of point cloud data, mapping the user data attributes of point cloud data to different color values, and more intuitively distinguishing the point cloud data of different user data.

Steps

1.Click  button and the Display by User Data display dialog box should pop up, as shown in the figure.



2.Select different colors for different user data, click the OK button, and the color indicator in the lower left corner of the window automatically maps the different user data of the point cloud data to the corresponding color. At the same time, the point cloud data in the scene is displayed according to the user data. The display effect is enhanced with EDL, and the effect is shown in the figure.



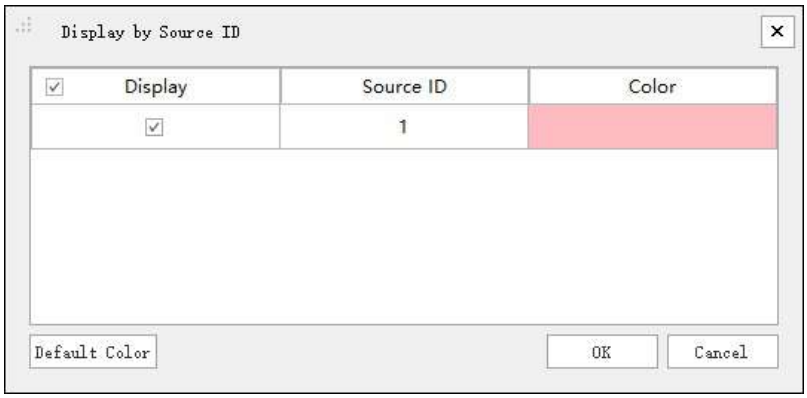
> Note: This function only works on point cloud data.

Display by Source Color

Function Description: This tool can be used for the display of point cloud data, mapping the point source ID attributes of point cloud data to different color values, and more intuitively distinguishing point cloud data of different point source IDs.

Steps

1.Click  button and the Display by Source dialog box should pop up, as shown in the figure.



2.Select different colors for different point source IDs, click the OK button, and the color indicator in the lower left corner of the window automatically maps the different point source IDs of the point cloud data to the corresponding colors. At the same time, the point cloud data in the scene is displayed according to the point source ID, which is enhanced with EDL display, and the effect is as shown in the figure.

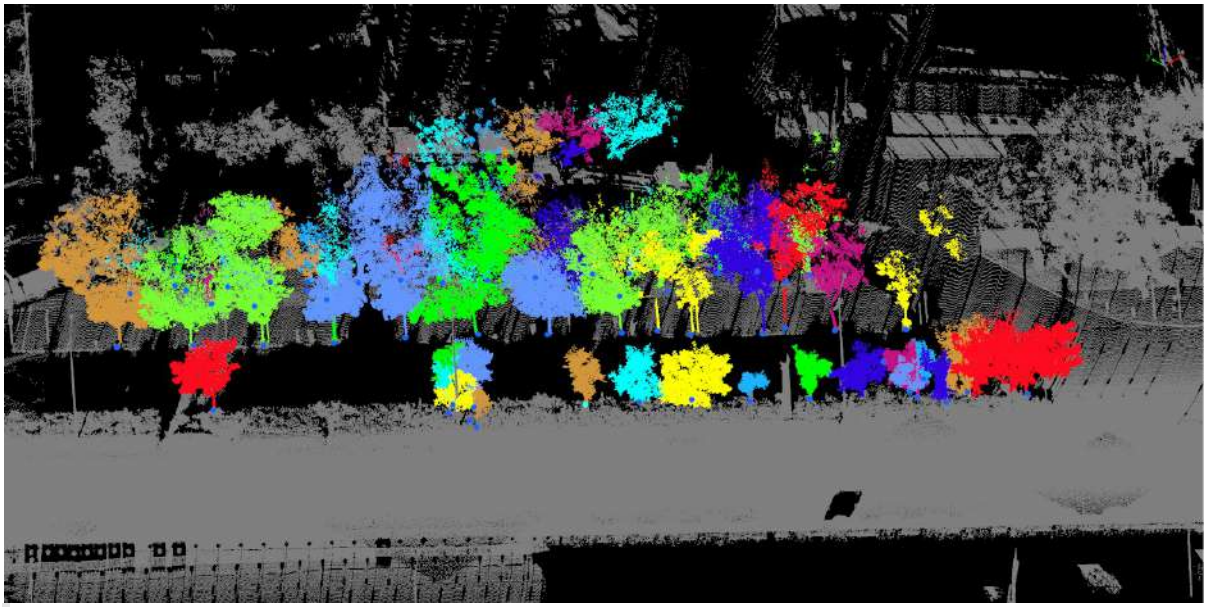
> Note: This function only works on point cloud data.

Display by Tree ID

Function Description: This tool can be used for the display of point cloud data, mapping the tree ID attributes of point cloud data to different color values, and more intuitively distinguishing point cloud data of different tree IDs.

Steps

1.Click  button, select different colors for different tree IDs, click the OK button, the color indicator in the lower left corner of the window automatically maps the different tree IDs of the point cloud data to the corresponding colors, and at the same time, the point cloud data in the scene is displayed according to the tree ID, which is better with EDL display, the effect is shown in the figure.



> Note: This function only works on point cloud data.

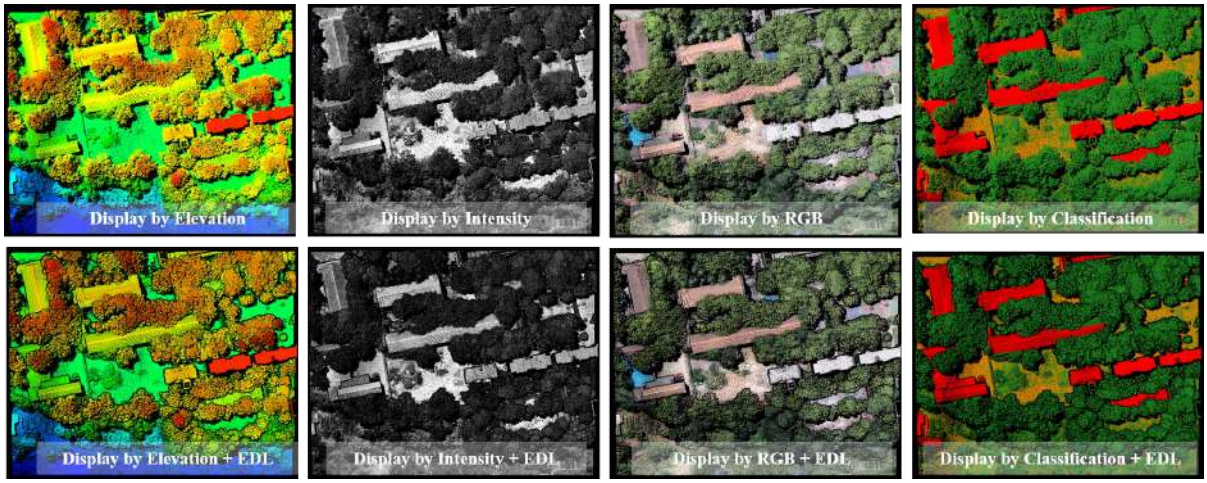
Display by EDL

Function Description: This tool can be used for the display of point cloud data and used in conjunction with other display methods to enhance the display of the contour characteristic of point cloud objects.

Steps

1.Click 

button, the point cloud data in the scene will be displayed in the EDL manner to enhance the display effect. The comparison of the point cloud display effect before and after using EDL in different display methods is shown in the figure.



> Note: This function only works on point cloud data.

Project Management

The project management includes five parts: layer management, layer setting, window management, display mode and element attribute.

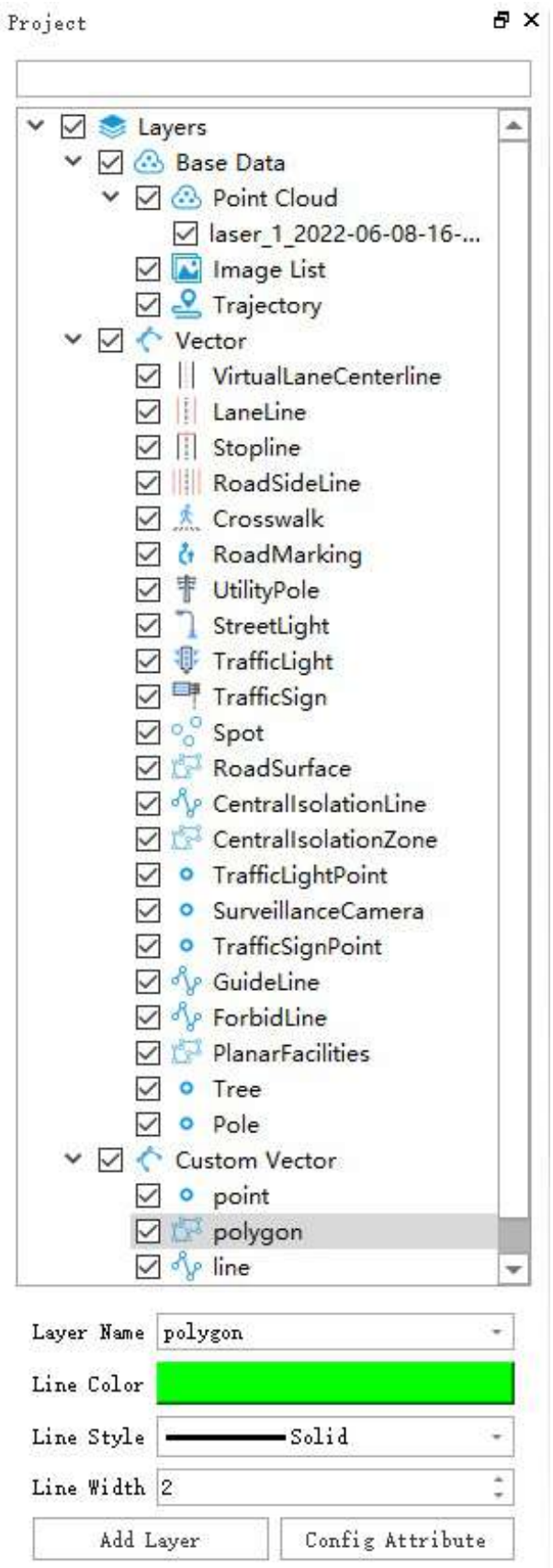
- Layer Management
- Layer Settings
- Window Management
- Display Mode
- Feature Attribute

Layer management

Function Description: Layer management manages the data contained in the software in groups, and the functions include the visibility control of the data in the entire software system (all windows).

> By checking or unchecking the checkbox of the tree node, you can control the display and hiding of the current data in the entire software, and the right-click menu of the data node is mainly responsible for data query, display, statistics, export, removal and other operations. The right-click menu for different data types (including point clouds, vectors, etc.) are different.

1.The layer management window is open by default. If the window is closed, you can click on  button in the shortcut toolbar to display a list of layers, as shown in the figure below：



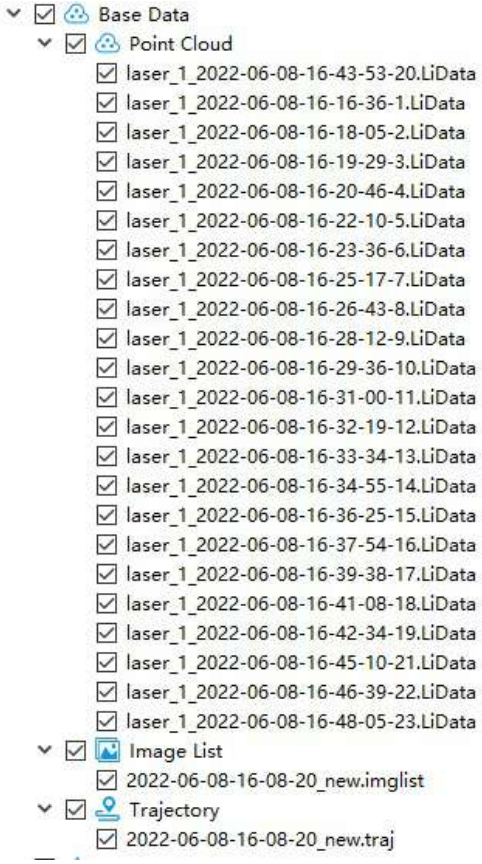
2.According to the data source, the layer can be divided into four parts: the basic data layer and the vector data layer. The vector data layer includes software built-in layers, custom vector layers, and table data layers：

- Basic data

- Vector
- Custom vector
- Tables

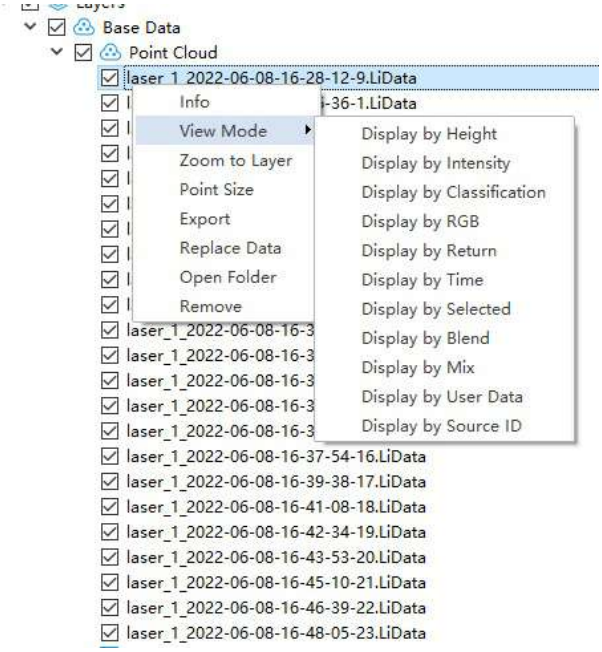
Basic Data

Function Description： The basic data layer includes point clouds, image lists, and trajectories. The following will introduce the right-click functions in different source data layers.



Point Cloud Data Context Menu

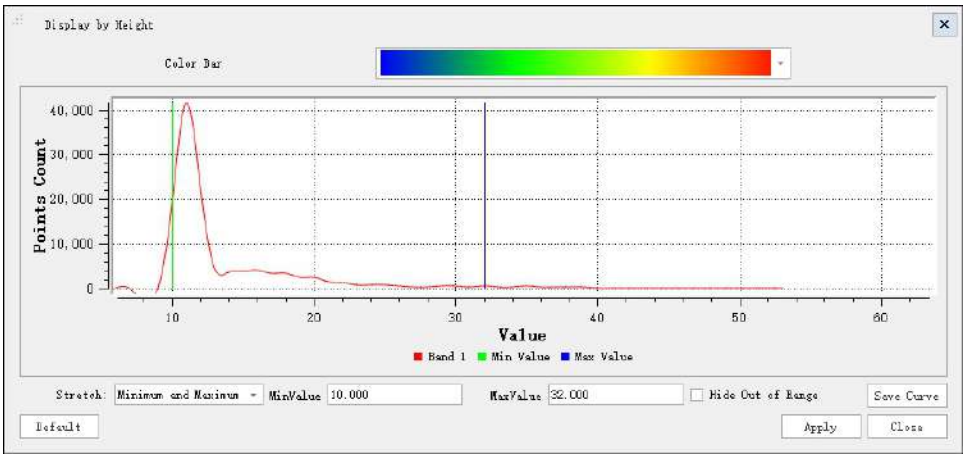
The right-click menu of point cloud contains the content shown in the figure below：



1.**Information**： View the basic information of the point cloud, including the path where the data is located, coordinate information, the minimum and maximum values of X, Y, and Z coordinates, the mean and standard deviation of Z, the minimum and maximum GPS time, the minimum, maximum, average and standard deviation of the intensity, the point cloud bounding box, the total number of points, the statistics of point cloud categories, and the number of echoes. Click the "Export" button to export the basic information of the point cloud as a txt file.

2.**View Mode**: Sets the display mode for individual point cloud files, including the following types:

- **Display by Height**: The interface as shown in the figure pops up, which can be stretched by the minimum and maximum values or standard deviations to improve the display effect.



Parameter setting：

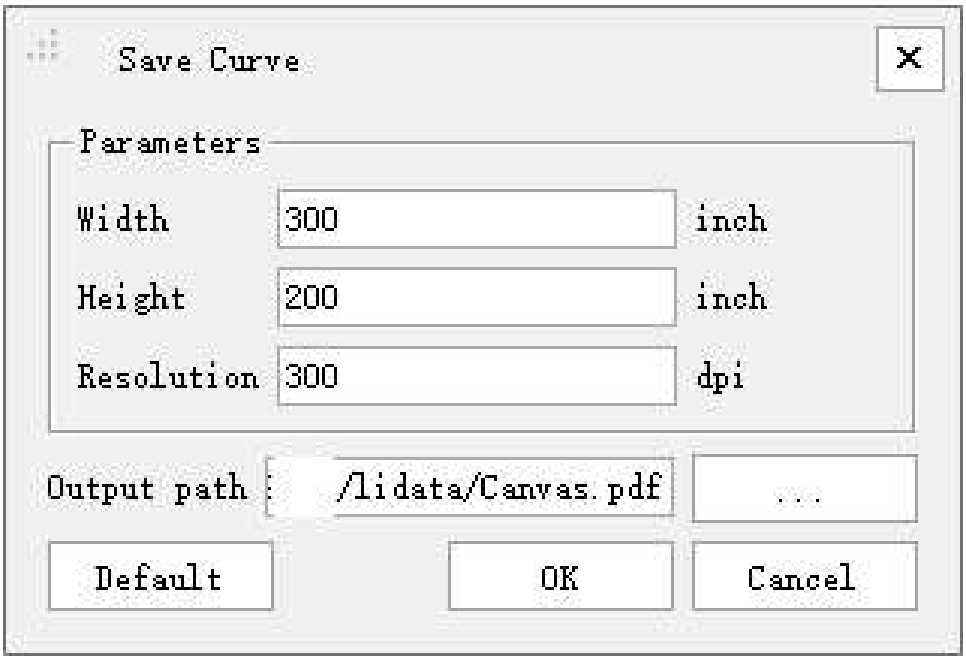
- **Color Bar**: A color bar used to map the elevation properties of a point cloud.
- **Stretch**: Set the stretching method of the histogram.
- **Minimum maximum value (default)**: Linear stretching is applied based on the minimum and maximum pixel values, with the minimum and maximum pixel values as the endpoints of the histogram. For example, the minimum and maximum values of the image are 2488 and 2656, respectively. Linear stretching distributes the pixel values between 0-255. By distributing the pixel values over the entire histogram range, the brightness and contrast of the image are improved, and the features in the image are easy to distinguish.
- **Standard deviation**: The standard deviation stretch type applies linear extension between values defined by standard deviation n. For example, the minimum and maximum values of an image are 2488 and 2656 respectively, and if n is 2, the value exceeding ±2 standard deviations becomes 0 or 255, and the other values are stretched between 0-255.

> The histogram displayed on the interface can be exported in PDF format, click the "Save Curve" button to pop up the "Save Curve" dialog box.

Parameter setting：

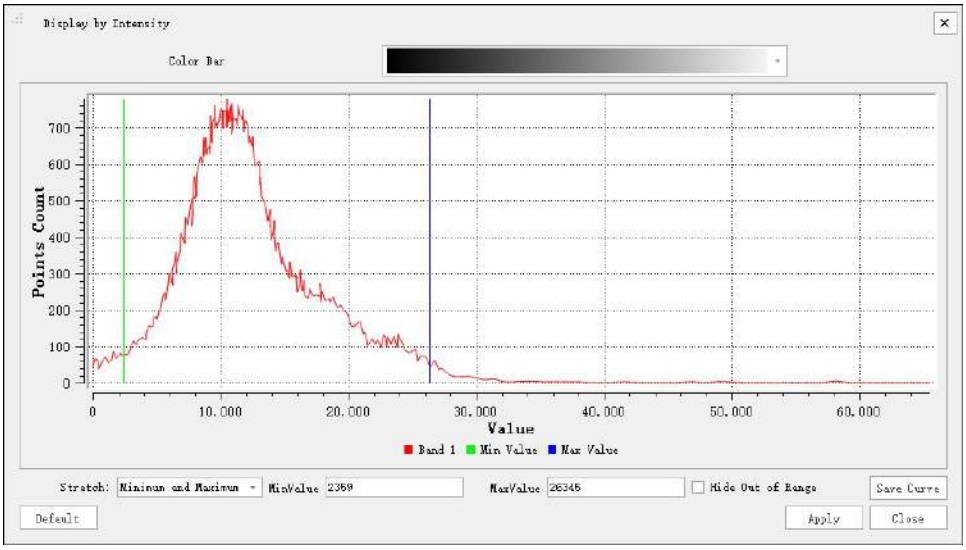
- **Width**: Save the pixel width of the curve.
- **Height**: Save the pixel height of the curve.
- **Resolution**: Save the resolution of the curve.
- **Output Path**: Save the output path of the curve.

Select the width, height, resolution and output path, click the OK button, and save the curve.



For specific display effects, see Display by elevation.

- **Display by Intensity**: The interface as shown in the figure pops up, which can be stretched by the minimum and maximum values or standard deviations to improve the display effect.



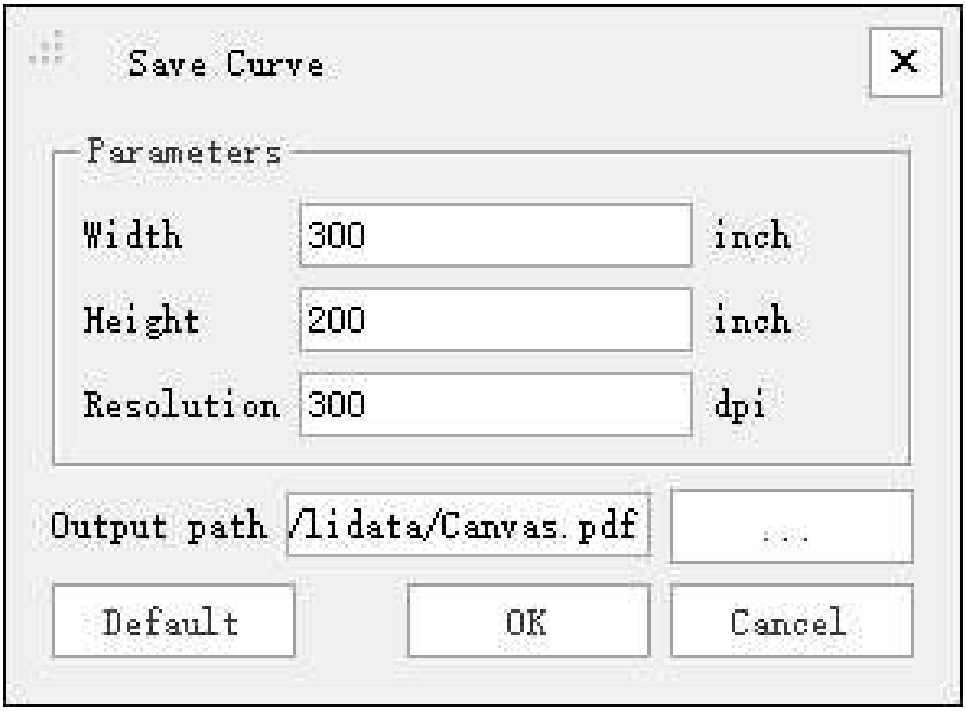
Parameter setting

- **Stretch**: Set the stretching method of the histogram.
- **Minimum and maximum values (default)**: Linear stretching is applied based on the minimum and maximum pixel values, with the minimum and maximum pixel values as the endpoints of the histogram. For example, the minimum and maximum values of the image are 2488 and 2656, respectively. Linear stretching distributes the pixel values between 0-255. By distributing the pixel values over the entire histogram range, the brightness and contrast of the image are improved, and the features in the image are easy to distinguish.
- **Standard deviation**: The standard deviation stretching type applies linear extension between the values defined by the standard deviation n. For example, the minimum and maximum values of the image are 2488 and 2656, respectively. If n is 2, the value that exceeds the second standard deviation will become 0 or 255, and the other values are stretched between 0-255.

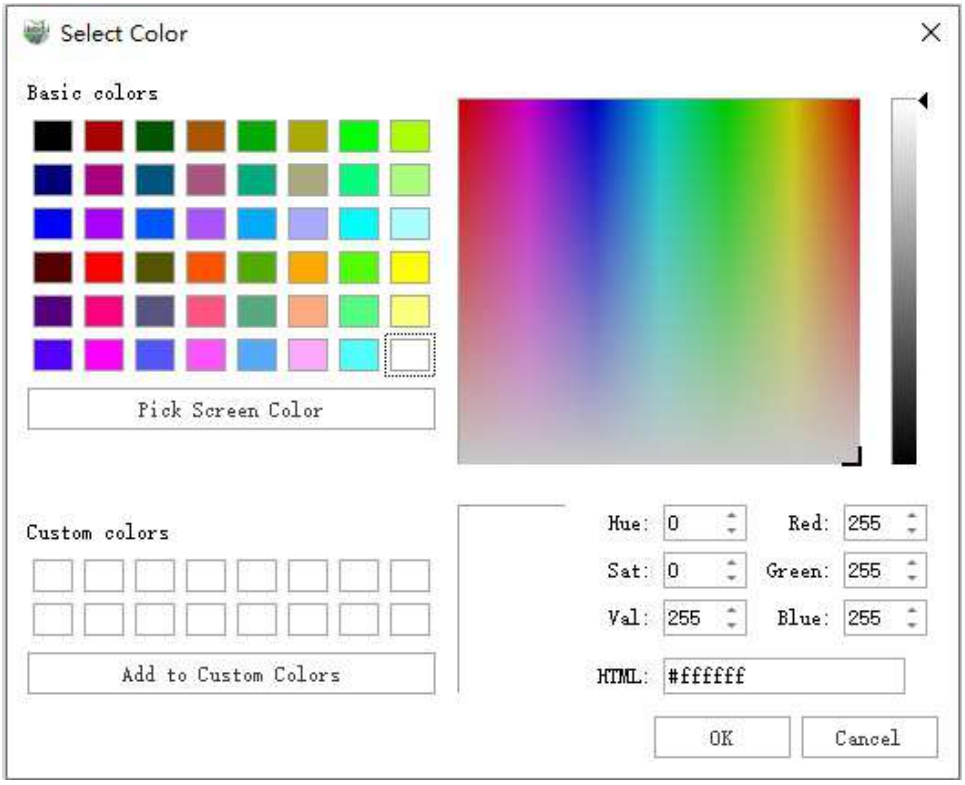
- **Save Curve**：

For specific display effects, see Display by intensity.

> The histogram displayed on the interface can be exported in pdf format. Click the "Save Curve" button to make the "ave Curve" dialog box pop up. As shown in the figure, select the width, height and resolution of the exported curve, select the output path, and click the OK button to save the curve.



- **Display by Classification**: For specific display effects, see Display by category.
- **Display by RGB**: For specific display effects, see Display by RGB.
- **Display by Return**: For the specific display effect, see Display by number of echoes.
- **Display by Time**: For specific display effects, see Display by GPS time.
- **Display by Selected Color**: Display by selected color allows you to select a color in the color dialog box, and the selected point cloud data is displayed in a uniform color, as shown in the selection color dialog box. For specific display effects, see Display by selected color.



- **Display by Blend**: For specific display effects, please refer to Mixed Display.
- **Display by Mix**: For specific display effects, please refer to Display by Combination.
- **Display by User data**: For specific display effects, see Display by User data.
- **Display by Source ID**: For specific display effects, please refer to Display by Point Source ID.

3.**Zoom to Layer**: Calculate the bounding box of the current point cloud data, and display all windows that open the data globally in this bounding box range.

4.**Point size**: The user sets the size of the point symbol displayed by the point cloud.

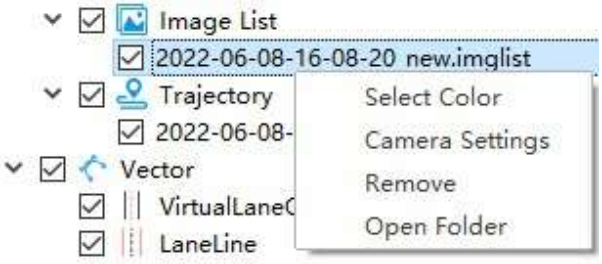
5.**Export**: Export a single point cloud data as LAS (*.las, *.laz) format.

6.**Replace data**: Replace the current point cloud data.

7.**Open Folder**: Open the folder where the current point cloud data is located.

8.**Remove**: Move the current point cloud data.

Image context menu

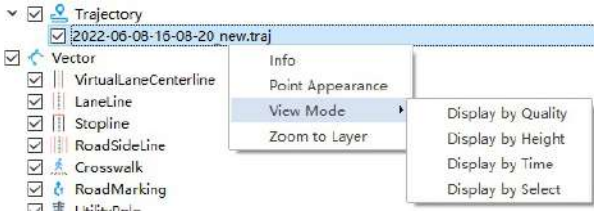


1.**Select Color**: Set the display color of the frame.

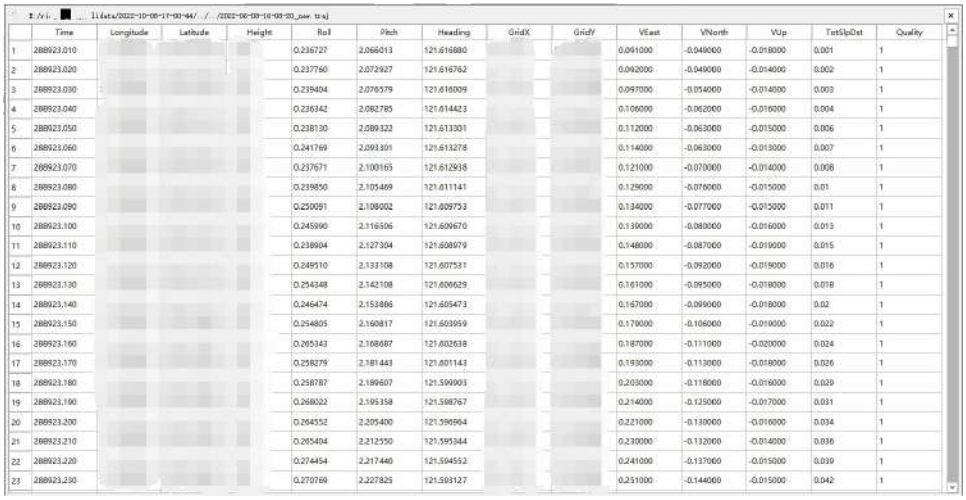
2.**Remove**: Removes the current image data.

3.**Open Folder**: Open the save path of the current image data.

Trajectory context menu



1.**Info**: View the basic information of the point cloud, including the path where the data is located, coordinate information, the minimum and maximum values of the X, Y, and Z coordinates, the average and standard deviation of Z, the minimum and maximum GPS time, the minimum, maximum, average and standard deviation of intensity, point cloud enclosures, total points, point cloud category statistics, and echo frequency statistics. Click the "Export" button to export the basic information of point cloud as a txt file.



	Time	Longitude	Latitude	Height	Roll	Pitch	Heading	GridX	GridY	VEast	VNorth	VUp	TotSlpDst	Quality
1	288923.010				0.236727	2.066013	121.616880			0.091000	-0.046000	-0.018000	0.001	1
2	288923.020				0.237760	2.072927	121.616762			0.092000	-0.049000	-0.014000	0.002	1
3	288923.030				0.239404	2.076379	121.616009			0.097000	-0.054000	-0.014000	0.003	1
4	288923.040				0.236342	2.082785	121.614423			0.106000	-0.062000	-0.016000	0.004	1
5	288923.050				0.238130	2.089322	121.613301			0.112000	-0.063000	-0.015000	0.006	1
6	288923.060				0.241769	2.093301	121.613278			0.114000	-0.063000	-0.013000	0.007	1
7	288923.070				0.237671	2.100165	121.612938			0.121000	-0.070000	-0.014000	0.008	1
8	288923.080				0.239850	2.105469	121.611141			0.129000	-0.076000	-0.015000	0.01	1
9	288923.090				0.250091	2.109002	121.609753			0.134000	-0.077000	-0.015000	0.011	1
10	288923.100				0.245990	2.116506	121.609670			0.139000	-0.080000	-0.016000	0.013	1
11	288923.110				0.238904	2.127304	121.608979			0.148000	-0.087000	-0.019000	0.015	1
12	288923.120				0.249510	2.133108	121.607581			0.157000	-0.092000	-0.019000	0.016	1
13	288923.130				0.254348	2.142108	121.606629			0.161000	-0.095000	-0.018000	0.018	1
14	288923.140				0.246474	2.153886	121.605473			0.167000	-0.099000	-0.018000	0.02	1
15	288923.150				0.254805	2.160817	121.603959			0.179000	-0.106000	-0.019000	0.022	1
16	288923.160				0.265343	2.168687	121.602638			0.187000	-0.111000	-0.020000	0.024	1
17	288923.170				0.258279	2.181443	121.601143			0.193000	-0.113000	-0.018000	0.026	1
18	288923.180				0.258787	2.189607	121.599903			0.203000	-0.118000	-0.016000	0.029	1
19	288923.190				0.268022	2.195358	121.598767			0.214000	-0.125000	-0.017000	0.031	1
20	288923.200				0.264552	2.205400	121.596964			0.221000	-0.130000	-0.016000	0.034	1
21	288923.210				0.265404	2.212550	121.595344			0.230000	-0.132000	-0.014000	0.036	1
22	288923.220				0.274454	2.217440	121.594552			0.241000	-0.137000	-0.015000	0.039	1
23	288923.230				0.270769	2.227825	121.593127			0.251000	-0.144000	-0.015000	0.042	1

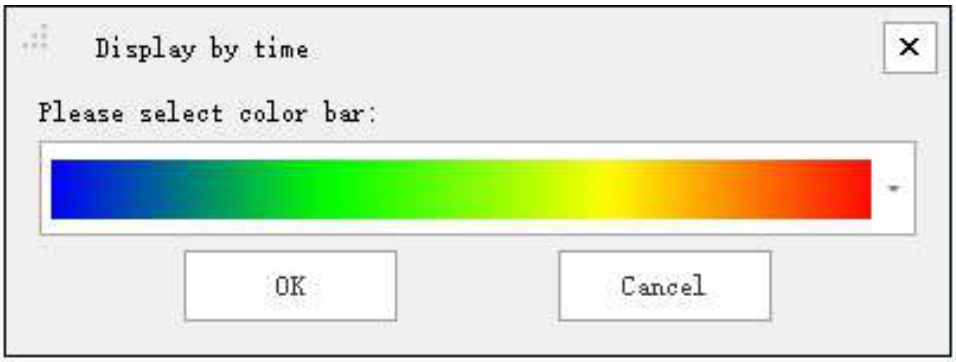
2.**Point Appearance**: Set the size of the display mode of a single track：

3.**View Mode**: Set the display mode of the track, including the following types：

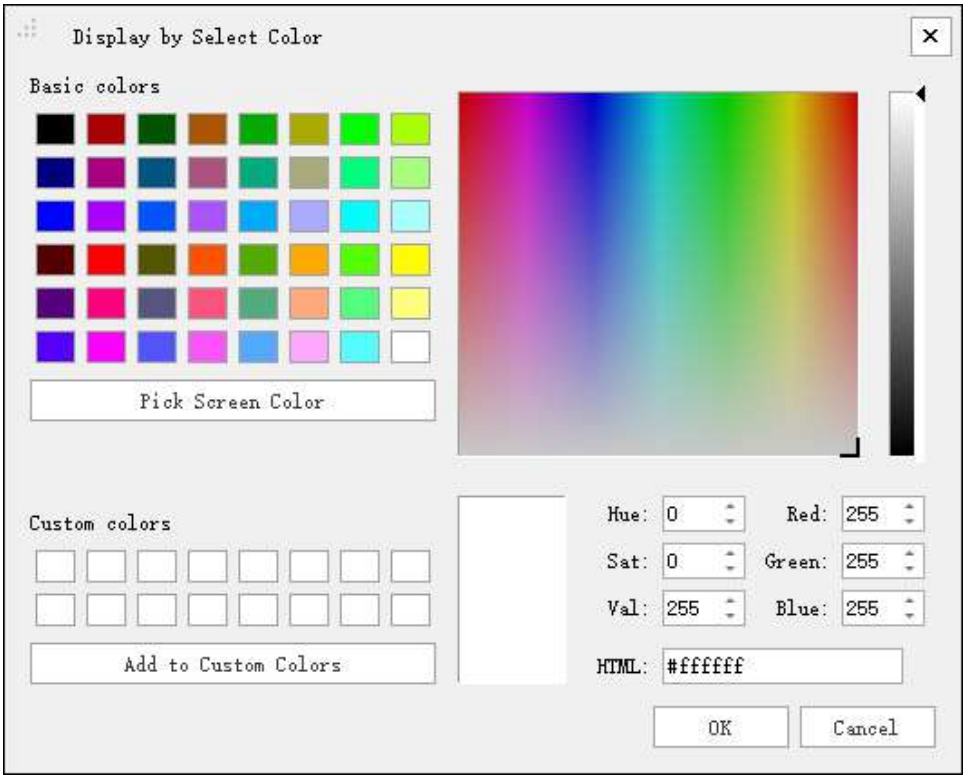
- **Display by Quality**: The interface as shown in the figure pops up, and the track can be displayed by randomly selecting the color and selecting the color bar.

- **Display by Height**: The interface as shown in the figure pops up, and the track can be displayed by selecting different color bars.

- **Display by Time**: Can be used for the display of tracks, mapping the user data attributes of the track data to different color values, and more intuitively distinguishing the track data of different user data.



- **Display by Select Color**: Each track data is displayed in the specified color.

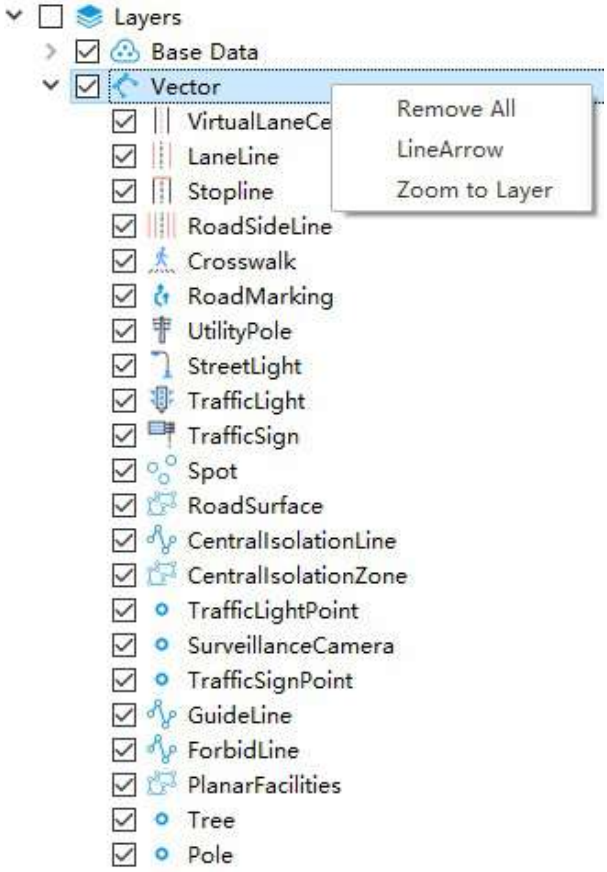


4.**Zoom to Layer**： Centered by current track range.

> The right click of the basic data root node, as well as the point cloud, image list, and track root node under it, also includes the zoom to layer menu, which has similar functions, and will not be repeated here.

Vector data layer

Function Description: The vector data layer is shown in the figure below, including the point, line, and polygon layers. Right-click on the vector layer tree to remove all and zoom to layers. The right-click menu of the three layers is the same.

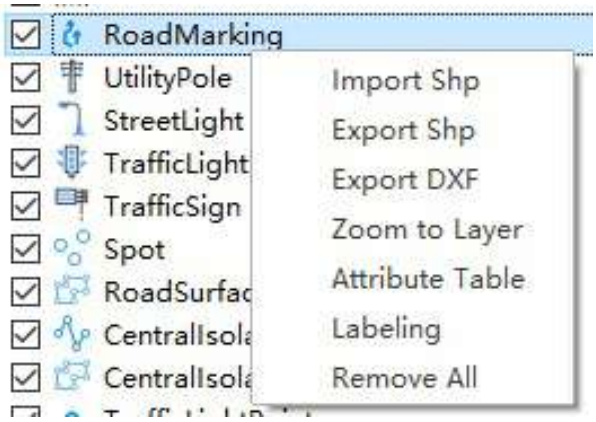


Description

- **Remove All**: Delete all vector data.
- **Zoom to Layer**: Display all vectors in the middle of the window.

Layer right-click operation

Take **polygon layer** as an example: the right-click menu is shown in the figure below, containing：

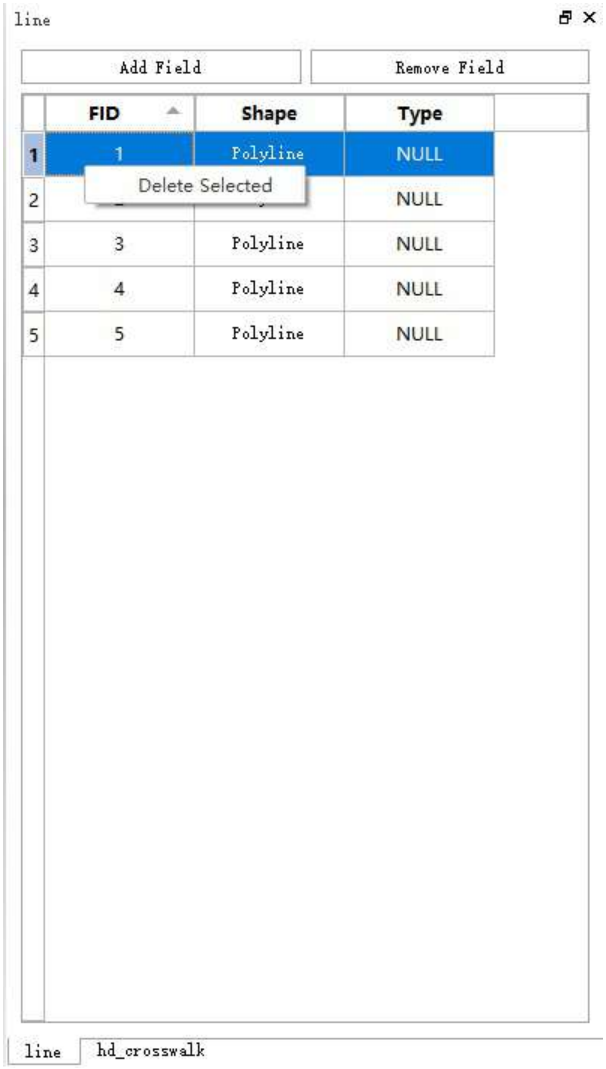


Surface layer

- **Import Shp**: Import external polygon layer SHP files separately into one of the project's polygon layers, and give a prompt if the geometry type does not match.

  > Note: Only geometry is imported, not attributes.

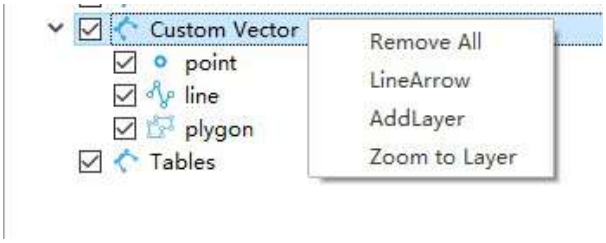
- **Export Shp**: Export the map feature layer to an Shp file separately.
- **Export Dxf**: Export this vector layer to a Dxf file separately.
- **Zoom to Layer**: Displays vector data for the layer that was selected first to the middle of the window.
- **Attribute Table**: View the properties of the vectors in each point layer, as shown in the image below, and you can also edit the attributes.
  - **Add Field** : Click the Add Field button to add the desired properties to the currently selected vector layer.
    - **Delete Field** : Click the Delete Field button to delete a column attribute field in the currently selected vector layer.
    - **Delete Selection**: Select the vector you want to delete in the attribute table, and then right-click on the left side of the attribute table box to make "Delete Selection" pop up to delete it. At the same time, hold down the Ctrl key to make multiple selections and delete them.



- **Labeling**: Activate the label function to display different attribute values for the layer.
- **Remove All**: Deletes all vector data in the currently selected layer.

Custom Vector

Function Description: The custom vector data layer is shown in the figure below, including the points, lines, polygons, and marker layers. Right-click on the vector layer tree to remove all, zoom to layers, and add layers. All four layers The right-click menu of the is the same, which will be described in detail below.

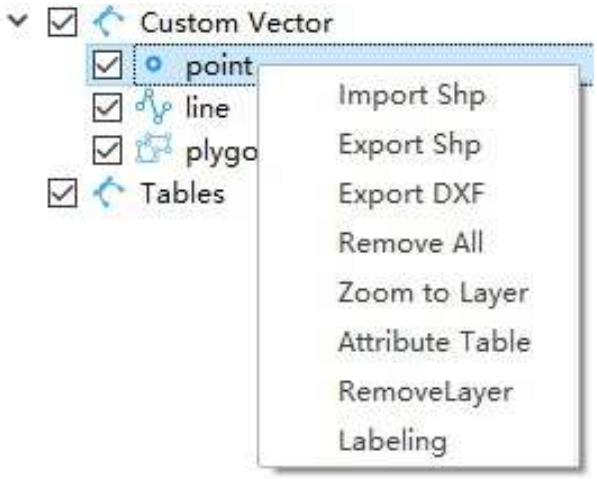


Description

- **Remove All**: Delete all vector data.
- **Add Layer**: Create a new custom vector layer.
- **Zoom to layer**: Display all vectors in the middle of the window.

Layer right-click operation

Take **Point layer** as an example: the right-click menu is shown in the figure below, containing：



- **Import Shp**: Import the external shp file into the currently selected layer, and give a prompt if the geometry type does not match.

  > Note: Only geometric information is imported, not attribute information is imported.

- **Export Shp**: Export the vector layer to a shp file separately.
- **Export Dxf**: Export the vector layer to a Dxf file separately.
- **Remove All**: Delete all vector data in the currently selected layer.
- **Zoom to Layer**: Displays the vector data of the currently selected layer in the middle of the window.
- **Attribute Table**: View the attributes of the vector in each point layer, as shown in the figure below, you can also edit the attributes.

- **Add Field**: Click the Add Field button to add the required attributes to the currently selected vector layer.
- **Delete Field**: Click the Delete field button to delete a column of attribute fields in the currently selected vector layer.
- **Delete selected content**: Select the vector that needs to be deleted in the attribute table, then right-click on the left side of the attribute table box to make “Delete Selected content” pop up to perform the delete operation. At the same time, hold down the Ctrl key to multi-select and delete.

- **Remove Layer**: Activate the remove layer function, and a pop-up box will pop up to choose whether to remove the file.
- **Labeling**: Activate the label function, the different attribute values of the layer have been displayed.

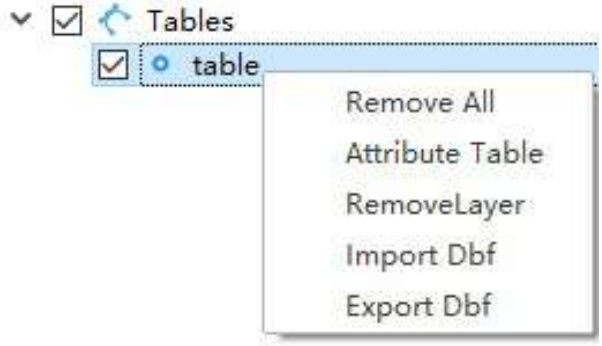
List

Function Description: Manage user-defined table layers (without geometry information).

Description

- **Create a new table**: Click Tables, right-click the pop-up box ((Add table)), enter the table name (for example, 11), press the OK button to complete the new creation;

Table right-click operation



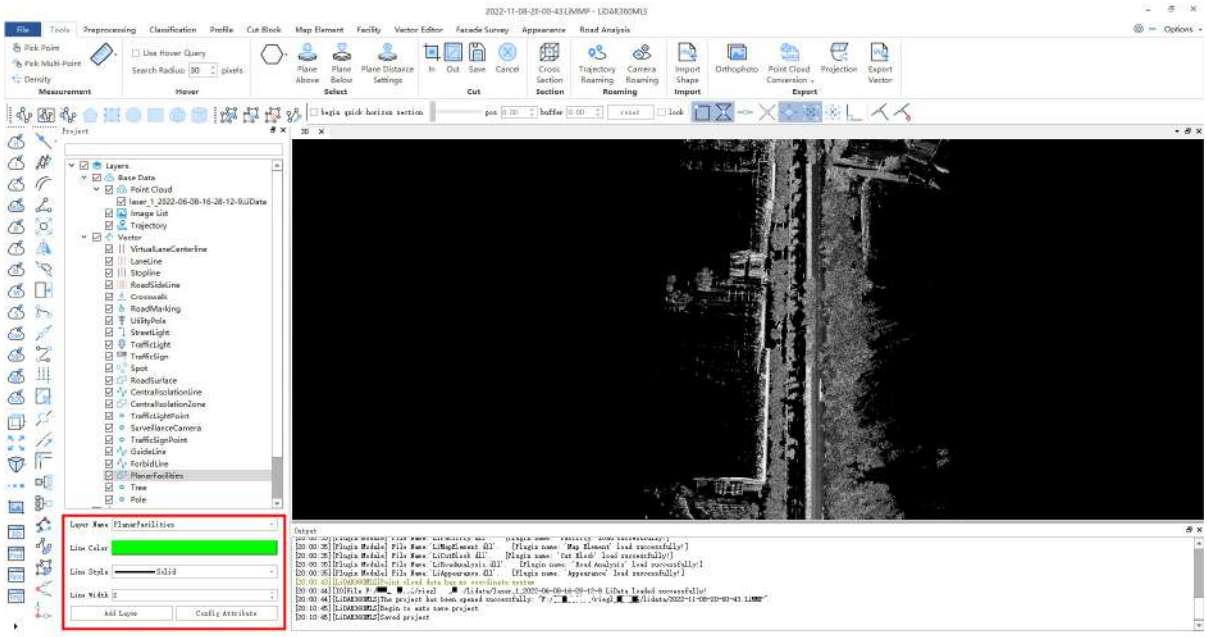
- **Remove All**: Select Remove All to clear all the contents in the table;
- **Attribute Table**: Select the attribute table and the attribute table interface will pop up. Users can edit the fields and content of the attribute table, add/delete fields or add a row of records.;
- **Remove Layer**: Select Remove layer to remove the current table from the directory tree;
- **Import Dbf**: A dialog box pops up to select the Dbf file to import it to the currently selected layer.
- **Export Dbf**: Export the current table into a Dbf file;

Layer settings

Layer explanation: Used to manage different types of vectorization results. According to the geometry shape, it is mainly divided into three categories: point layers, linear layers, and area layers. For example, layers such as lane centerline, lane line, and stop line belong to linear layers, traffic lights, surveillance camera points, etc. belong to point layers, and no-stop lines, pedestrian crossings, etc. belong to area layers.

> Note: The point vector results can only be saved in the point layer, the linear vector results and the polygon vector results are the same.

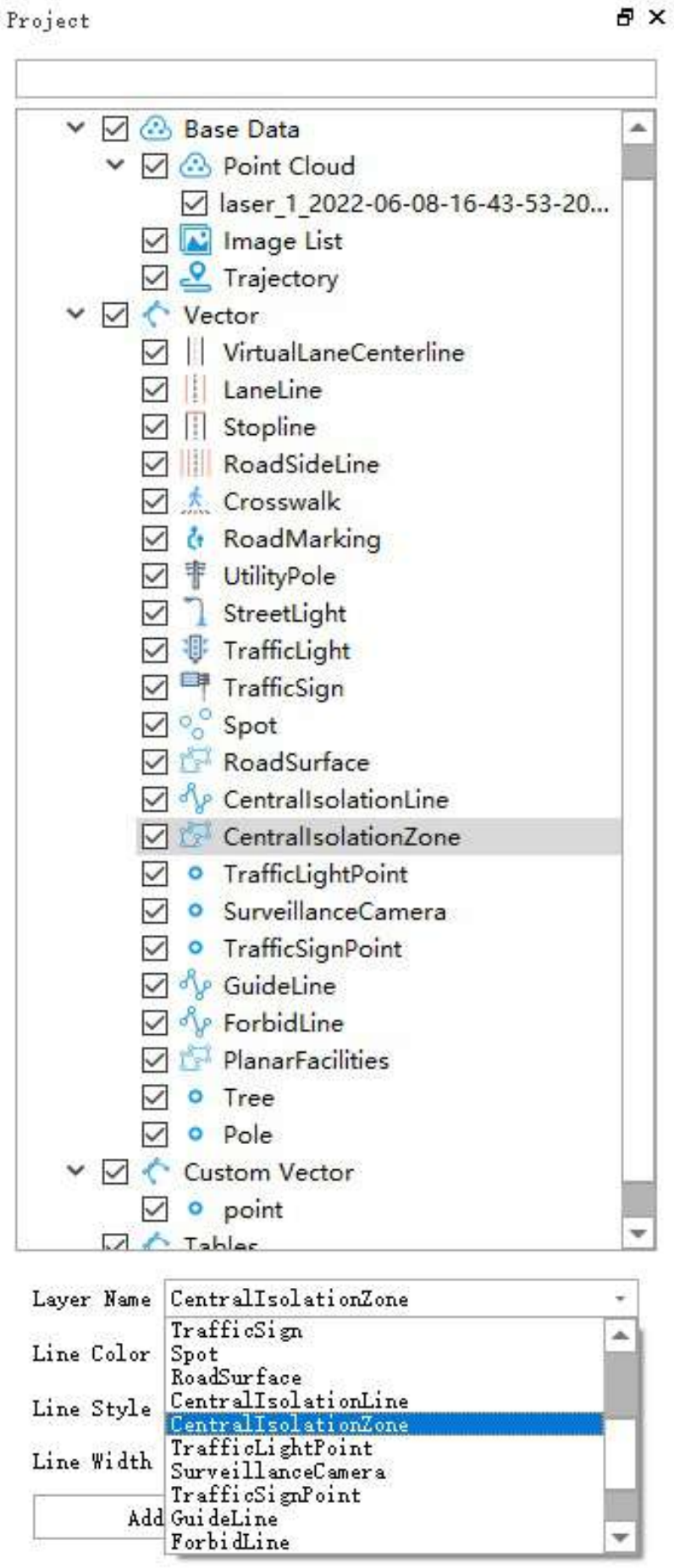
Function Description: The role of the layer management module is to select the currently activated layer, set the line type, line width, and color of the currently selected layer, and also add custom layers and configure the selected layer attribute table functions. You can also switch the current layer by clicking the vector layer tree in window management with the left mouse button.



Layer management interface

Description

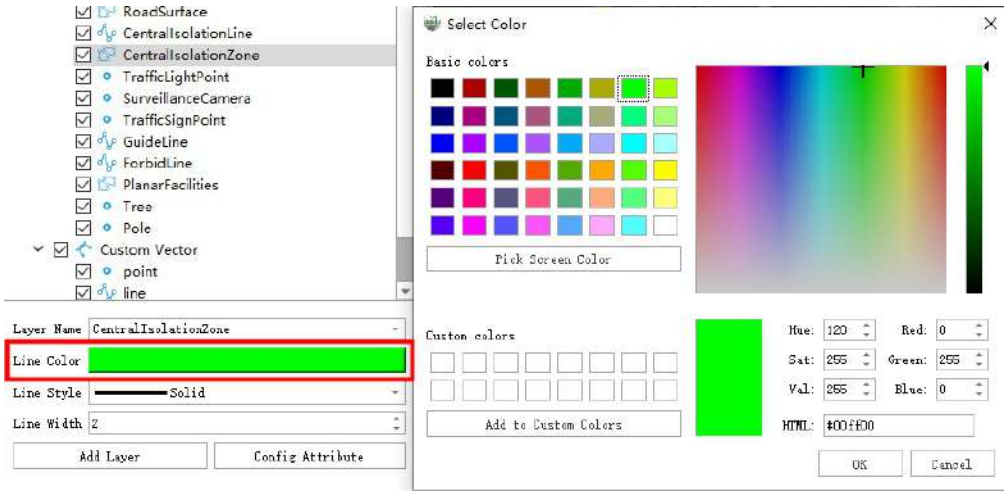
1.**Layer selection**: Click the drop-down arrow under the layer name in the layer manager to select the layer you want to switch, or click the layer on the node tree with the left mouse button to achieve the same effect.


Project
Base Data
Point Cloud
laser_1_2022-06-08-16-43-53-20...
Image List
Trajectory
Vector
VirtualLaneCenterline
LaneLine
Stopline
RoadSideLine
Crosswalk
RoadMarking
UtilityPole
StreetLight
TrafficLight
TrafficSign
Spot
RoadSurface
CentralIsolationLine
CentralIsolationZone
TrafficLightPoint
SurveillanceCamera
TrafficSignPoint
GuideLine
ForbidLine
PlanarFacilities
Tree
Pole
Custom Vector
point
Tables
Layer Name
CentralIsolationZone
Line Color
Line Style
Line Width
Add
TrafficSign
Spot
RoadSurface
CentralIsolationLine
CentralIsolationZone
TrafficLightPoint
SurveillanceCamera
TrafficSignPoint
GuideLine
ForbidLine

Layer switching interface

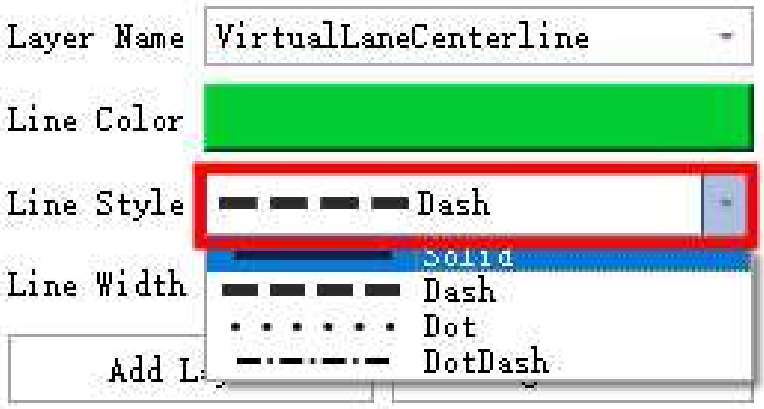
2.**Layer property settings**：

- **Vector color settings**: Click on the color of the midline in the layer manager to make the display palette pop up. Users can choose existing colors or customize the various colors they want.



Layer color selection setting interface

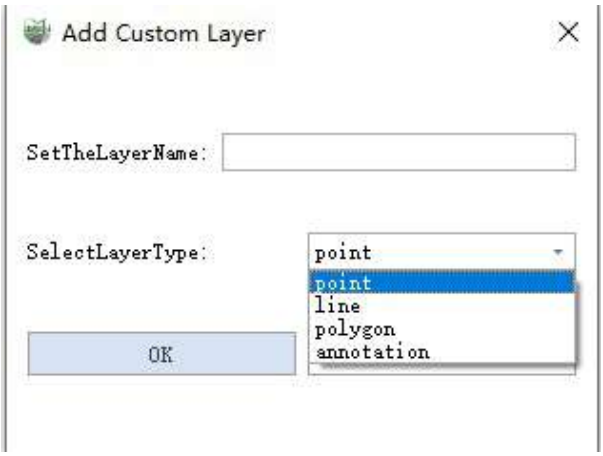
- **Vector line type setting**: Click the line type drop-down arrow to switch the desired line type and the width of the line.



Layer line type setting interface

- **Vector line width setting**: Click the adjust button on the right side of the line width to set the line width.

3.**Add layer**: 3.Click the Add Layer button to make the Add user-defined layer dialog box pop up.



Add layer interface

Parameter setting:

- **Set the layer name**: Customize the layer name, a combination of recommended characters and numbers, and no special characters.
- **Select the geometry type of the layer**: According to the geometric characteristics of the vector result (point vector, line vector, plane vector, annotation), set the geometric properties of the custom layer.
- **OK**: After clicking, create the corresponding layer based on the set information, and update it in real time on the **Custom layer node tree**.
- **Cancel**: Cancel the new layer.

Window management

Function overview

The window management tool controls the display of the project window.

Detailed

Display a 3D window.

Display the profile window.

Display the panoramic window.

Display a planar images window.

Display the log window.

Display the layer (project tree) window.

Display mode

Function Description: The display mode provides a single attribute display and a multi-attribute combination display for the visualization of point cloud data, and the best display method can be selected for different analysis functions.

Single attribute display

1. Display by Height: For specific display effects, please refer to Display by Elevation.

2. Display by Intensity: For specific display effects, please refer to Display by Intensity.

3. Display by Blend: For specific display effects, please refer to Blend Display.

4. Display by Classification: For specific display effects, please refer to Display by Classification.

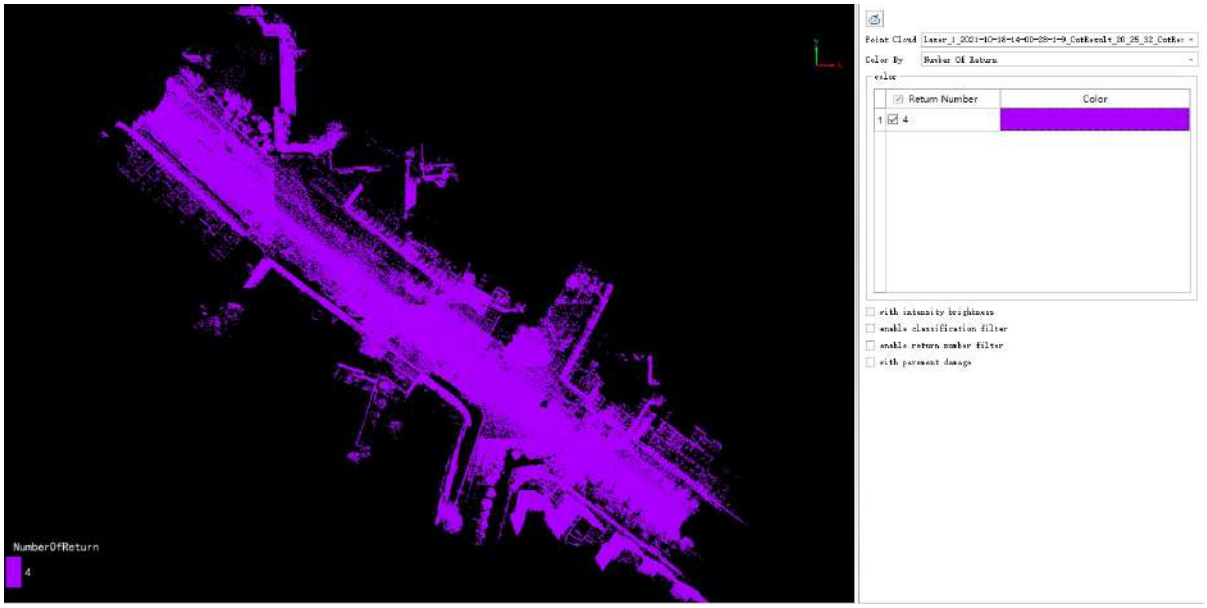
5. Display by RGB: For specific display effects, please refer to Display by RGB.

6.Display by Return Num: For specific display effects, please refer to Display by Return Num.

7.Display by Number of Return: It can be used for the display of point cloud data, and the last attribute of the number of echoes of the point cloud data is mapped to the point cloud data.

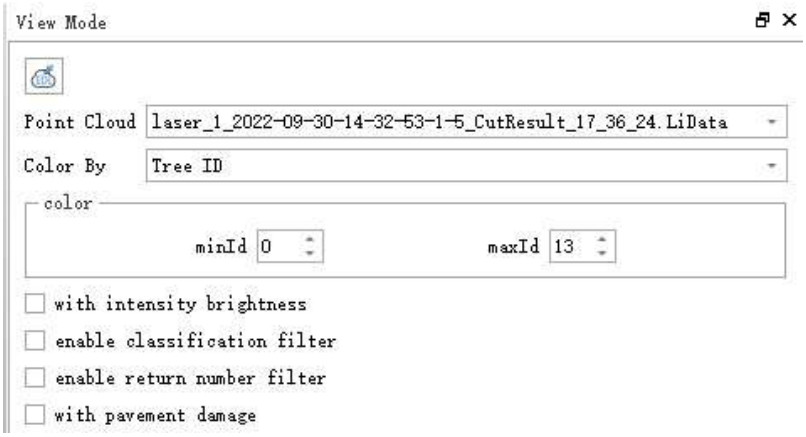
7.1 Click the number of echoes in the drop-down box, and a dialog box will pop up to display the number of echoes.

7.2 The display color can be modified, and the display effect is better with EDL. The effect is shown in the figure.



8. Display by GPS Time: For specific display effects, please refer to Display by Time.

9.Display by Tree ID (Tree ID): For specific display effects, please refer to Display by Tree ID. The drop-down selection tree ID display box is shown in the figure below. You can choose to display the minimum and or maximum ID of the tree ID.



10.Display by Flight Line: If the point cloud data records the route edge information, it can be displayed and rendered according to this attribute.

11.Display by Point Source ID: If the point cloud data records data source information, it can be displayed and rendered according to this attribute.

12.Display by Scan Angle Rank: 12. If the point cloud data records the scan angle attribute, it can be displayed and rendered according to this attribute

13.Display by Scan Direction: If the point cloud data records the scan direction information, it can be displayed and rendered according to this attribute.

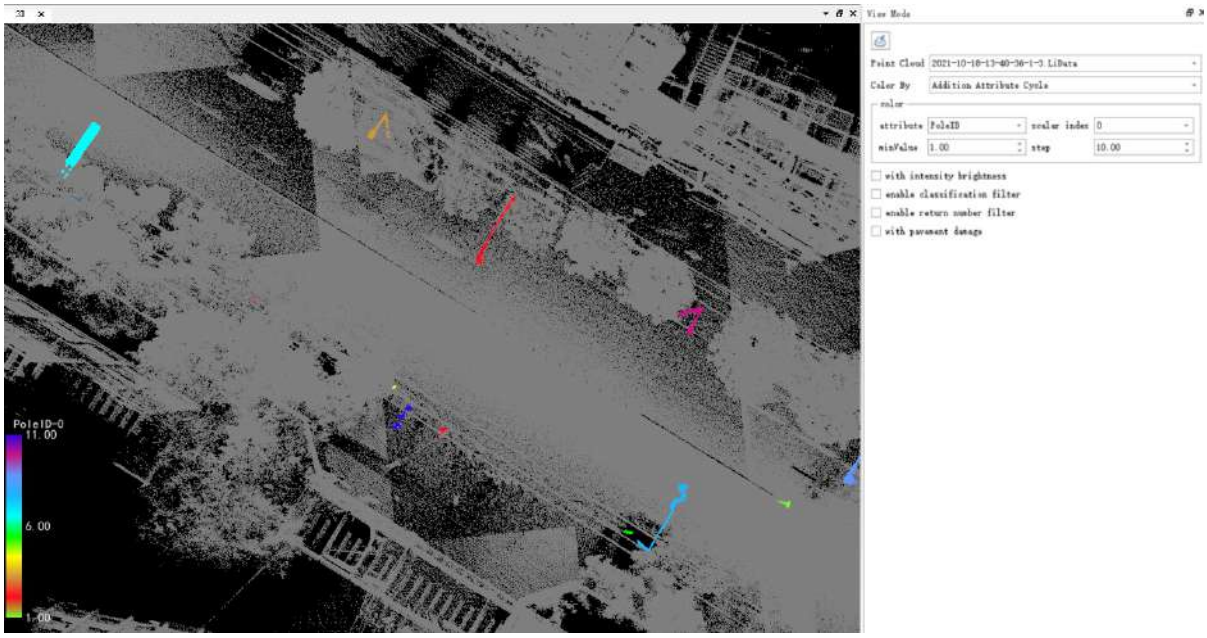
14.Display by Scan Channel: If the point cloud data records the scan channel information, it can be displayed and rendered according to this attribute.

15.Display by Near Infrared: If the point cloud data records infrared information, it can be displayed and rendered according to this attribute.

16.Display by User Data: For specific display effects, please refer to Display by User Data.

17. Display by Selected: For specific display effects, please refer to Display by selected color.

18.Addition Attribute Cycle: 18. If there are additional attributes in the point cloud data, as shown in the figure below, there is a single feature ID (PoleID), it can be displayed in a color cycle according to a certain additional attribute value, and the display effect is as shown in the TreeID attribute display effect.



19.Display by Additional Attribute: If there are additional attributes in the point cloud data, a color band can be set to render incrementally according to a certain additional attribute value.

20.Display additional attributes by RGB: If there are additional attributes in the point cloud data, it can be mapped to RGB information for rendering according to a certain additional attribute value.

Multi-attribute combination display

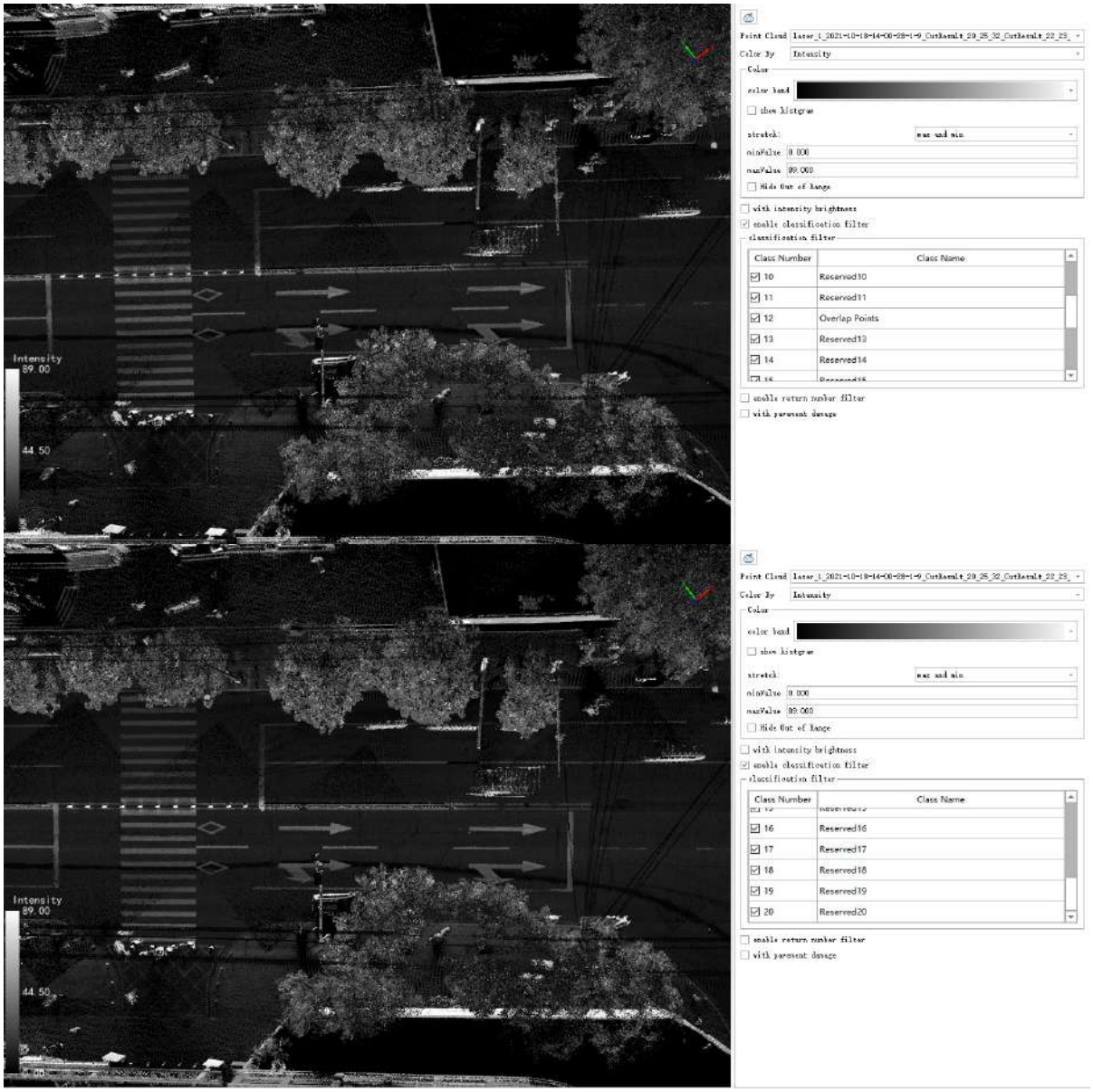
In the display mode window, you can set the categories that need to be displayed in the color display, and in the check box at the bottom of the window, check the categories that need to be superimposed.

Description

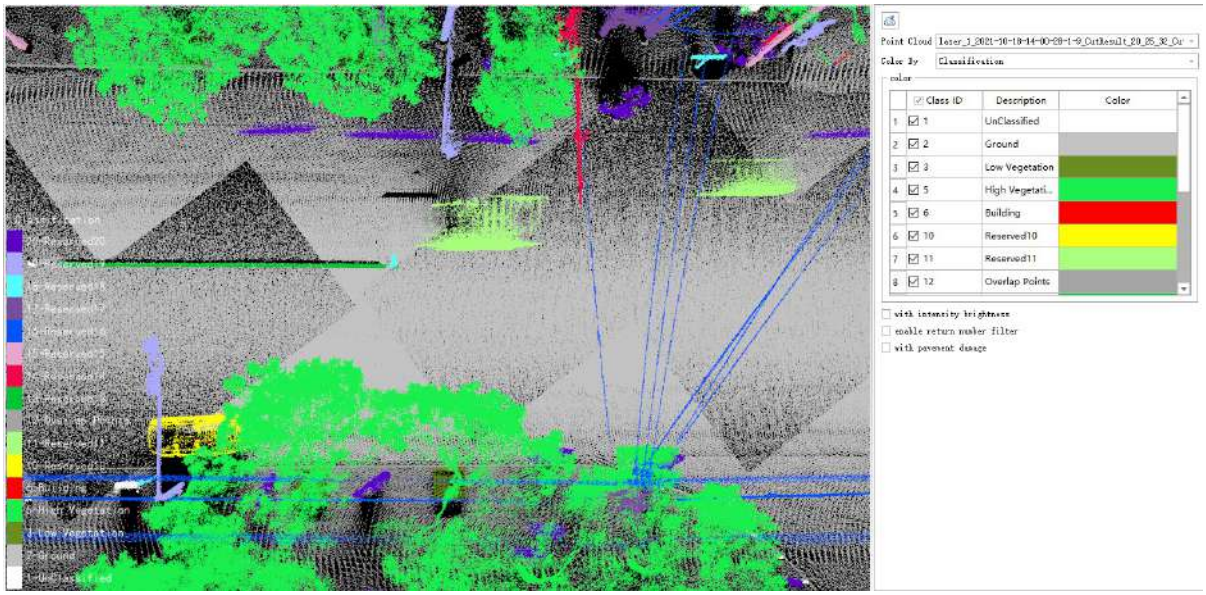
- **Turn on intensity filtering**: After checking, the intensity filtering attributes of the point cloud data are mapped to different color values to more intuitively distinguish between different intensity filtering point cloud data.
- **Enable category filtering**: After checking, filter the categories of point cloud data.
- **Enable echo filtering**: After checking, it can be used for the display of point cloud data, mapping the echo frequency attributes of point cloud data to different color values, and more intuitively distinguishing point cloud data with different echo frequency.
- **Turn on road damage filtering**: After checking, it can be used for the display of point cloud data to map the road damage attributes of point cloud data to different color values to more intuitively distinguish road damage.

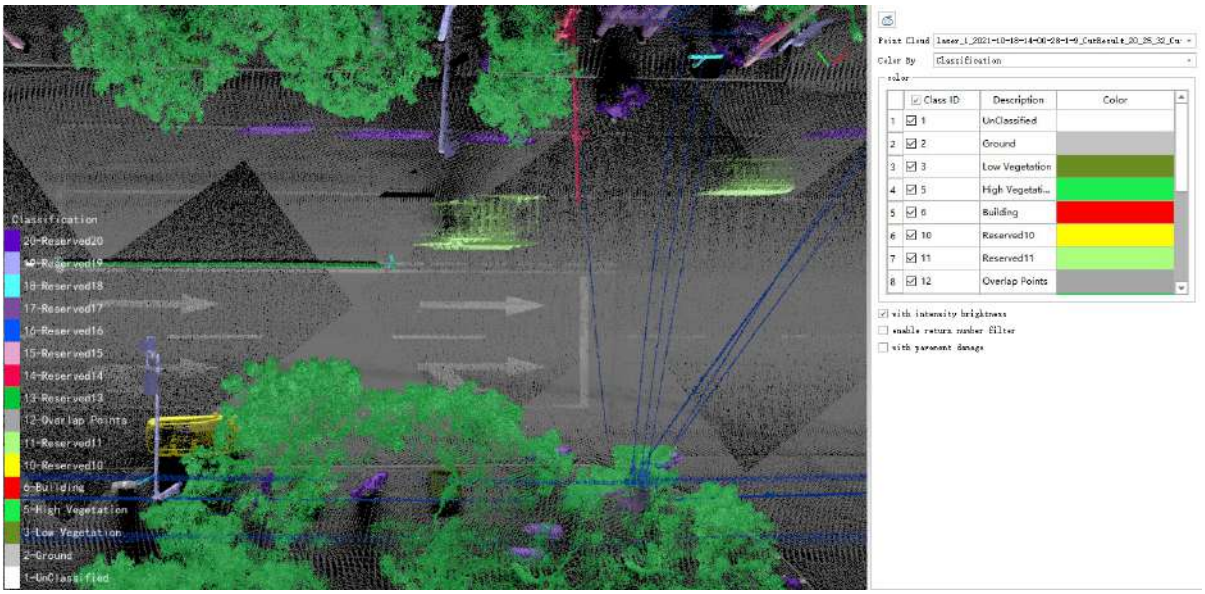
The commonly used combination display modes are：

(1) Intensity superposition Classification filtering：

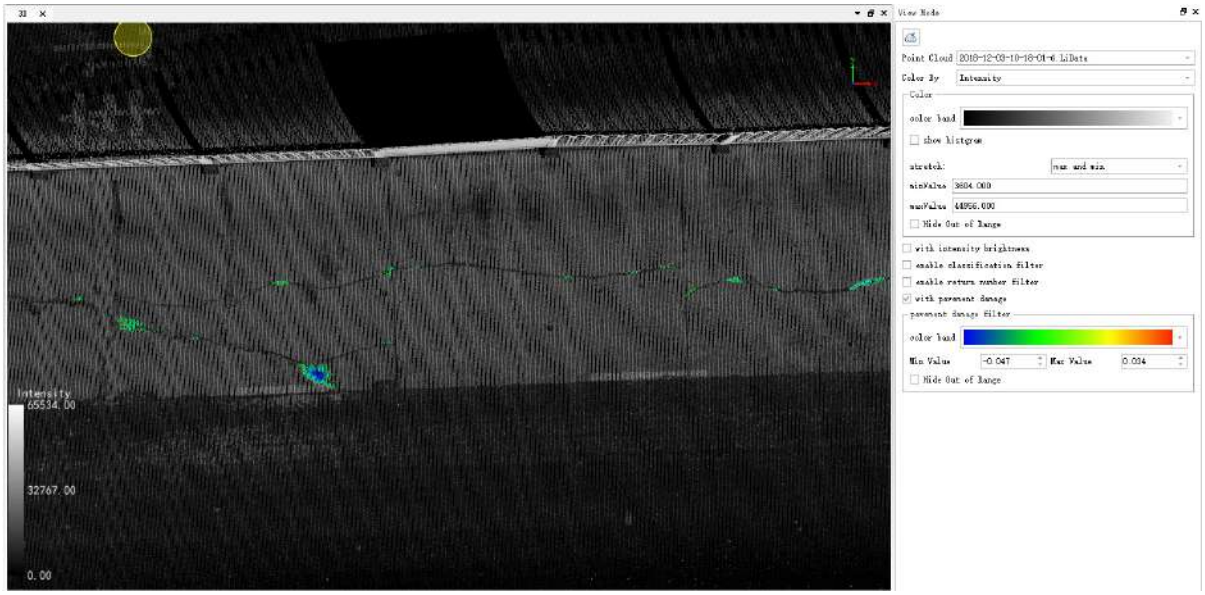


(2) Classification with intensity brightness：





(3) Intensity with pavement damage：

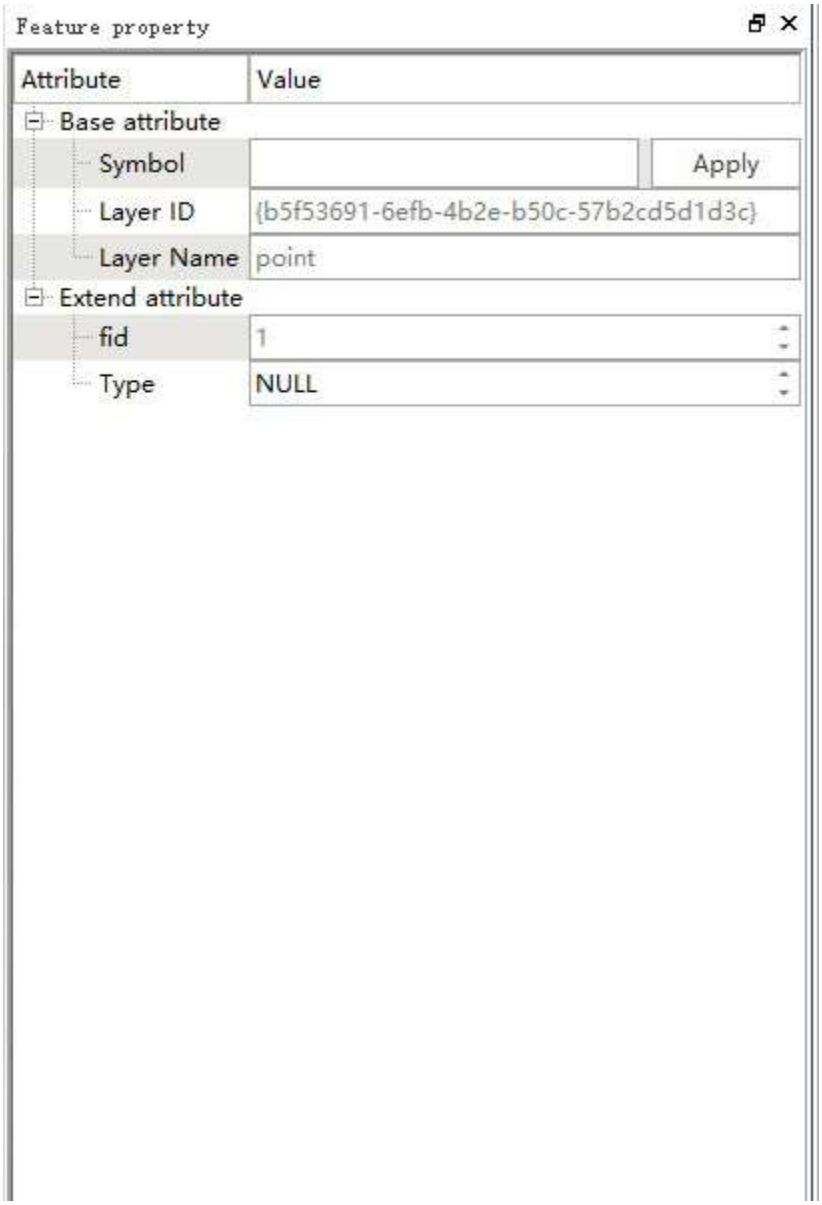


Feature Attribute

Function Description: It is used to display and edit the currently selected vector features. When the layer attribute is a point layer, the added symbol can be displayed.

Description

1.**Feature Attribute：** Select a vector to display vector-related information.



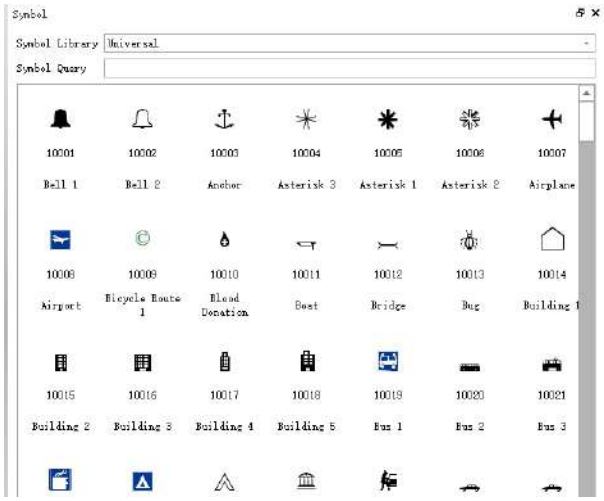
Feature attributes

Basic attributes

- **Symbol：** You can add the desired symbols on the point layer. Displays the symbol code of the point feature, or empty if not set.
- **Layer ID：** The ID of the layer cannot be modified.
- **Layer Name：** The name of the corresponding vector layer.

- **Extended attributes**：Other field descriptions of the vector, and the fields added by the layer are displayed accordingly.
- **fid**：The ID generated by the layer cannot be modified.

2.When the layer geometry type is point layer: Calibrate the symbol on the point vector.
Select the point layer and click the symbol long blank box for the basic attribute in Feature Attributes to make the symbol table pop up:

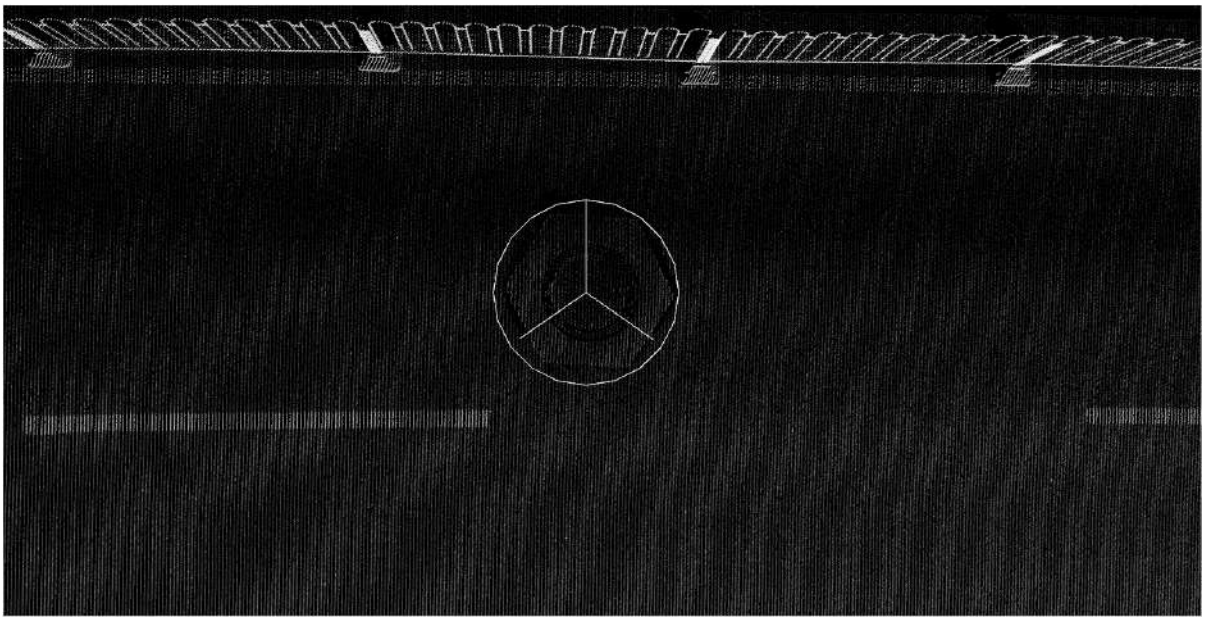


Symbol table

Parameter setting

- **Symbol Library**：It is divided into two types of symbols, standard and general, and you can choose according to your needs.
- **Symbol Query**：Symbols can be accurately found based on the name and code name.

After selecting the desired symbol with the mouse, click "Apply". To see the effect of the symbol, click Display symbol in the Appearance function.


Symbol effect

Modify attribute values

You can directly click the edit control in the second column of the extended attribute to modify it.

Appendix

As shown in the list below, the file page contains the following functions:

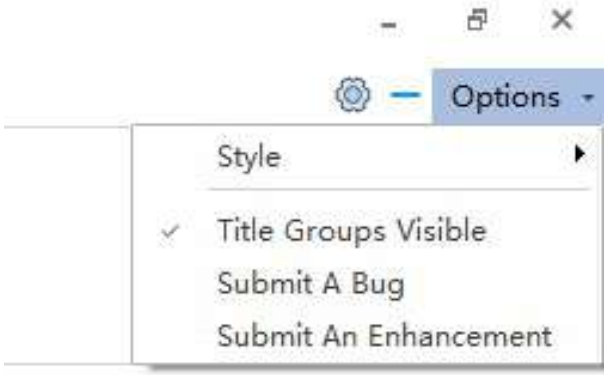
- Options
- Display Options
- Adjusting to High-Performance Display Mode
- FAQ
- Version Update Subscription

Options

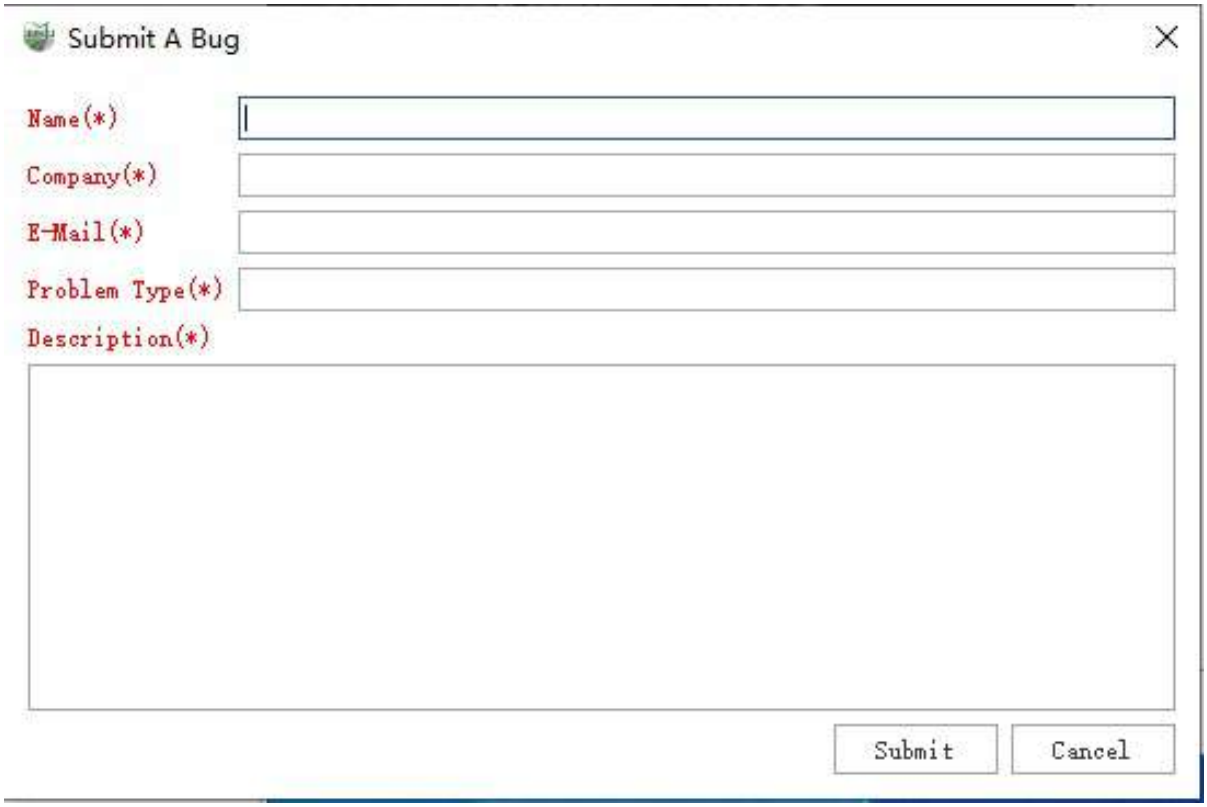
Function Description： The Options function in the main interface of the software provides auxiliary operations such as style setting, group name display and hidden, bug and demand feedback.

Steps

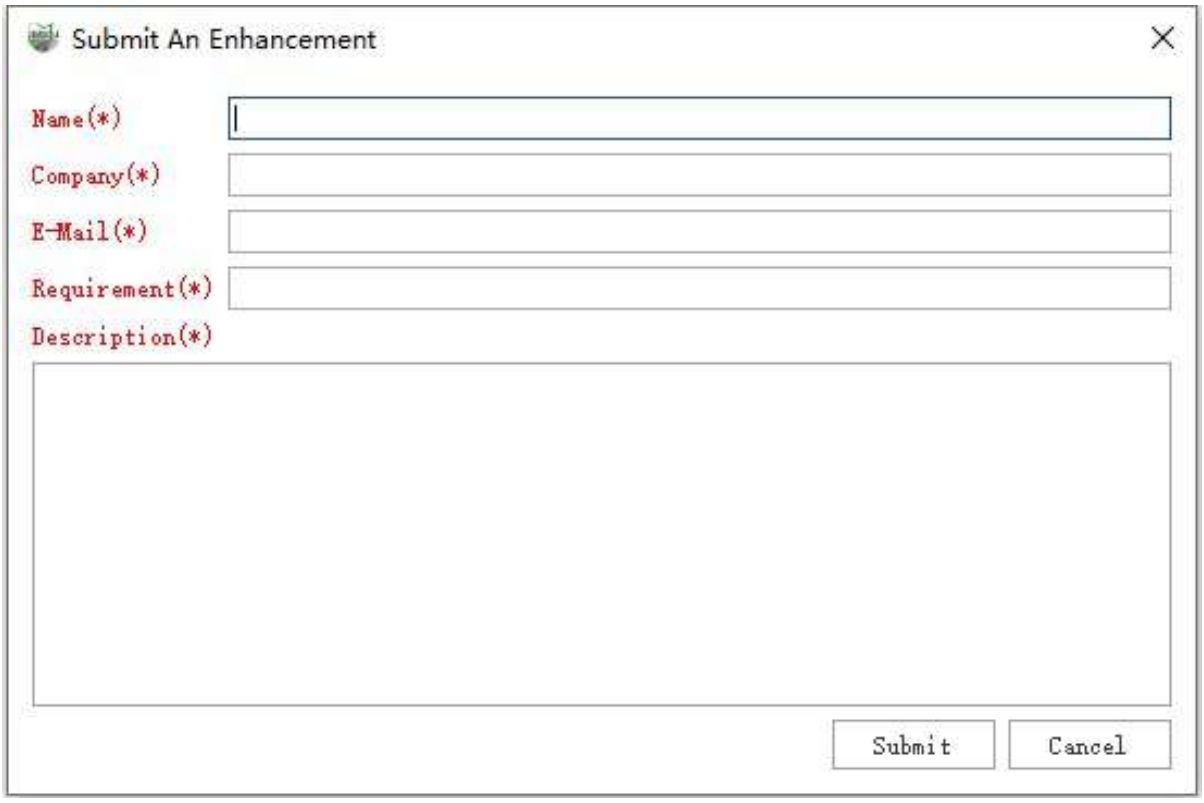
1.In the upper right corner of the main interface, click **Options**, where you can modify the interface style and show or hide the menu bar group names.



2.Submit a bug: Click **Submit A Bug**, in the pop-up window, please describe your basic information and the steps of the bug generation in detail, so that we can reproduce the problem and contact you.



3.Submit an enhancement: Click **Submit An Enhancement**, in the pop-up window, please describe your basic information and software requirements for business needs in detail, so that we can contact you easily.

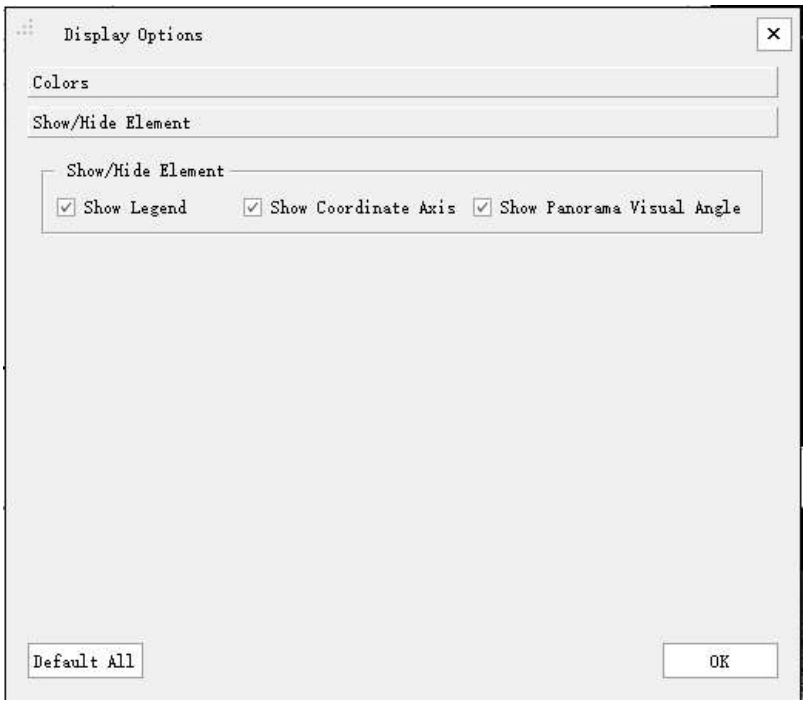


Display Options

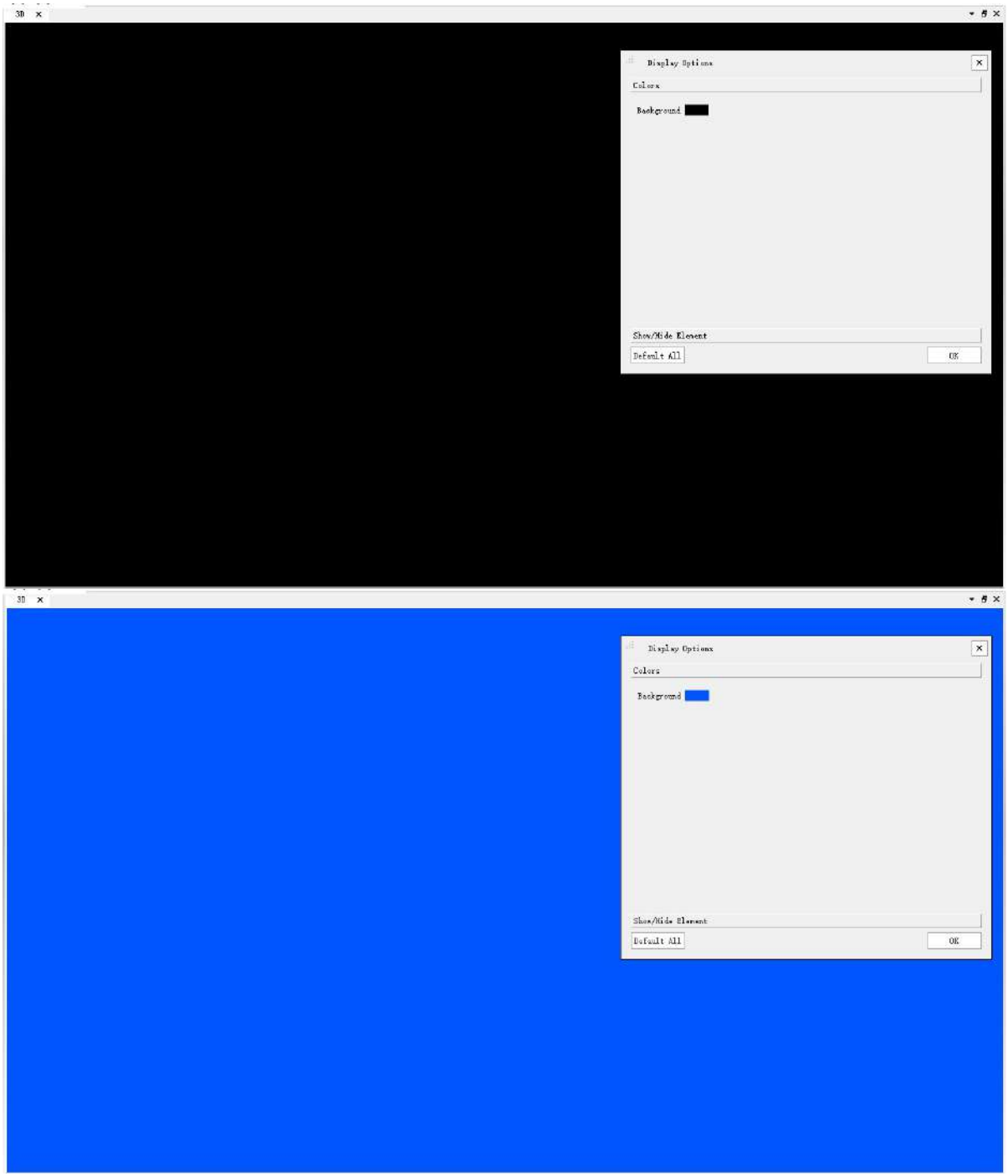
Function Description: The Display Options function in the main interface of the software provides settings for options such as background color, element display and hide, camera height and orientation.

Steps

1.Click the display options setting button in the upper right corner of the software to pop up display options settings dialog.



2.Colors: Determines the color settings on the 3D window background color.



3.Show/Hide Elements: Controls whether to show/hide the legend, coordinate axis and panoramic view of the 3D window.

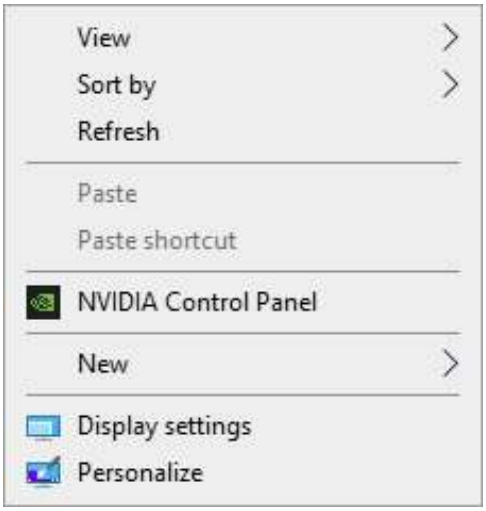

3D
Display Options
Colors
Show/Hide Element
Show/Hide Element
Show Legend
Show Coordinate Axis
Show Panorama Visual Angle
Default All
OK


3D
Display Options
Colors
Show/Hide Element
Show/Hide Element
Show Legend
Show Coordinate Axis
Show Panorama Visual Angle
Default All
OK

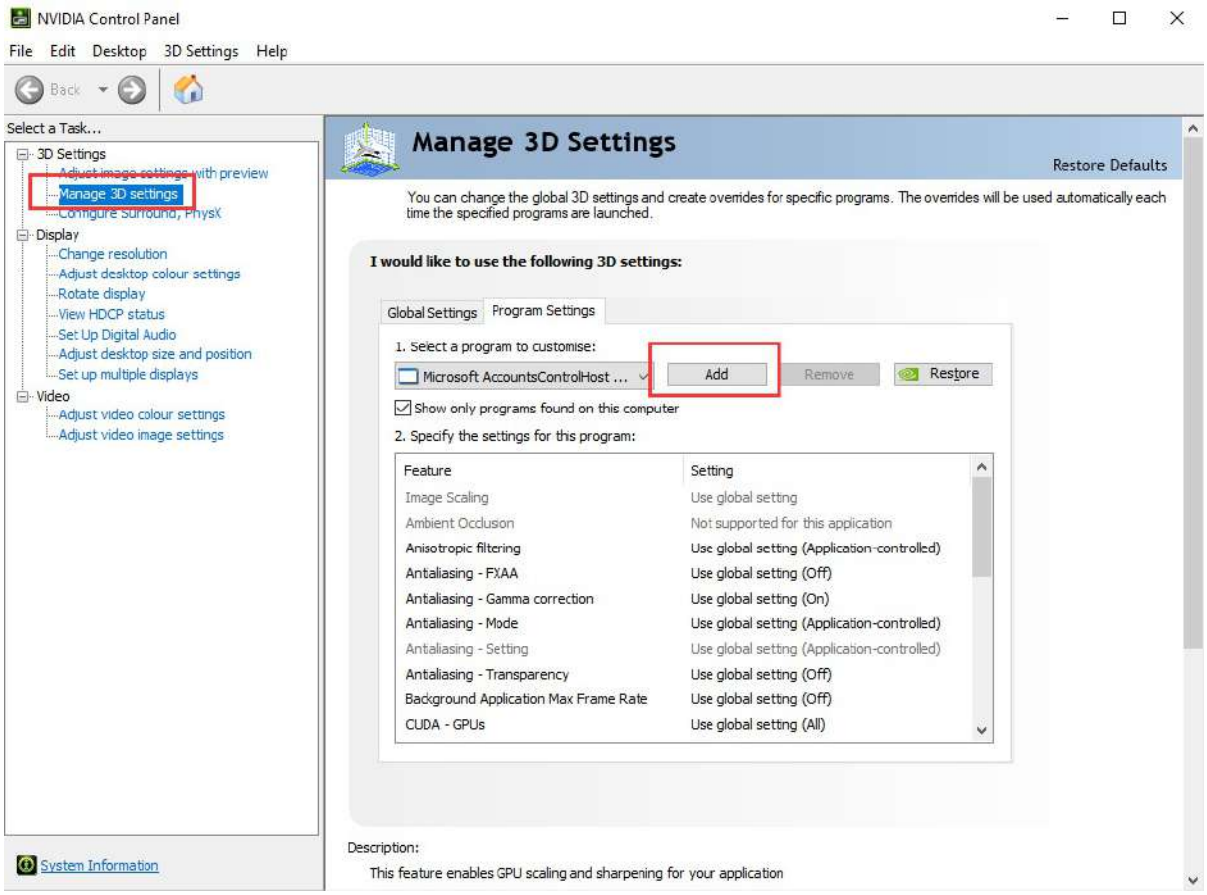
Adjusting to High-Performance Display Mode

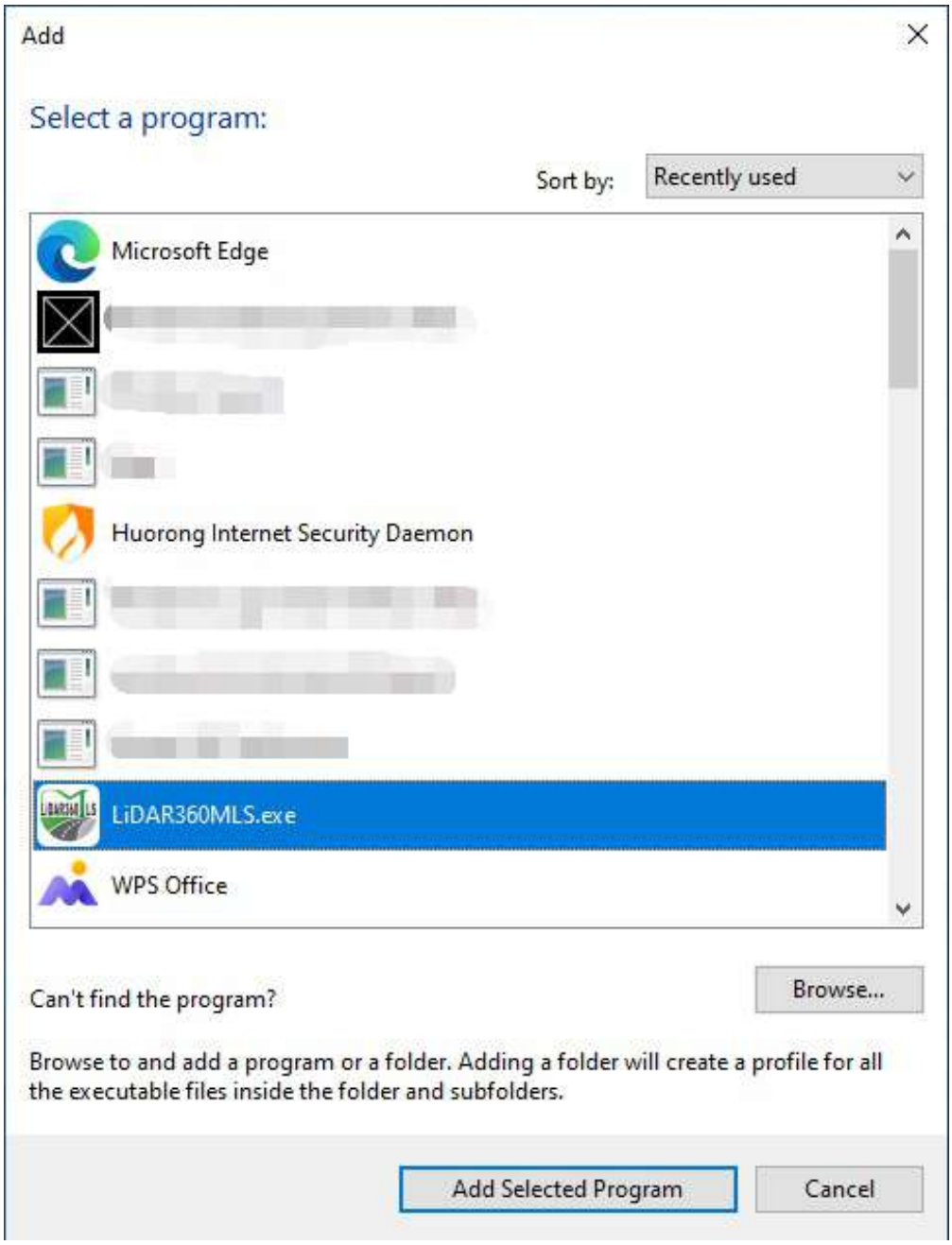
Follow the steps below to optimize the graphics mode of LiDAR360MLS.exe (for NVIDIA graphics cards).

1.Right click on the desktop and select NVIDIA Control Panel.



2.Select Manage 3D settings -> Program Settings -> Add **LiDAR360MLS.exe** to the list of high-performance graphics modes, click **Apply**.




Add
Select a program:
Sort by:
Recently used
Microsoft Edge
Huorong Internet Security Daemon
LiDAR360MLS.exe
WPS Office
Can't find the program?
Browse...
Browse to and add a program or a folder. Adding a folder will create a profile for all the executable files inside the folder and subfolders.
Add Selected Program
Cancel

FAQ

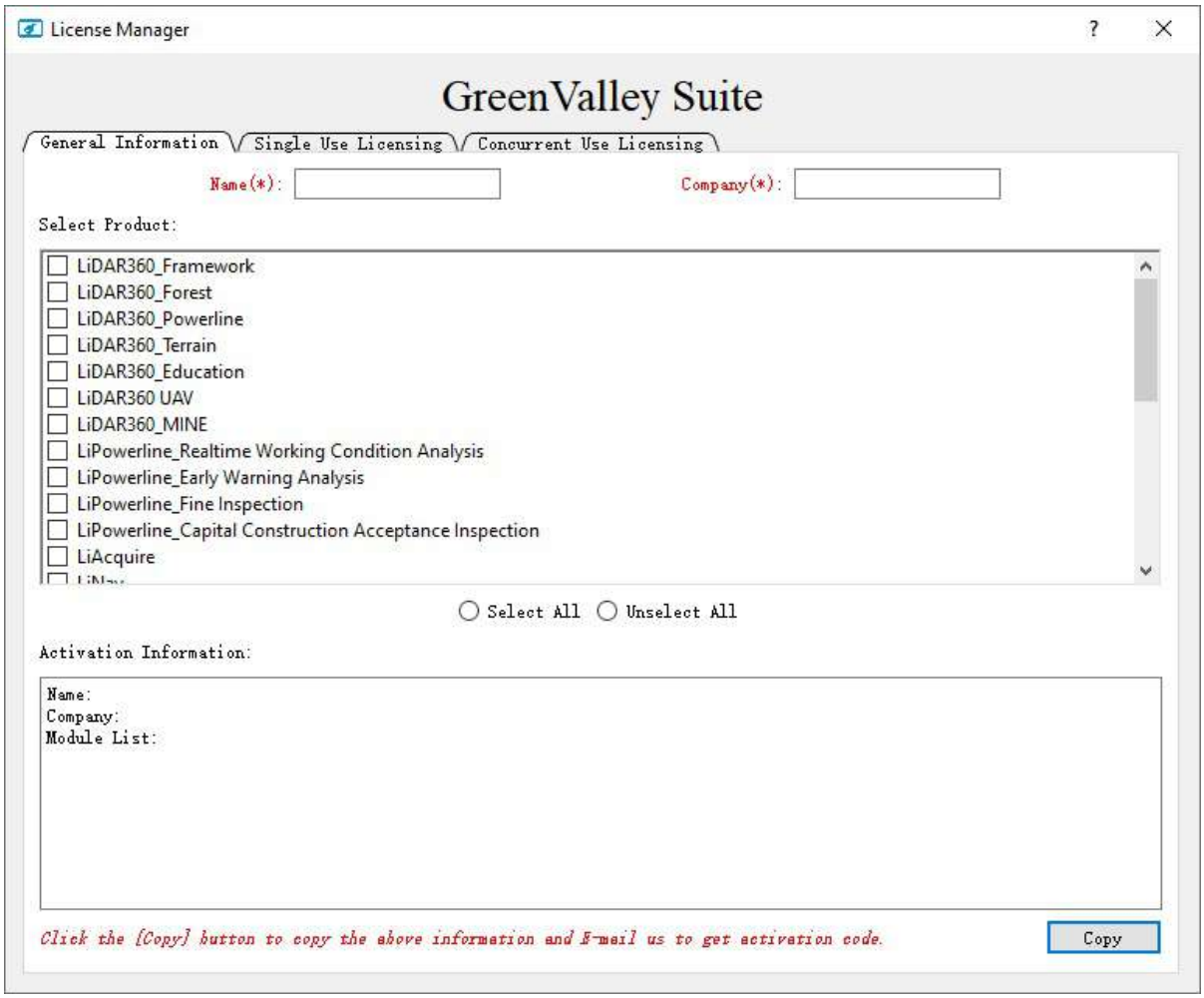
Function Description: This section introduces common problems and their respective solutions during the use of LiDAR360 MLS software.

Why does the software prompt that the trial license has expired after installing LiDAR360 MLS?

- LiDAR360 MLS cannot be tried out for the following reasons:
    - Change of system time: During the trial of LiDAR360 MLS, there may be some reasons that may cause the system time to change, causing the trial to expire.
    - Trial expiration: The trial period of each version of LiDAR360 MLS is 15 days. If the same version is installed for more than 15 days, the trial will expire.
    - Other reasons: Please send an email to info@greenvalleyintl.com to apply for an extension of the trial license.

How does LiDAR360 MLS send activation information? How to activate after receiving the license file?

- Send activation information: Click File > License or double-click License Manager.exe in the installation directory.



- enter name;
- enter the company;
- select the activation module;
- Click Copy;
- Ctrl+V paste the content into an email and send it to info@greenvalleyintl.com.
    - License activation:
        - Copy the authorization code in the email to the local, and the activation method can refer to the license manager.

What features can I still use after the LiDAR360 MLS trial expires?

- After the trial period ends, the LiDAR360 MLS software cannot be used normally, and the function needs to be activated before it can be used.

How to view the help manual?

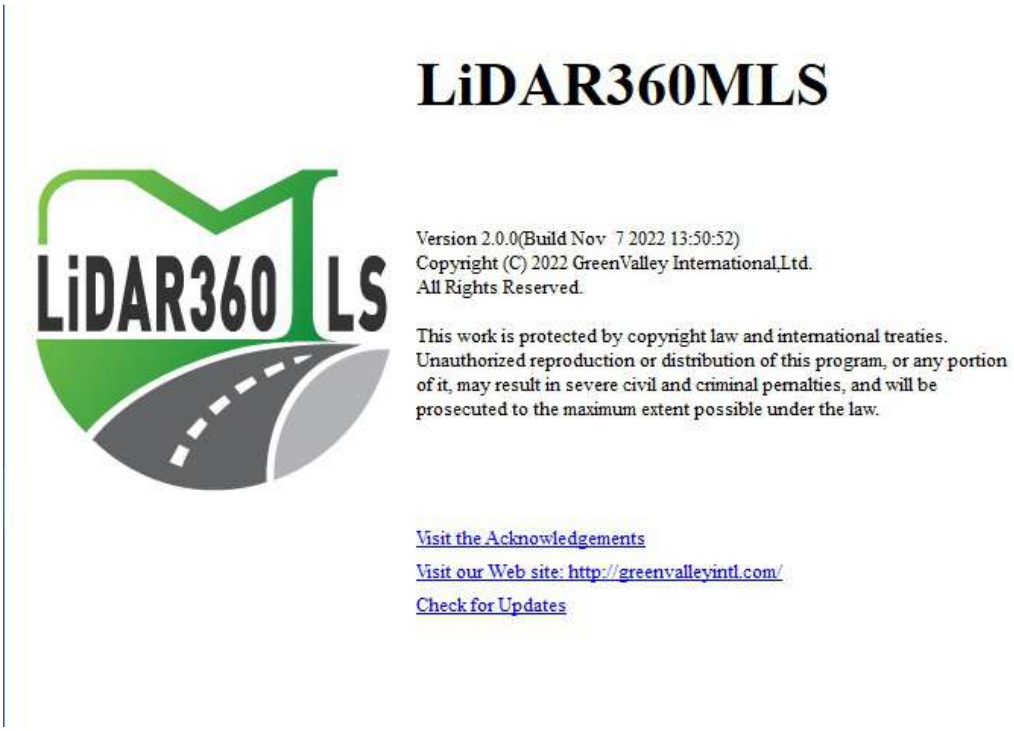
- Click the Help button in the upper right corner of the software interface.

If the computer is damaged and the authorization code cannot be checked out, how can I use the authorization code on other computers?

- Please contact info@greenvalleyintl.com, we will recover the authorization code from the background, and then you can use the authorization code on other computers.

How to check the version information of LiDAR360 MLS software?

- Click File > About to view the version information of LiDAR360 MLS software, as shown in the figure below, 2.0.0 is the software version number, and 7 2022 13:50:52 is the compilation date.



How to change the language settings?

- Click File > Options > Language to switch between English and Chinese.

Why does the software exit abnormally?

- When the software exits abnormally, please check: (1) Whether the on-screen word-taking software (such as Youdao Dictionary) is opened; (2) Whether there is enough hard disk space or memory space.

What data formats does LiDAR360 MLS support?

- The data types that can be imported into LiDAR360 MLS are divided into four categories: point cloud, trajectory, image and vector. The specific formats are as follows:
  - **Point cloud**: LiData file (.LiData custom point cloud format), LAS file (.las, .laz), PLY file (.ply), E57 file (*.e57*).
  - **Trajectory**: Trajectory data (*.traj*, *.pos*, *.txt*, *.csv*, *.asc*, *.xyz*, *.pts*).
  - **Image**: The image data is divided into panoramic camera and plane camera. It is necessary to ensure that the image recording file [Image List File(.imglist), Leica Pegasus File(.csv), Trimble MX9 File(.csv), Orbit(*.txt*)] and the image file storage location is accurate.
  - **vector**: vector data (*.shp*).
- The data formats that LiDAR360 MLS can export are as follows:
  - **Point cloud**: LiData file (.LiData custom point cloud format), LAS file (.las, .laz), ASCII file (.txt), PLY file (.ply), E57 file (.e57).

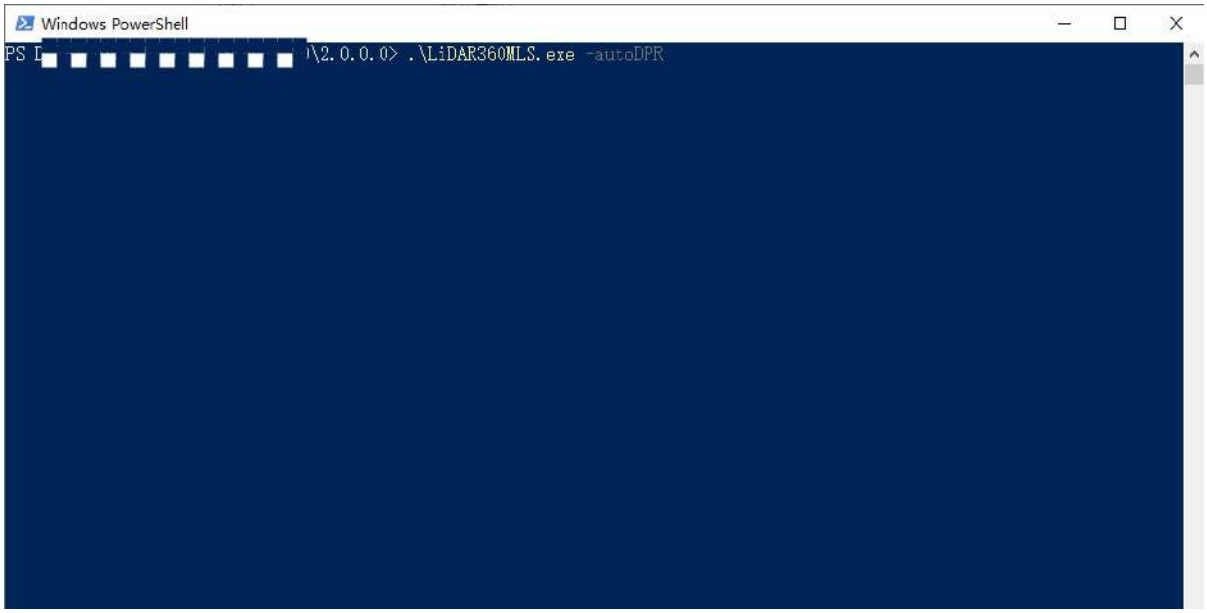
- **Vector**: Vector data (*.shp, .dxf*).

LiDAR360 MLS software crashes after startup?

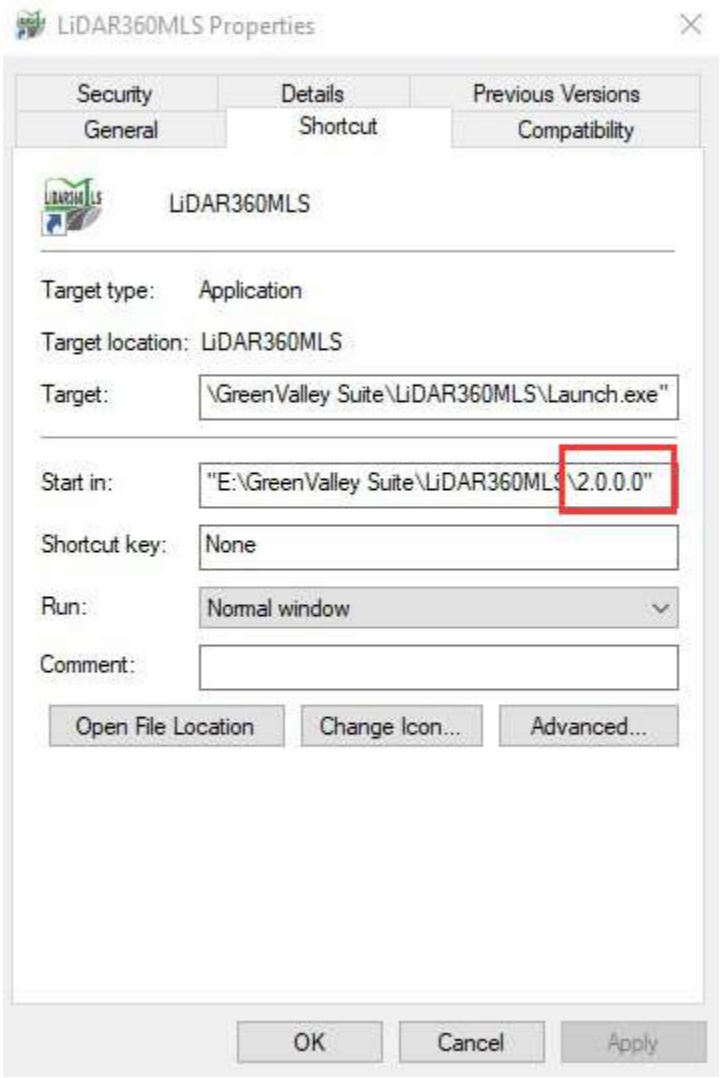
- Please check whether the computer screen is plugged in correctly. Normally, it should be plugged into the position of the graphics card connector of the host computer.

The high-resolution screen computer software cannot be started or the software is stuck?

- Method 1: In the software installation directory (eg: D/LiDAR360MLS/2.0.0.0/), press and hold the left button of the keyboard, click the right mouse button, click "Open Power Shell window here", and enter the command line in the window **.\LiDAR360MLS.exe -autoDPR**, you can start the software normally on the high-resolution screen computer.



- Method 2: In the software installation directory (eg: D/LiDAR360MLS/2.0.0.0/), send the LiDAR360MLS.exe in the directory to the desktop shortcut, then right-click the shortcut, open the properties, and at the end of the target. Add **-autoDPR**, then double-click the program to start the software normally.



After using the software for a long time, double-clicking to run it starts slowly？

- Prepare the software activation code
- In the ProgramData folder on the C drive, find the BitAnswer folder (C:\ProgramData\BitAnswer)

> Note: The ProgramData folder is a hidden folder, you need to check the "Hidden Items" box in the view section of the folder to see the folder.

- Delete the contents of the BitAnswer folder and re-run the software to re-activate it to solve the problem of slow start-up.

After running the software, some of the displays were found to be in the wrong place, such as the misplaced hints in the 3D window of the element extraction function

This usually happens when there are multiple graphics cards on the computer and the monitor is plugged into the integrated display.

- You can first check if the monitor socket is plugged into a unique graphics card (Nvidia is recommended), and it is recommended that the monitor is plugged into a unique graphics card.
- If you do not want to change the position of the monitor socket, you can refer to Adjust High-Performance Graphics Mode to run the software in high performance mode.

LiDAR360 MLS Release Notes

V2.0.4

- Optimized node editing function, Supported “Z value” and “move to” function for quick modification by picking Z
- Supported various vector selection tools to change the line direction, and Optimized shortcut keys and tips
- Supported Riegl MMS panoramic image file import and Optimized Orbit format
- Supported for South-North crown diameter (SN), East-West crown diameter (EW), crown length (CL) and crown width (CW) in the calculation of individual tree parameters
- Updated trajectory data
- Fixed the problem of Ligeo and LiFuser-BP project opening failure
- Fixed the problem of strip adjustment
- Fixed the problem of invalid horizontal section dragging
- Fixed other bugs in the software

V2.0.3

- New 2D mode
- Optimised horizontal section function
- Optimise edit vertices function
- Optimise drape to ground function
- Optimize vector objects and the corresponding panoramic image window linkage jumping
- Fix other bugs in the software

V2.0.2

- Add new project from template
- Add import gpkg function
- Support layer renaming
- Add road damage report export function
- Add clearance analysis report export function
- Add DXF and KML import and export functions
- Add dashed line batch drawing mode
- Supports lane line and road edge detection with shortcut keys to switch between semi-automatic and manual modes
- Add custom template drawing method for cross walk and parking spot
- Add line smoothing function
- Support vector objects and the corresponding panoramic image window linkage jumping
- Fix other bugs in the software

V2.0.1

- Fix the problem of database storage

- Optimize the field selection limit when import trajectory
- Optimize the right click menu of the directory tree
- Optimize the interface of vector export function
- Optimize the emptying tips of the directory tree layers

V2.0.0

- Added Modules and Features
  - New project
    - Supports drag and drop point cloud to create a new project, and save as to the specified path
    - Supports for importing original projects of LiFuser-BP and LiGeo
    - Supports point cloud and panoramic data entering software for more than 90% of mobile measurement devices on the market in formats such as Trimble MX9, Leica Pegasus, Orbit Pos, etc.
  - Database
    - Supports database for vector storage
    - Supports adding, deleting, modifying and querying layers in the database
  - Basic platform
    - Added point cloud and vector projection conversion function
    - Supports point cloud format conversion
    - Supports basic tools such as point cloud selection and cropping
    - Supports vector data format conversion
    - Supports category display 0-255 categories
    - Supports cross-sectional export of color orthophotos
    - Supports for mobile measurement device travel direction arrows and default view settings
  - Vector editing
    - The right-click menu has been added to the movement function, which supports moving in a specified distance and direction
    - Added arbitrary polygon, circle selection, ball selection, random selection, three-dimensional selection, and cylinder selection
    - Added polygon combination and split functions
    - Added one-to-many interrupt function
  - Symbol library and callout labels
    - Added feature symbolization function
      - Supports domestic standard road facility symbol library
      - Support for common symbol library
      - Supports symbol number storage
      - Support to set symbols for collected point features
      - Support for modification of symbol
    - Added annotation function
      - Support for adding annotation layers
      - Support for adding annotations to independent objects
      - Support for adding annotations to object nodes and edges
      - Support for adding annotations to 3D, elevation annotation
      - Supports annotation style modification
    - Added layer label function
      - Supports all layer properties for label display
      - Configurable styles for labels

        - Facilitate operations such as attribute entry
    - New vector line object direction display
- Road feature extraction module
    - Added the modularization extraction function of strip features
    - Added monomerization segmentation function
    - Added monomerization editing function
    - Added monomerization parameter extraction function
    - Added AI-based road surface marking recognition function
    - Added batch processing function for road surface marking recognition
    - Added AI-based batch extraction of road facilities
- Point cloud classification and extraction
    - Point cloud classification
        - Supports point cloud deep learning classification
        - Supports polygon-based point cloud classification
        - Supports point cloud classification based on vector lines
        - Supports air noise classification
        - Supports subsurface classification
        - Support above ground point classification
        - Support for separating lows
        - Supports outlier classification
        - Supports proximity point classification
        - Supports ground point classification
        - Supports classification by attribute
        - Supports classify by cluster size
    - Point cloud extraction
        - Supports extraction by category
        - Supports extraction by elevation
        - Supports extraction by intensity
        - Supports extraction by GPS time
        - Supports extraction by return number
- Planar camera browsing measurement
    - Added planar camera data and point cloud overlay browsing display
    - Added planar camera measurement
    - Added planar camera calibration
    - Added planar camera undistort
- Vehicle point cloud preprocessing
    - Supports trajectory segmentation
    - Supports laser placement inspection and calibration function
    - Supports trajectory quality check and repair
    - Added control point correction function
    - Added vehicle trajectory splicing function
- Road analysis module
    - Added road damage detection function
    - Added headroom analysis function
    - Added visual field analysis function
    - Added road section analysis function
    - Added road cross section parameter extraction function

V1.2 - 10/03/2022

- New modules and features:
  - Added Facade Measurement Module
    - Supports vertical elevation
    - Supports horizontal reference line drawing
    - Supports range line drawing
    - Supports taking a section based on a reference/extent line and making adjustments
    - Supports the drawing of the basic elements of the facade
    - Supports for construction line drawing
    - Supports array drawing
    - Supports translation and rotation
    - Supports for exporting DXF and orthophotos
    - Added shortcut key configuration function
    - Added layer manager function
    - Added combined display function
    - Added cross drawing and vertical drawing functions
    - Added capture module
- Optimization function
  - Optimize the drawing method of rectangular traffic signs

V1.1 - 10/03/2022

- New modules and features:
  - Added data block module
    - Supports blocking by scale
    - Supports track segmentation
    - Supports area block node editing
    - Supports area block merging
    - Supports point cloud segmentation based on block
    - Supports area block selective display
    - Supports display and hide of area blocks and their labels
      - Added more road feature template symbol library
    - Supports all types of arrow reticle drawing
    - Supports for manhole covers, parking spaces, and drainage grate to be drawn according to templates
    - Supports traffic signs to be drawn according to templates
    - Supports Chinese and English character drawing
    - Supports non-motor vehicle signs, no U-turn signs are drawn according to the template
      - Added template matching function
    - Supports automatic vectorization matching based on default templates
    - Supports custom feature templates and auto-vectorization or manual vectorization
      - Added multi-type vector editing function
    - Supports for modifying the shape of line features
    - Supports tracing other vectors when drawing vectors
    - Supports point break line, line break line, line break surface
    - Supports vector affixed to ground
    - Supports precise selection when vector glands

    - Added attribute table calculation and predefined functions
- Added attribute batch filling function
- Added line feature length calculation function
- Added character replacement function
    - Added node editing function
- Support for individually modifying node coordinate values
- Support batch modification of node Z value
    - Added the function of modifying the project
    - Added layer vector copy function
- Supports copying of elements on the same layer
- Support vector geometry copy between different layers