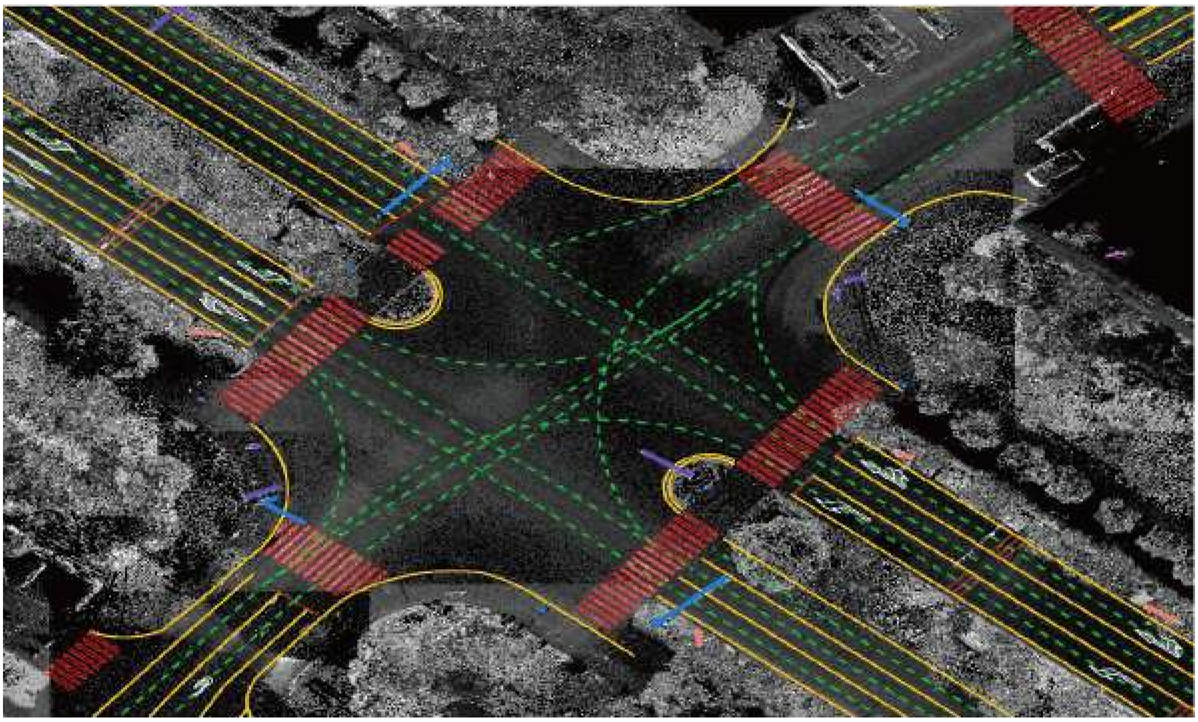
GreenValley International

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.

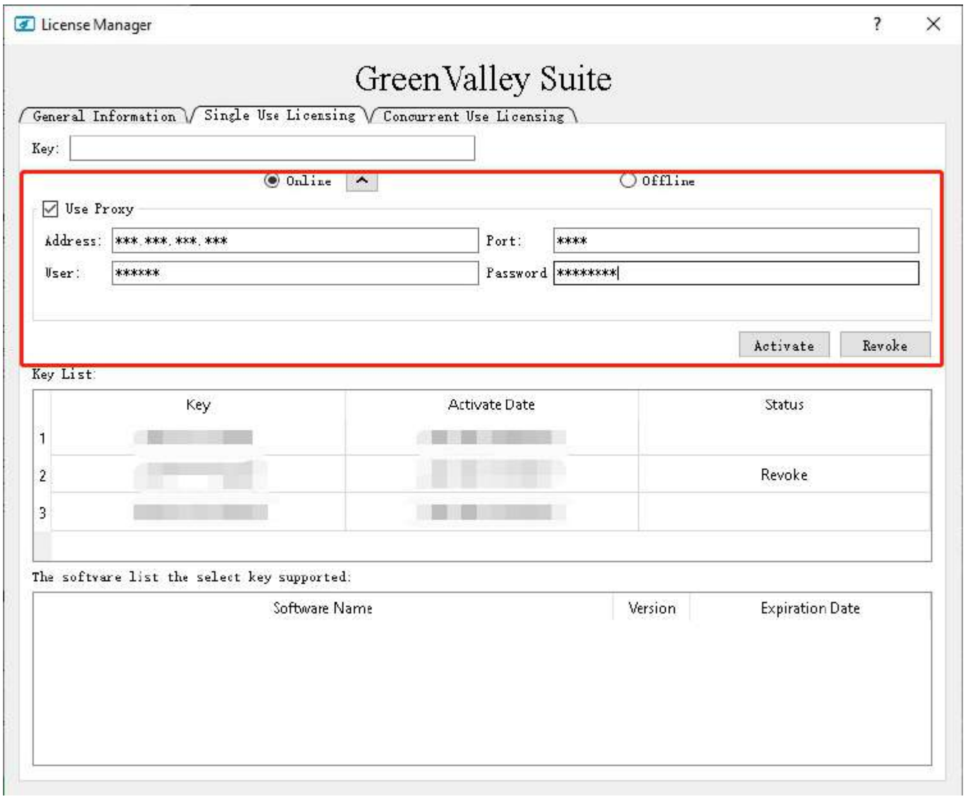
2）License key

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.

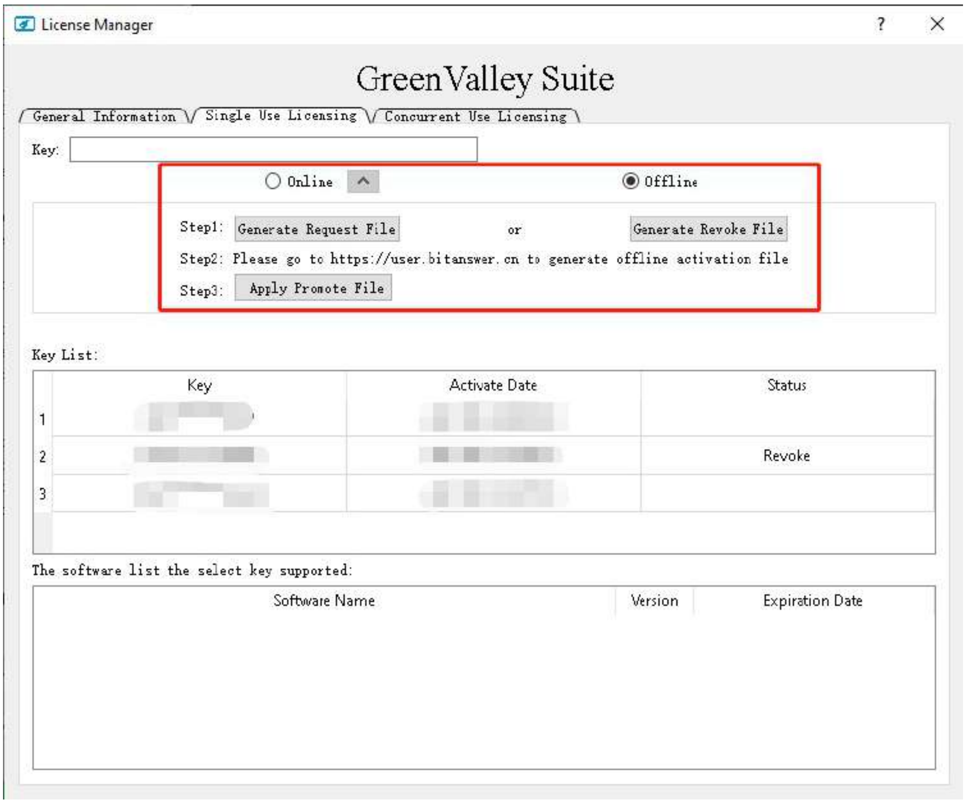
1. Run LiDAR360 MLS, the license management dialog box will appear.
2. Fill in the name and company name, select the module that needs to be activated, and then click "Copy".
3. Paste the copied information into the body of the email and send it to info@lidar360.com.
4. The software authorization mode 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.



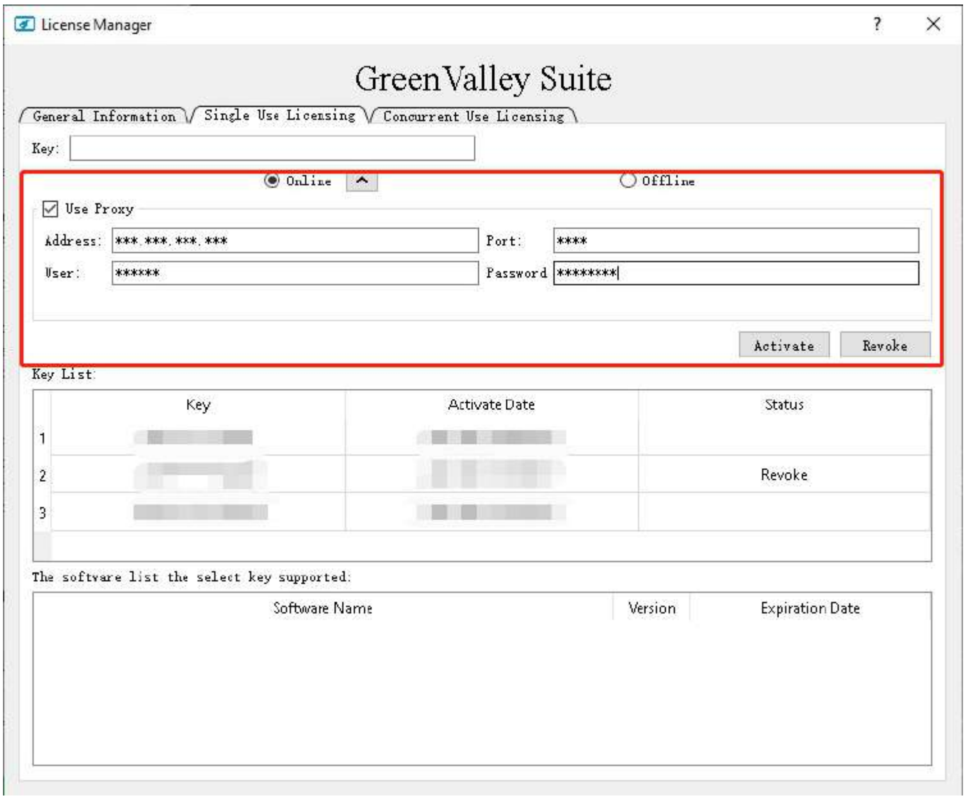
Offline Activatin/Update: Enter the key under Single Use License tab, select "Offline", and click"Generate request file"(.req). Use a computern with internet access, type https://user.bitanswer.cn in an internet browser, enter the license key to log in and click "Offline update". Upload .req file, and 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.



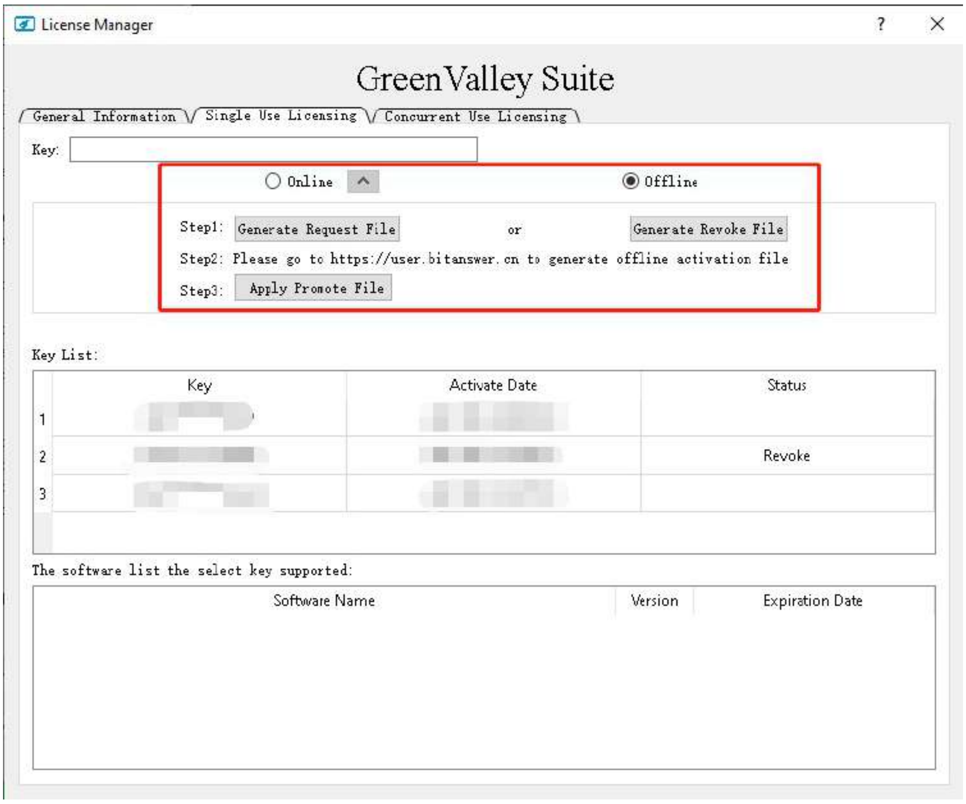
- Revoke

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.

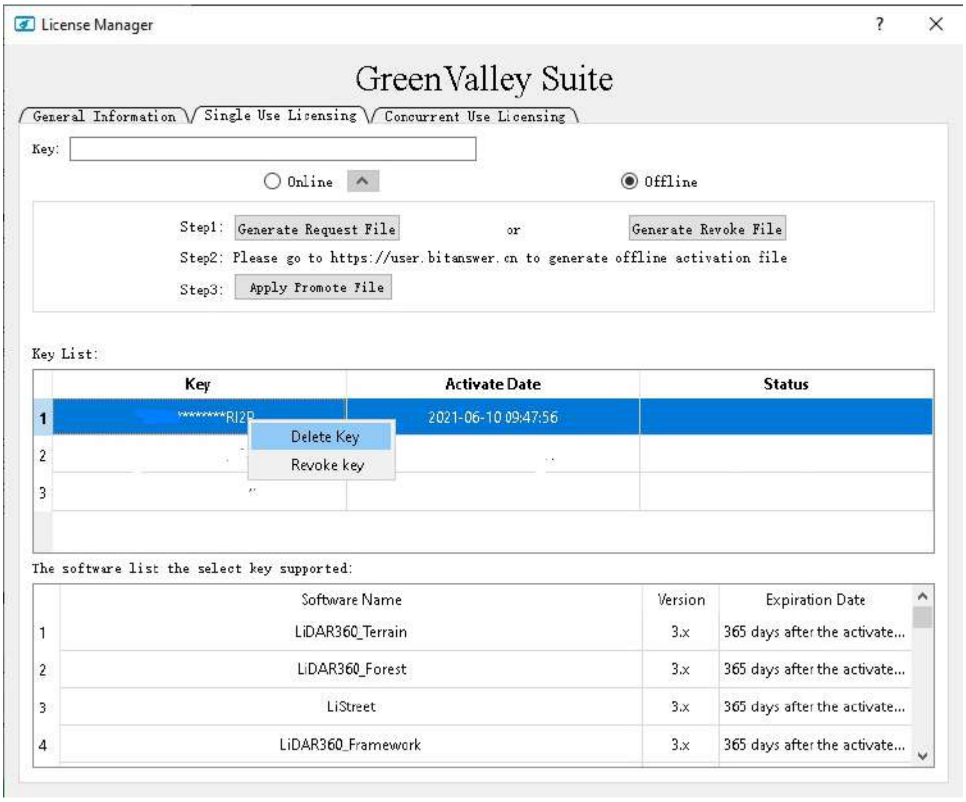


Revoke offline: Enter the authorization code, select "Offline", click "Generate Move Out File" to generate move out request file (.req). With the help of a computer that can be connected to the Internet, enter https://user.bitanswer.cn in the browser, enter the authorization code to log in, click Offline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



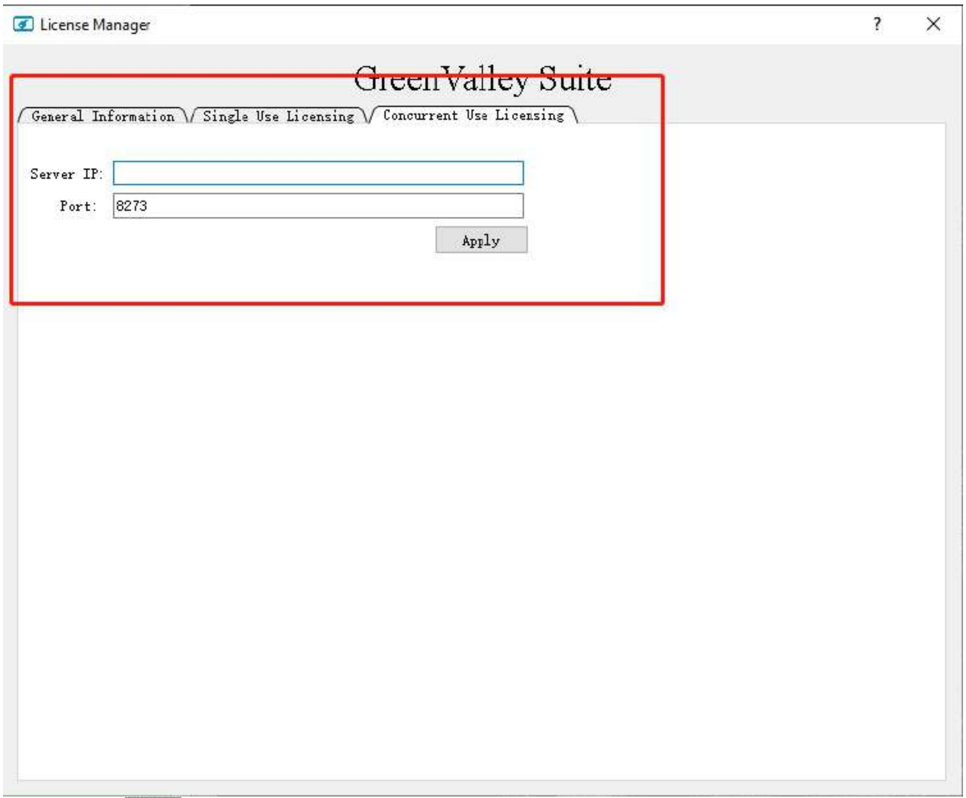
- Delete

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:：

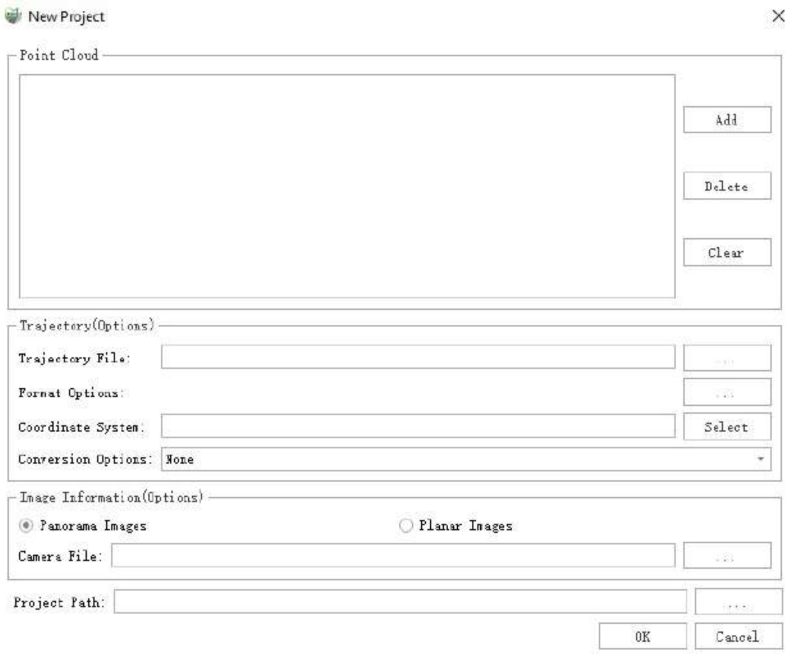
- New Project
- Open Project
- Modify Project
- Save Project
- Save As Project
- Close Project
- Option
- About
- License
- Help
- Exit

New project

Function Description: LiDAR360MLS project is created 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**: You 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**: You can choose to convert the track 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. You can also set the coordinate system yourself
- **Conversion option**: conversion option of track coordinate system, whether to use seven parameters can be selected

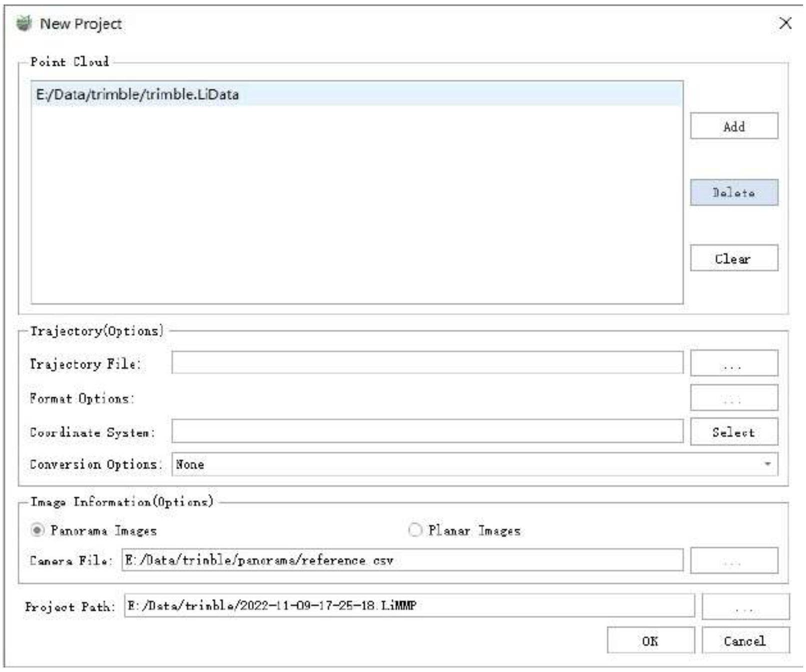
Set Camera File Path

- **Image information**: support panoramic image and plane image

Panoramic image

- **Camera file**: fill in panoramic image file

> Software can suppor ***.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 suffix. imglist are supported
- **Camera parameters**: You can select our own cal calibration file to import as parameters, or you can click the Add button in the camera parameters to manually add the calibration parameters of the planar camera. You 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

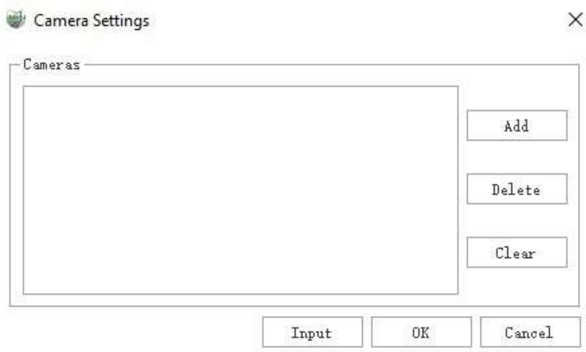
the coordinate center of the projection plane; Wherein, k1, k2, k3 are the coefficients of Taylor series expansion, which are used to correct the radial distortion caused by the lens; P1 and p2 are used to correct the distortion caused by the tangential distortion of the lens; B1 and b2 are generally 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

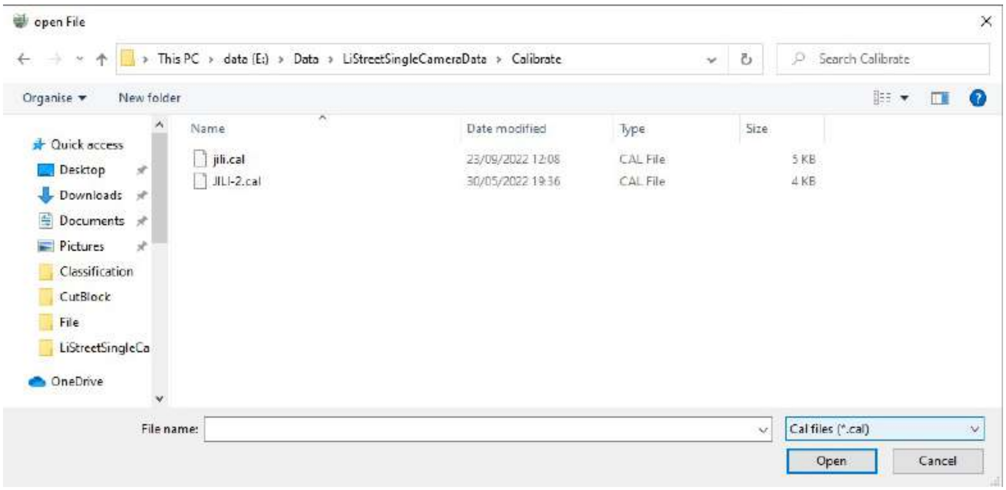
Steps

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

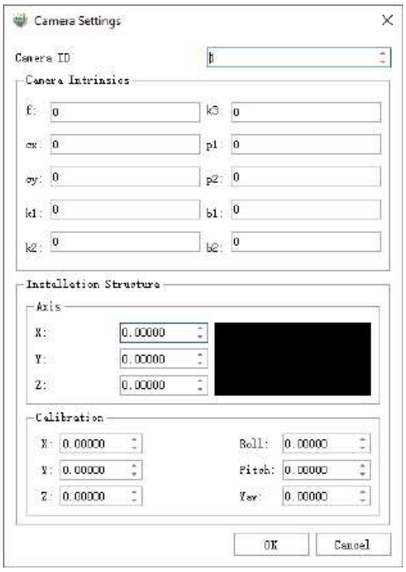
2.Click Settings to pop up the **Camera Settings** dialog box



3.Cal calibration files can be imported directly or calibration parameters for flat cameras can be added manually.

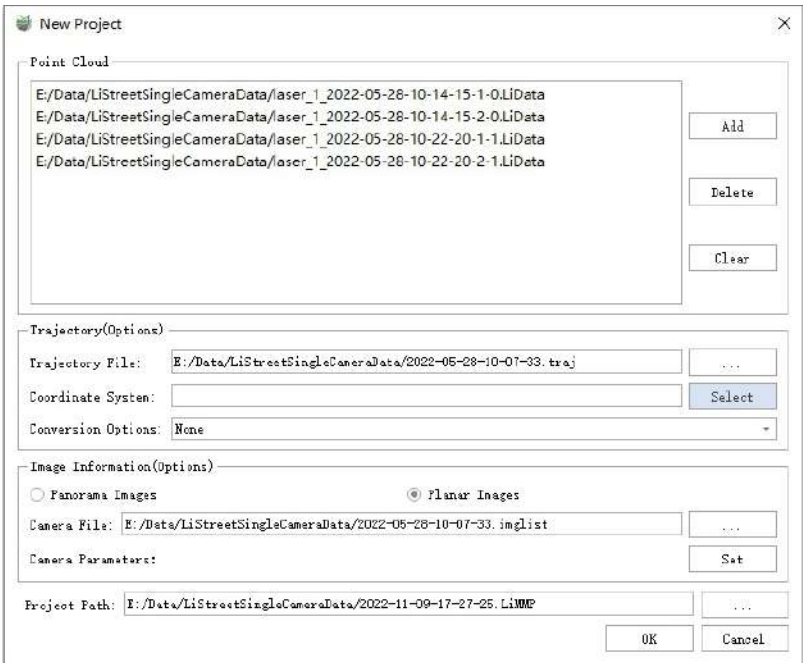


Import cal calibration file directly



Manually adding calibration parameters for flat cameras

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



Select a planar image

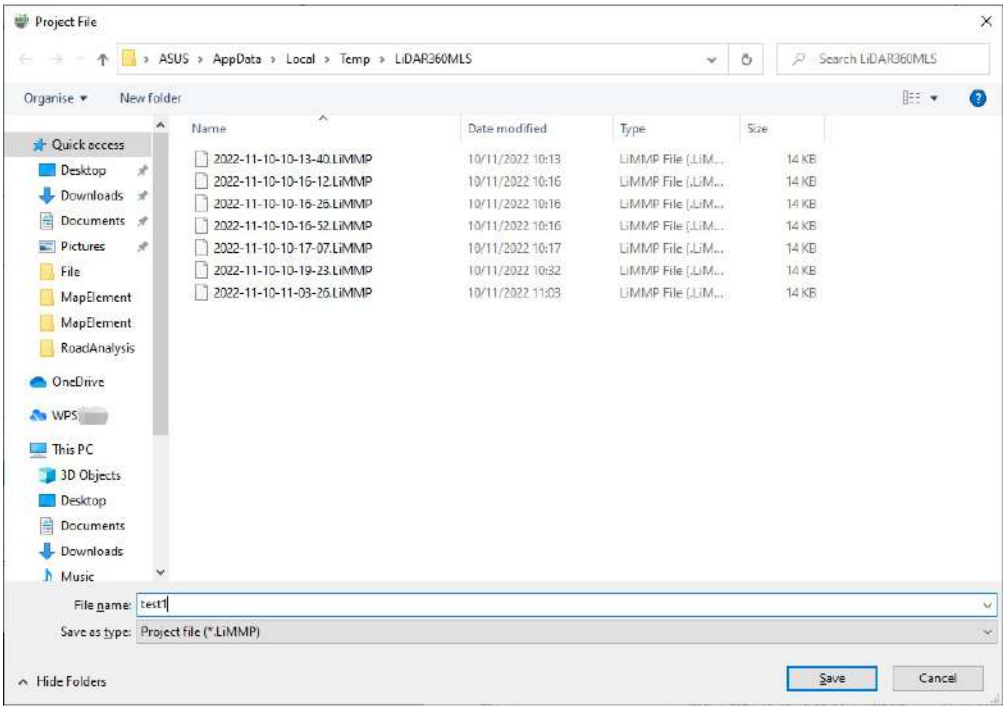
Set project file path

- **Project Path**: the storage path of the project file (suffix is. 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

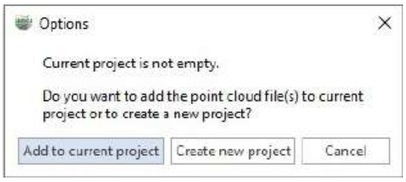
Function Description: You can create or modify a project. Select several point cloud files with the mouse, drag them to the main window of the software, and then release the left button to complete relevant operations.

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



Engineering documents

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 a new project based on the selected point cloud and open it;
- **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.

Open Project

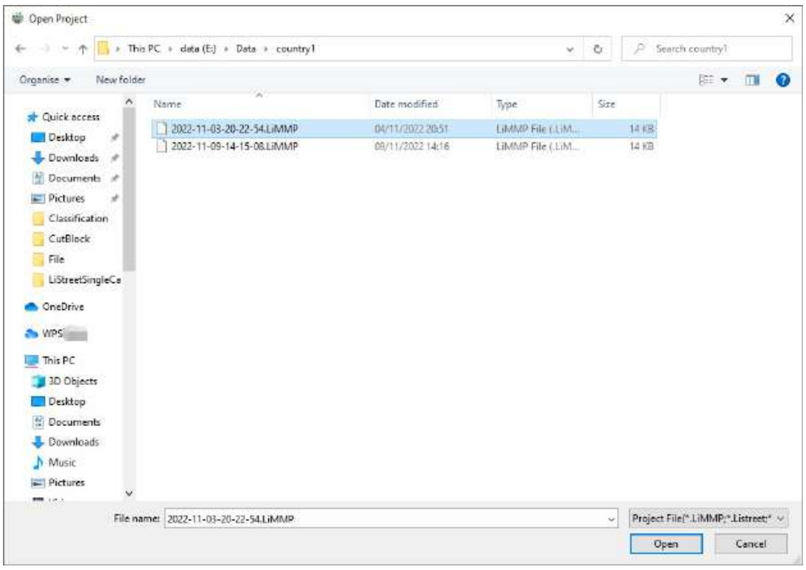
Launch the software, click Open, select a recently saved project or open a project saved at a specific directory.

> 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.



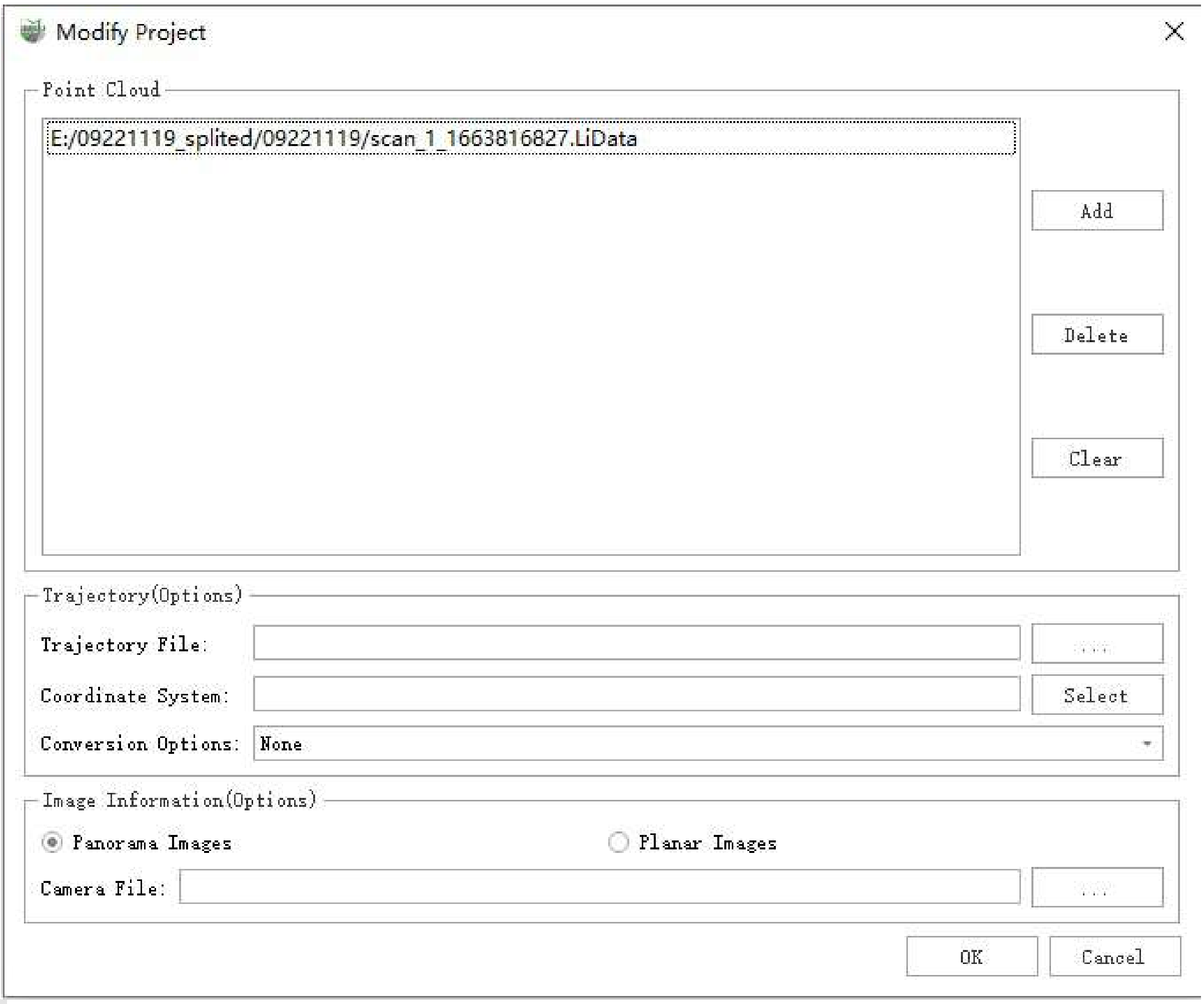
Modify Project

Description：You can click Modify Project to modify opened project, such as point cloud, trajectory or image.

Steps

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

2.Then, click ， and you can add, delete, remove point cloud in current project; add, delete or adjust coordinate system of the trajectory; modify image file.



Note：

- Currently, software doesn't 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 directory of laser scanner data."

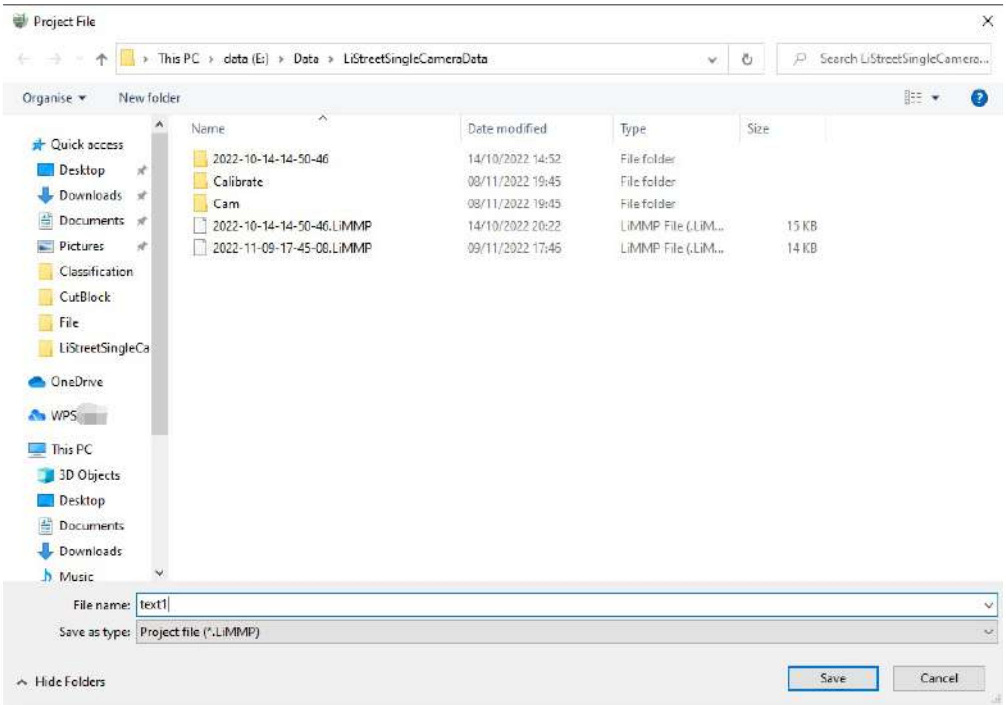

Warning
Please check the laser file path
Yes

Save Project

Click **Save**, the software will automatically save the project file being edited, the shortcut key for this function is: Ctrl+S.

Save As

Function description: Click  button, the software can rename the project file being edited and save it.



Close project

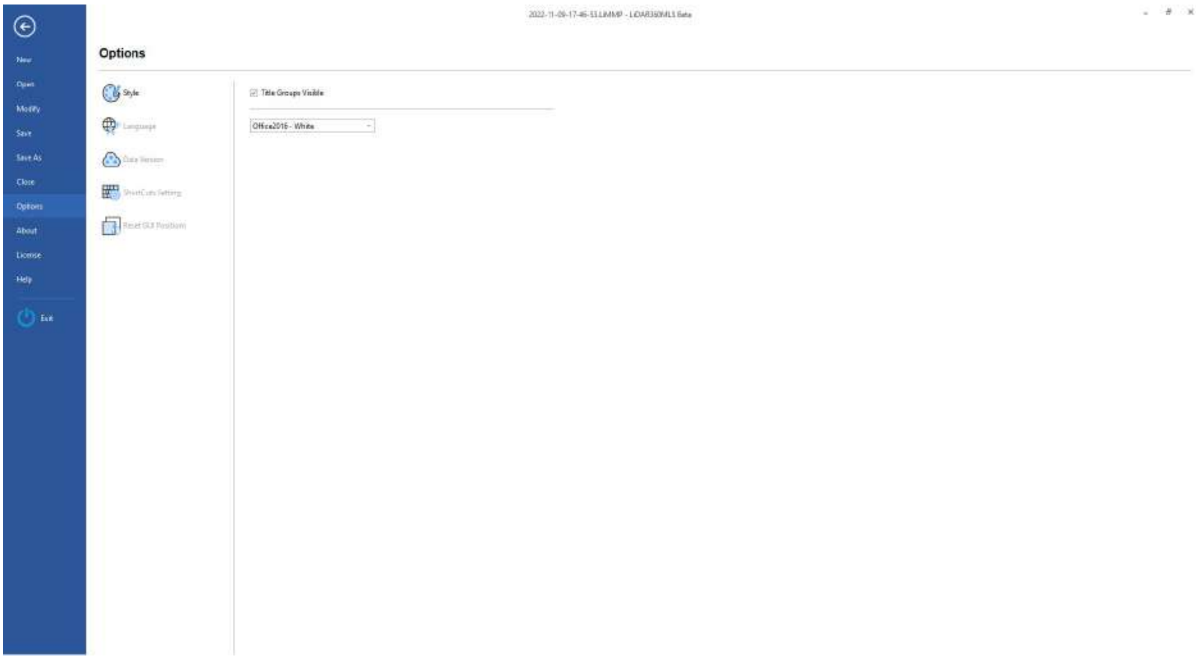
Function description: Click to close, the software will automatically exit the project file being edited.

Options

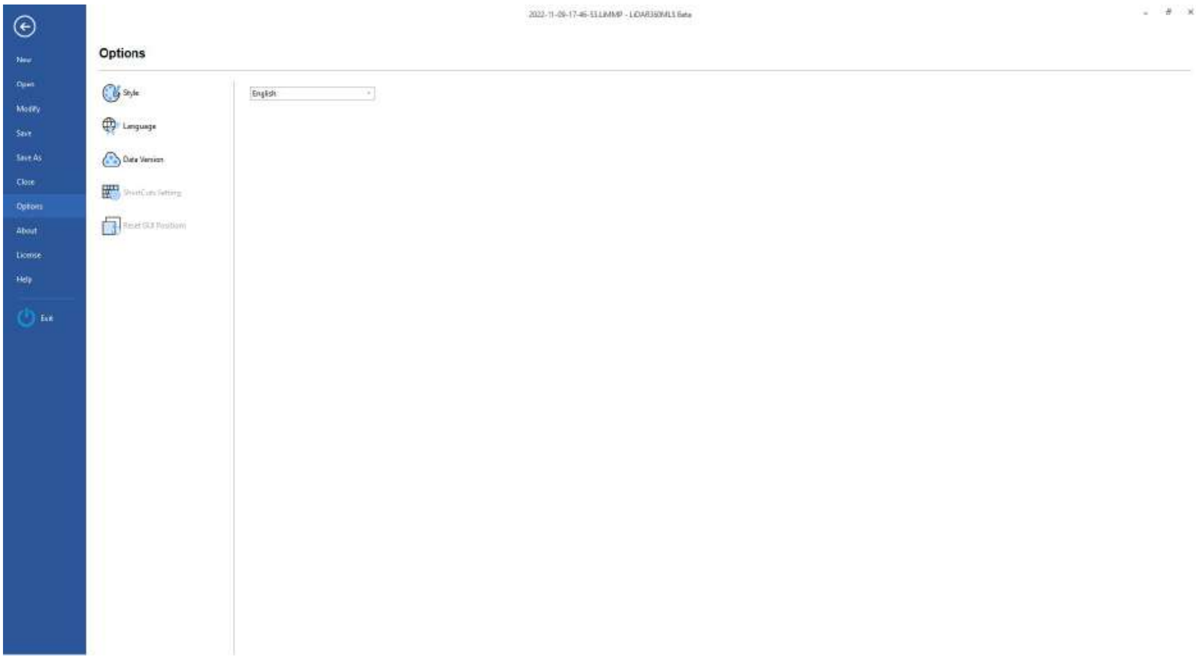
Click on the options to make some settings for the software.

Steps

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



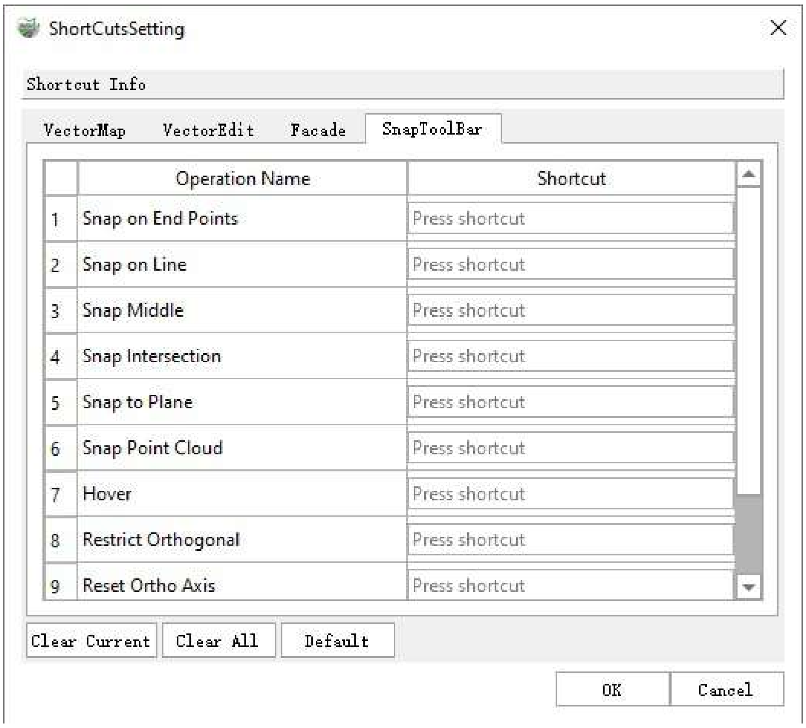
- **Language**: Set the language of the software



- **Data Version**: Set the data input and output to the data version under the las 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 Configuration  button to bring up the Shortcut Configuration dialog box.



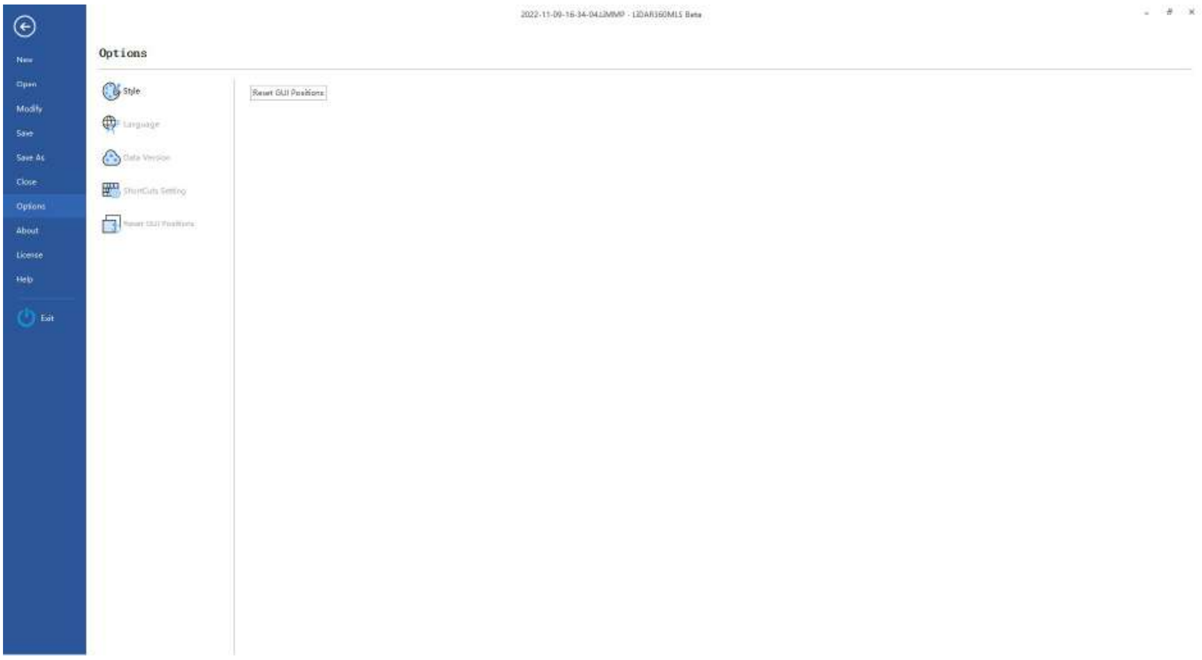
2.In the Shortcut Configuration dialog box, you can freely combine the letters, numbers, Shift, Ctrl, Alt and other symbols of the keyboard, and click OK to take effect.

3.Click Clear this Line 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 draggable toolbar out of the interface and cannot be restored.



Software Default Shortcuts

1.Common Platform Shortcut Keys

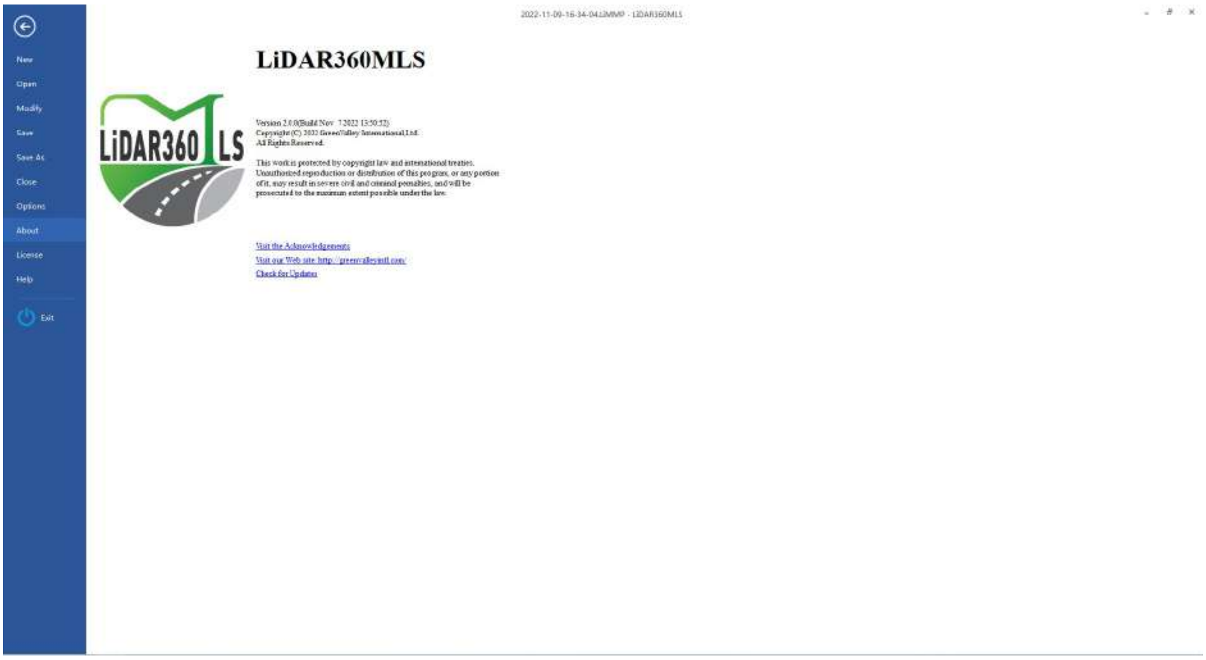
Function	Shortcut keys
Exit LiDAR360 MLS software	Alt+F4
Save LiDAR360 MLS project	Ctrl+S
Orthographic/Perspective Projection Toggle	F3
Data shift up	↑
Data Pan Down	↓
Data shifted to the left	←
Data shifted to the right	→
View zoom in	+
View reduction	-
Rotation	Left mouse button
Shift	Right mouse button 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

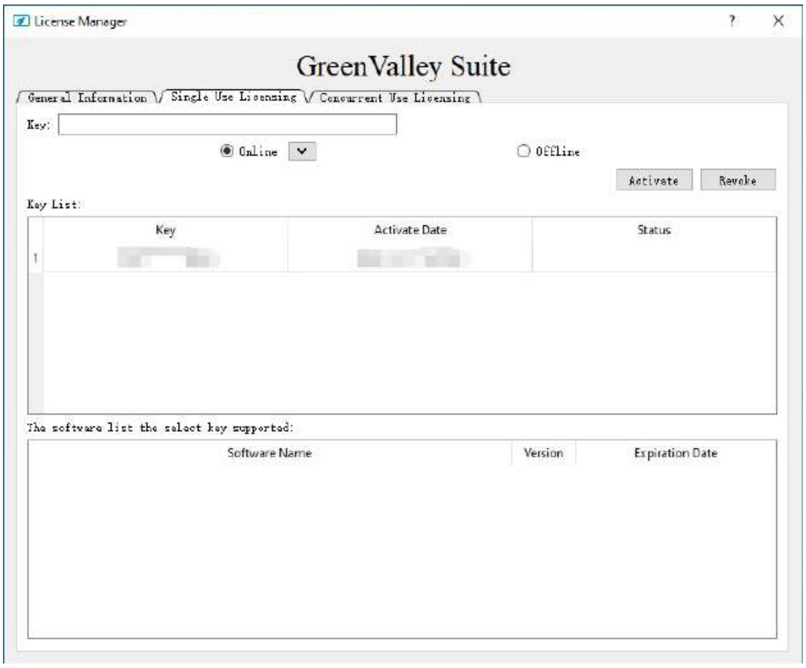
About

Function description: You can view the software version, visit the official website and check for updates.



License

Function description: Click License to activate and move out the activation code.



Help

Function description: Click Help, the software will automatically open the user manual.

Exit

Function description: Click Exit to exit the software.

Tool

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

Measure

Measurement includes 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.The following are introduced separately.

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

Single point selection

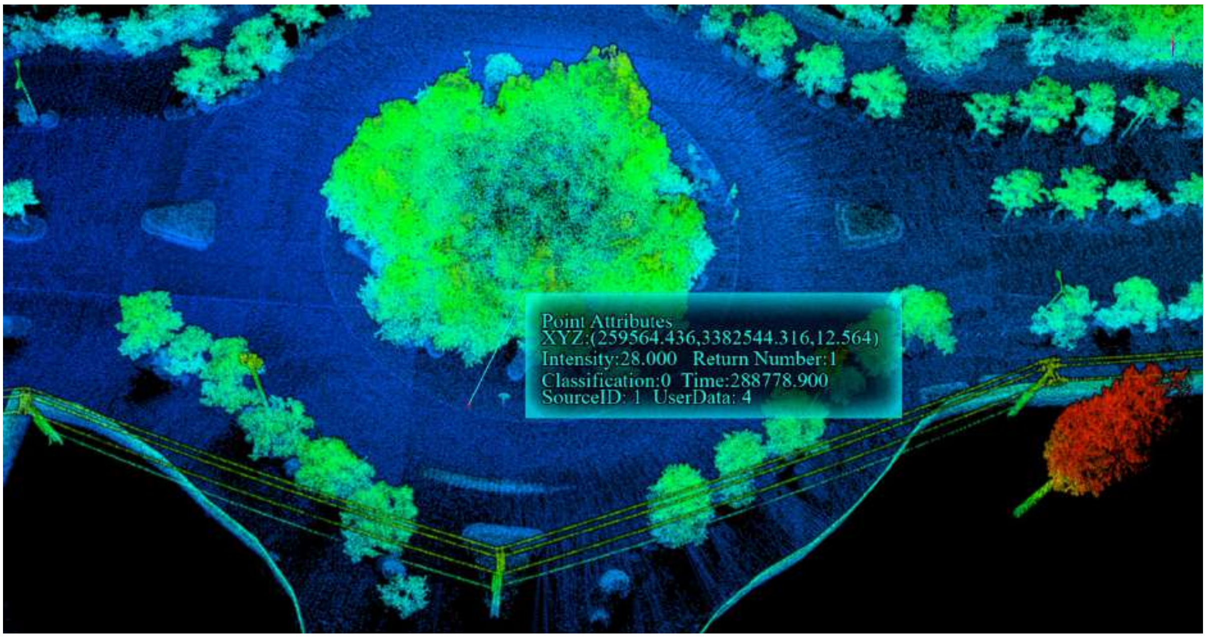
Function description : The single point selection tool can interactively query the attribute information of a single point in the point cloud data, including the location, intensity, echo frequency, category, GPS time, etc. of the point.

Steps

1.Click on the **single point in the** measurement **item to select**  button.

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 click completion window to display the selected point 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 feature can only be used in the 3D view window.

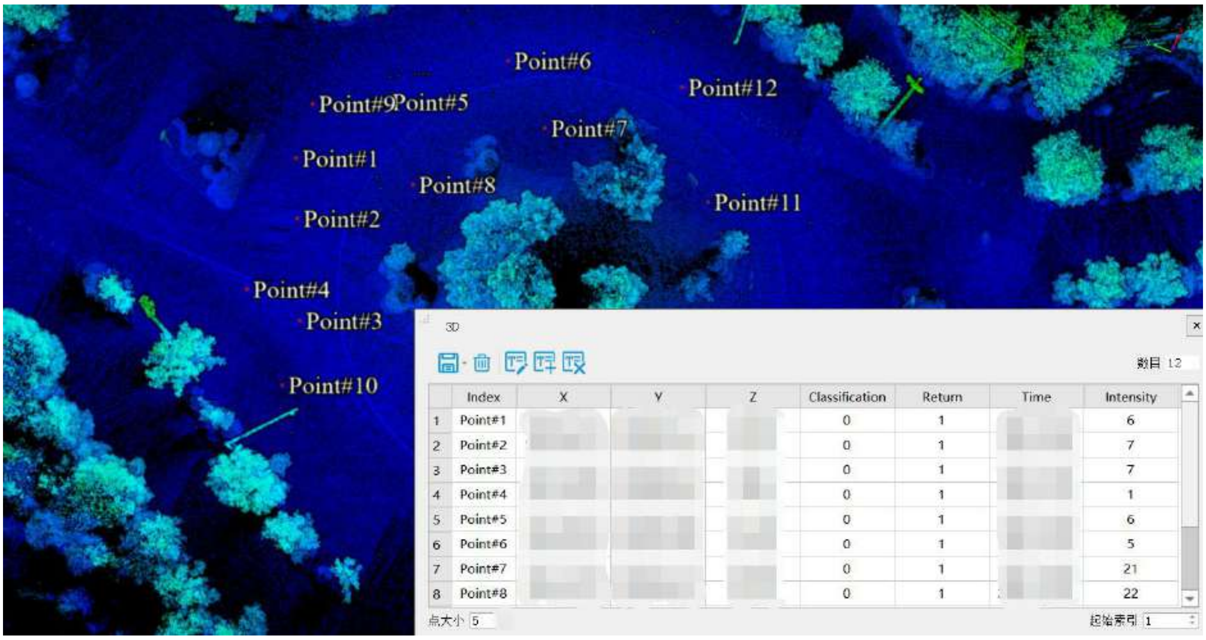
Multi-point selection

Function description : The multi-point selection tool uses mouse clicks to interactively query the attribute information of multiple points, and supports the selection of point sets to .Export 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.

Step

1.Click on the **multi-point selection in the** measurement **item** Button, use the left mouse button to click on the single points in the scene in turn, the selected points are marked in the form of dot labels in the scene, and the interface list pops 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.

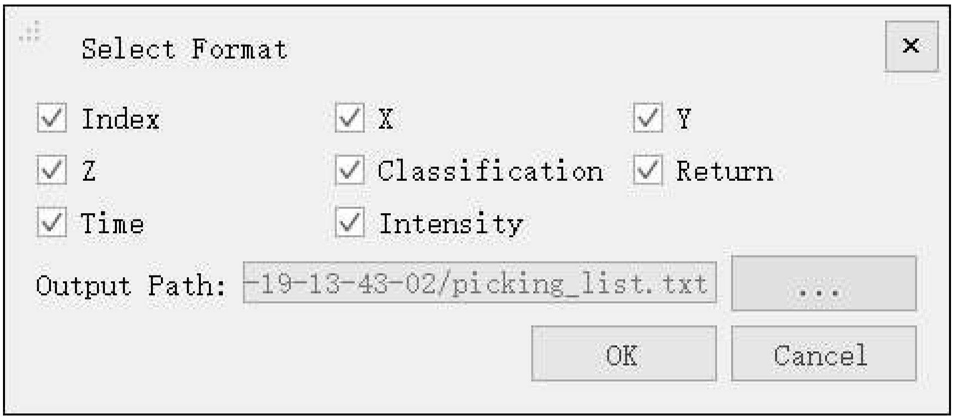
3.Click any row in the selection list and click The button deletes the point.

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

5.Click the Add field button After that, 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 It is unavailable before 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.The selected point set is marked with *.Export in txt* format. Click  The button drop-down menu pops up the "Select method" dialog box (as shown in the figure).You can click Save 3D point to save the coordinates and other attribute information as *.txt* file.

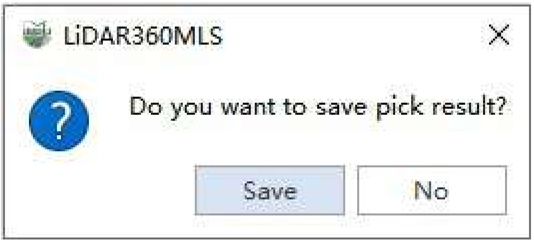


Parameter setting

- **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(Output Path)**: The output path of the exported file.

8.Click  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 feature can only be used in the 3D view window.

Length measurement

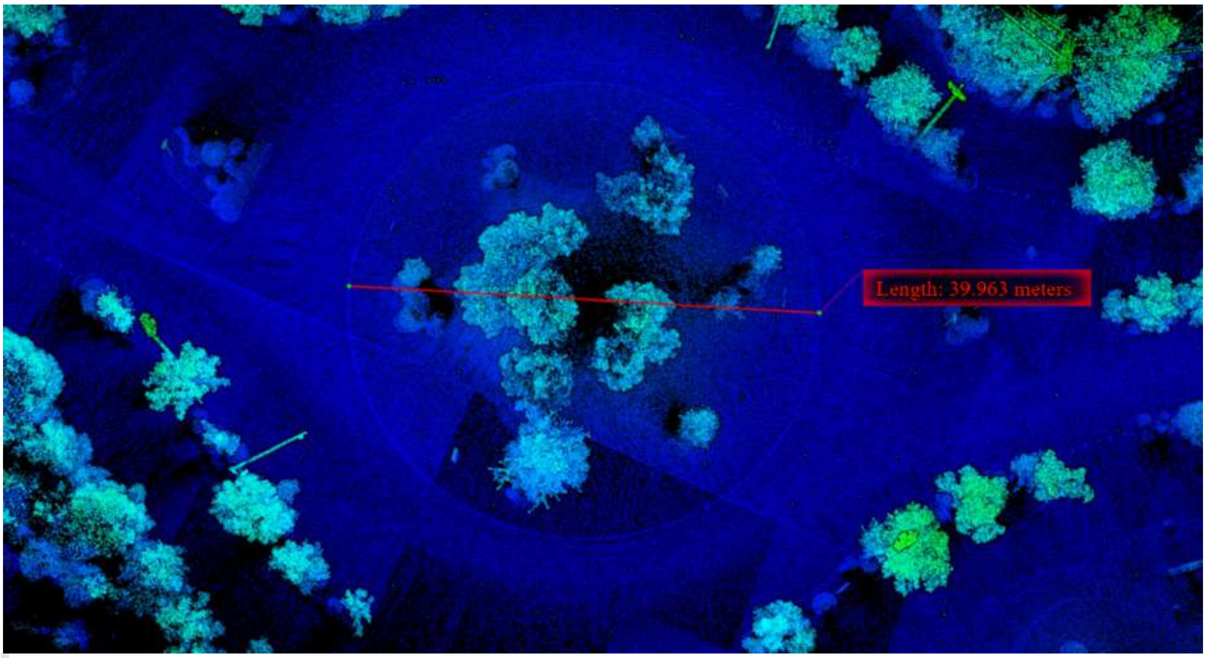
Function description: The length measurement tool uses a mouse click to interactively query the distance information between multiple points.The measurement result represents the Euclidean distance of the point in the three-dimensional space.

Step

1.Click on the **length** in the **measure** item button.

2.Use the left mouse button to click on the data points (at least two points) in turn, and multiple line segments are drawn in real time in the scene, and the measurement results are 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 feature 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 area measurement of 3D data, the current window will automatically switch to orthogonal projection mode.

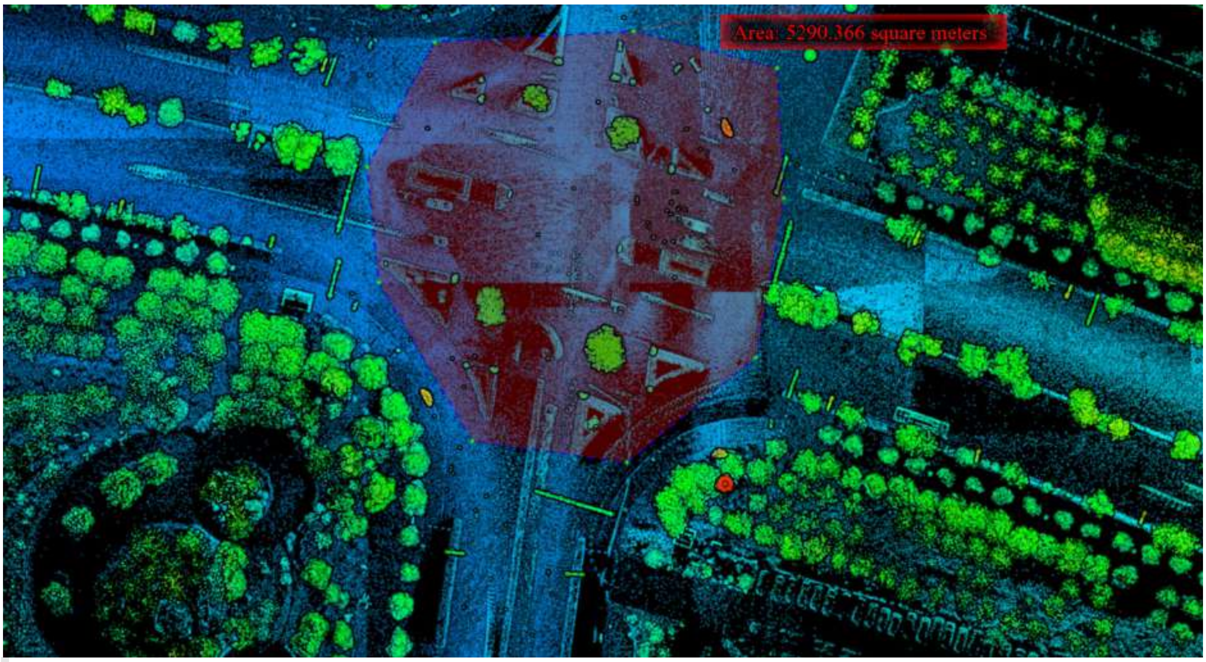
Steps

1.Click on the **area** in the **measure** item  button.

2.Use the left mouse button to click to select the data points in turn (at least three points are selected), polygons are drawn in real time in the scene, and the measurement results are 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.

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.

Step

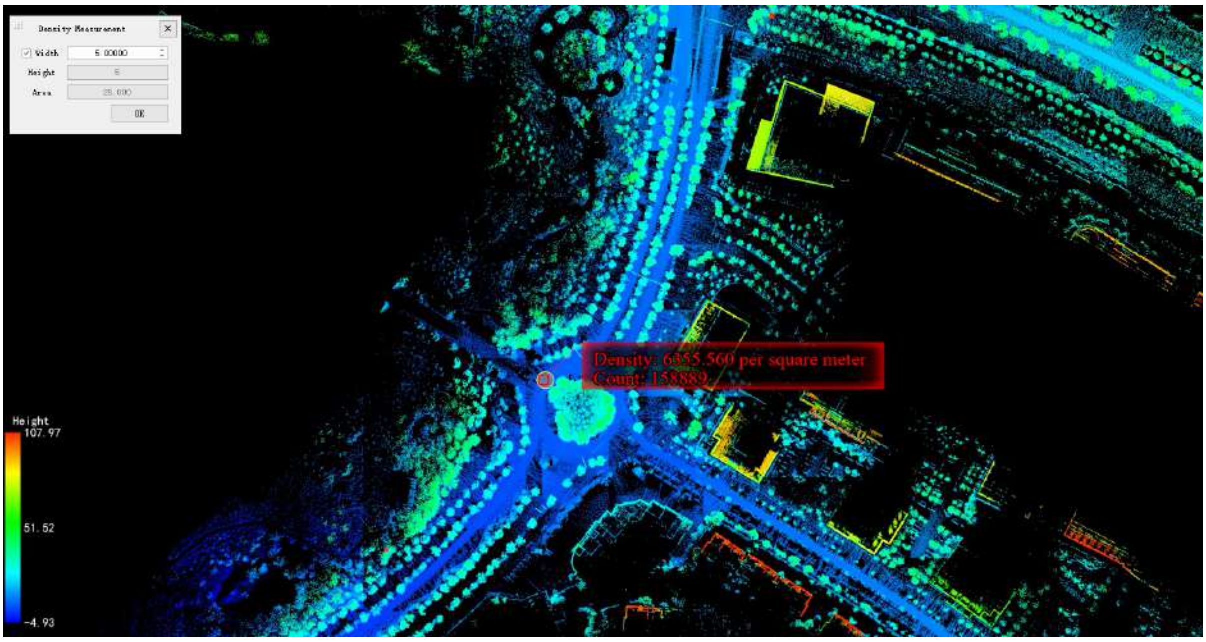
1.Click on the **density** in the **measure** item  Button, open the point density measurement tool, and the 3D display window will automatically switch to orthogonal projection.

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.

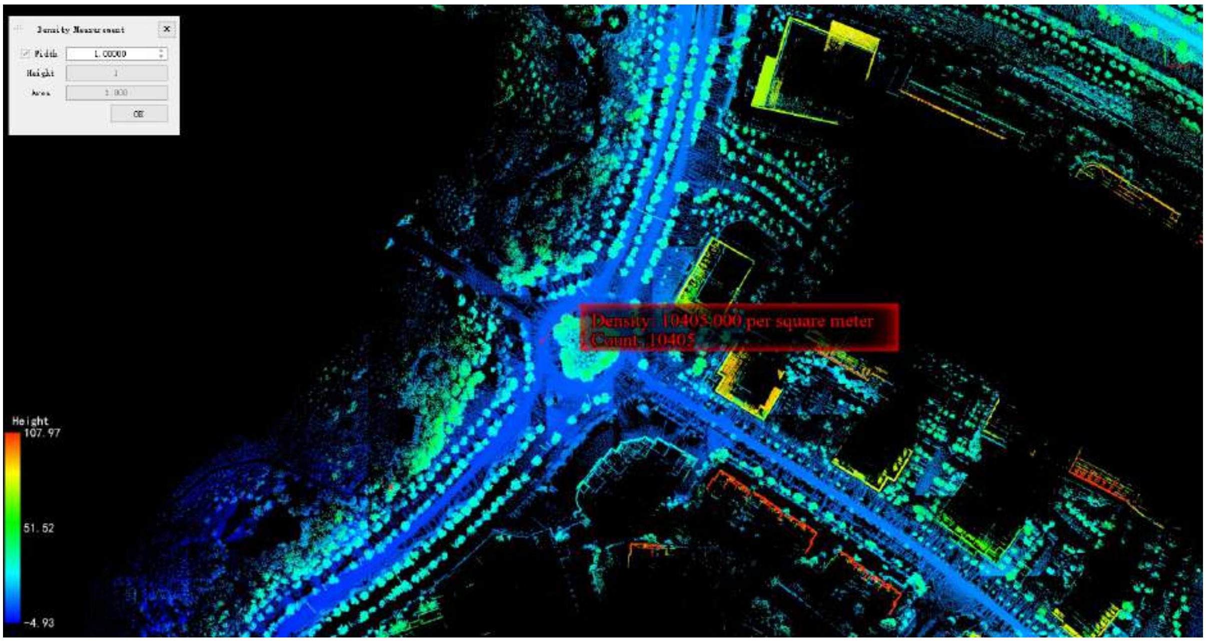
3.If "Width" is checked, you can enter the width value, the height value will be automatically set to the same as the width value, and the area is set to the product of width and height. Use the left mouse button to click to select the measurement area; if "Width" is not checked, the height value and 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.

Angle measurement

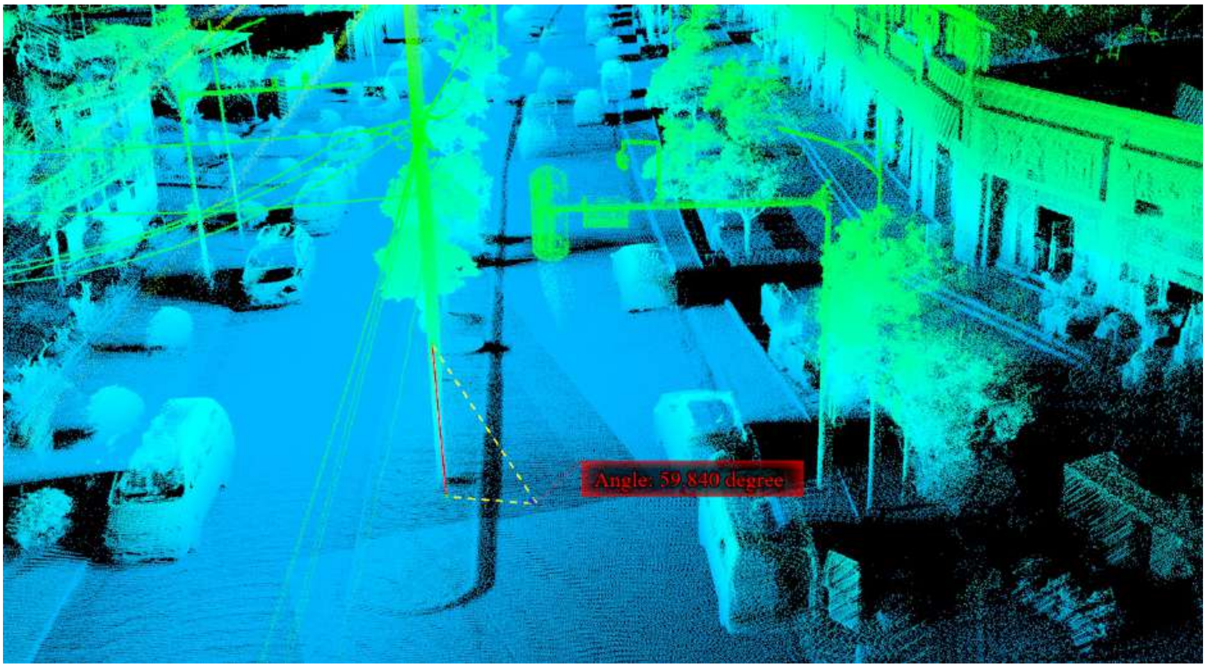
Function description : The angle measurement tool uses the mouse to click interactively to select the measurement point. In the 3D view, the pitch angle between the two points is queried, that is, the angle between the starting point to the end point connection and the horizontal plane, and the projection angle of the three-point connection on the horizontal plane is queried in the 2D view.

Steps

1.Click on the **angle** in the **measurement** item button.

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, the measurement is over, the measurement angle is drawn in real time in the scene, and the measurement results are 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

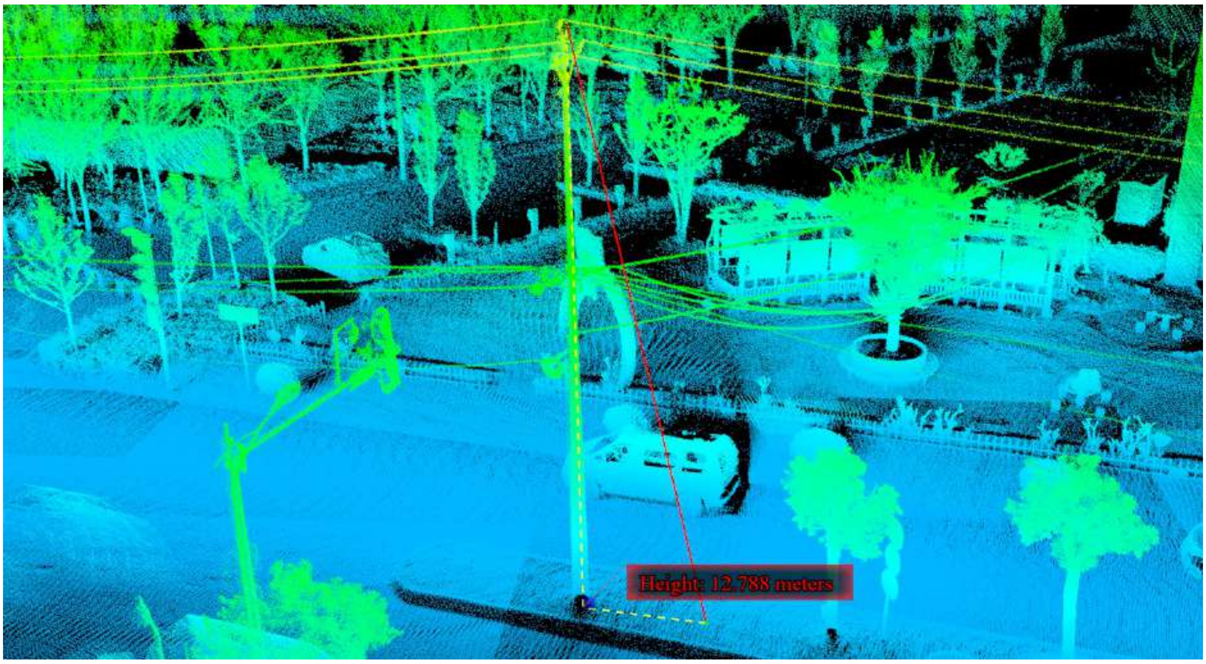
Function description: The height measurement tool interactively selects the measurement point with a mouse click and queries the relative height between the two points.

Step

1.Click on the **height** in the **measure** item  button.

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, the measurement is over, the relative height between the two measurement points is drawn in real time in the scene, and the measurement results are 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.

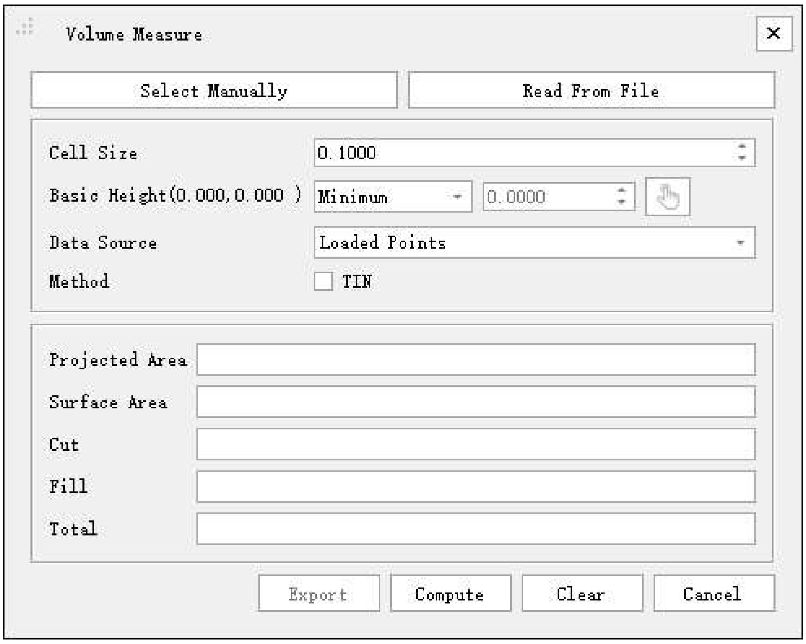
> Note: This function can only be used in the 3D view window.The fallback function can only be used before double-clicking the mouse.

Volumetric measurement

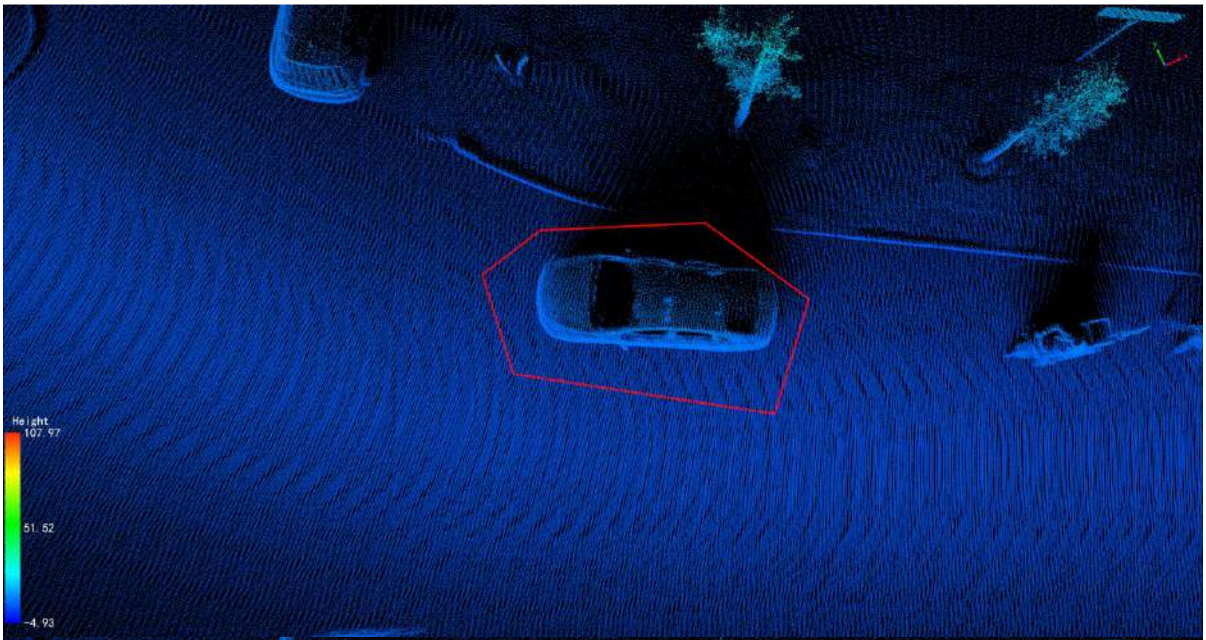
Function description : The volume measurement tool interactively selects the measurement reference plane by clicking with the mouse, and calculates the amount of filling, digging, and filling relative to a certain height. It is generally used in coal pile volume measurement, hull volume measurement,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

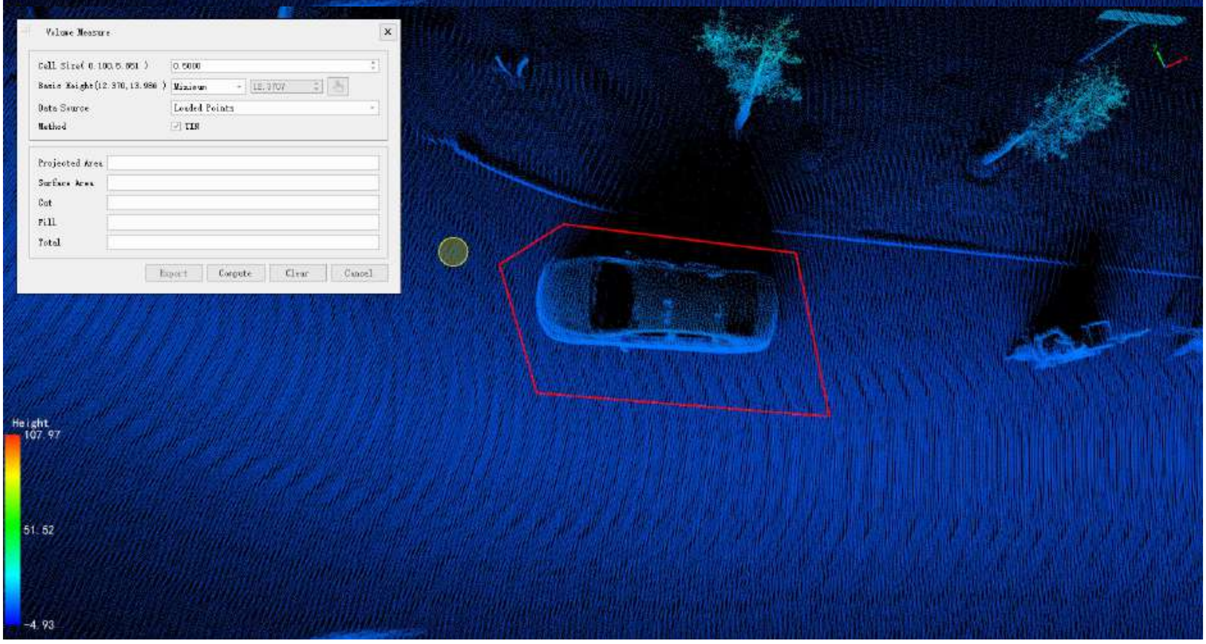
Steps

1.Click on the **volume** in the **measure** item  Button, 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 in the window where the point cloud data is loaded to continuously click on the area where the data is located (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 is drawn with a solid red line in the scene, and the dialog box shown in the figure pops up.

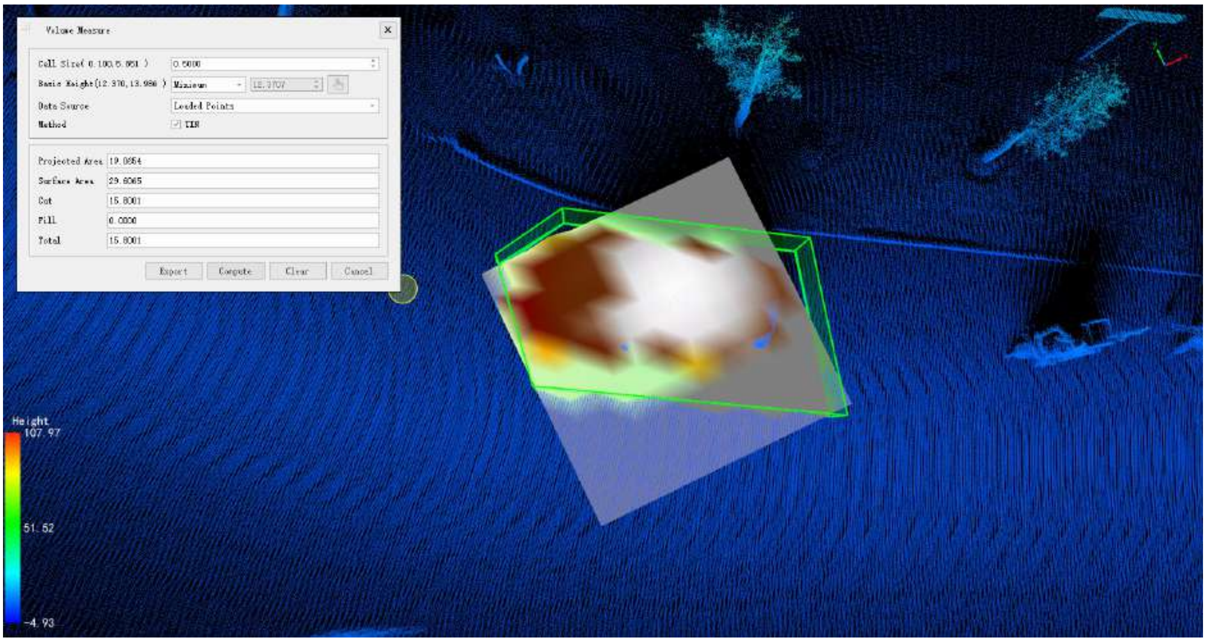




3.Set the cell size.

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

5.Click the calculate button to generate the digging, filling, and filling results. At the same time, the space occupied by the current measurement data is plotted in the scene, 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 filling and digging party.
  - **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 volume measurement data source.
  - **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

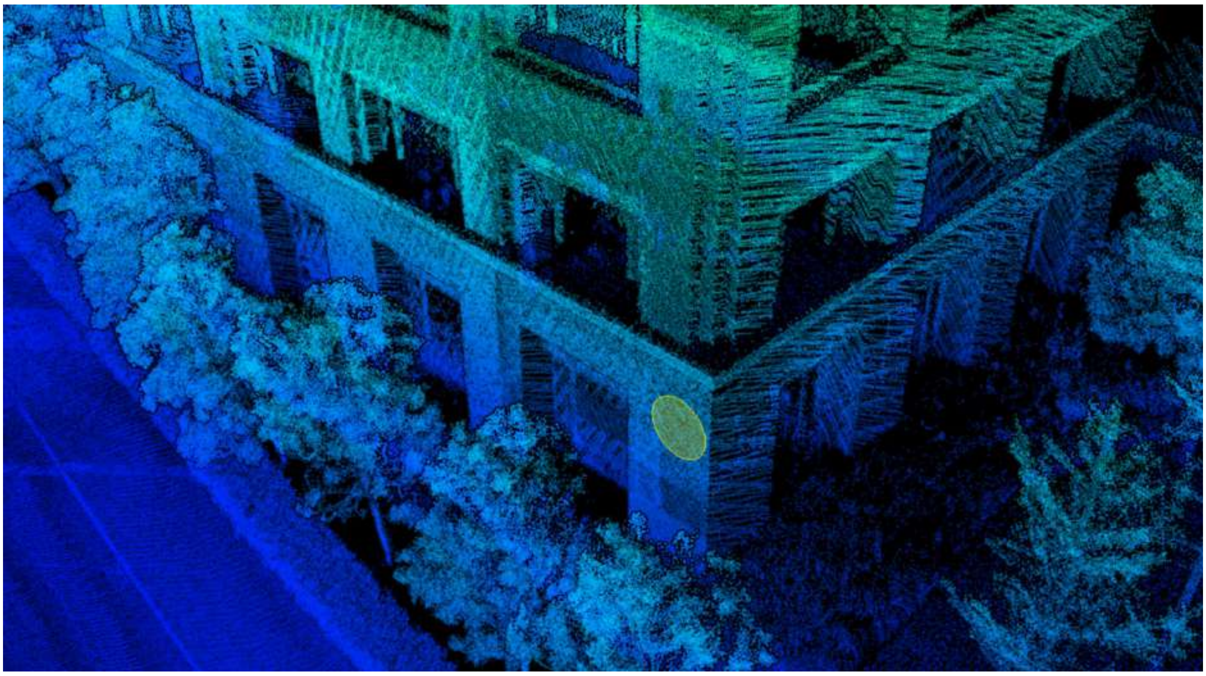
Function description: The mouse hover query tool takes the point cloud within a certain pixel range at the position where the mouse is currently sliding over for feature fitting, which is used to assist in measuring planes, ridges, corners and other local features.

Steps

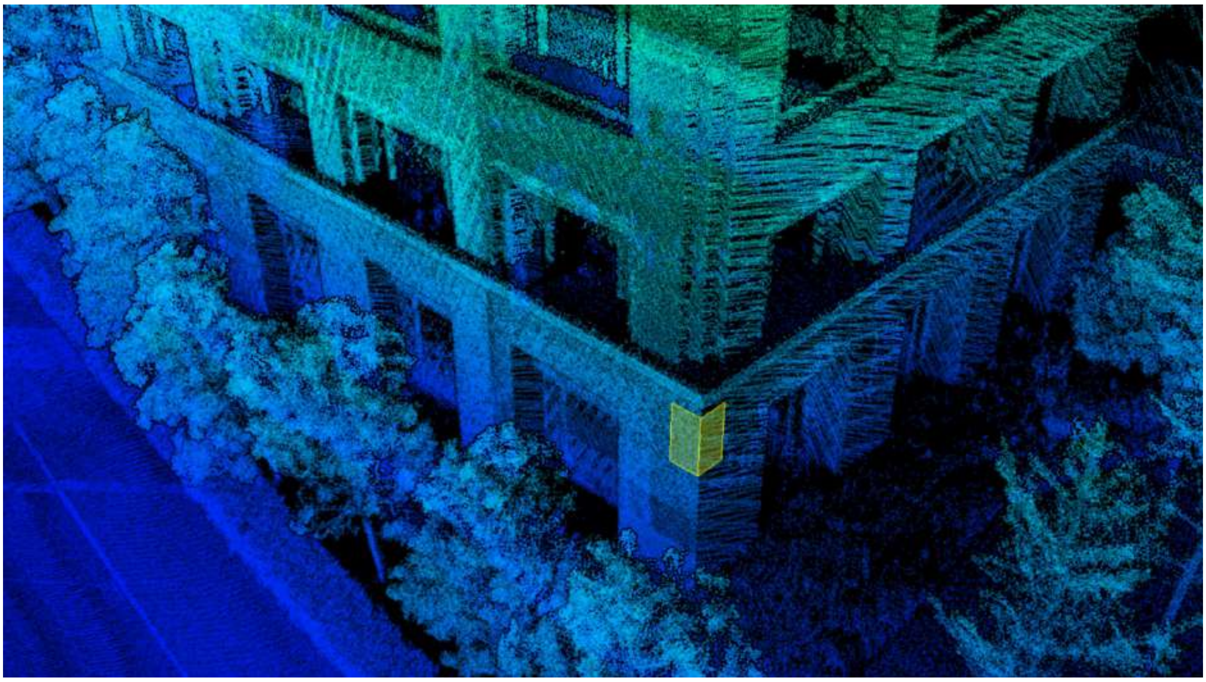
1.Click the **Use mouse hover query** check box in the **Mouse hover** item to turn on the hover tool.

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

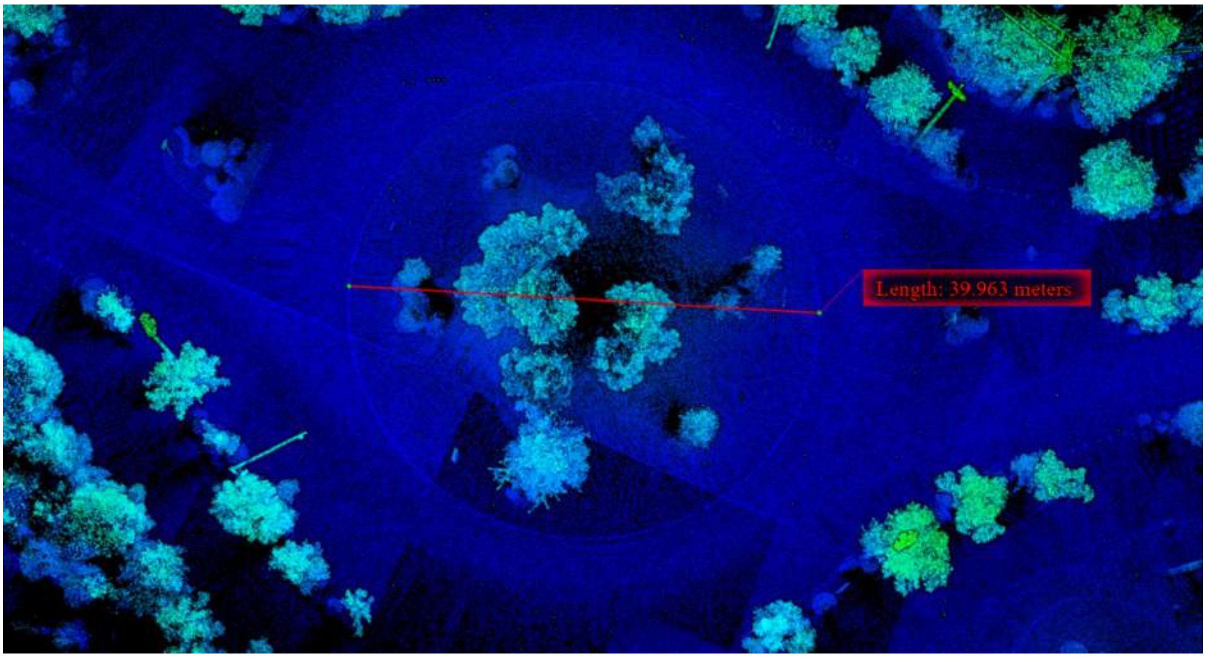
3.The effect of hovering the mouse to detect the plane is shown in the figure.



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.



Selection tool

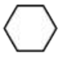
Select and save the point cloud object of the current window.

- Polygon selection
- Rectangle selection
- Spherical selection
- Round selection
- Online selection
- Offline selection
- Plane
- Minus
- On the surface
- Under the surface
- Plane distance setting
- Inner cutting
- External cutting
- Save and cancel
- Cross section

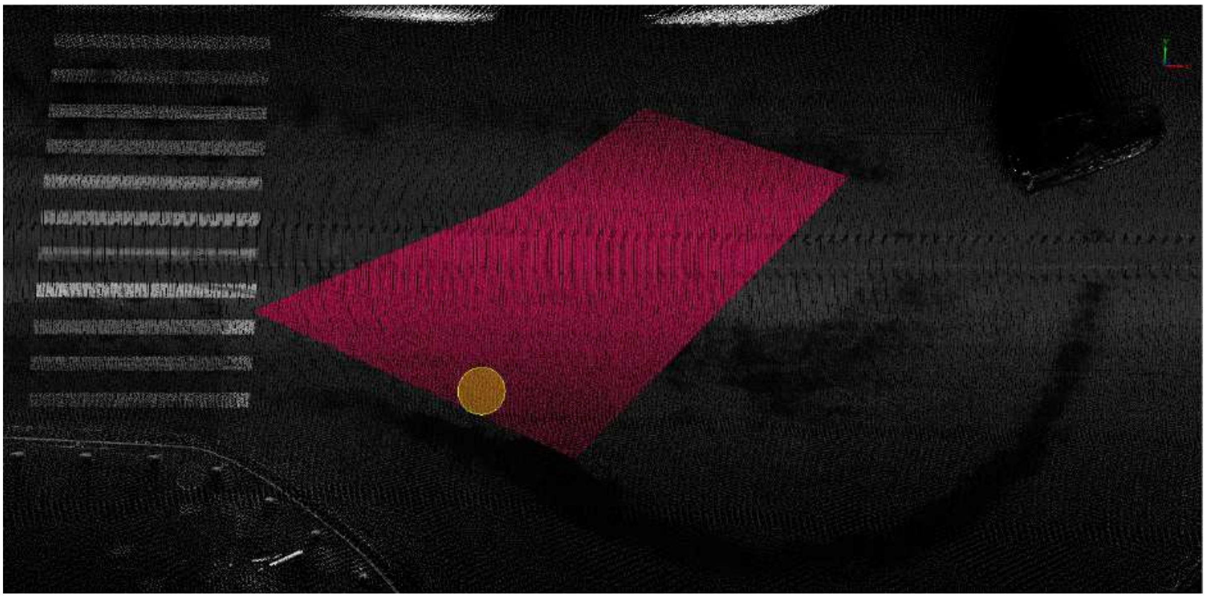
Polygon selection

Function description: Select the point cloud by polygon.

Step

1.Click **to select**  After the function 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, another 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 feature is only for point cloud data.

Rectangle selection

Function description: Select the point cloud in a rectangular manner.

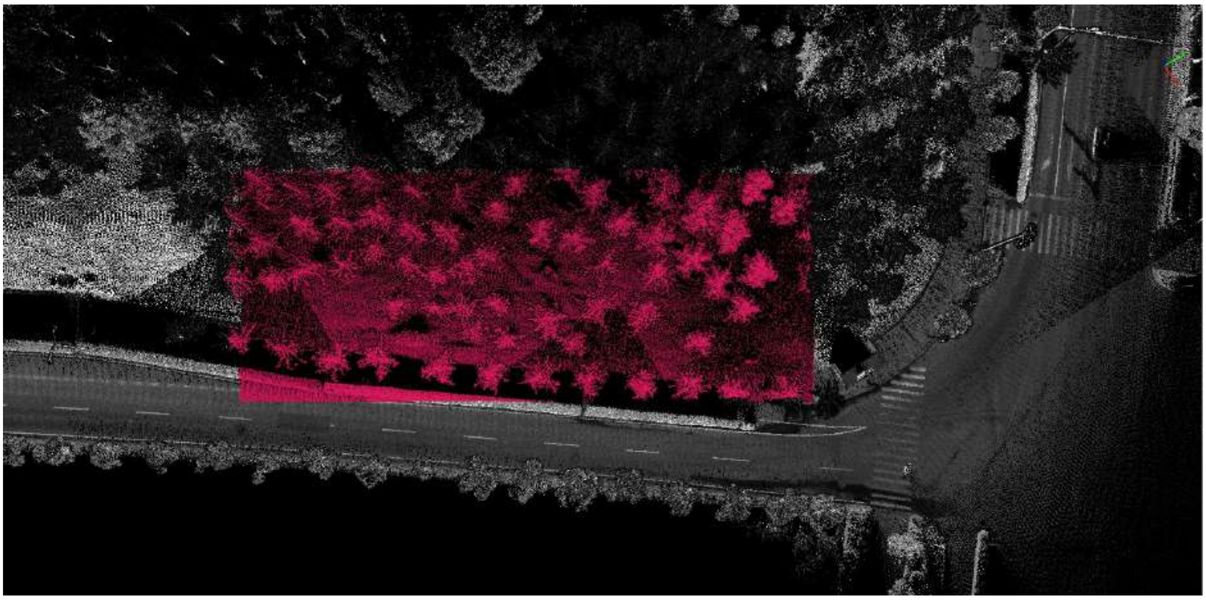
Steps

1.Click **rectangle to select** After the function 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, another selection can be made on the basis of the first selection.

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

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

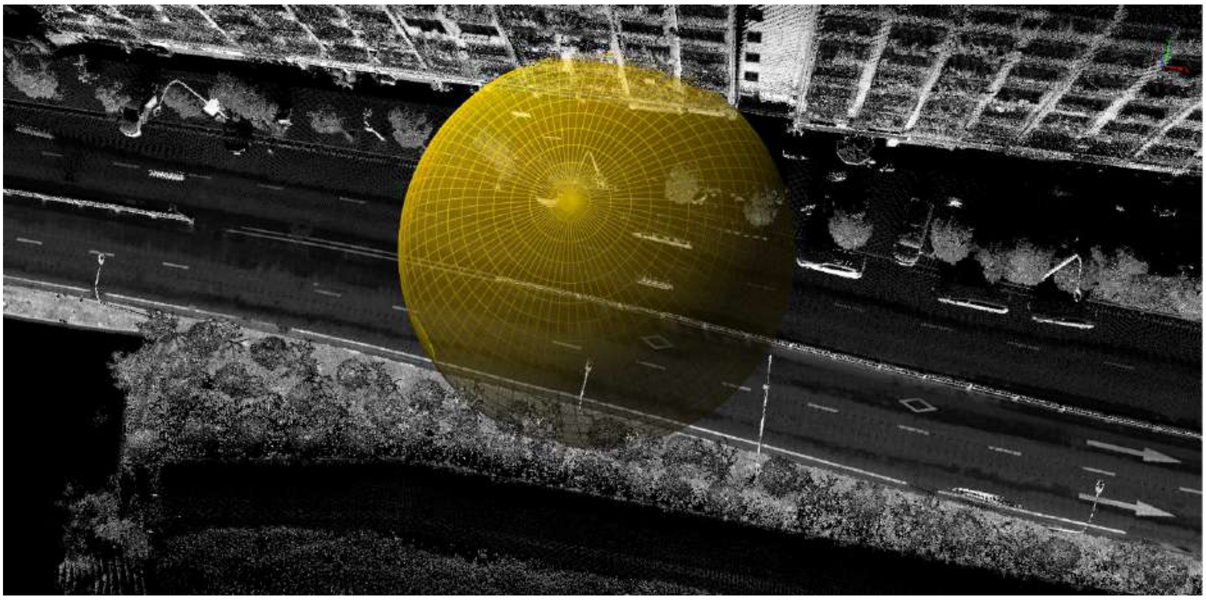
Spherical selection

Function description: Select the point cloud in a spherical manner.

Step

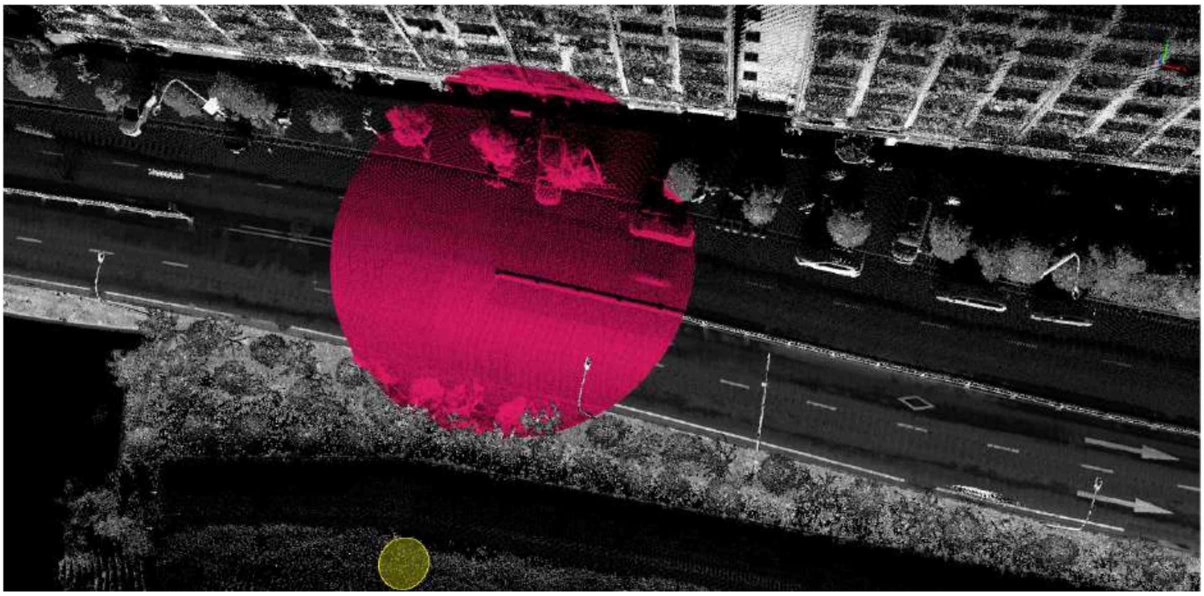
1.Click **to select**  After the function button, the button is in the selected state and the function is activated.

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, another selection can be made on the basis of the first selection.

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

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

Round selection

Function description: Select the point cloud in a circular manner.

Steps

1.Click **circle to select**  After the function 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, another selection can be made on the basis of the first selection.

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

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

Online selection

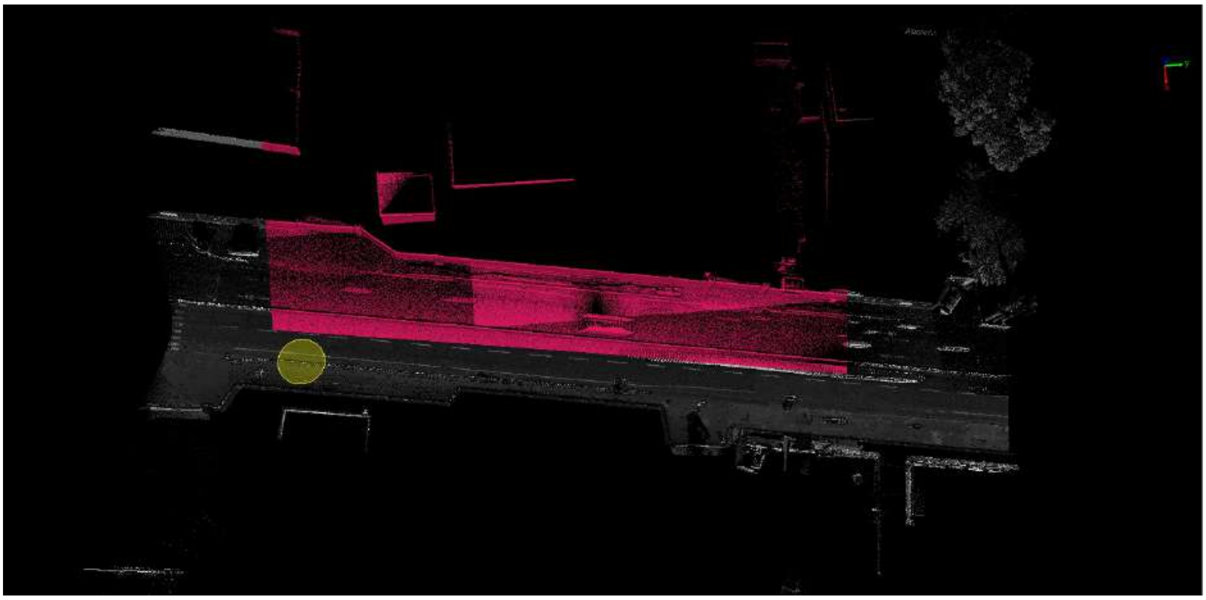
Function description: Select the point cloud according to the online method.

Step

1.Click **Online to select**  After the function button, the button is in the selected state and the function is activated.

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, another selection can be made on the basis of the first selection.

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

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

Offline selection

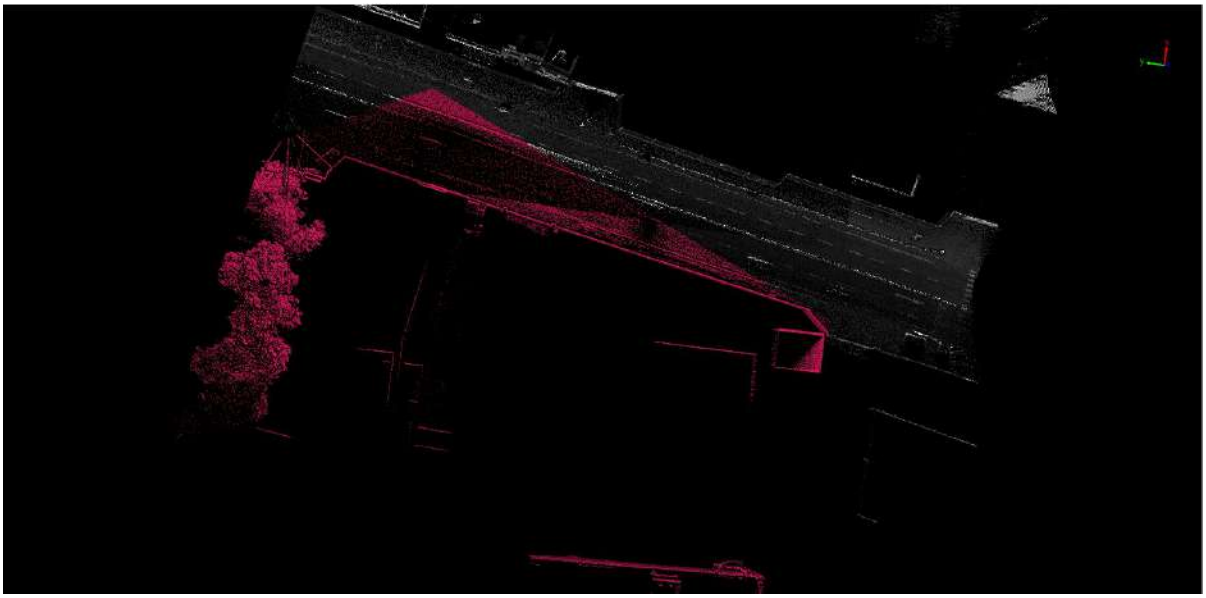
Function description: Select the point cloud according to the offline method.

Steps

1.Click **Offline to select**  After the function button, the button is in the selected state 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, another selection can be made on the basis of the first selection.

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

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

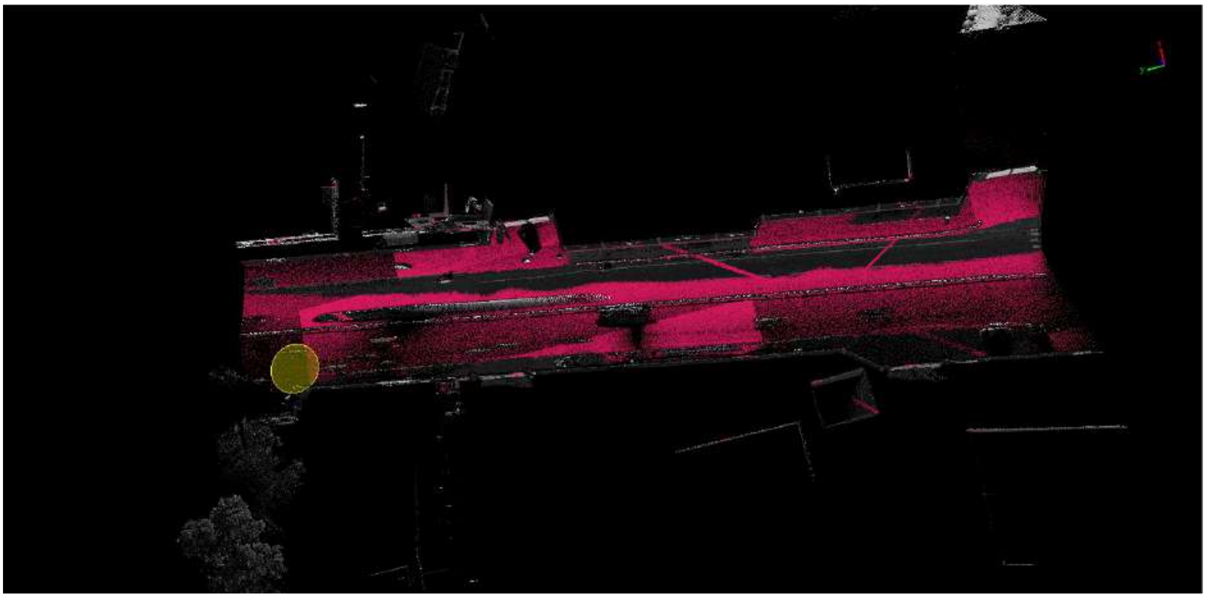
Plane

Function description: Select the point cloud by plane.

Step

1.Click **Plane**  After the function button, the button is in the selected state 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, another selection can be made on the basis of the first selection.

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

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

Deselect

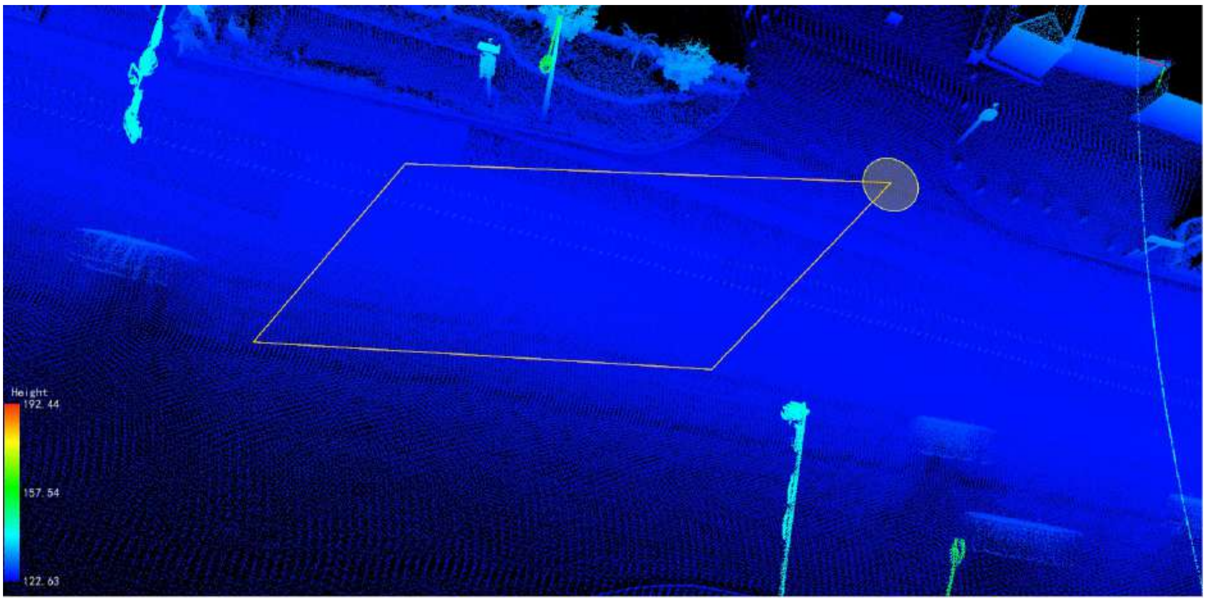
Function description: It works on the current selection tool, indicating the current selection state, and is used to control whether the selected area is added or subtracted. This function cooperates with polygon selection, Rectangle selection, Spherical selection, Round selection, Online selection, Offline selection, plane, on the surface, below It works.

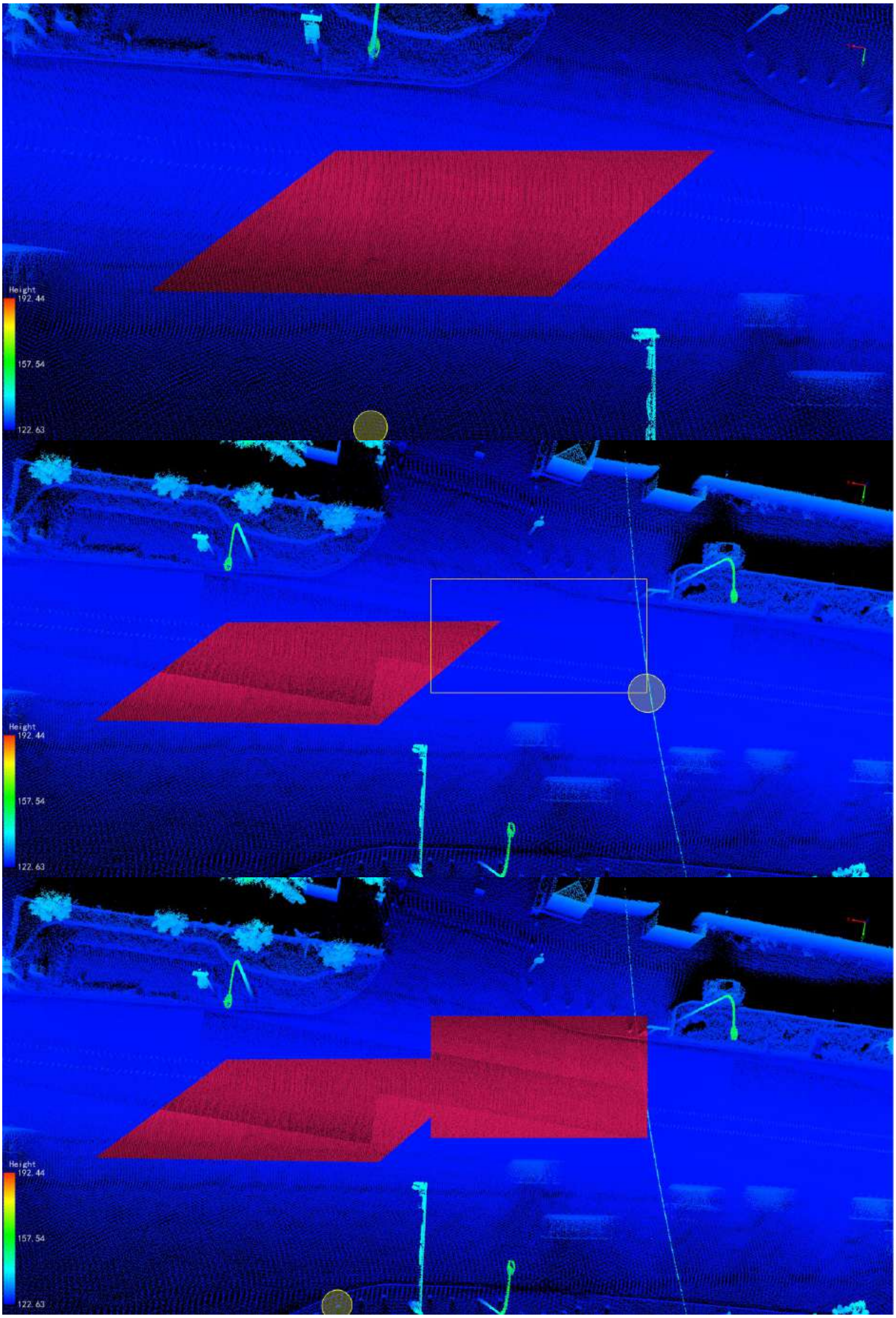
Step

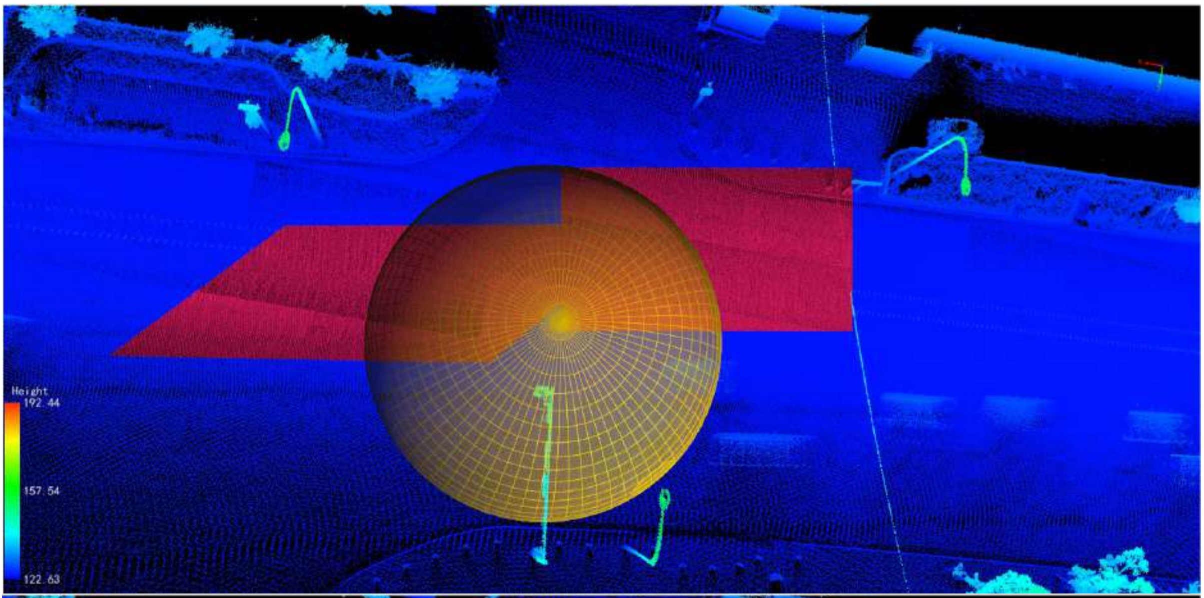
1.Click **to sub-select**  After the function button, the button is in the selected state and the function is activated.

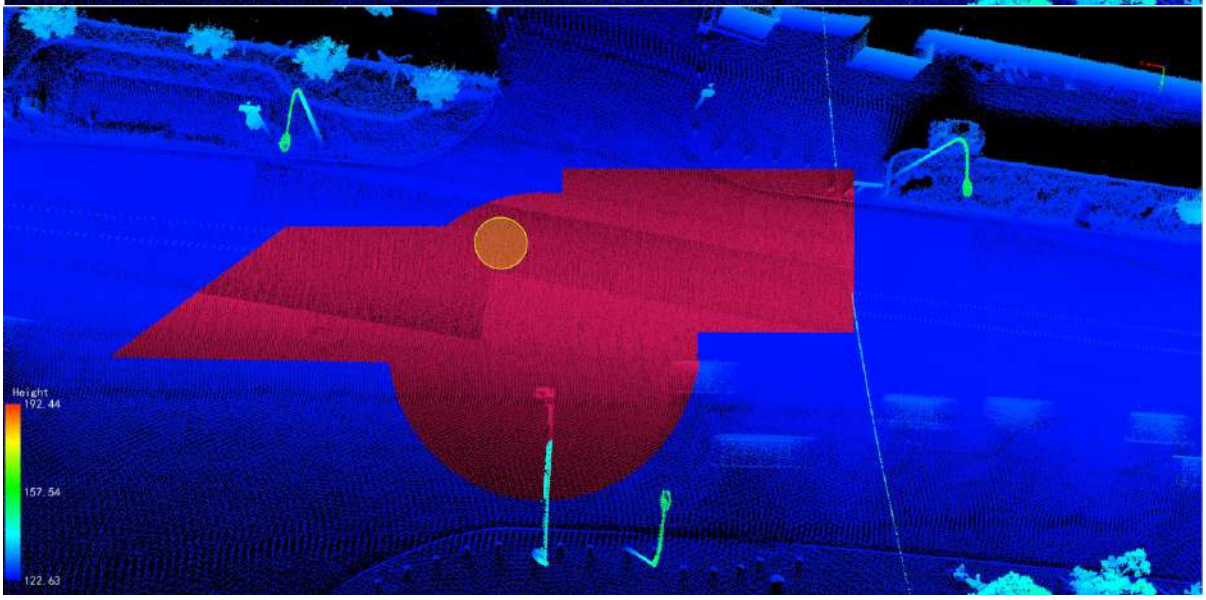
> Note: This function cannot work alone, when a certain selection tool (Polygon selection, Rectangle selection, Spherical selection, Round selection, Online selection, Offline selection, plane, on the surface, below This function can only be activated when it is in the active state.

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.



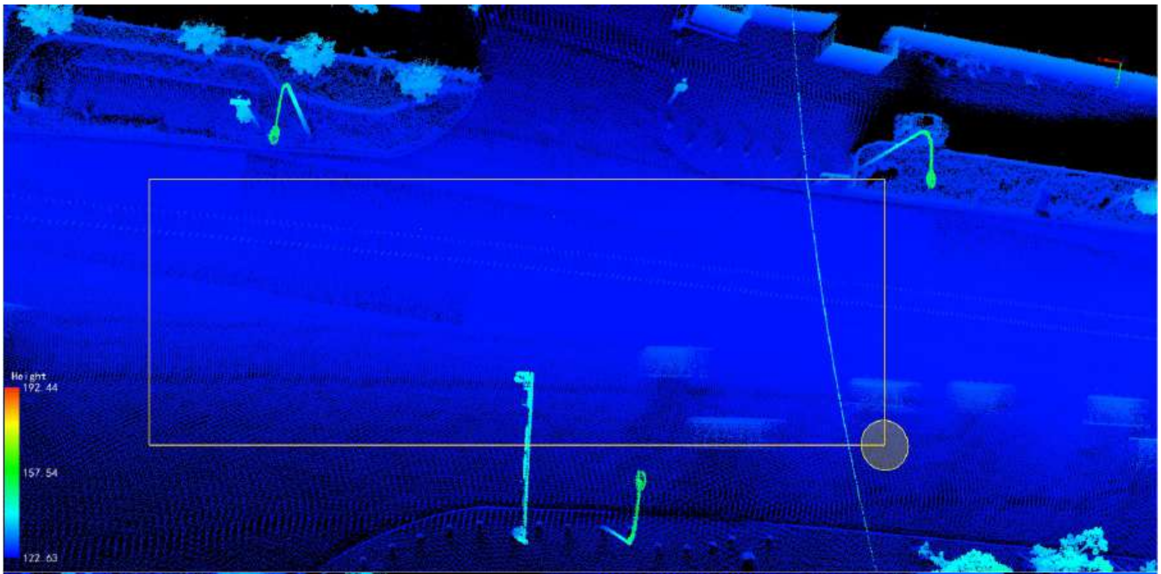

Height
192.44
157.54
122.63
Height
192.44
157.54
122.63
Height
192.44
157.54
122.63

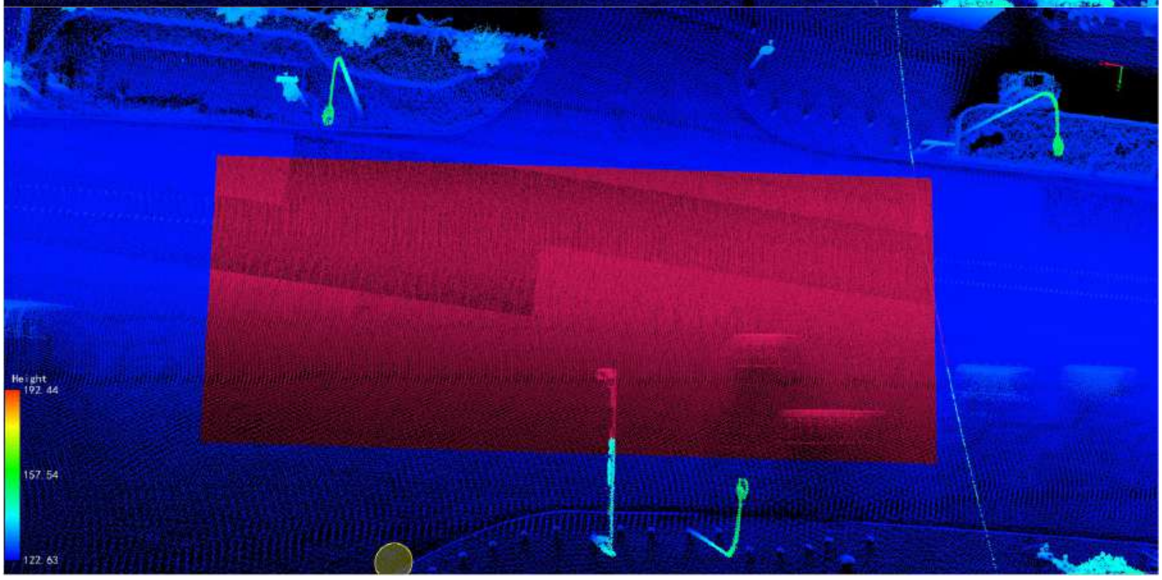




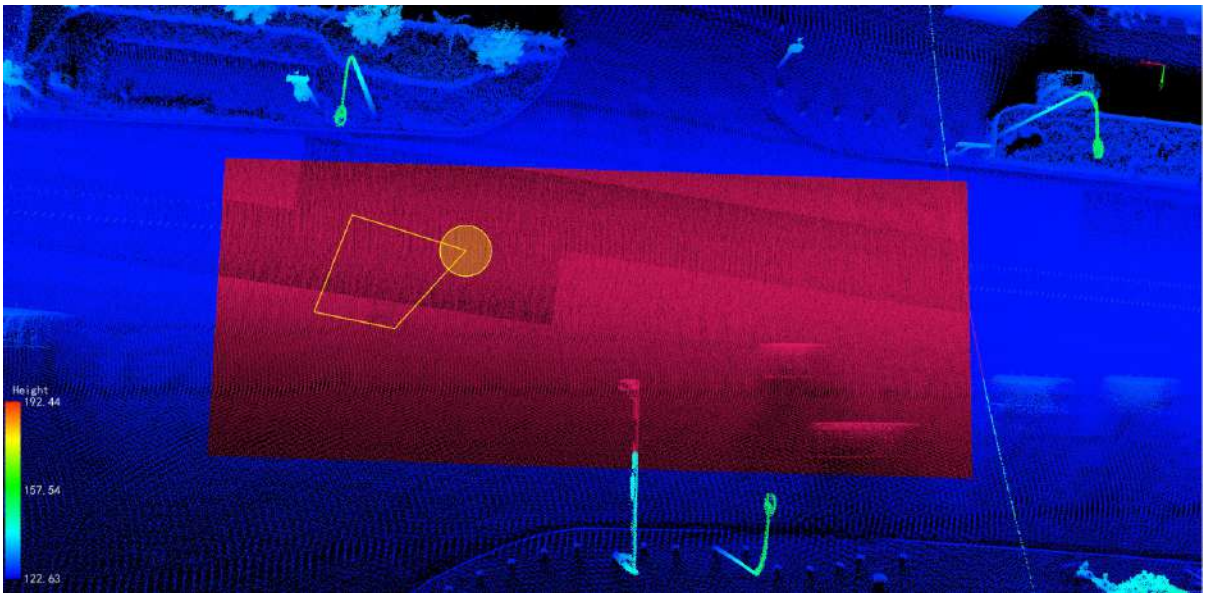
3.Press and select this function. When the button is in the selected state, the selection result is in the minus state, and the selected area is subtracted from the common part of the currently newly selected area as the final selection result.

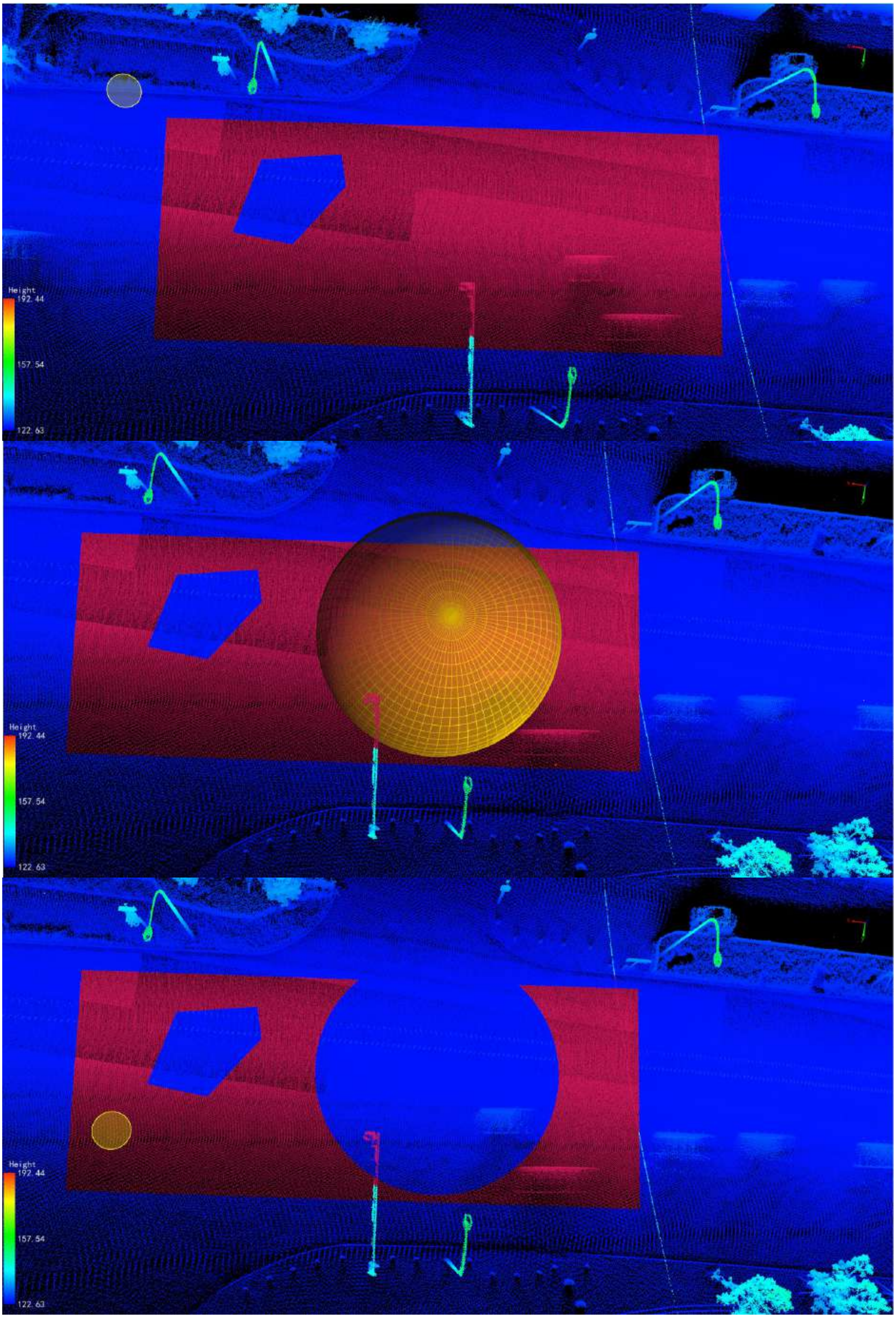
- 3.1 Select an initial selection area in the plus selection state.





- 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 feature is only for point cloud data.

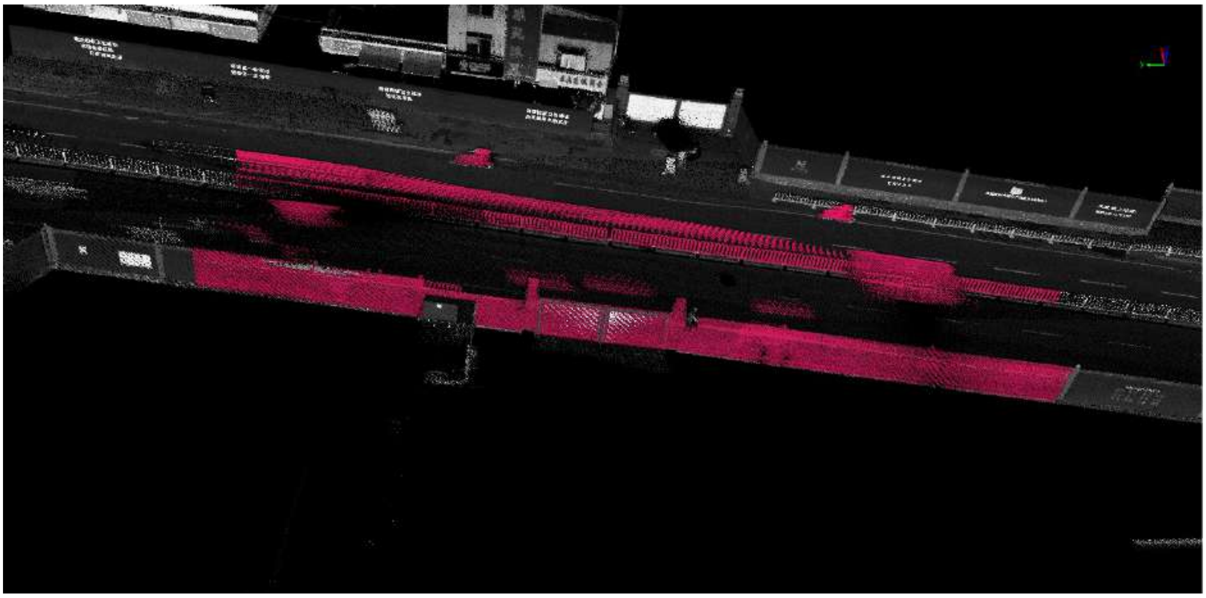
On the surface

Function description: Draw a plane, and select the point clouds above the plane.

Steps

1.Click **on the surface**  After the function button, the button is in the selected state 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, another selection can be made on the basis of the first selection.

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

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

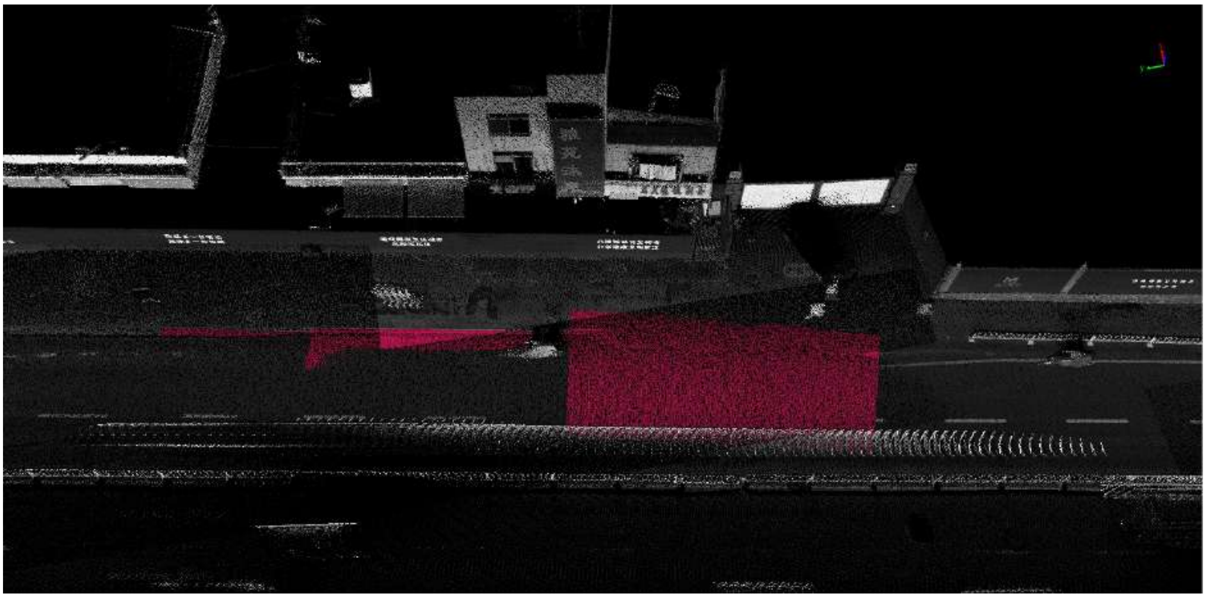
Under the surface

Function description: Draw a plane, and select the point cloud below the plane.

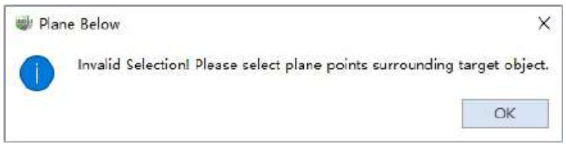
Step

1.Click **Below**  After the function button, the button is in the selected state 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：



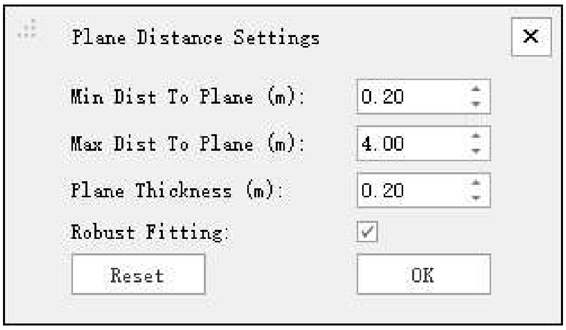
4.After the first selection is over, another selection can be made on the basis of the first selection.

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

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

Plane distance setting

Function description: Set the plane distance.



Description

- **Minimum distance from the plane (meters)**: Set the minimum distance from the plane.
- **Maximum distance from the plane (meters)**: Set the maximum distance from the plane.
- **Plane thickness (meters)**: 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.

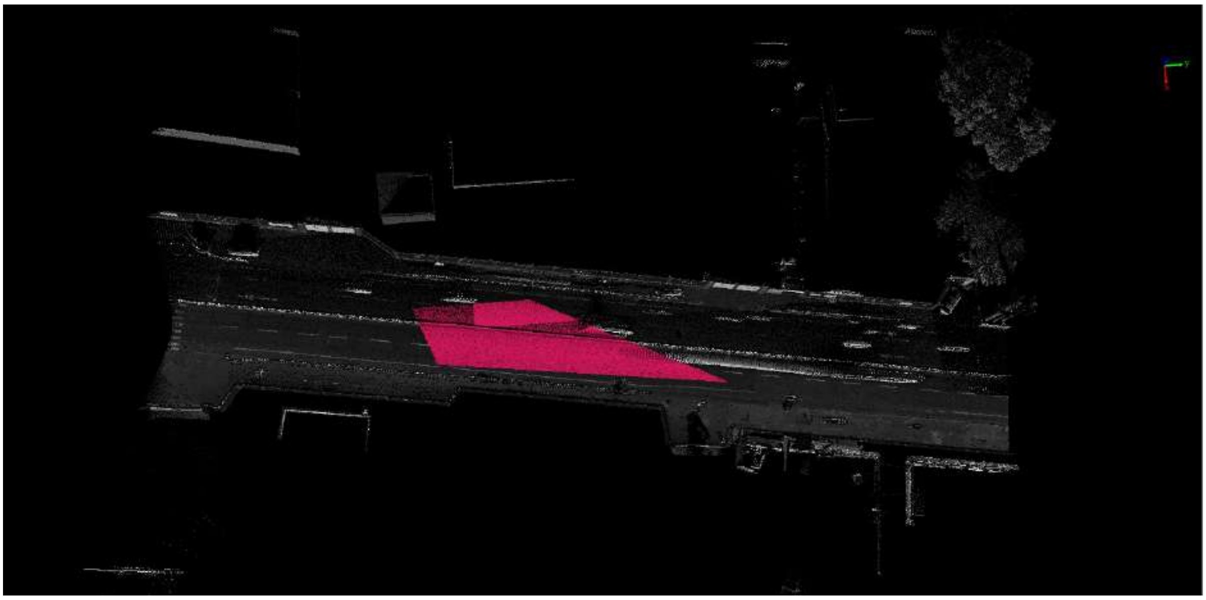
Internal cutting

Function description: According to the currently selected area, crop all point cloud data in the window, select the point cloud in the area, and the point cloud outside the area is hidden.

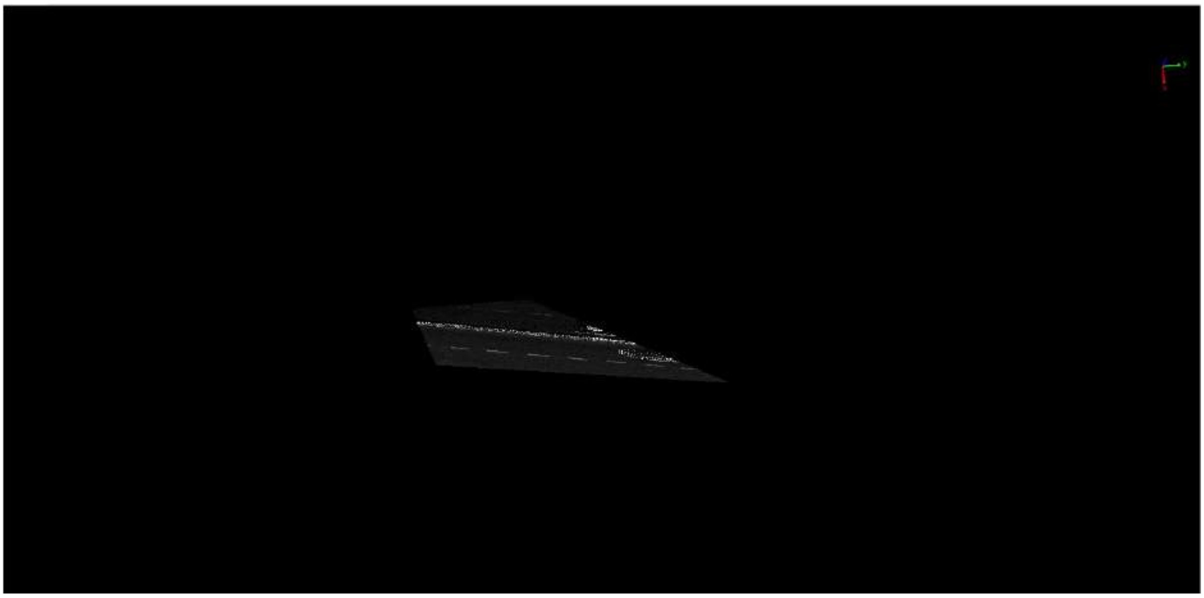
Step

Click **to crop inside**  After the 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 cropping, as shown in the figure.



3.After one cut, the resulting point cloud of the cut can be selected multiple times (polygon selection, rectangle selection, spherical selection, etc.) and cut (external cut).

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 feature is only for point cloud data.

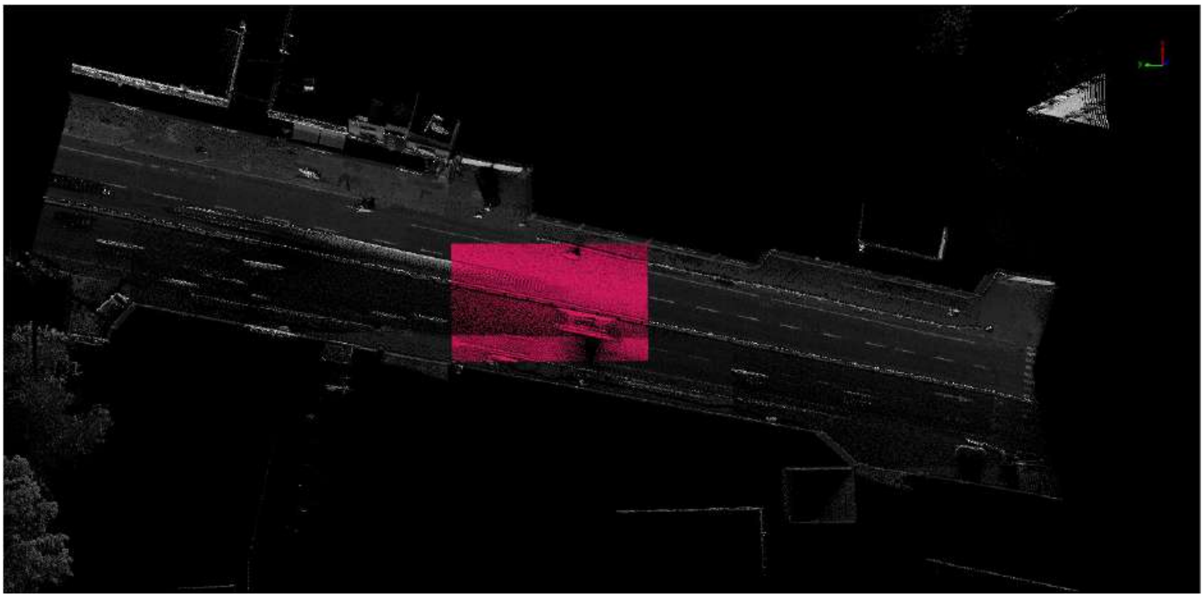
Outer cut

Function description : According to the currently selected area, crop all point cloud data in the window, select the point cloud in the area, and hide the point cloud outside the area.

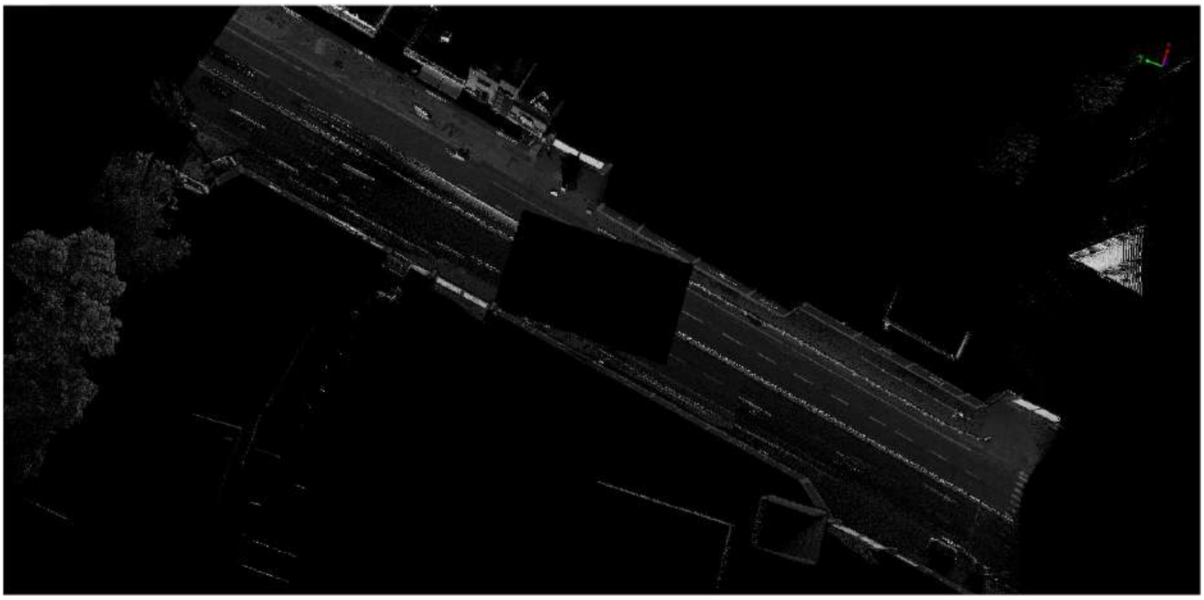
Step

Click **to crop out**  After the 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 external cropping, as shown in the figure.



3.After one cutting, the resulting point cloud can be selected multiple times (polygon selection, rectangle selection, spherical selection, etc.) and cut (including internal cutting).

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 feature is only for point cloud data.

Save and cancel

Save

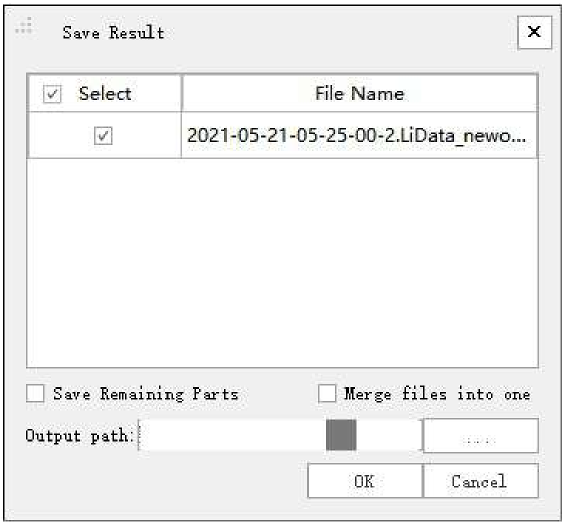
Function description : Save the resulting point cloud after cropping into a new point cloud file.

Step

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 you want 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**  After the button, cancel the result of the last cropping, and the selection area corresponding to the cropping 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.

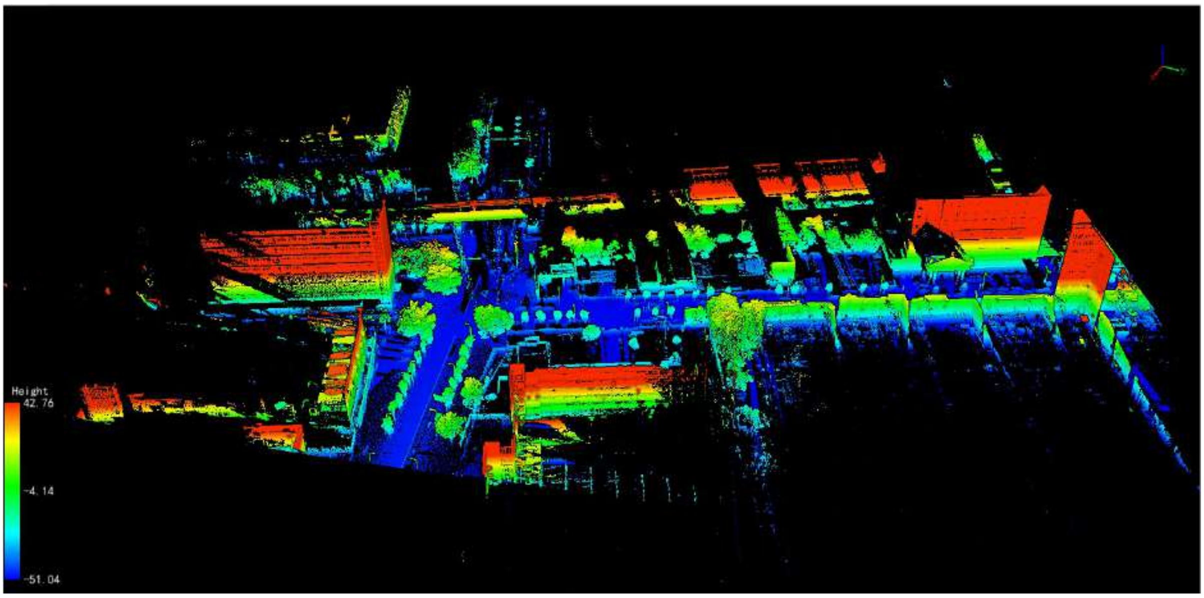
Cross section

Function description: The cross-section tool is based on the bounding box for data cropping.

Step

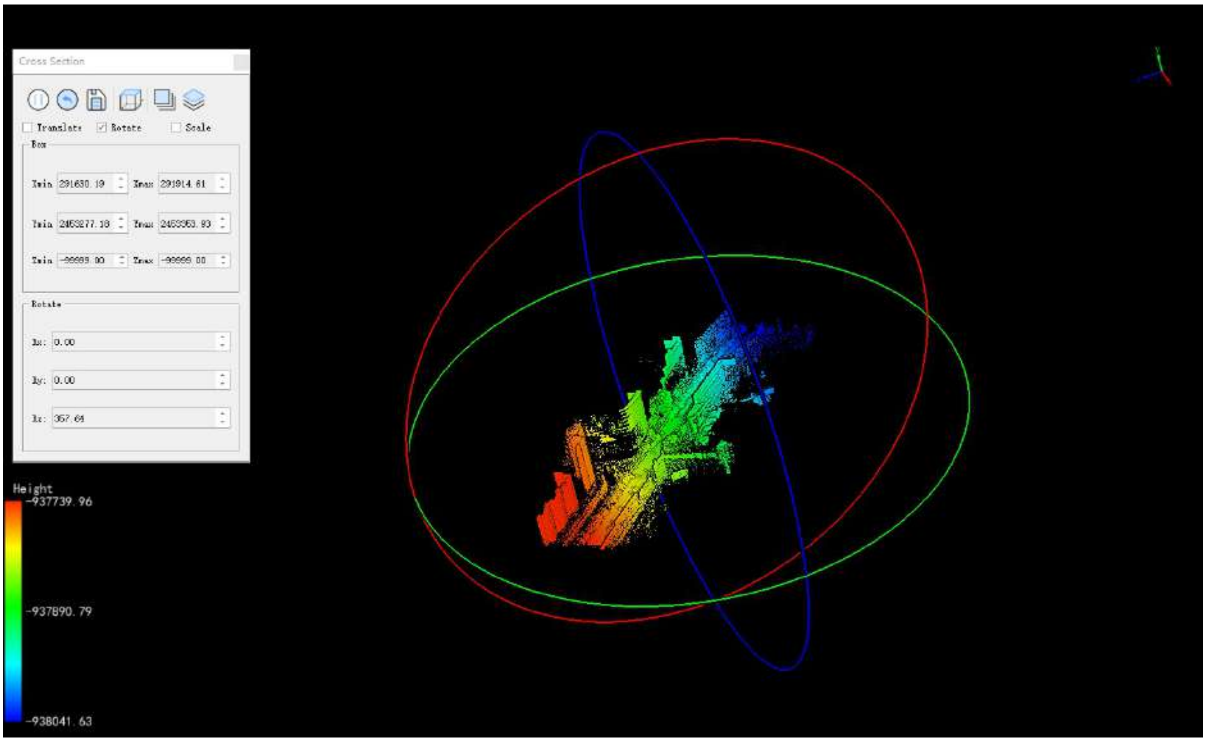
1.Open the LiDAR360 MLS software and load the corresponding project file.

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 (**pan**,**rotate**, **zoom**) 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 enclosure (green square) in the view and adjusts the position. You can adjust the zoom of the enclosure 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 **pan** 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).



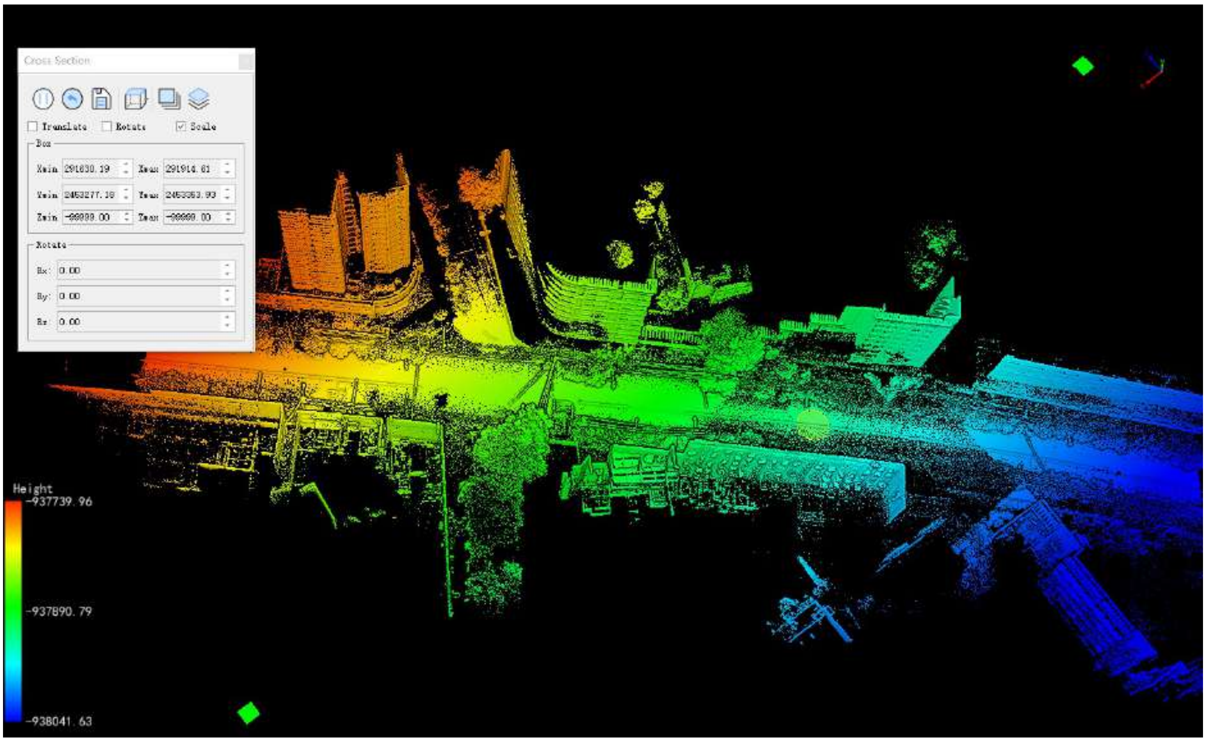
Adjust the enclosure

Parameter setting

- **Translate**: Controls whether the enclosure can be moved.
- **Rotate (Rotate)**: Control whether the enclosure can be rotated.
- **Scale**: Controls whether the enclosure 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 surround box can be precisely adjusted.

4.(Optional) Click  The button surrounds the box and returns to its initial state.

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



Cut according to the z axis

6.Click  The button saves the cropped point cloud.

7.Click  The button pauses the cross-selection.

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

Roaming tool

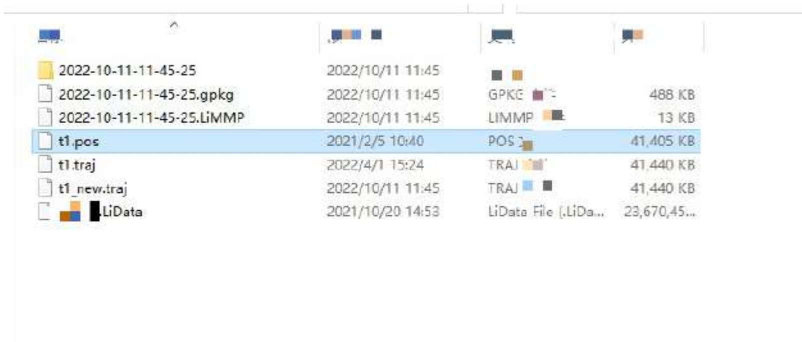
The roaming tools mainly include track roaming and camera roaming. Just switch the top menu bar to **Tools**.

Track roaming

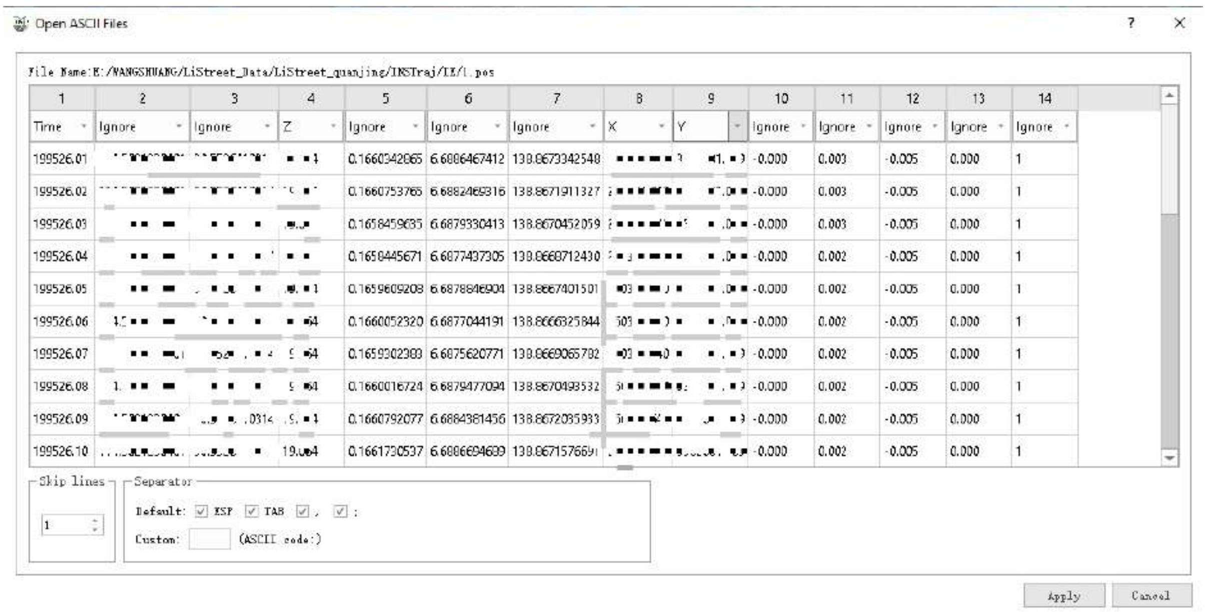
Function description : Based on the roaming of the data trajectory, the camera can be controlled to move according to the trajectory points, and the process of imitating the vehicle collecting data in the real scene can be realized.

Step

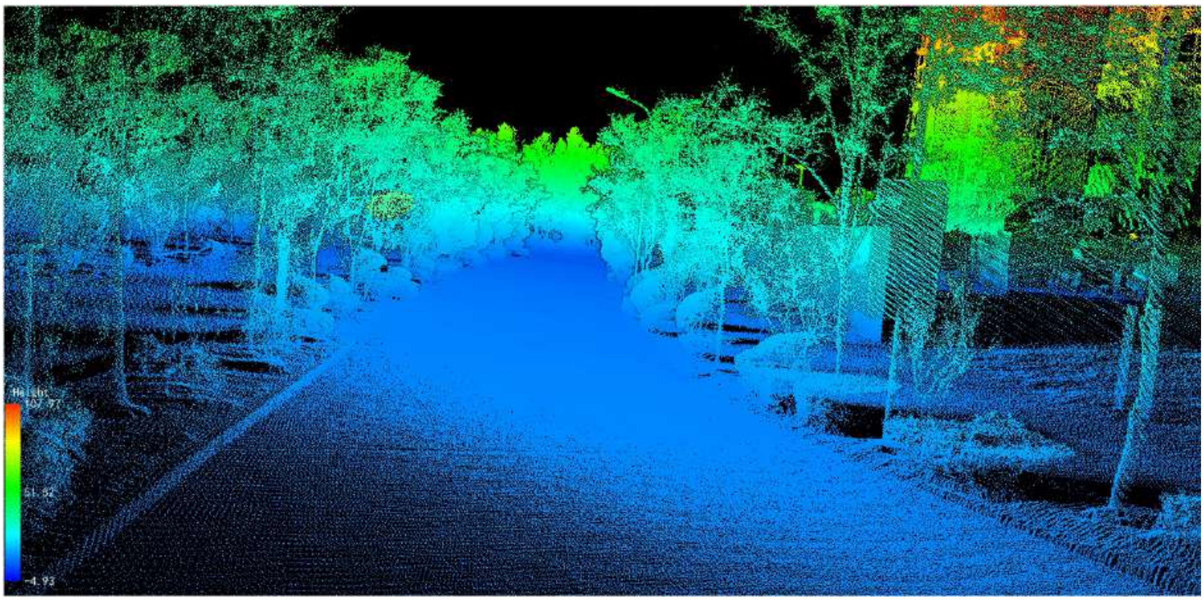
1.Click **Track to roam** button, the track selection interface pops up, which supports the import of track files in multiple formats, and generally selects the solved *by default.pos* format is enough.



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.



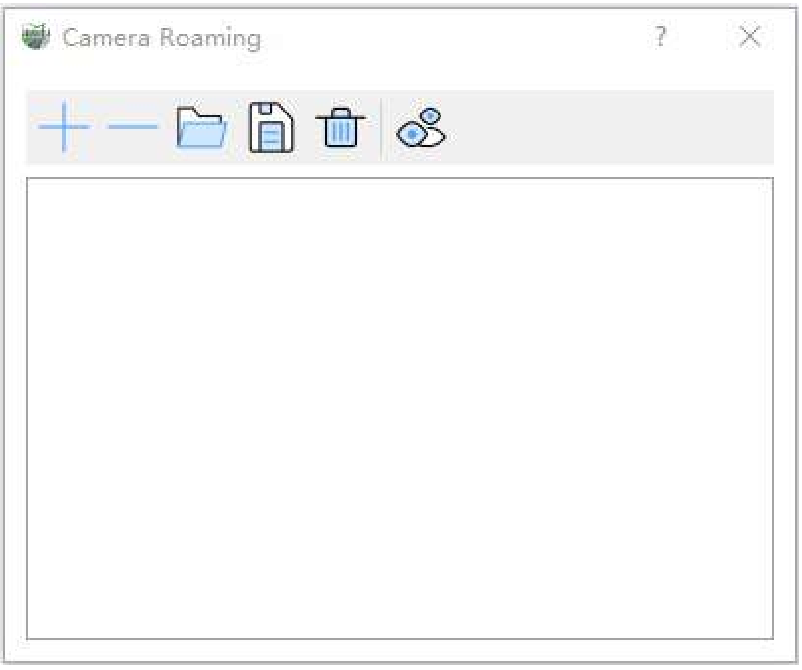
4.The user can click **track to roam again** Button to end the track roaming operation.

Camera roaming

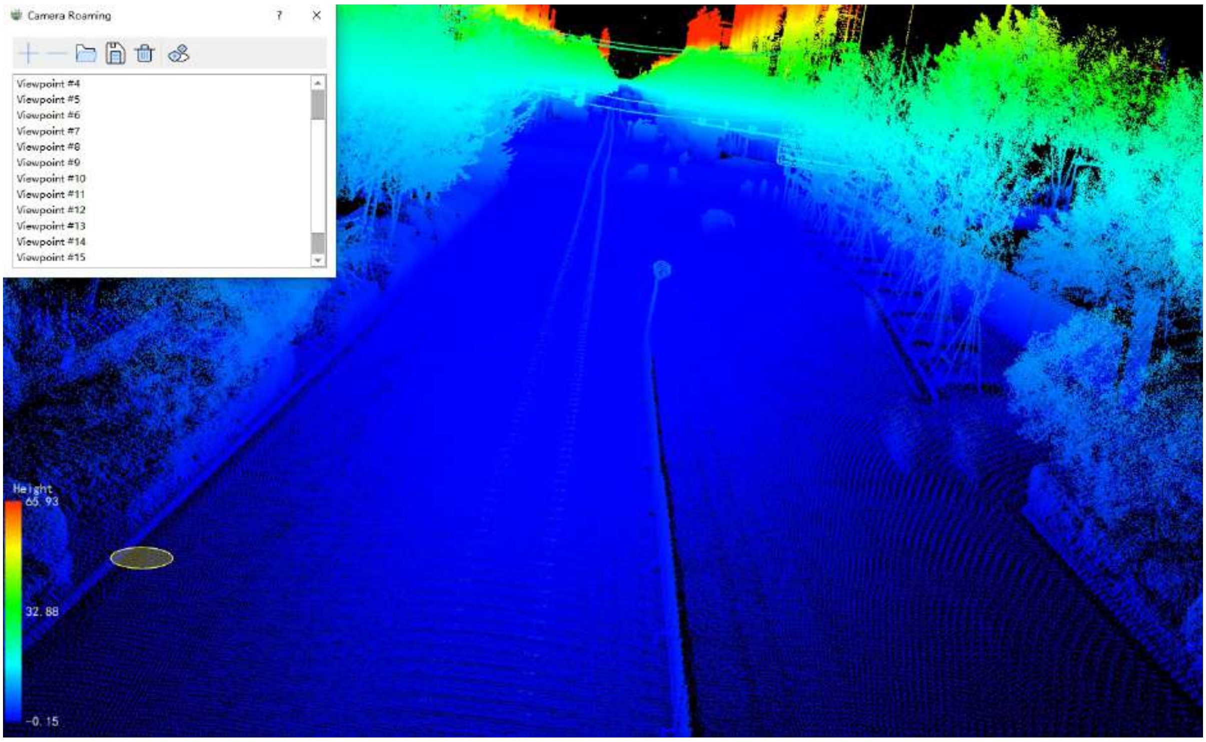
Function description : Based on the roaming of user-defined viewpoints, the camera can be controlled to move according to the viewpoints, and the process of imitating people observing data in real scenes is realized, which enhances the immersion and presentation of the scene.

Steps

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

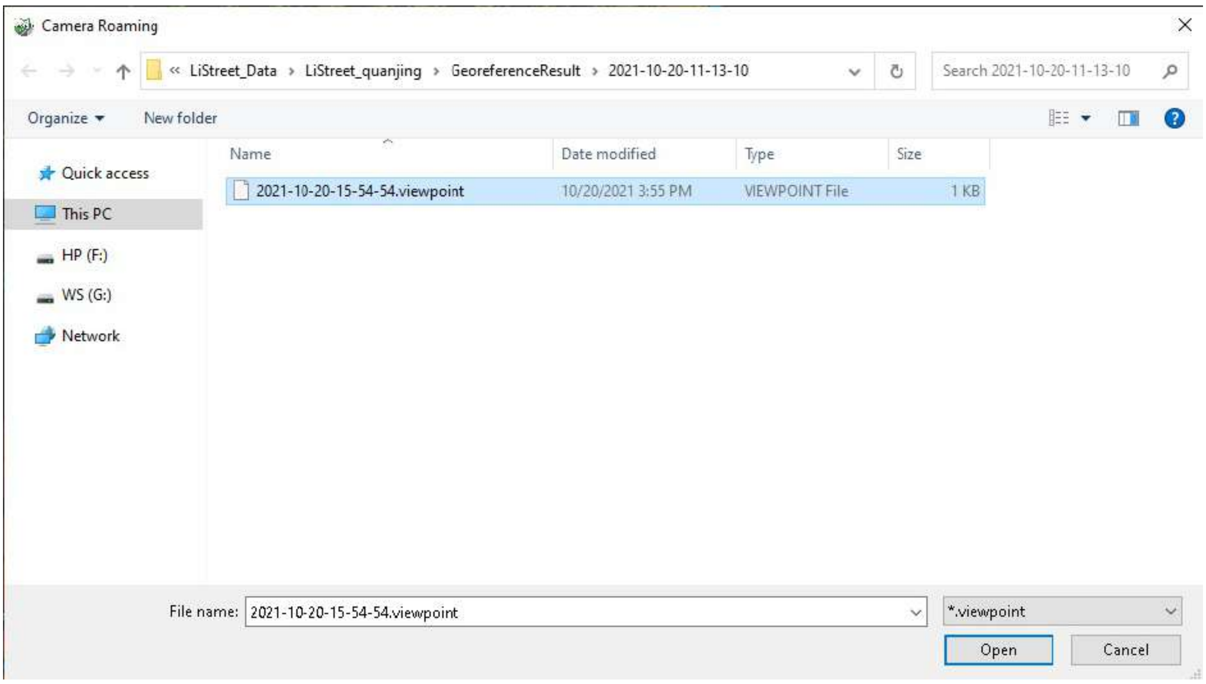


2.By adjusting the appropriate viewing angle, the user clicks **To increase the viewpoint** Button, add the current viewpoint position to the roaming path, as the keyframe of the roaming, the new viewpoint will appear in the viewpoint list box.

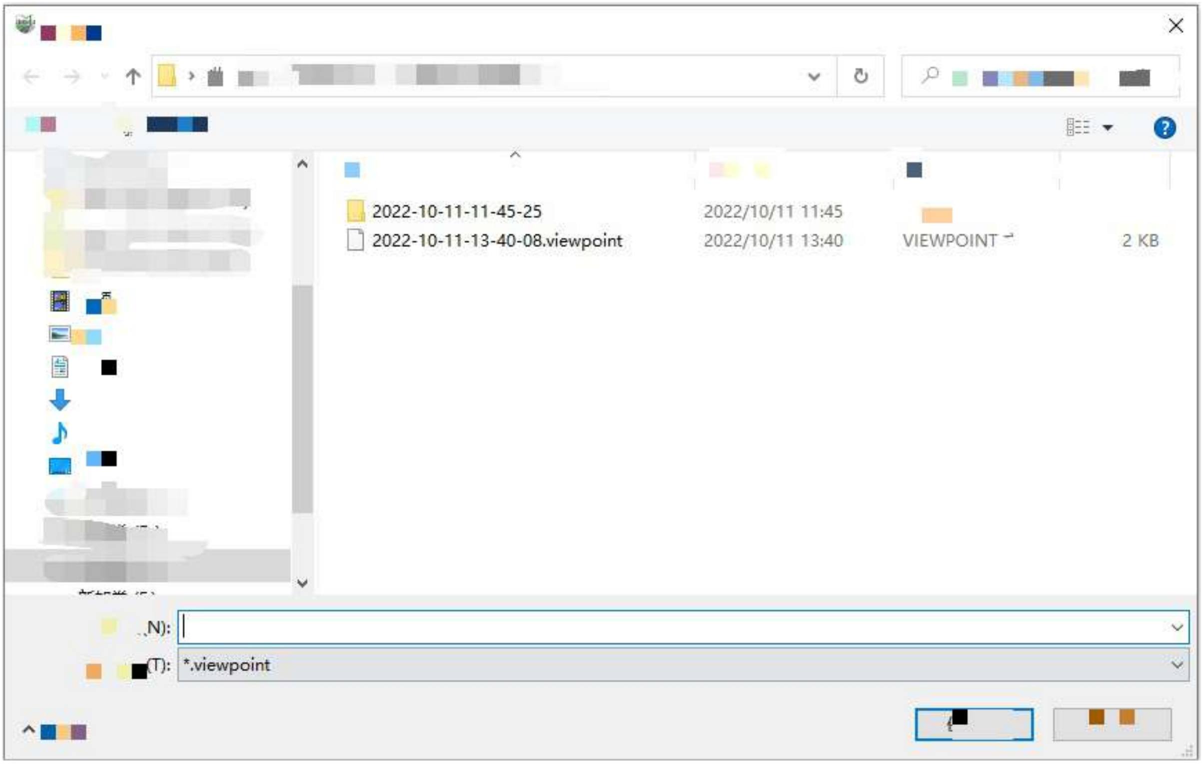


3.Users can also click **to delete the viewpoint (optional)** Button to delete unwanted or problematic viewpoints.

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

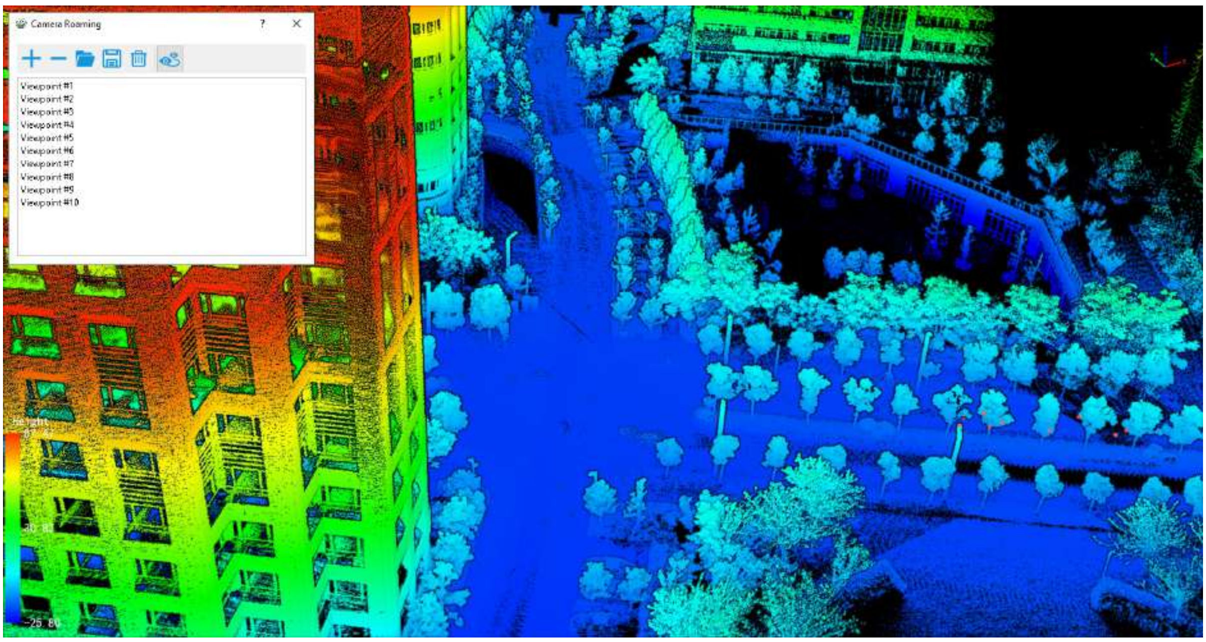


5.After adjusting the viewpoint, the user can click **to save the viewpoint file (optional)**  button, save all viewpoints in the viewpoints list box as keyframe information into a viewpoints list file.



6.Users can click **to delete all viewpoints (optional)**  Button, clear all viewpoints in the viewpoints list box.

7.After confirming all viewpoints, the user clicks on **Camera 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 on **Camera roaming** in the camera roaming box again  button to stop roaming.

Import Shape vector

Function description: The dialog box layer for importing vectors is shown in the figure below. The main function is to import shp files and add them to the software. The following will be described in detail.

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.

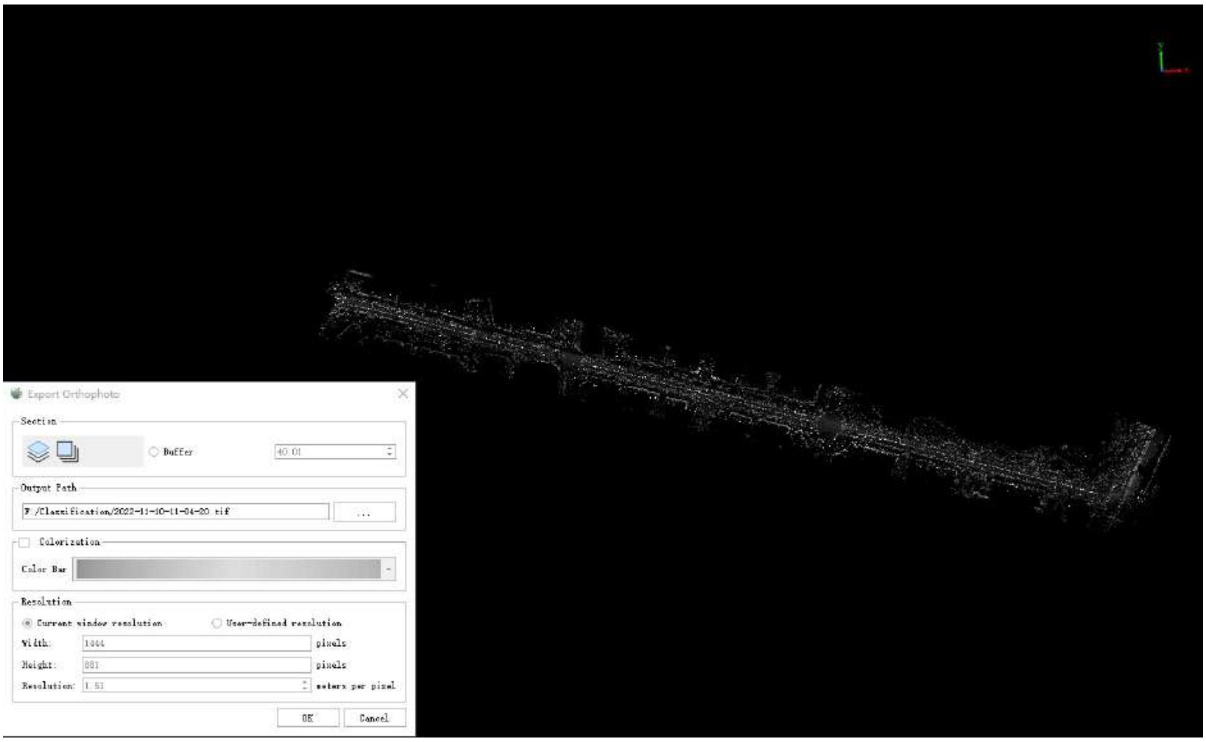
Export tool

Function Description: The export tool can 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 image
- Point cloud format conversion
- Projection conversion
- Export vector

Export orthophoto image

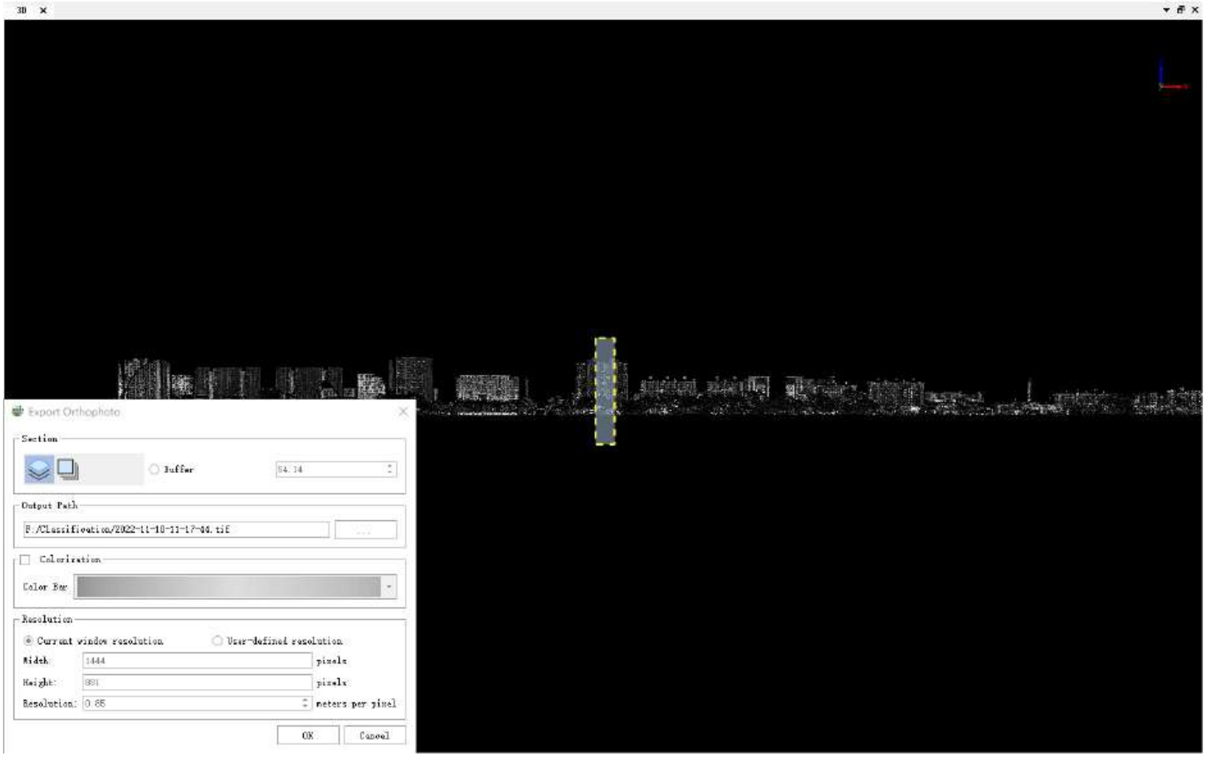
1.Click **Orthophoto image**  button, the export orthophoto image dialog box will pop up, as shown in the figure, the export orthophoto image can be combined with the horizontal cross-section tool and the vertical cross-section tool, and the exported orthophoto image can be opened and measured in other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



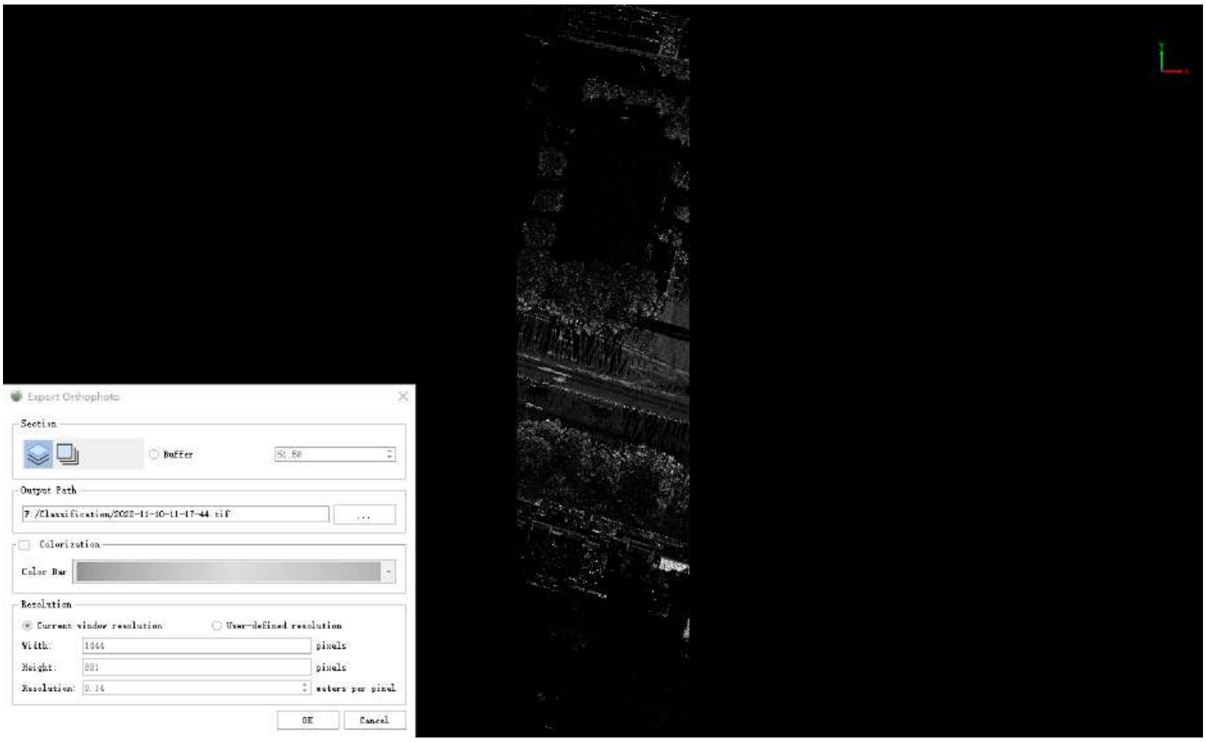
Horizontal section tool

1.Click  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.

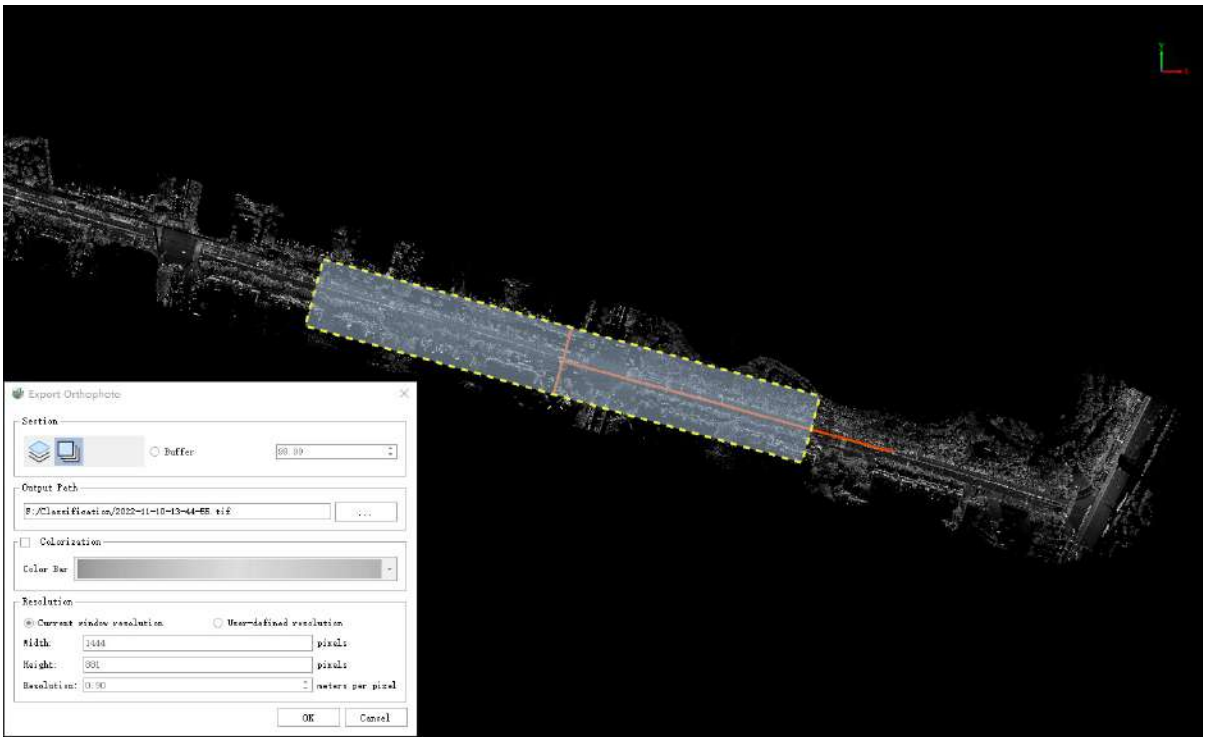


4.If you need to fix the cross-section buffer, you can click the Buffer radio box and adjust the value of the buffer.

Vertical section tool

1.Click  Button, the 3D scene automatically switches to the top view

2.The mouse interactively 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.

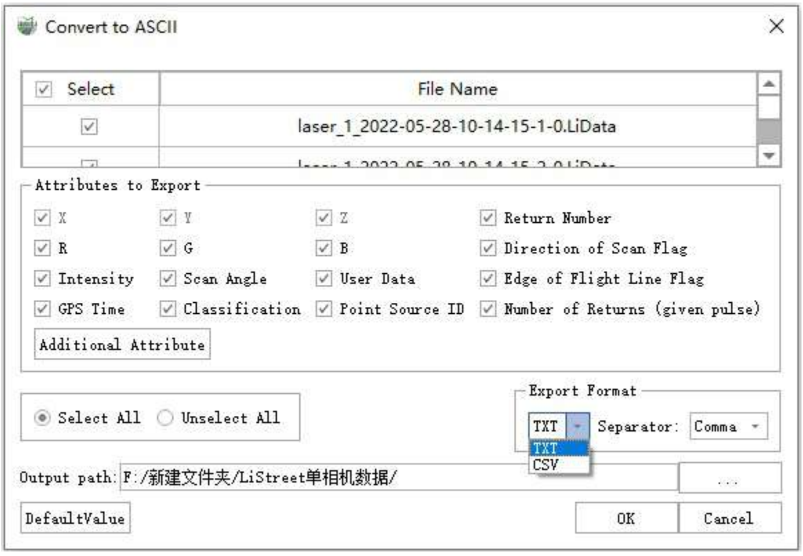
Point cloud format conversion

Function description : Point cloud conversion supports the 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 **to 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



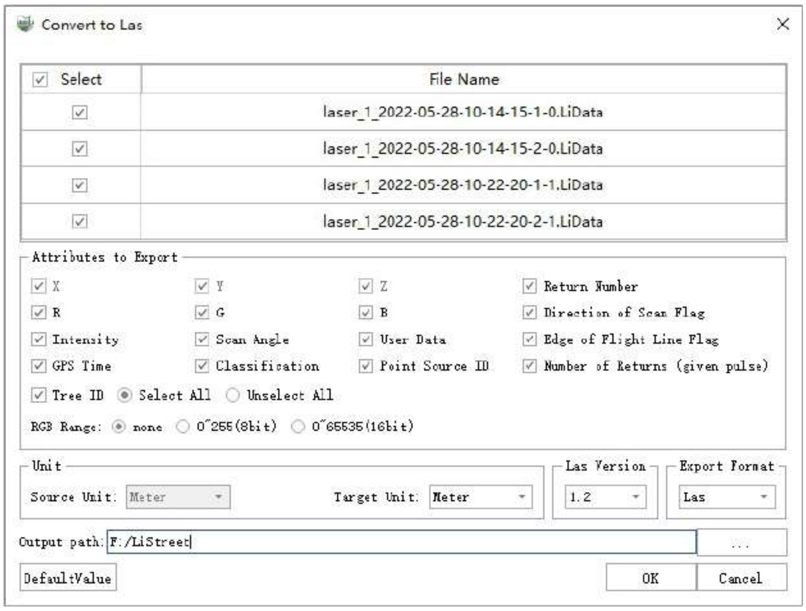
Parameter setting

- **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.
- **Exported attributes**: 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 executed.

Convert to Las

Click **to 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

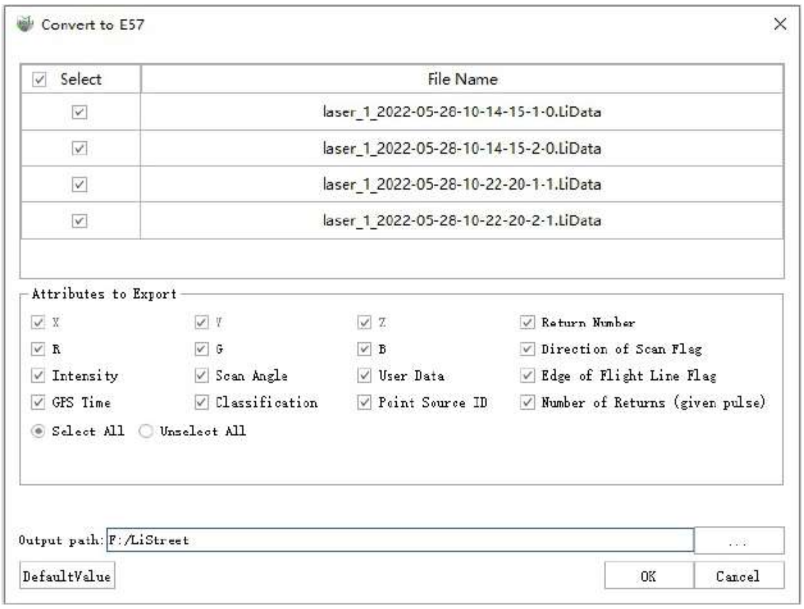
- **Input data**: Select the LiData file you want to export. The input file can be a single point cloud data file or multiple data files.
- **Exported attributes**: Support all Las attribute types.
- **Color range**: There are three RGB ranges to choose from.
  - **None**: Output according to the original 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 **to 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



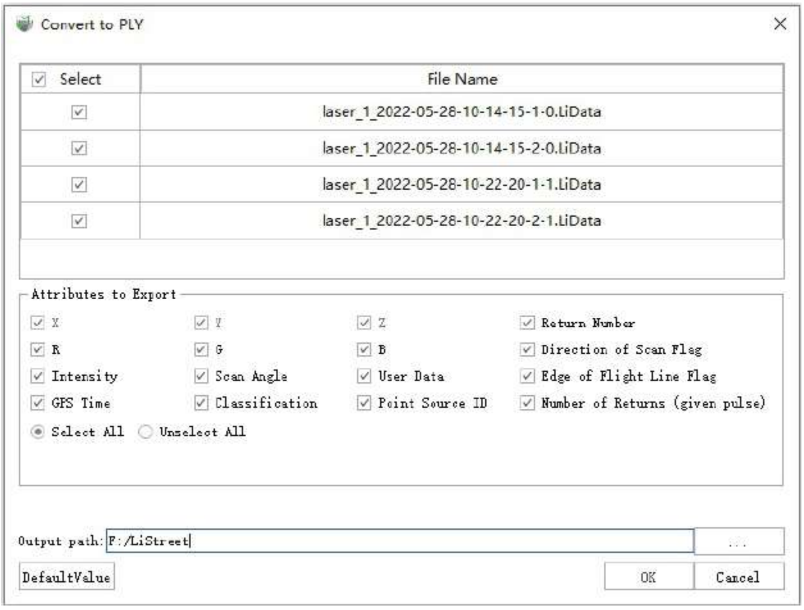
Parameter setting

- **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.
- **Exported attributes**: 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 **to 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



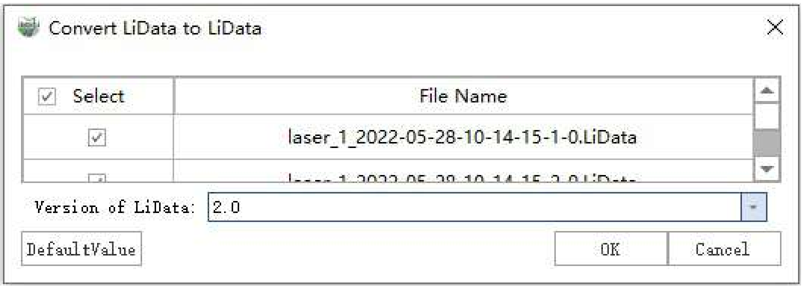
Parameter setting

- **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.
- **Exported attributes**: 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



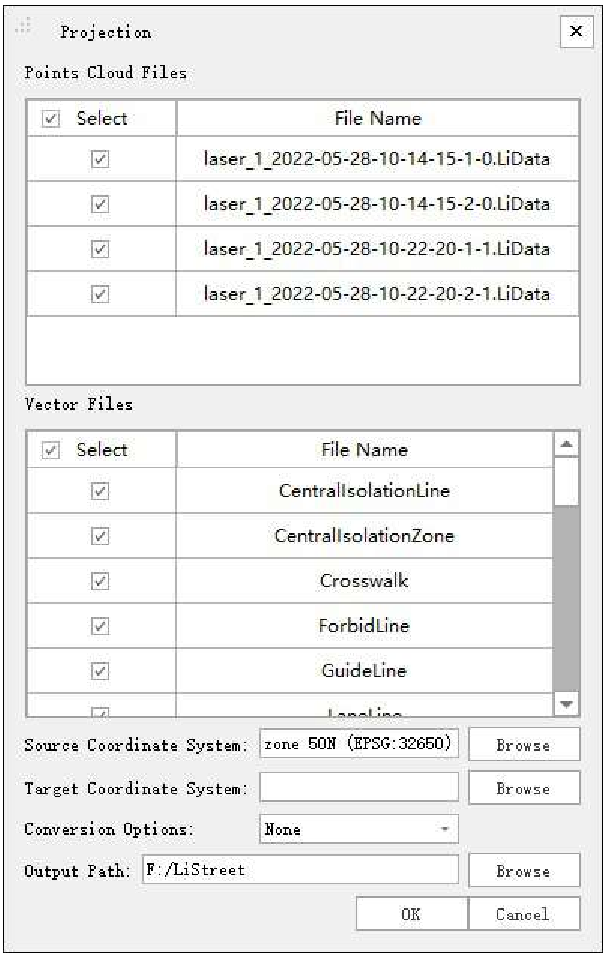
Parameter setting

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

Projection conversion

Function description : The dialog box layer of the projection function is shown in the figure below, which is used to project the coordinate system of the point cloud and vector data in the software. It will be described in detail below.

1.Click **Point cloud format conversion**  Button



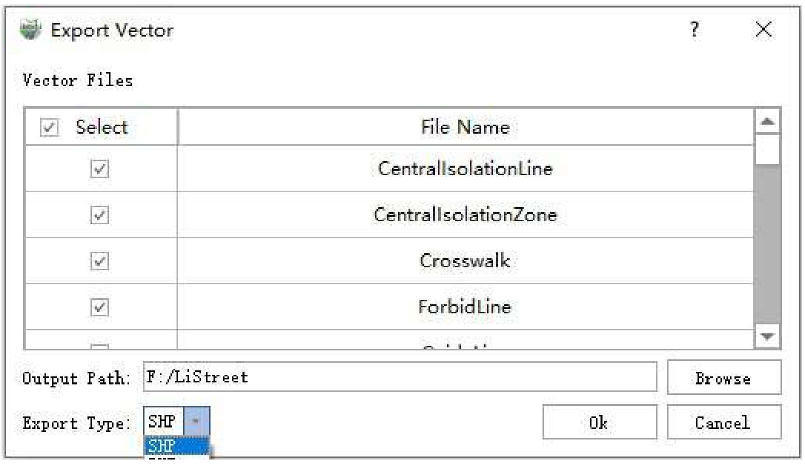
Description

- **Point 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 it 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 you want to export.
- **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 : The dialog box layer for exporting vectors is shown in the figure below. The main function is to select the layer to be exported, select the output directory and the exported vector format, and click the ok button to export. The following will be described in detail.

Click **Export vector**  Button



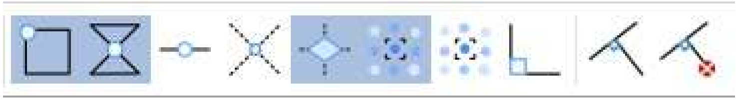
Description

- **Vector file**: 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.

Capture tool

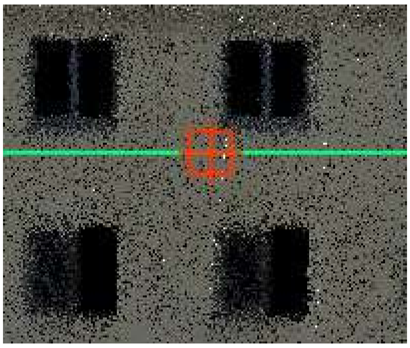
Function description : Capture the required scene requirements.

Steps

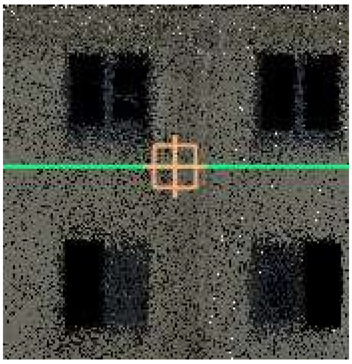


The software capture functions are shown in the figure above, in the order from left to right: endpoint capture, line capture, midpoint capture, intersection capture, reference surface capture, point cloud capture, point cloud fitting, orthogonal restriction, orthogonal axis capture, cancel orthogonal axis capture.The capture description is as follows： Click the "New" button, in the Point cloud dialog box, you need to enter the following files：

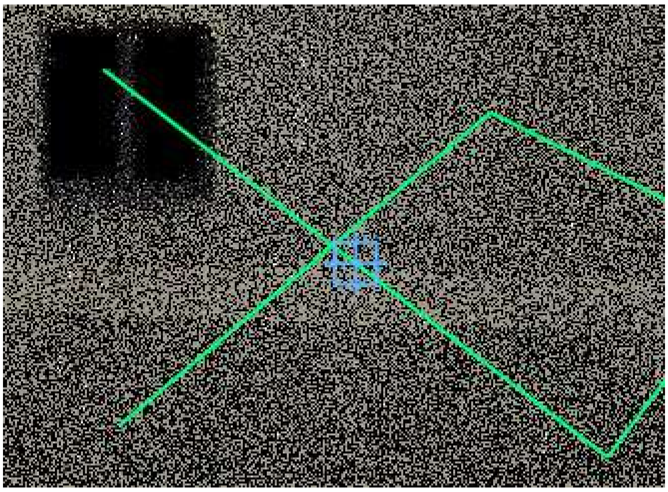
1.Endpoint capture: Capture each node of the vector data, and the state of the cross wire captured by the node is red



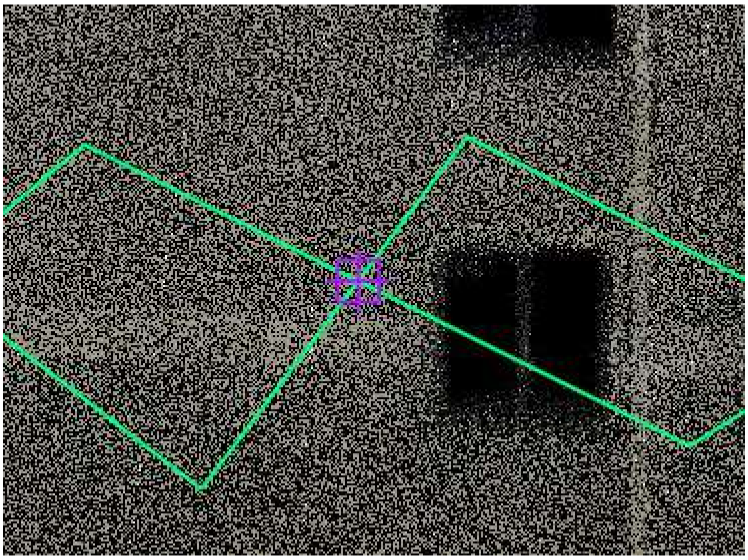
2.Line capture: Capture the edges of vector data, such as any position on the edges of lines and polygons, the state of the line captured to the cross wire is orange



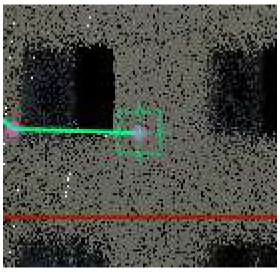
3.Midpoint capture: Capture the midpoint position of the vector line, and the state of the cross wire captured by the midpoint is blue



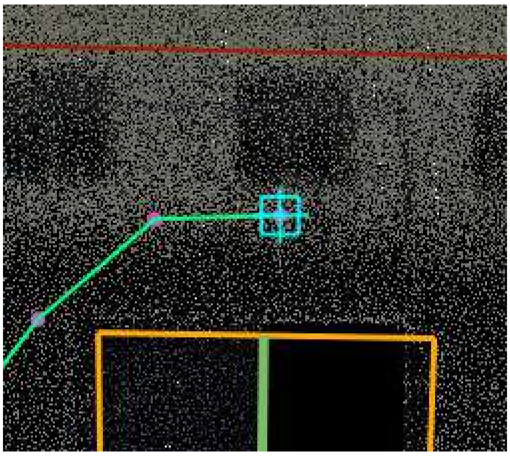
4.Cross capture: Capture the intersection of the cross line, and the state of the cross wire captured at the intersection is purple



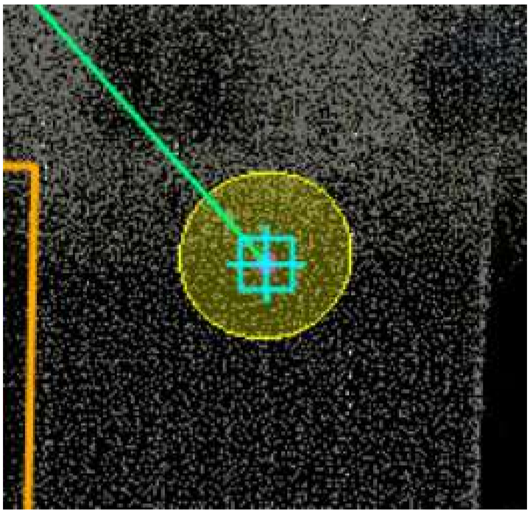
5.Plane capture: Capture any point on the reference plane, and the state of the cross wire captured by the plane is green

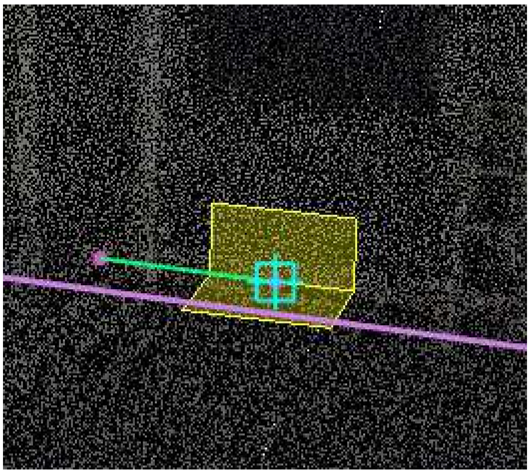


6.Point cloud capture: Capture the real point cloud coordinate points on the point cloud, and the state of the cross wire captured by the point cloud is light blue



7.Point cloud fitting: 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.Orthogonality limit: 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.Select the 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 the 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
- Classify Low Points
- Classify Isolated Points
- Classify Close by Points
- Classify Ground Points
- Classify By Attribute
- Classify by Polygons
- Classify by Polylines
- Classify By Cluster Size
- Classify by Deep Learning
- Classify

Classify Air Points

Function description : This function classifies point clouds that are significantly higher than surrounding points as noise points in the air.The algorithm idea is the same as the principle of denoising in the data management module.

Steps

1.Click **Classify Air Points** Button, pop up the **Classify Air Points dialog box**

Parameter description:

- **Input data**: The input file can be a single point cloud data file or a point cloud data set.
- **From Class**: Category to be classified.
- **To Class**: Categorize the target category.
- **Neighbor points**: The default is “10”. These neighboring points will be used to determine whether a point is an aerial noise.Calculate the distance from each point to neighboring points, and calculate the standard deviation of the average distance.
- **Multiples of std deviation**: The default is “5”. Points that exceed the minimum allowable deviation range of the average distance are considered to be aerial noise.When this threshold value increases, the number of noise classified as air noise becomes smaller, and vice versa, it becomes larger.
- **Default value**: Click this button to restore the default values of all parameters.

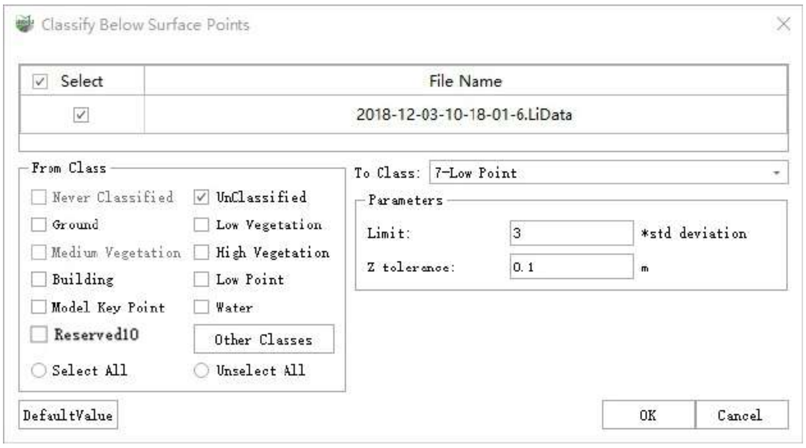
Classify Below Surface Points

Function description : This function classifies points in the starting category that are lower than the elevation of the surrounding adjacent area.For example, when the starting category is ground points, this method can be used to classify points with a certain height difference below the surface into points below the surface.

- The main algorithm idea of this function is：
  1. Look for a certain number of adjacent points of the current point in the starting category.
  2. Fit the plane with adjacent points.
  3. Calculate the absolute value of the height difference between the current point and the plane. If this value is less than the set tolerance, it will not be classified. If it is greater than the tolerance, then proceed to the next step.
  4. Calculate whether the difference between the current point elevation and the average elevation of the adjacent point is greater than the limit threshold of a multiple of the standard deviation. If it is greater than, it is classified as the target category, and vice versa, it is not classified.

Steps

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



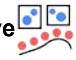
Parameter description：

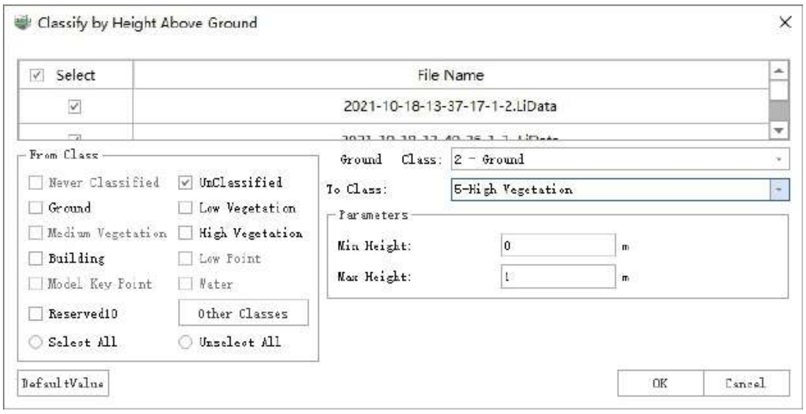
- **Input data**: The input file can be a single point cloud data file or a point cloud data set.
- **From Class**: Category to be classified.
- **To Class**: Categorize the target category.
- **Limit**: The default is “3", the multiple of the mean square error of the fitting plane of the neighborhood point of the current point to be classified.According to the principle of the algorithm, when this value is increased, the number of points classified as the target category becomes less.When this value is adjusted, the number of points classified as the target category becomes more.
- **Z tolerance (meters)**: The default is “0.1”, the height difference threshold, and the distance from the point to the fitting plane is less than this value, it will not be classified.When this value becomes larger, the number of points in the classification target category becomes less.When this value becomes small, the number of points classified as the target category becomes more.
- **Default value**: Click this button to restore the default values of all parameters.

Classify by Height Above

Function description : Classify points of a certain height on the surface of the terrain.This function can quickly classify vegetation of different heights.

Steps

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



Parameter description:

- **Input data**: The input file can be a single point cloud data file or a point cloud data set.
- **From Class**: Category to be classified.
- **Ground point category**: 2-Ground point.
- **Minimum height (meters)**: The default is “0”, the minimum height difference in the area to be classified above the ground point.
- **Maximum height (meters)**: The default is “1”, the maximum height difference in the area to be classified above the ground point.

> Note: This function requires the ground point category to be included in the point cloud.

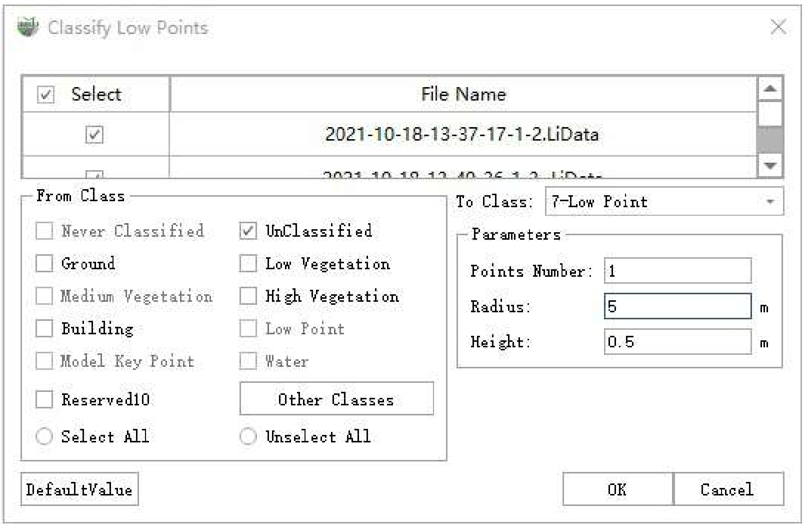
Classify Low Points

Function description : The low point refers to the coarse noise point that is lower than the actual terrain. The existence of the low point will affect the extraction of ground points, because the progressive triangular network filtering algorithm is based on the lowest point of the gridded point cloud as the seed point.Therefore, filtering out low points is a preprocessing operation of point cloud data, which directly affects the filtering of subsequent ground points, the establishment of digital models, and the generation quality of contour lines.

- The distribution of low points is divided into the form of a single point or cluster. The algorithm flow of this function is：
  1. Traverse the point cloud and search for points to be classified within a certain radius of the current single point or point cluster.
  2. Calculate the maximum elevation difference between the current point and the adjacent point and compare it with the threshold value.
  3. If it is greater than the threshold value, the current point is considered a low point, and vice versa, 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：

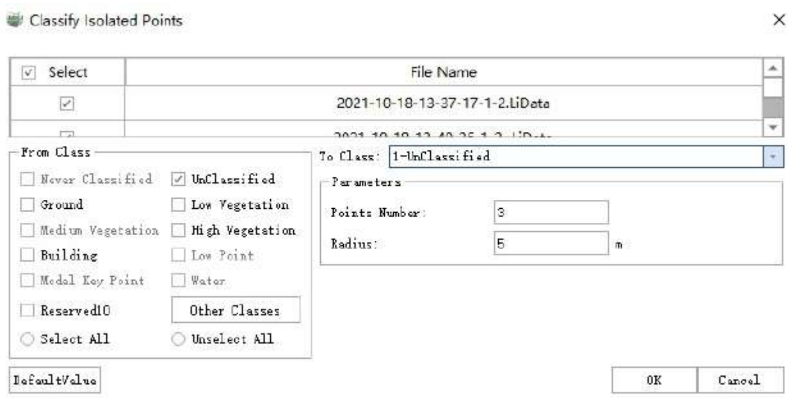
- **Input data**: The input file can be a single point cloud data file or a point cloud data set.
- **From Class**: Category to be classified.
- **To Class**: Categorize the target category.
- **Points Number**: The default is “1”. When the number of points is set to 1, a single low point is classified, and if it is greater than 1, it is classified as a cluster low point.
- **Radius (meters)**: The default is “5”, the radius threshold between the point to be classified and the adjacent point.
- **Height (meters)**: The default is “5”, the threshold value of the elevation difference between the point to be classified and the adjacent point.
- **Default value**: Click this button to restore the default values of all parameters.

Classify Isolated Points

Function description : This function classifies points with less than a certain value of points in a certain area of point cloud data as isolated points.Generally used to find isolated points in the air or below the ground.

Steps

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



Parameter description：

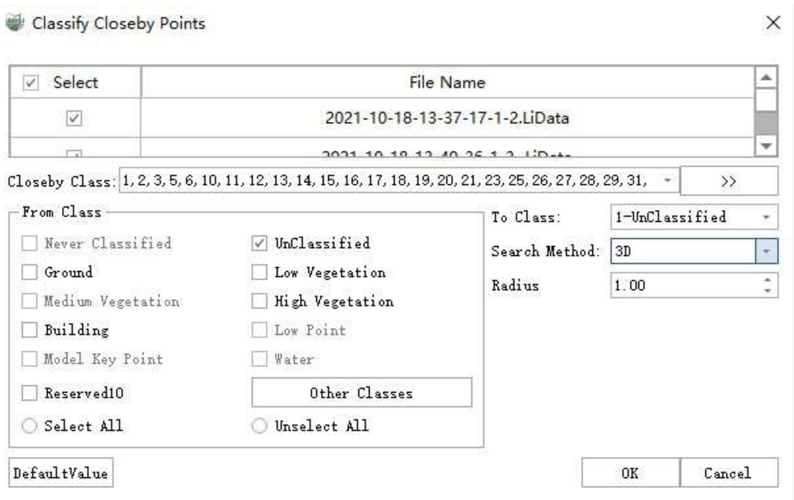
- **Input data**: The input file can be a single point cloud data file or a point cloud data set.
- **From Class**: Category to be classified.
- **To Class**: Categorize the target category.
- **Number of points**: The default is “3", the threshold value for the number of points in the radius neighborhood, less than or equal to this value is considered an isolated point.
- **Radius (meters)**: The default is “5”, the neighborhood search radius.
- **Default value**: Click this button to restore the default values of all parameters.

Classify Close by Points

Function description : This function can classify point clouds close to other categories.For each point in the source category, the software looks for point clouds in the specified 2D or 3D neighborhood, and determines whether these points meet certain conditions (such as containing a specified category). If the conditions are met, the point is assigned to the target category.

Steps

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



Parameter description:

- **Input data**: The input file can be a single point cloud data file or a point cloud data set.
- **Closeby Class**: For points in each source category, if the specified category appears in the search range, it will be classified.
- **From Class**: Category to be classified.
- **To Class**: Categorize the target category.
- **Search method**: Neighborhood search method, support 2D or 3D neighborhood.
- **Radius**: Neighborhood search radius.
- **Default value**: Click this button to restore the default values of all parameters.

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 **Ground point classification**  Button, pop up the **Ground point classification dialog box**

Parameter description：

- **Input data**: The input file can be a single point cloud data file or a point cloud data set.
- **From Class**: Category to be classified.
- **To Class**: Categorize the target category.

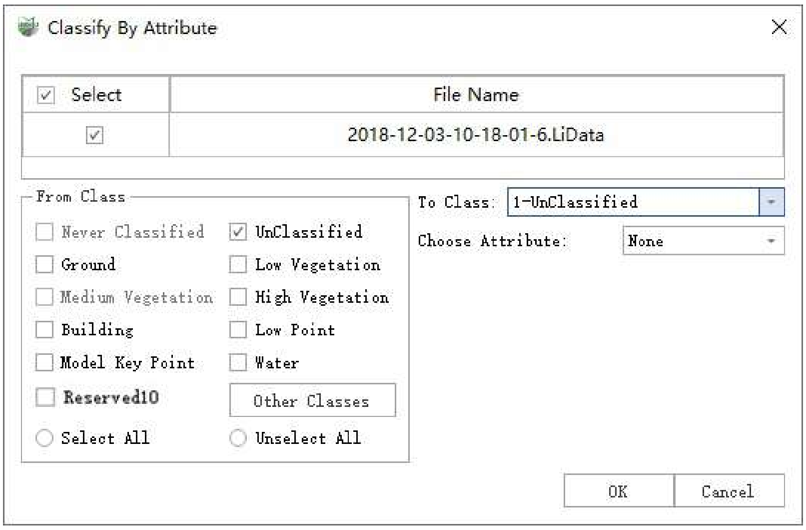
- **Scene (optional)**: For different terrain scenes, you can choose steep terrain, gentle terrain, and flat terrain.Check different terrain features to correspond to different default parameters
- **Grid size (meters)**: The default is “1.0”, the resolution of the fabric node.1.0 is suitable for most 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 iterations**: The default is “500”, and the iteration is completed when the algorithm reaches the maximum number of iterations or all fabric nodes are immovable.
- **Smoothing 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.
- **Default value**: Click this button to restore the default values of all parameters.

Classify By Attribute

Function description : Through this function, a category in the point cloud can be classified into another category according to attribute characteristics.The attributes currently available for classification include: absolute elevation, echo intensity, GPS time, scanning angle, and echo number.For data whose classification effect is not ideal, if you need to reclassify, you can use this function to restore all categories.

Steps

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



Parameter description:

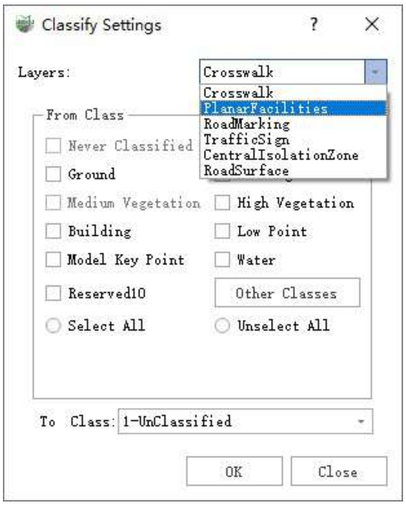
- **Input data**: The input file can be a single point cloud data file or a point cloud data set.
- **From Class**: Category to be classified.
- **To Class**: Categorize the target category.
- **Attribute selection**: Classify according to the selected attribute.
  - None (default): Change all points in the "Initial Category" option to the "Target Category".
  - Elevation: Classified by the range of elevation values.If the elevation value of a point falls within the specified range, it will be assigned to the “target category”.
  - Intensity: Classified by intensity range.If the intensity value of a point falls within the specified range, it will be assigned to the “target category”.
  - Time: Classified by time.If the time value of a point falls within the specified range, it will be assigned to the “target category”.
  - Angle: Classified according to the scanning angle.If the scanning angle value of a point falls within the specified range, it will be assigned to the “target category”.
  - Return: Classified by echo frequency.If the number of echoes of a point falls within the specified range, it will be assigned to the “target category”.

Classify by Polygons

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

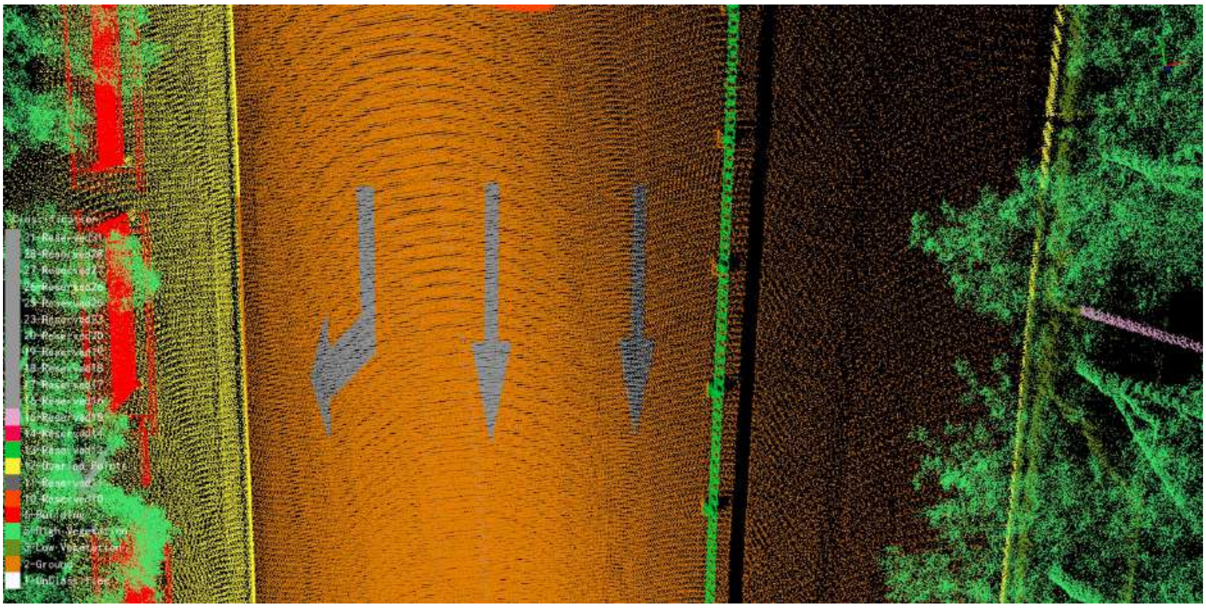
Steps

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

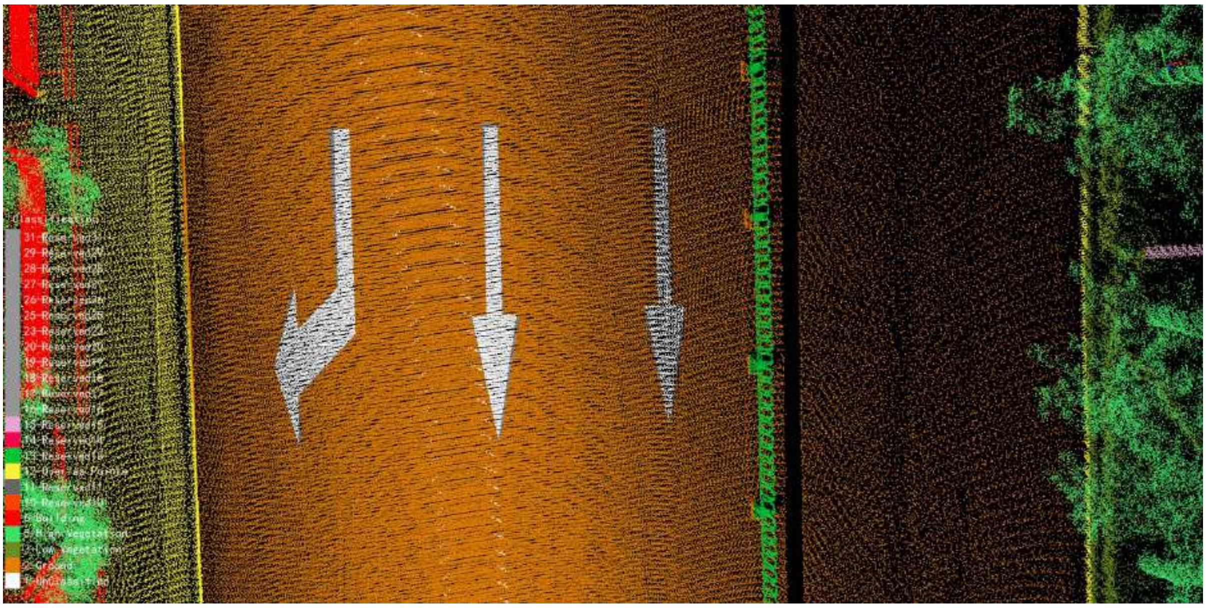


Parameter description:

- **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



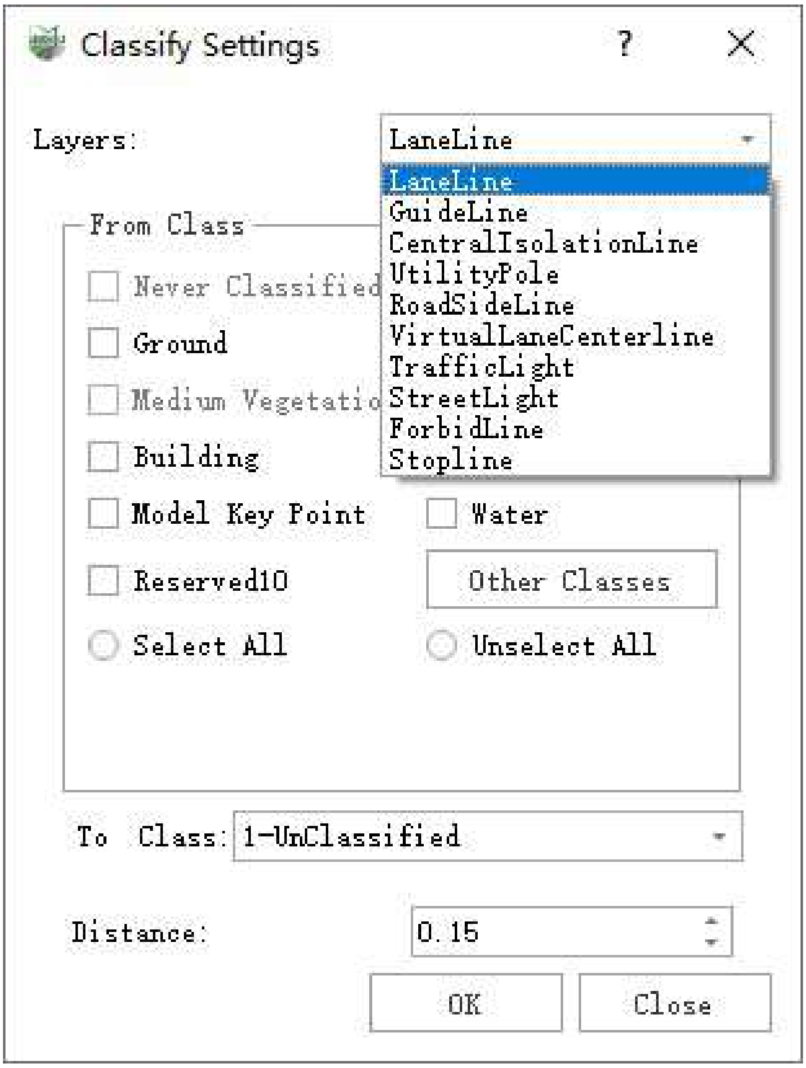
Effect diagram after Classify by Polygons

Classify by Polylines

Function description : Classify point clouds based on vector lines.

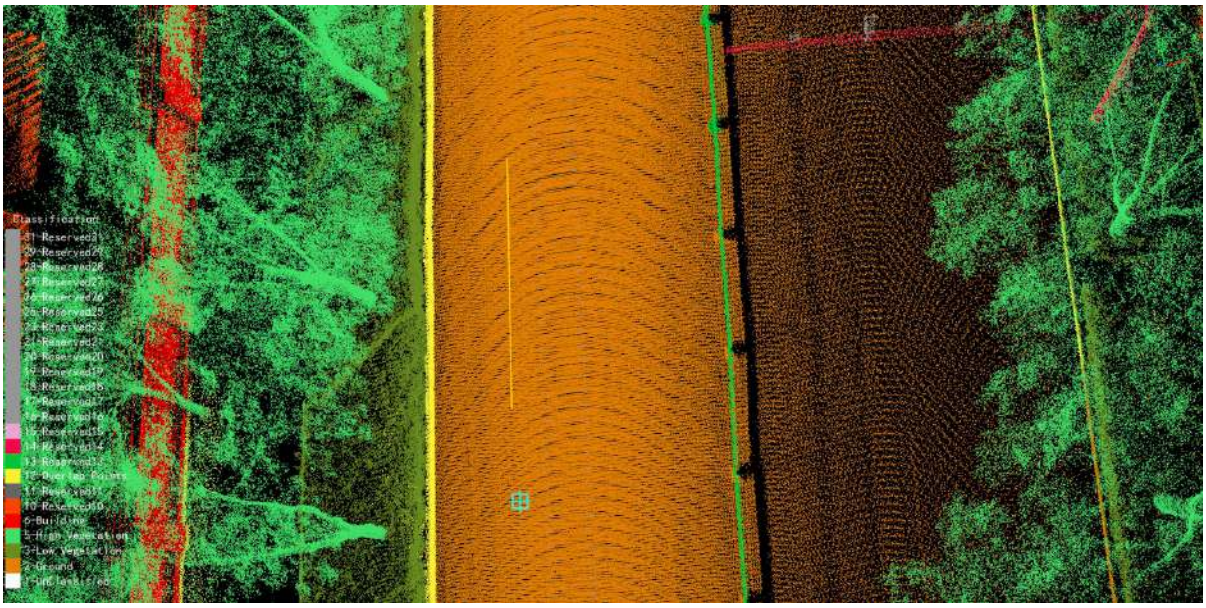
Steps

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

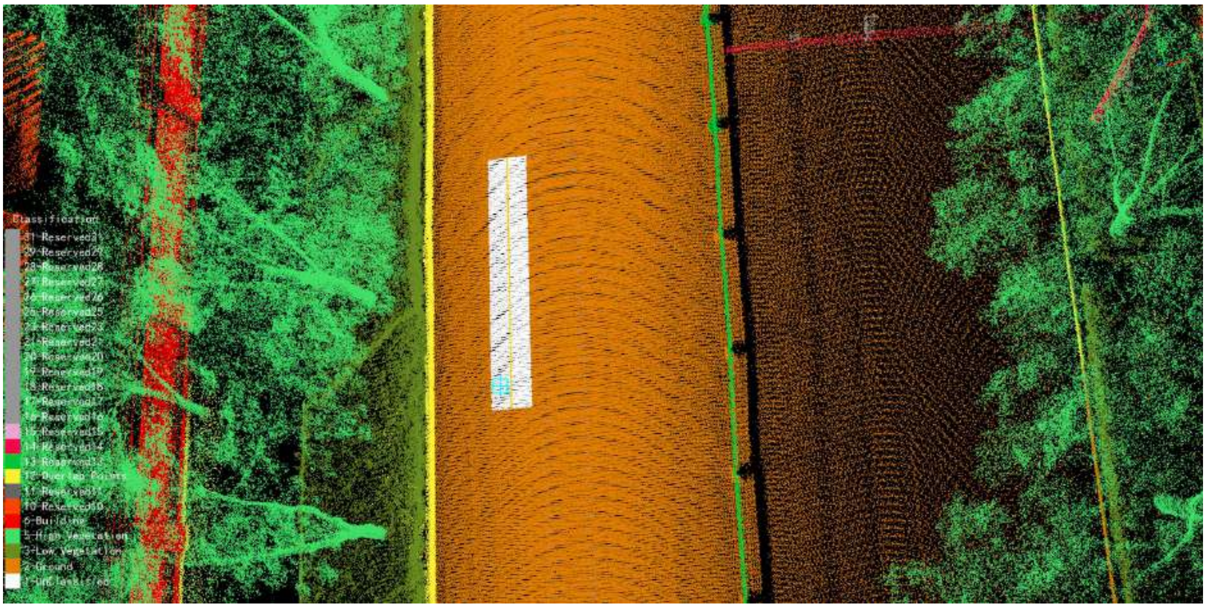


Parameter description:

- **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



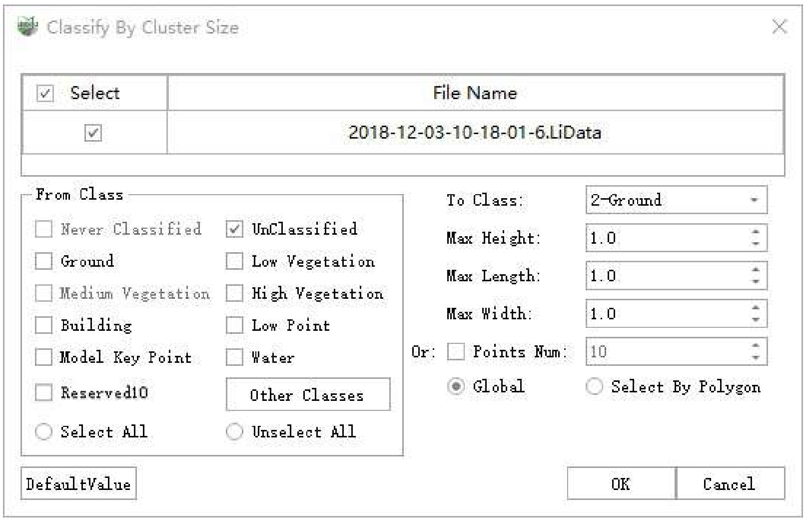
Effect diagram after Classify by Polylines

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 Number**: 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 polygon box selection option is checked, the point clouds within the polygon range will only be processed.

> Multiple polygons can be framed at once,

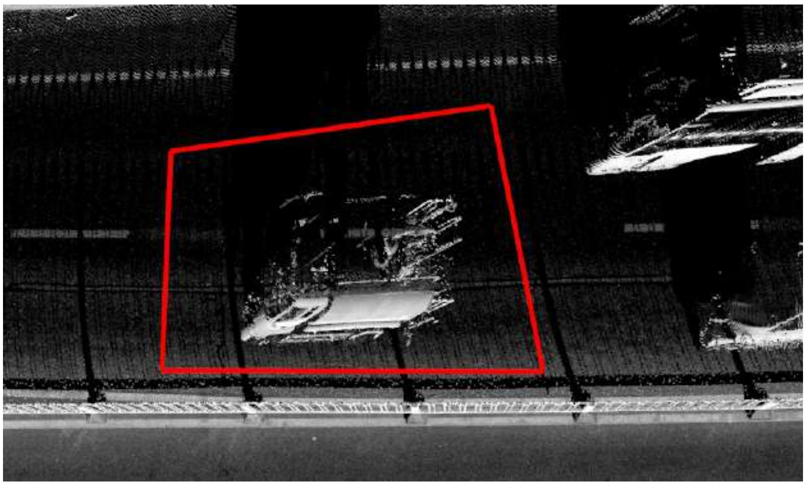
- **OK**: After the parameters are set, click the OK button to start automatic detection
- **Cancel**: Exit function

Steps

1. In the **Classification**panel, click **Classify By Cluster Size**  button in the Classification drop-down box.
2. Pop up the**Parameter setting 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

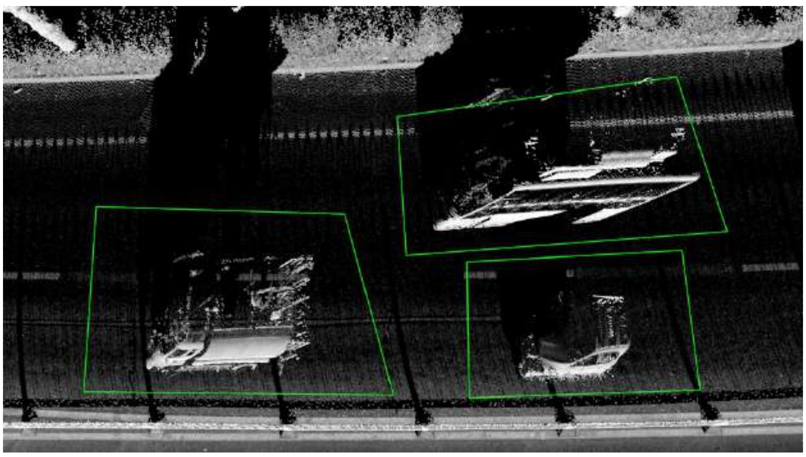
Polygon box selection

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



Polygon box selection

2.Repeat step 1 until the desired range



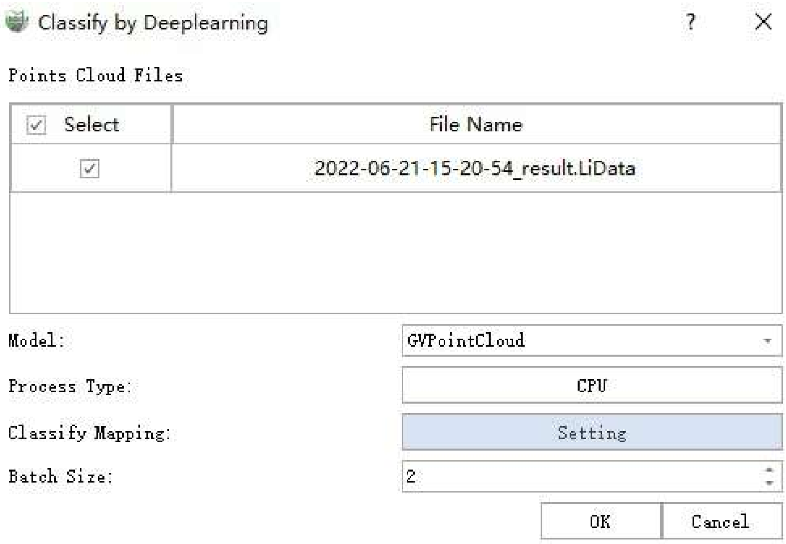
Polygon box selection

Classify by Deep Learning

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

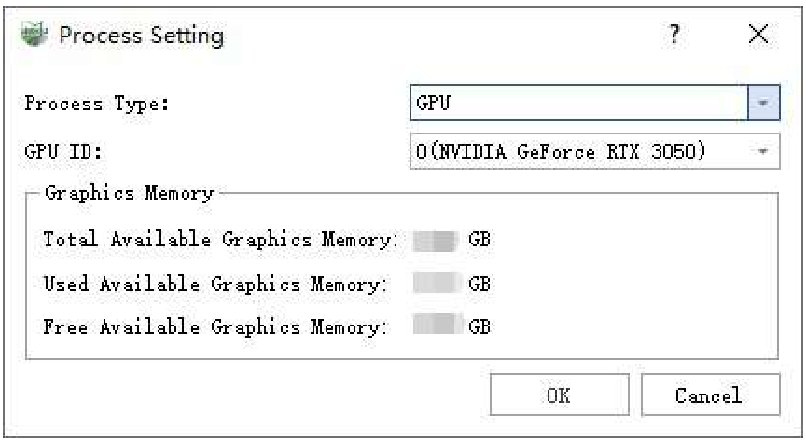
Steps

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

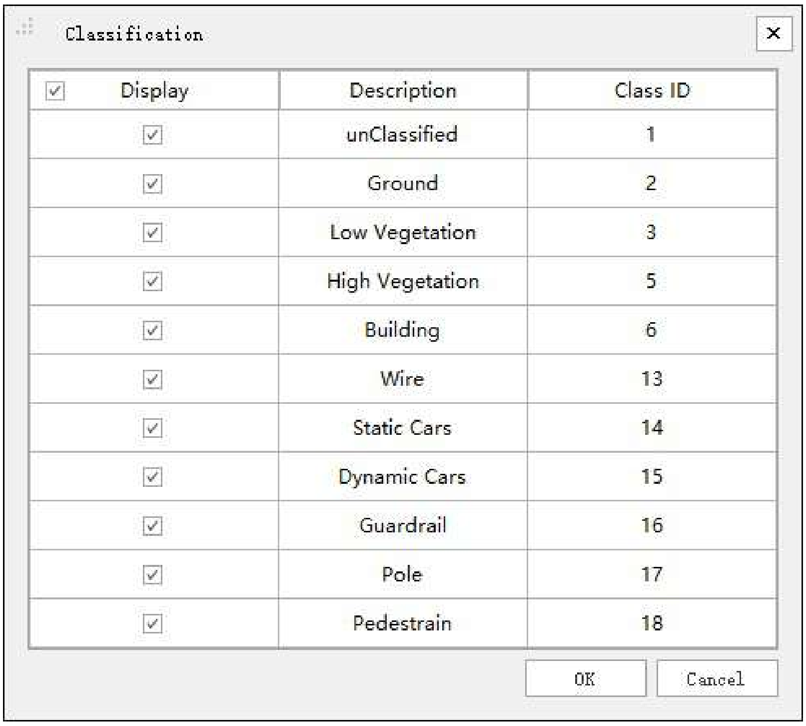


Parameter description:

- **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.
- **GPU**: 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.



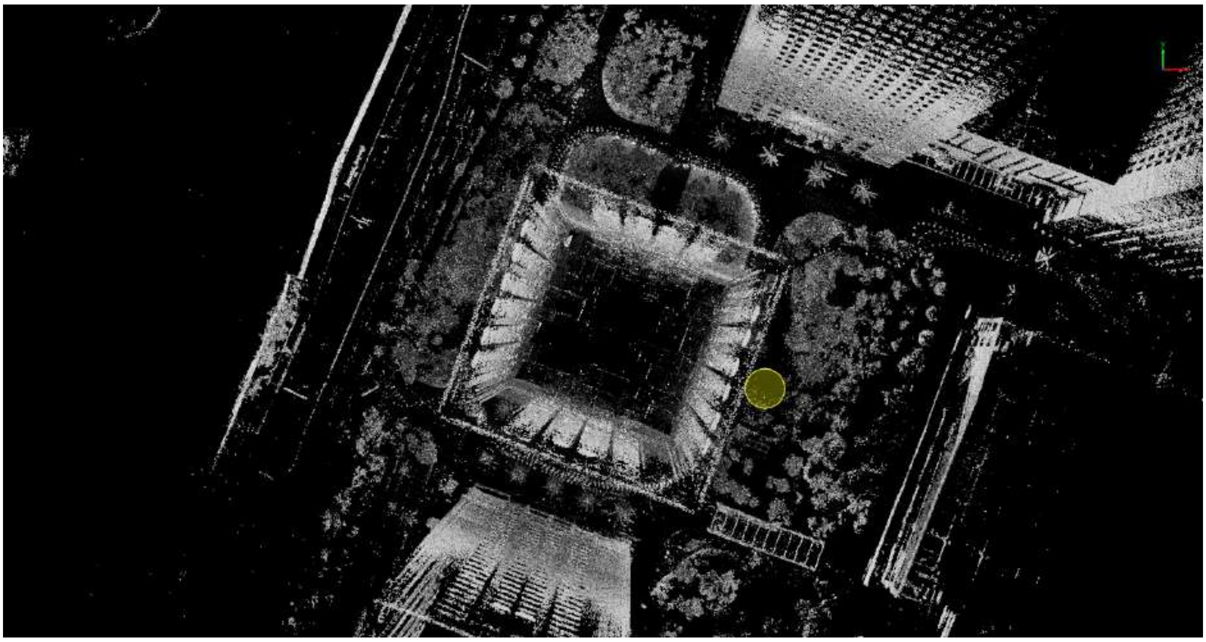
- **Classify mapping**: Map the category and map the category to the corresponding category label.



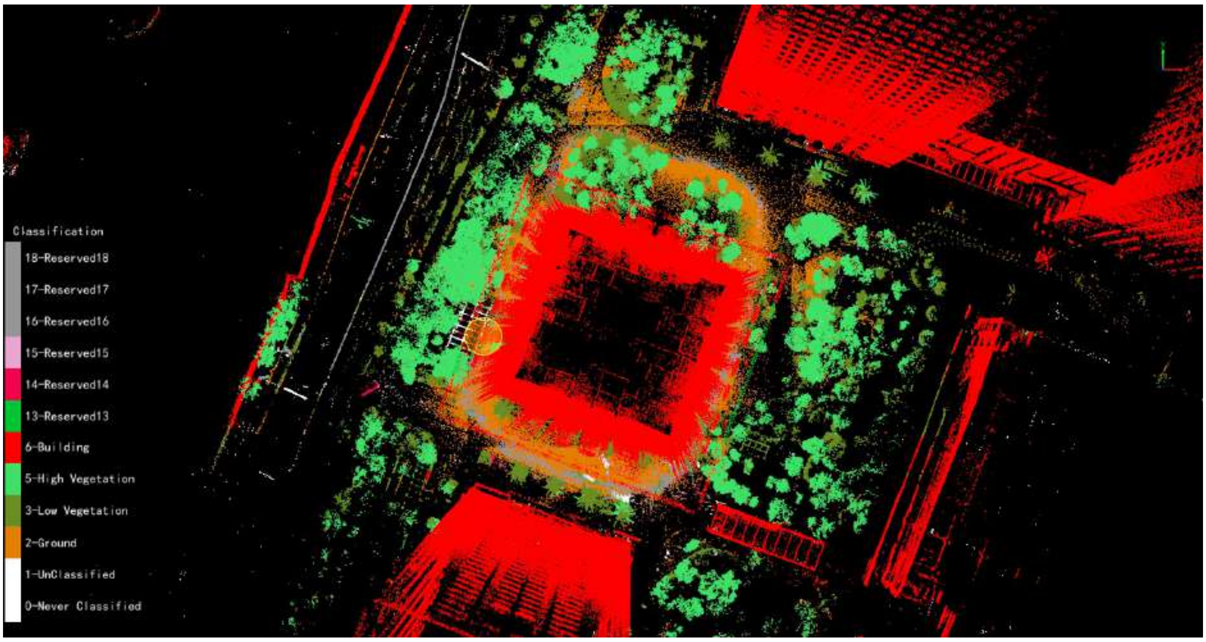
Display	Description	Class ID
☑	unClassified	1
☑	Ground	2
☑	Low Vegetation	3
☑	High Vegetation	5
☑	Building	6
☑	Wire	13
☑	Static Cars	14
☑	Dynamic Cars	15
☑	Guardrail	16
☑	Pole	17
☑	Pedestrain	18

Here you can set your own categories and labels according to user needs, such as: The corresponding category 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 The 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 Deep Learning



After Classify by Deep Learning

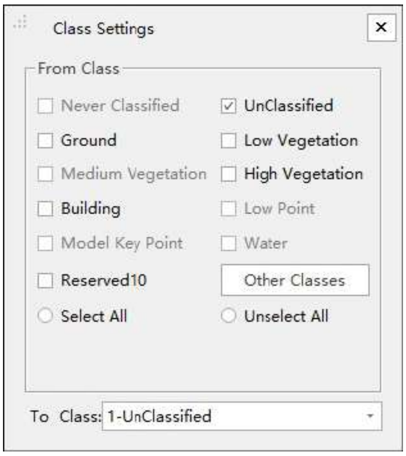
Classify

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**: Category to be classified.
- **To Class**: Categorize the target category.

Classify selection area

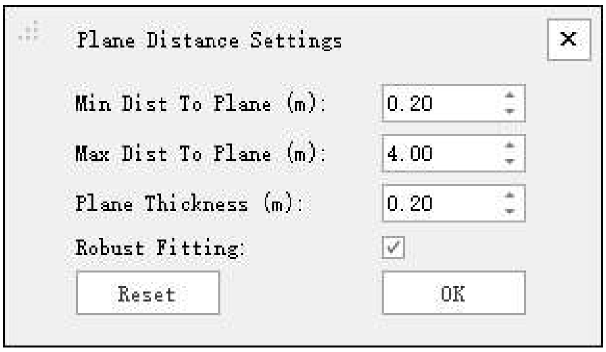
Function description : Select the area through the box selection, you can classify the selected area.

Steps

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

Parameter description:

- **polygon**: Select a polygon area.
- **Rectangle**: Select a rectangular area.
- **Spherical**: Select a ball 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 setting to pop up the **Plane distance setting dialog box**



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

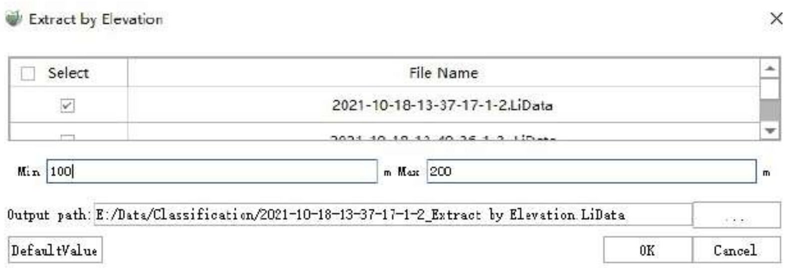
- Extract By Elevation
- Extract By Intensity
- Extract By Return
- Extract By Time
- Extract By Class

Extract by Elevation

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

Steps

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



Parameter description:

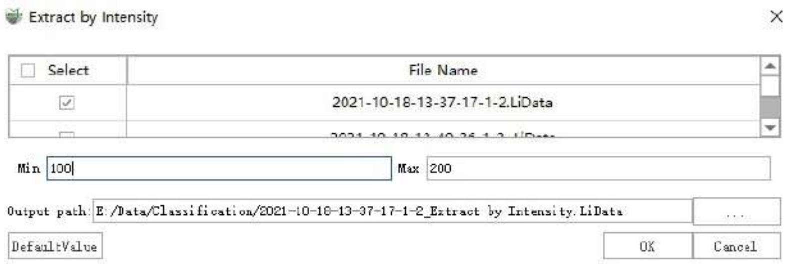
- **Enter point cloud data**::Enter one or more point cloud data files.File format***.LiData**.
- **Min (meters)**: The default is “100”, the user needs to enter the minimum elevation value of the point cloud data to be extracted.
- **Maxi (meters)**: The default is “200”, the user needs to enter the maximum elevation value of the point cloud data to be extracted.
- **Output path**: The path of the output folder, and the extracted new file is generated after the function is executed.

Extract by Intensity

Function description : Extract by Intensity tool can extrac all the point cloud data within the user-defined intensity range. This function supports multiple fleoperations.

Steps

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



Parameter description:

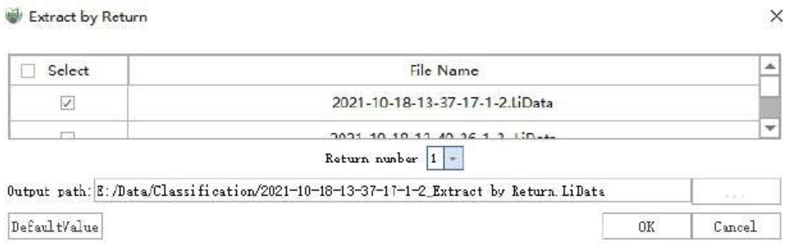
- **Enter point cloud data**::Enter one or more point cloud data files.File format***.LiData**.
- **Min (meters)**: The default is “100”, the user needs to enter the minimum elevation value of the point cloud data to be extracted.
- **Max (meters)**: The default is “200”, the user needs to enter the maximum elevation value of the point cloud data to be extracted.
- **Output path**: The path of the output folder, and the extracted new file is generated after the function is executed.

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 fle.This functionsupports multiple file operations.

Steps

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



Parameter description:

- **Enter point cloud data**::Enter one or more point cloud data files.File format***.LiData**.
- **Return Numbe**: 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**: The path of the output folder, and the extracted new file is generated after the function is executed.

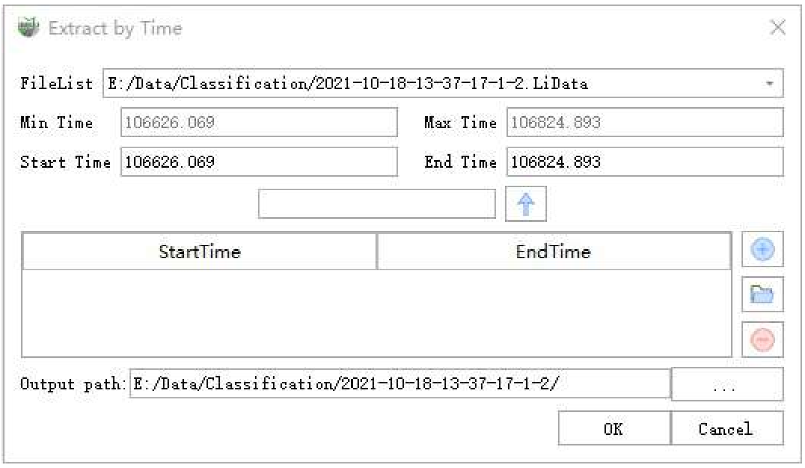
  > If the data does not have the number of echoes selected by the user, the data cannot be extracted.

Extract by time

Function description : The time extraction tool can extract all point cloud data within that range and save it in a file according to the time range entered by the user.

Steps

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



Parameter description:

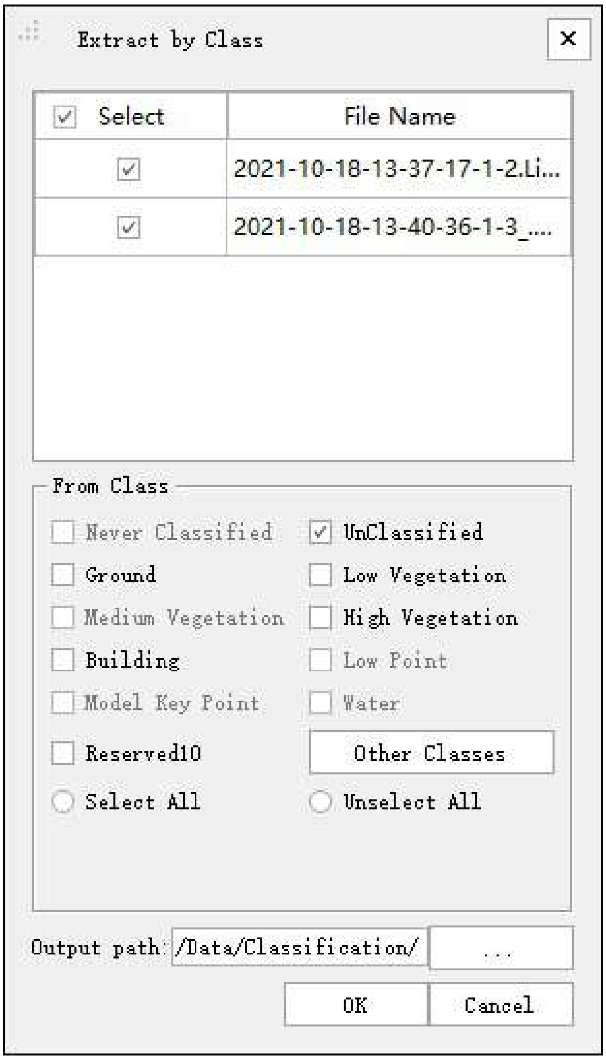
- **File list**: The user needs to select the file to be processed from the drop-down list.
- **Min time**: Displays the minimum time value in the point cloud file selected by the user, and the user can set the extracted time range based on this value.This value does not need to be set by the user.
- **Max time**: Displays the maximum time value in the point cloud file selected by the user, and the user can set the extracted time range based on this value.This value does not need to be set by the user.
- **Start time**: The default is the minimum time, and the user needs to enter the minimum time value of the point cloud data to be extracted.
- **End time**: The default is the maximum time. The user needs to enter the maximum time value of the point cloud data to be extracted, which needs to be greater 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 the button. The values of the start time and end time will be incremented at the set interval.
- : Add the input time range to the range list, and all point cloud data within the time range will be extracted into a file.
- : Users can click this button to load existing external data files, and the time range in the file needs to be between the minimum time and the maximum time.
- : The user selects a row in the time range list and clicks the button to remove the row from the list.
- **Output path**: The path of the output folder, and the extracted new file is generated after the function is executed.

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 fle. This function supportsmultiple file operations.

Steps

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



Parameter description:

- **Enter point cloud data**::Enter one or more point cloud data files.File format***.LiData**.
- **From Class**:The user needs to select the category to extract from the check box. Categories that do not exist in the file are not available in the check box.
- **Output path**: The path of the output folder, and the extracted new file is generated after the function is executed.

Profile Tools

Profile editing tool allows users to view the profile of the point cloud data in the selected rectangle area. Users can view, meansure, and edit the data in the profile window.

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 profile window to measurement tools interface, the measurement tools will 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 profile window to select tools interface, the measurement tools will activate. To reuse the profile tools, it is necessary to reclick the profile tools. The sub-interface of profile tools only support selected tool in the profile interface,

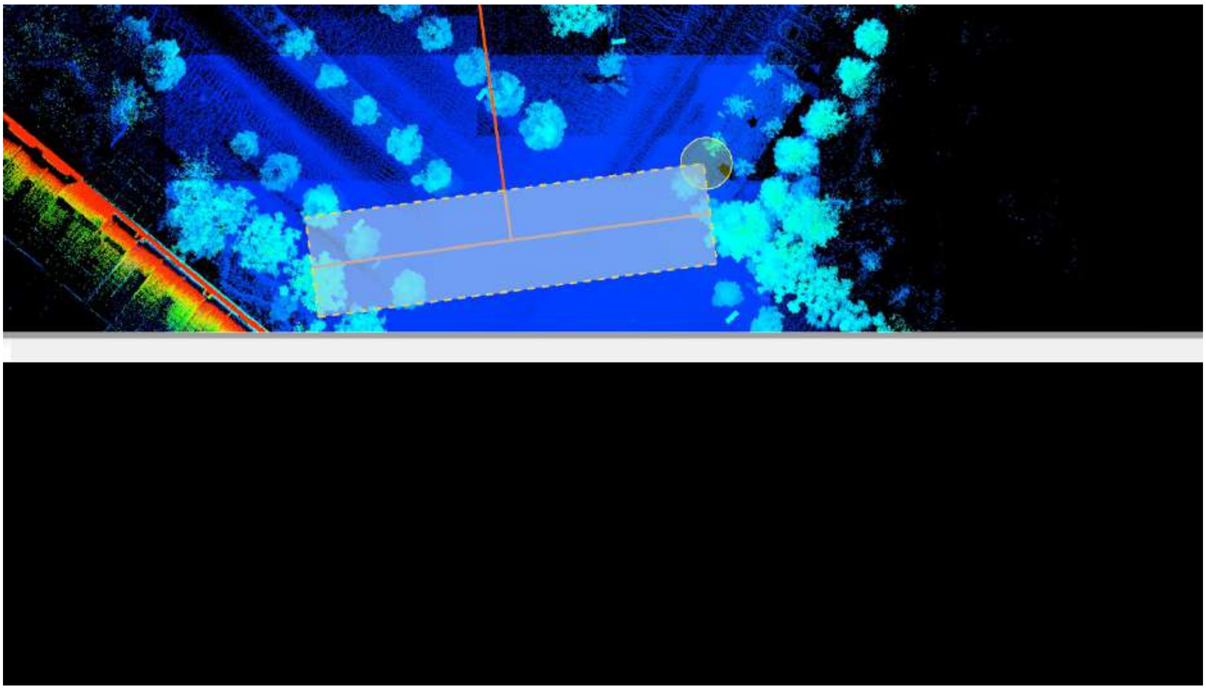
Select Profile Region

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

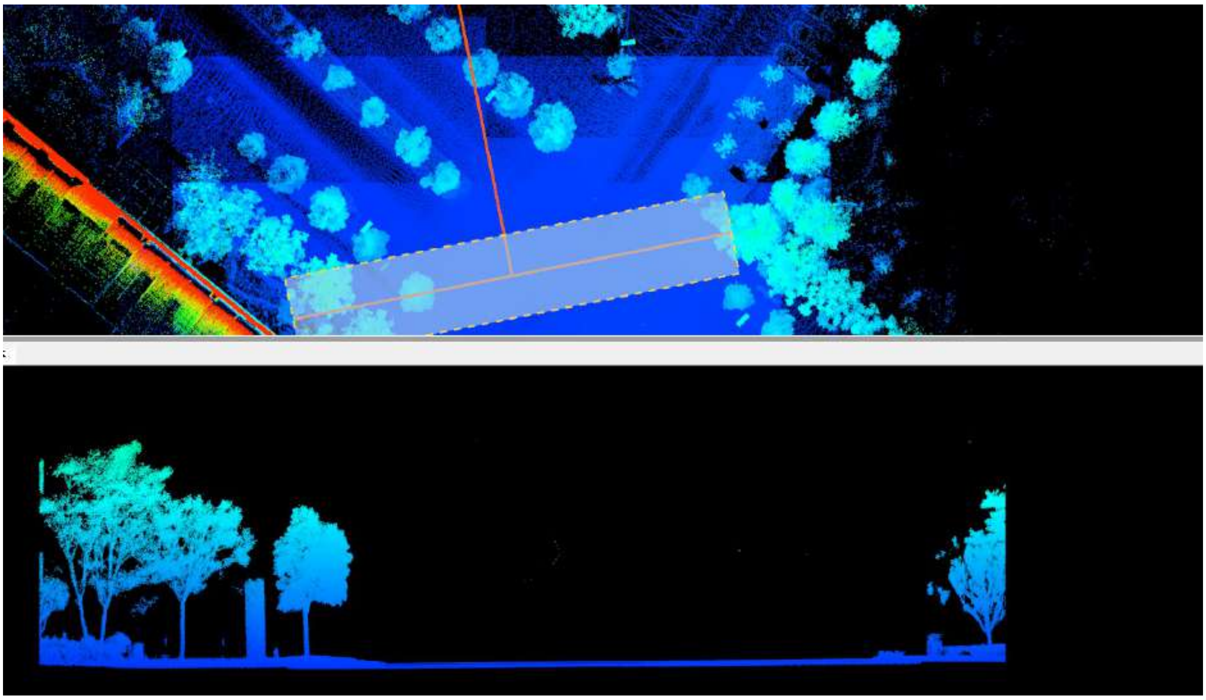
Steps

1.Move the mouse to 3D window to zoom to interested 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. In profile window, the selected profile area will display.



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

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

Fixed Buffer

Move

Rotate

Expand

> Note: It is supported to mixed use of tools mentioned above.

Fixed Buffer

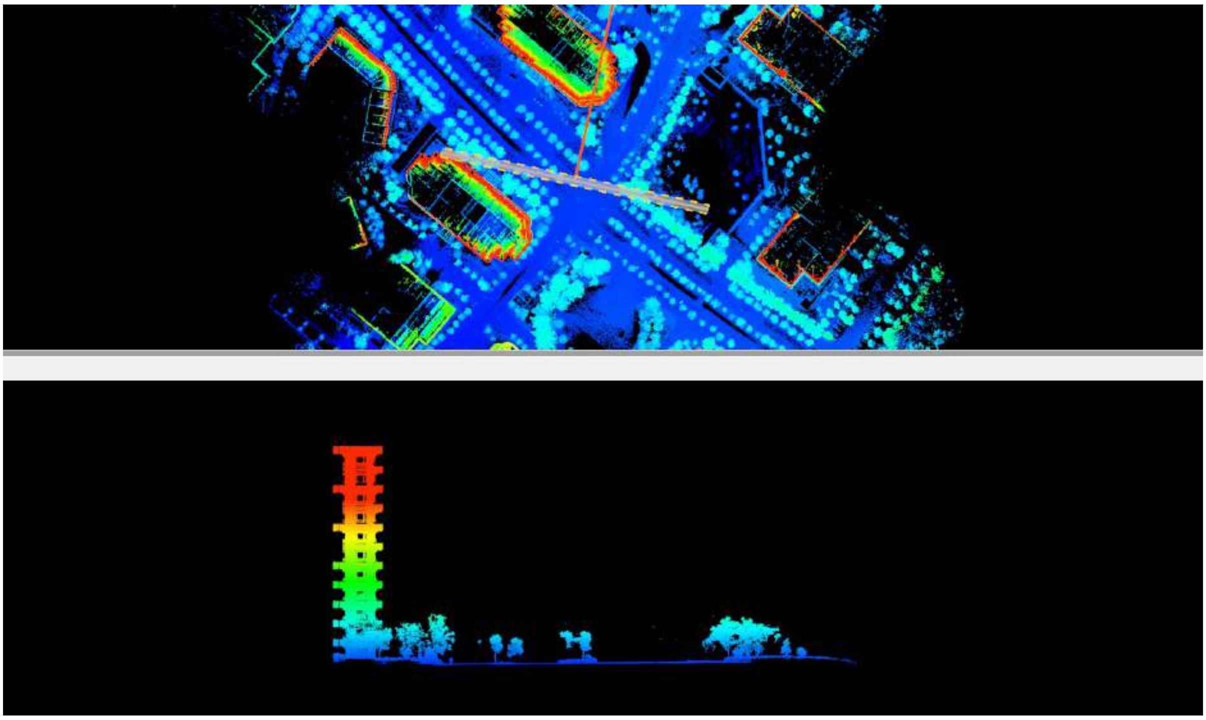
Fixed rectangular width is supported to use.This function can help users to fix the size of the buffer area.

Steps

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

2.Click the **buffer setting** button.

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



Measure Tools

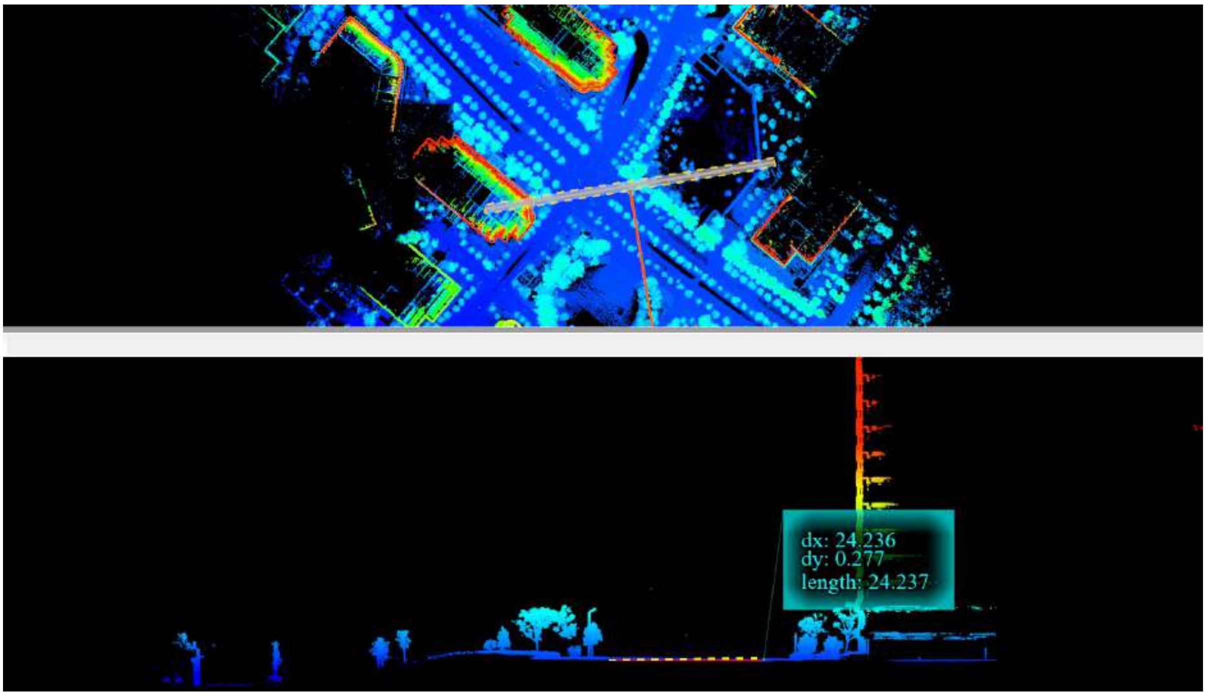
Function Description: Orthogonal projection is as default for profile window, and it is convenient to calculate the horizontal and vertical distance.

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

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.

Steps

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

2.**optional** Click  to move up the profile.

3.**optional** Click  to move down the profile.

Rotate

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

Steps

1. Click Angle: 5 edit box, adjust the angle of each rotation, defaulting to "5".

2. Click  button to rotate.

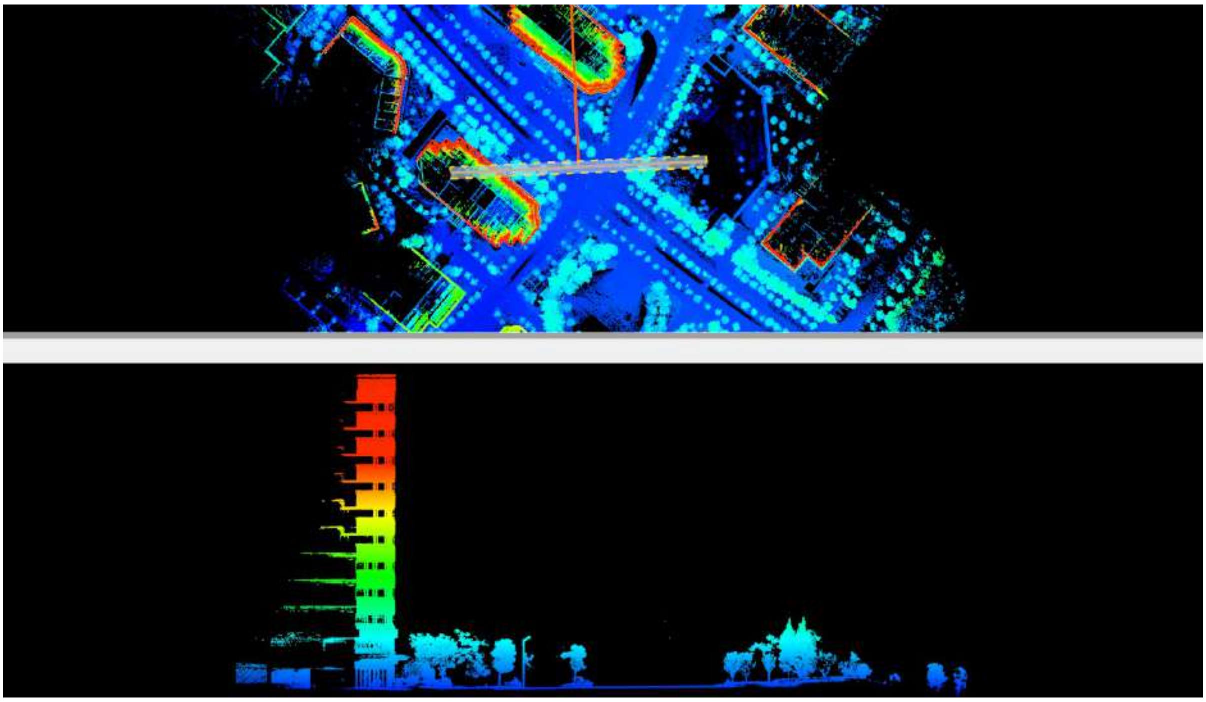
Expand

Function Description：Used to expand the width of profile area by the entered number.

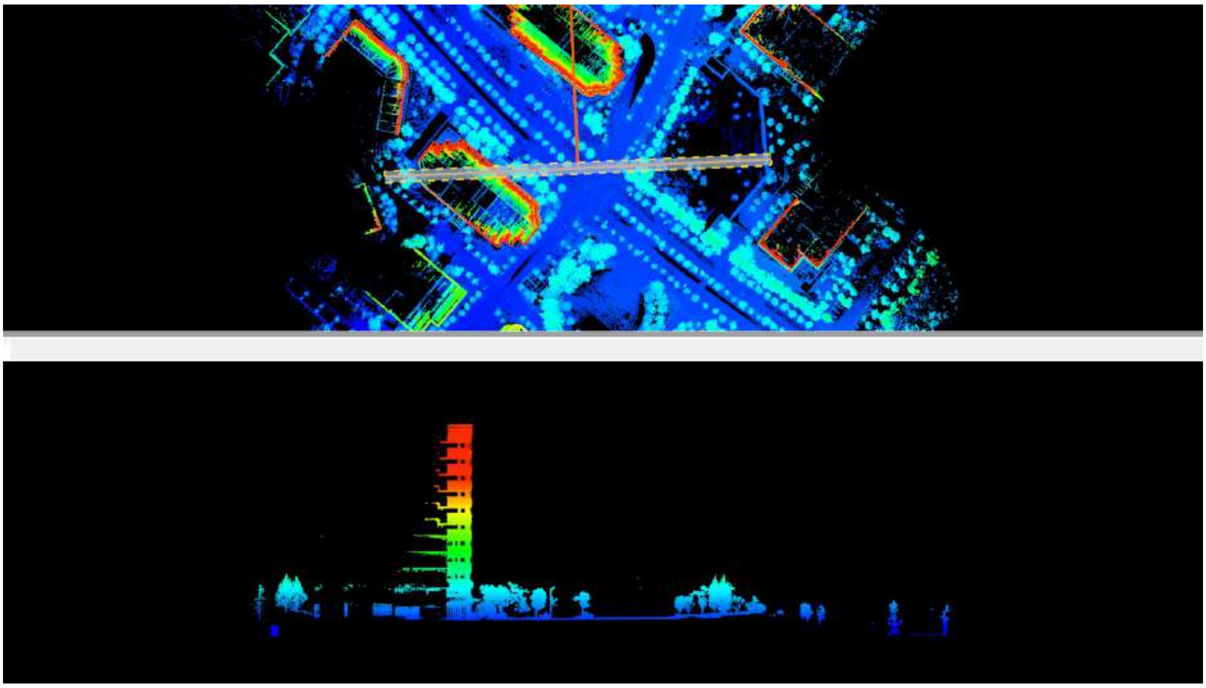
Steps

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

Before expand：



After expand:



2.Click **Expand** 

button to change the expand length.

Switch View

Function Description：Regarding the section rotation mode, in addition to switching the section to the front view, back view, left view, and right view, it also provides interactive operation rotation mode to view the point cloud of other angle sections:

Steps

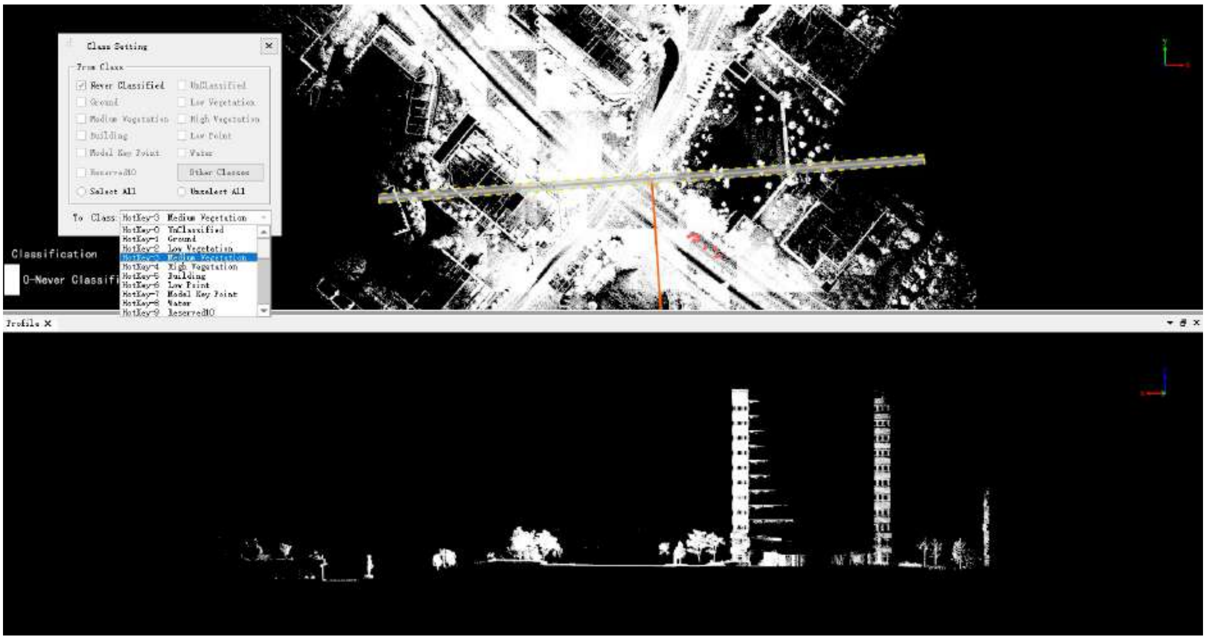
1. By default, profile view shows the front view.
2. **optional** Click  button to switch to front view.
3. **optional** Click  button to switch to rear view.
4. **optional** Click  button to switch to left view
5. **optional** Click  button to switch to right view.
6. **optional** Click  button to start 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 Classify

Function Description：Use tools supported by profile window 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.

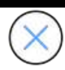
Rectangle Selection ：In some cases where the shape is relatively simple, or the requirements are not particularly strict, you can directly use the rectangular selection. Compared with polygon selection, the sides of the rectangle selection can only be parallel to the sides of the window. In other words,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

Below the Polyline Select : Select below the line.

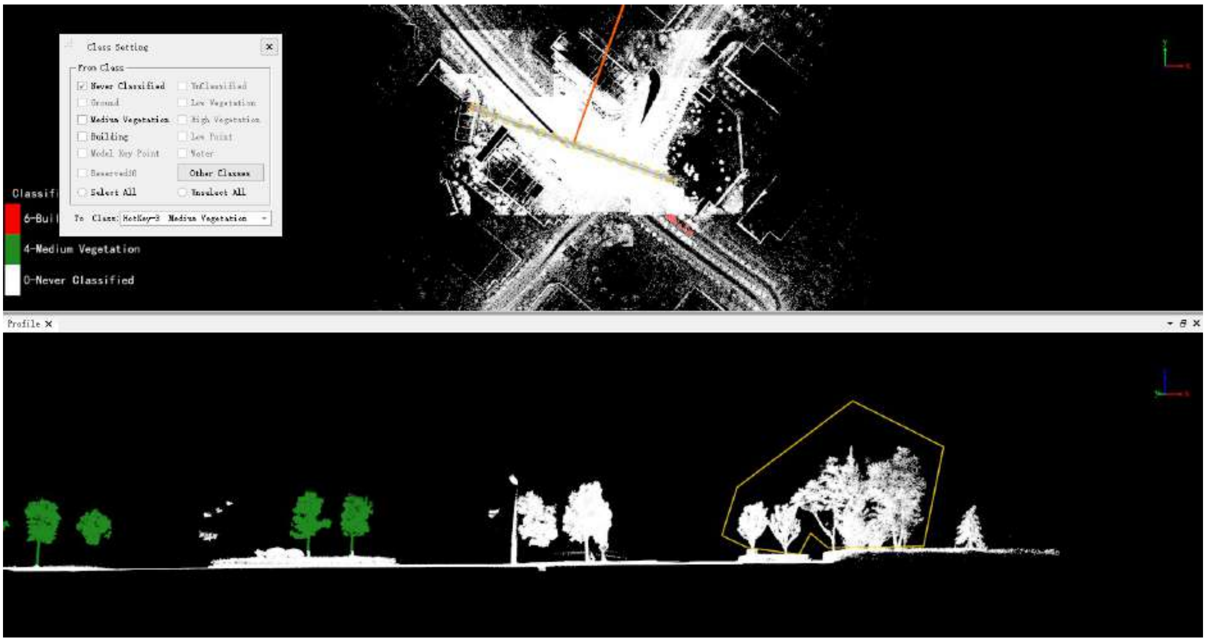
3.Display by class to see previous selected points.

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.

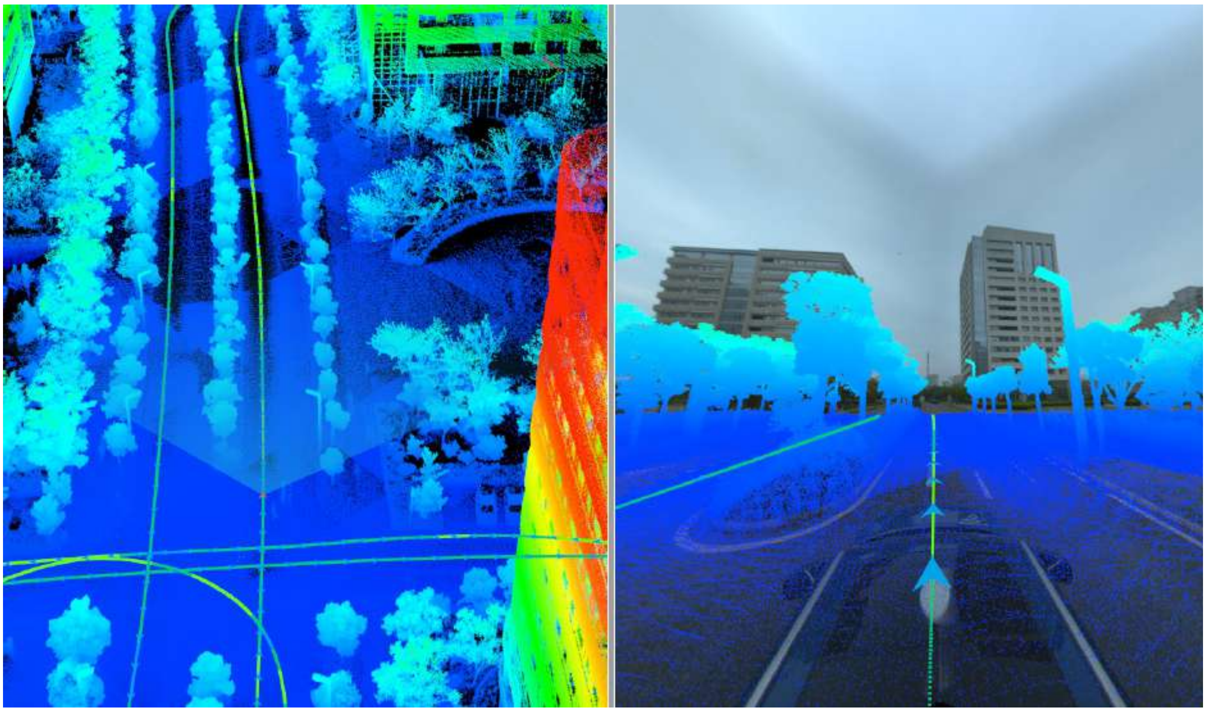
The panorama menu is displayed only after image data is added when a new project is created. Before measurement, switch the top menu of the software to the panorama page.

Point Cloud and Panoramic Image Roaming

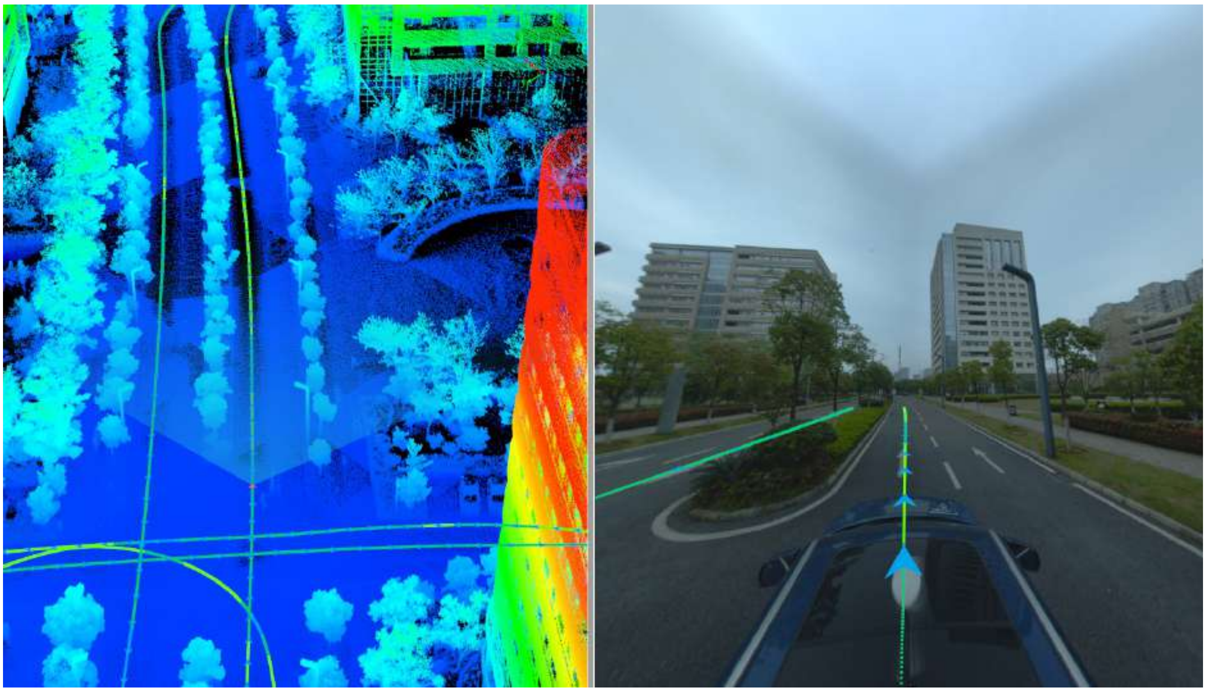
Function Description：The software supports point cloud and panoramic image roaming. The original camera file must be added when creating a new project before the menu bar will be activated.

Steps

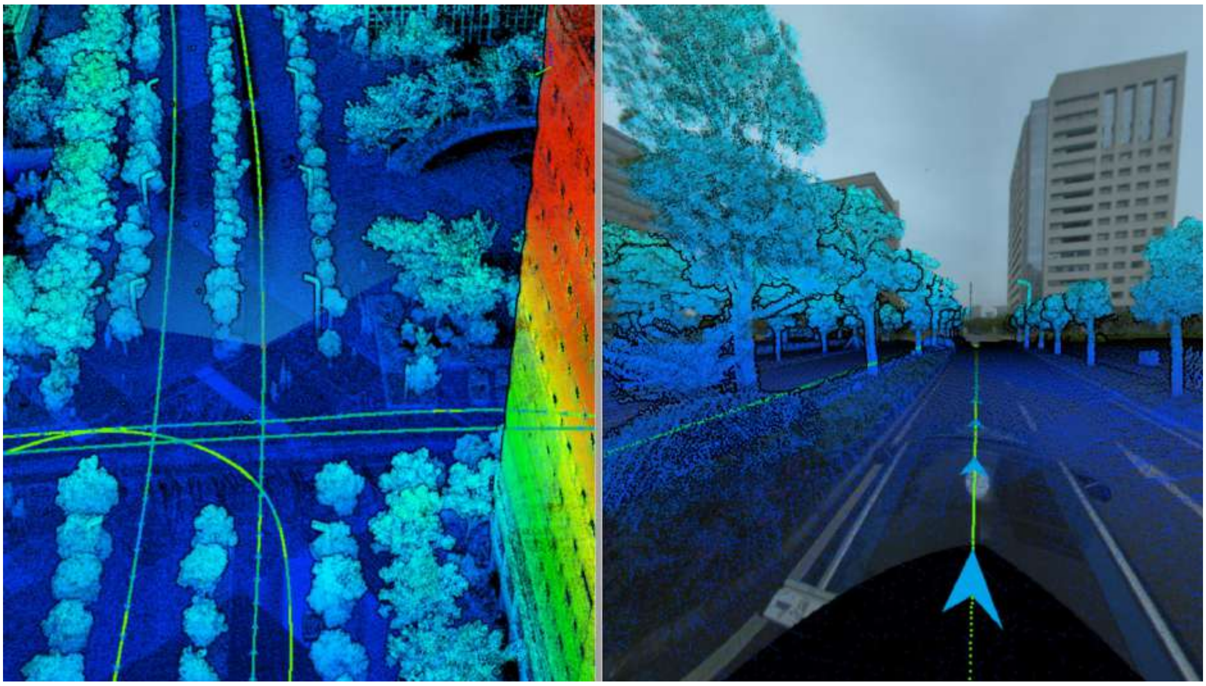
1.After a new project is created, a 3D window and a panoramic window will be opened. The point cloud data will be displayed in the 3D window and the panoramic window at the same time, and the panoramic image will only be displayed in the panoramic window. You can choose to show or hide the corresponding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



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



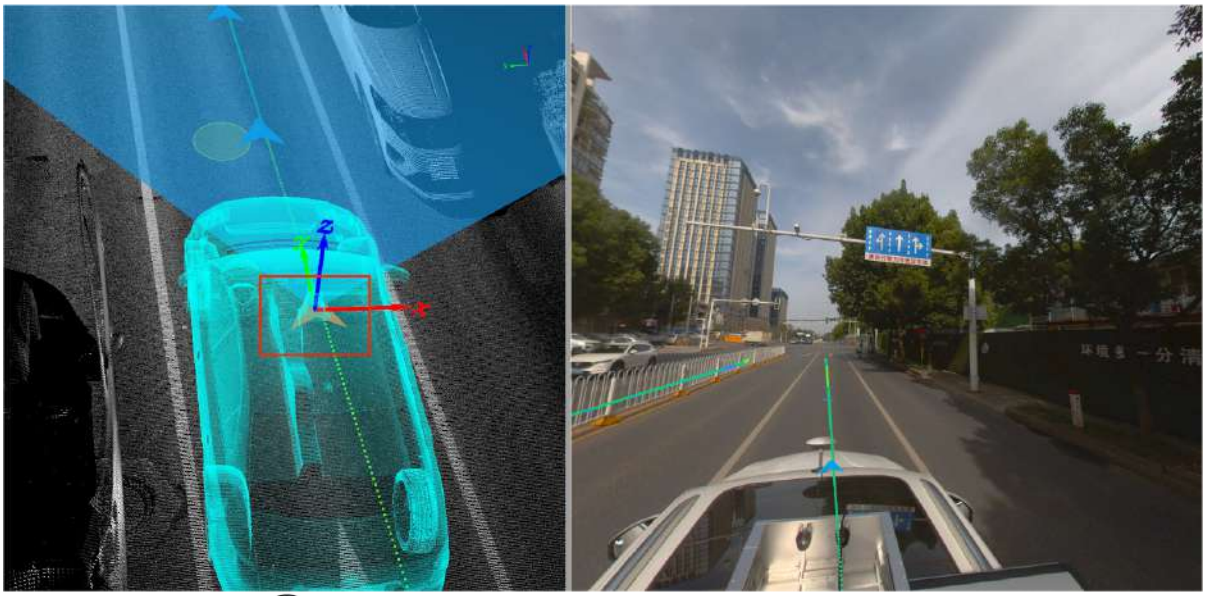
3.Click the color bar tool on the left side of the window to switch the display mode of the point cloud, including by elevation, intensity, category, RGB, echo times, time, mixed display, combination display, display by selected color and EDL display, EDL can Used in conjunction with other display methods 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 point cloud, click the right mouse button on the data name and select **View mode**> Press elevation/intensity/category, etc.

> - 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.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 is clicked) 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.

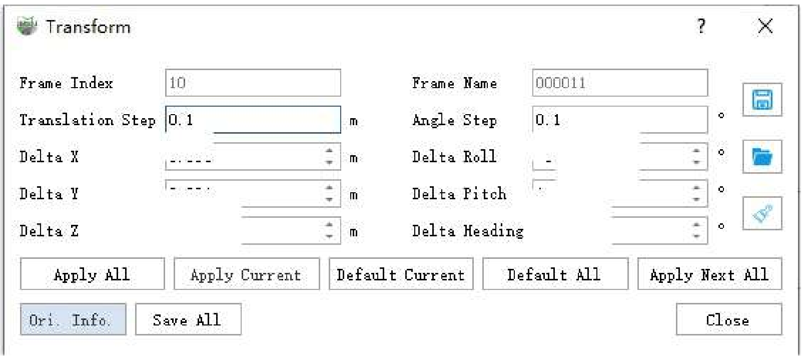
> Note: Shortcut of next frame and previous frame:

Shortcut key	Function
PageUp	Next Frame
PageDown	Previous Frame

6.Drag the slider to switch between different image positions or directly enter 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.

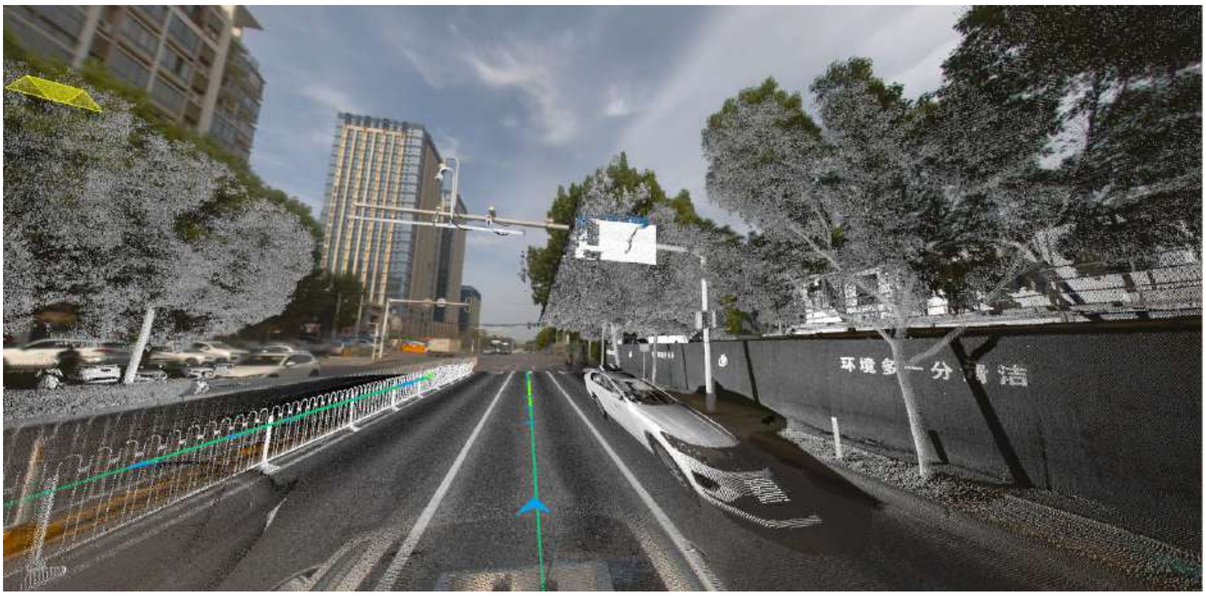
8.Click **Transform** button, the transformation is mainly to adjust the pose of the image in real time and superimpose the point cloud perfectly.



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.
- **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.

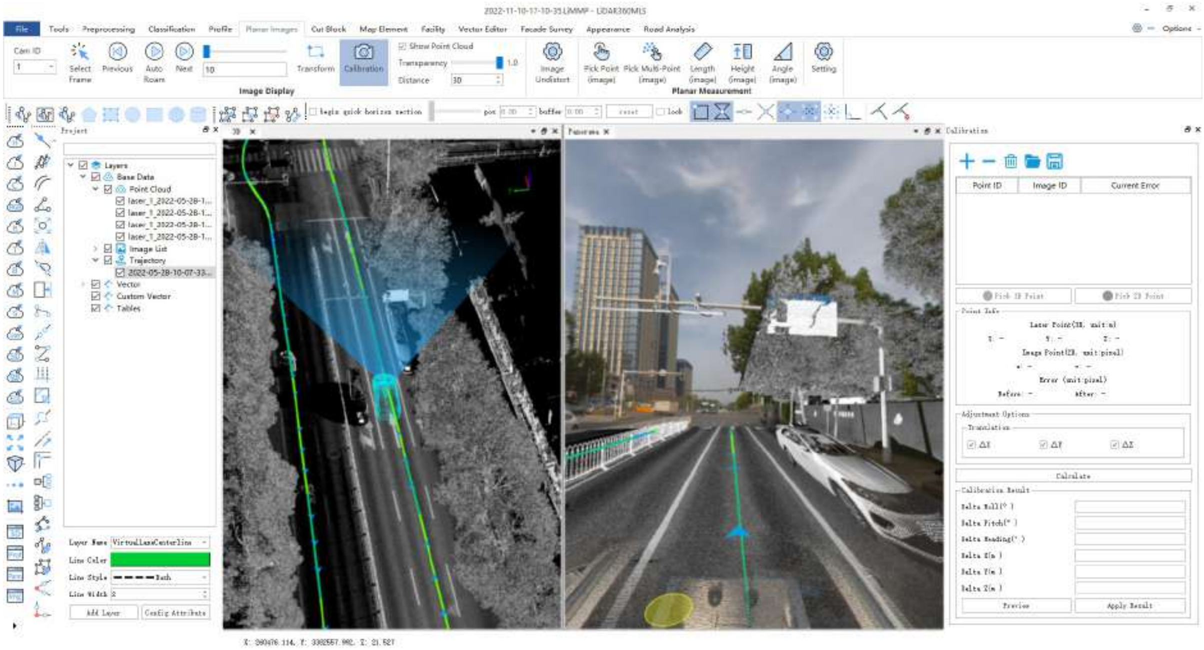
9.Due to the placement error between the image camera and the laser, there is still a certain deviation between the plane image and the point cloud data, which cannot be superimposed perfectly, as shown in the following figure:



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 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.

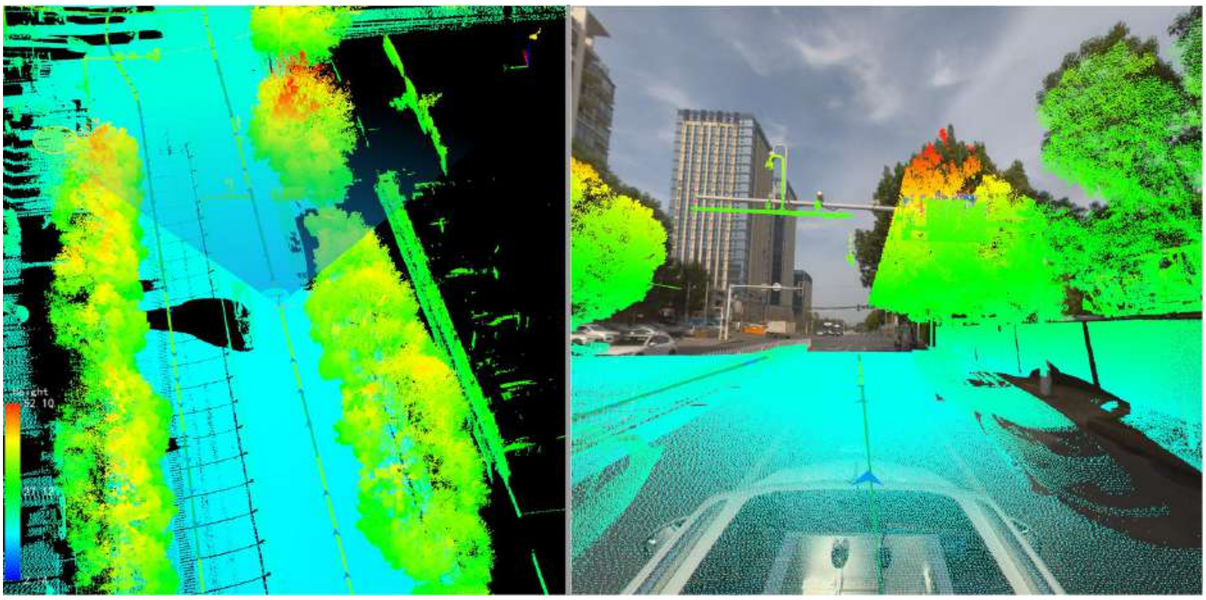
1.Switch to the **menu page and calibrate**  button, the calibration window will pop up on the right side of the user interface.



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 south, south, and northwest, that is, a total of 16 point pairs are selected. point pair).

The specific steps are:

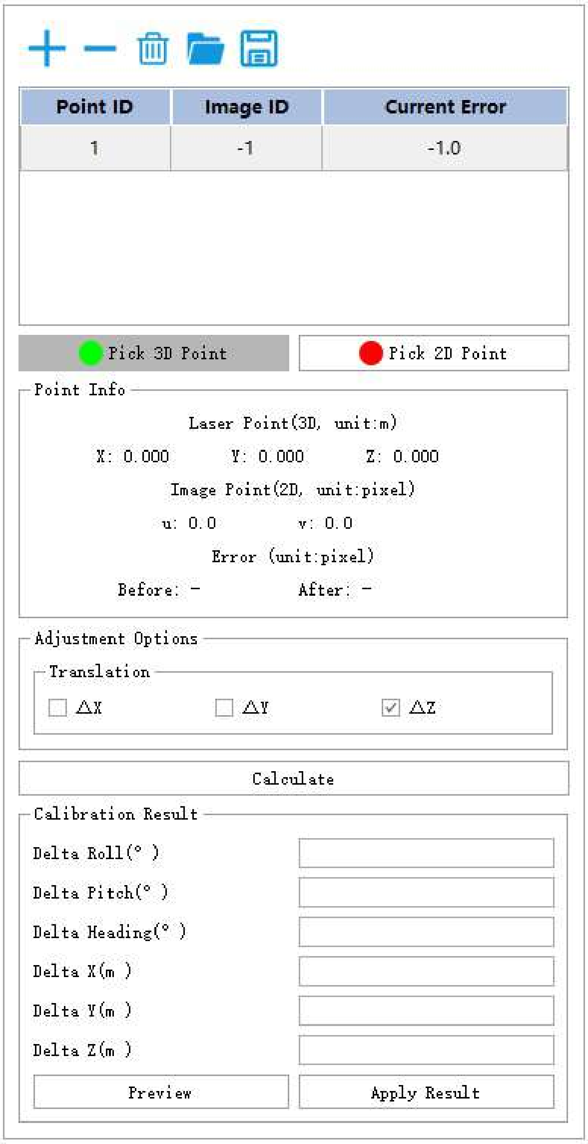
- Click the *Select Frame* button, click the triangle arrow in the 3D view window, and the image window will automatically switch to the corresponding image frame.



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

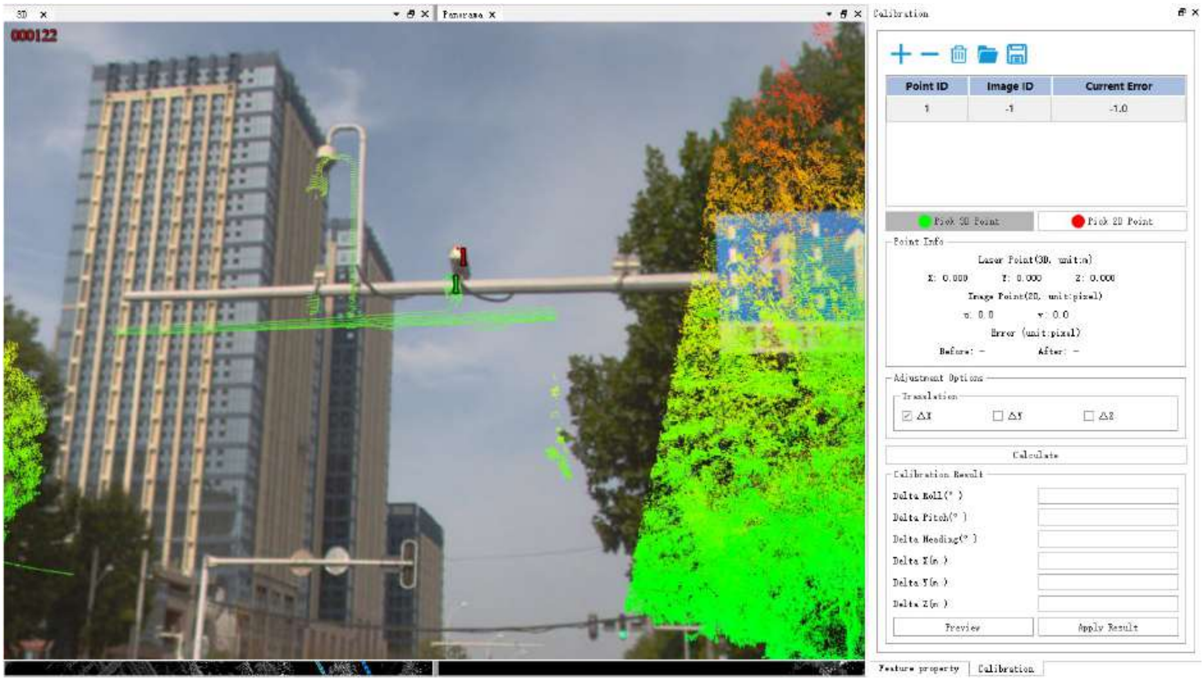
- Click the *Select 3D point* button to select the laser point corresponding to the image point in the image window.
- Repeat the above three steps until enough point pairs are selected on the current image frame (four point pairs are recommended).
- Select the corresponding image frames in the other 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.

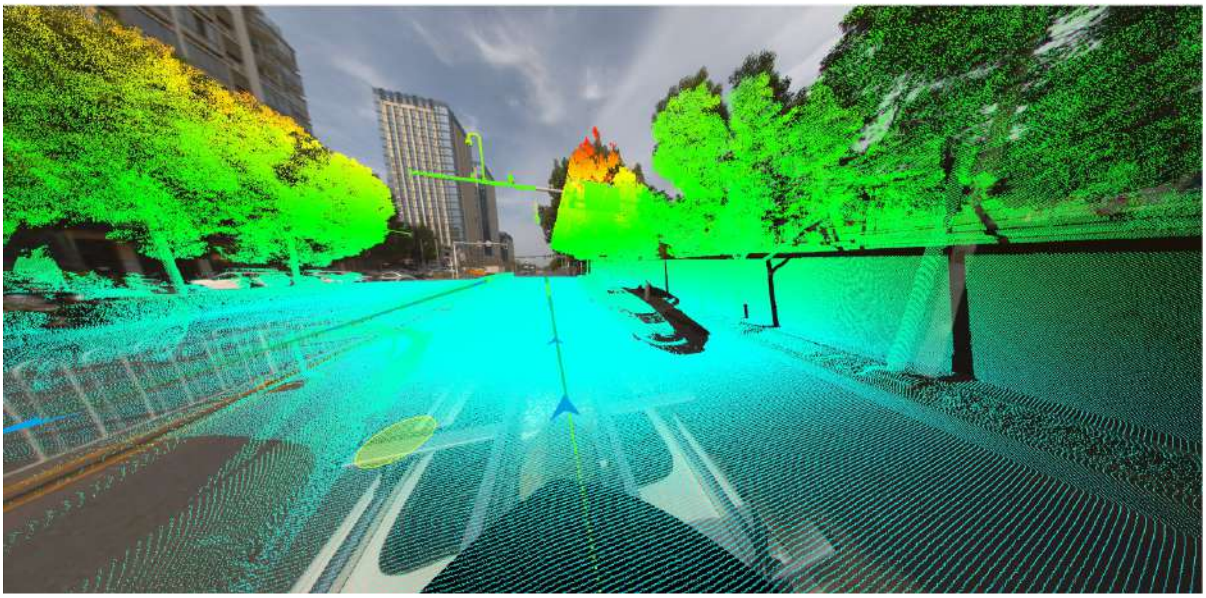


4.Click the *Preview* button to preview the calibration result.

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, set the point cloud display mode to display by intensity 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, 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.

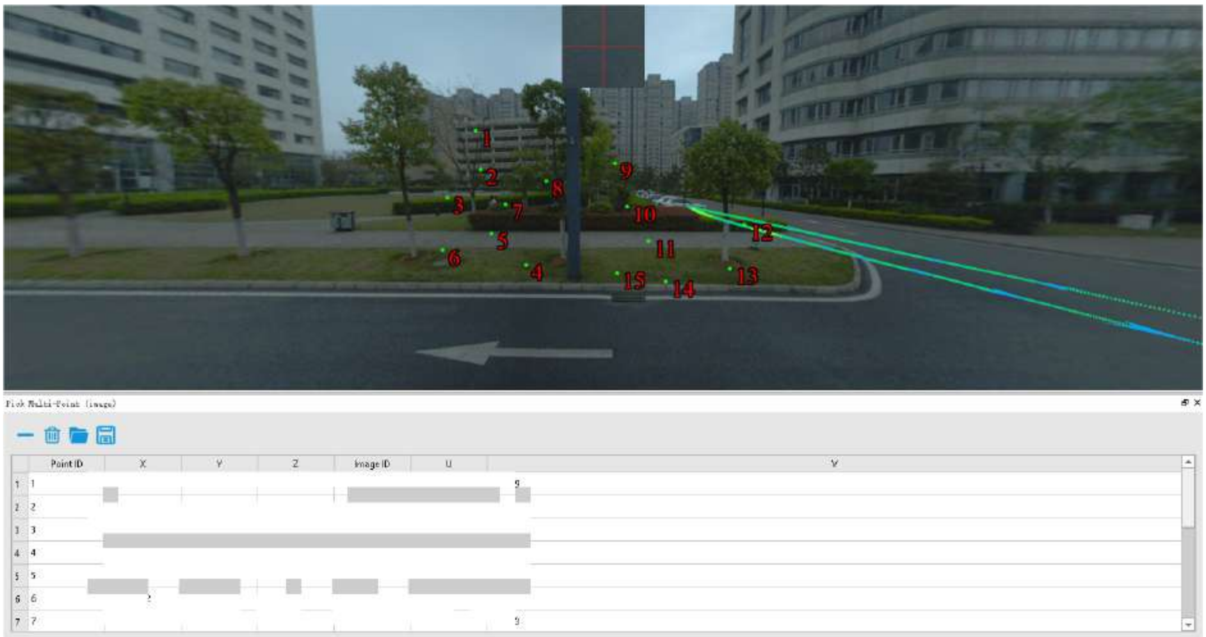
Multi Pick Point (Depth Interpolation)

Function Description: The multi-point selection tool uses the mouse to click interactively to 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 **multi-point selection (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.

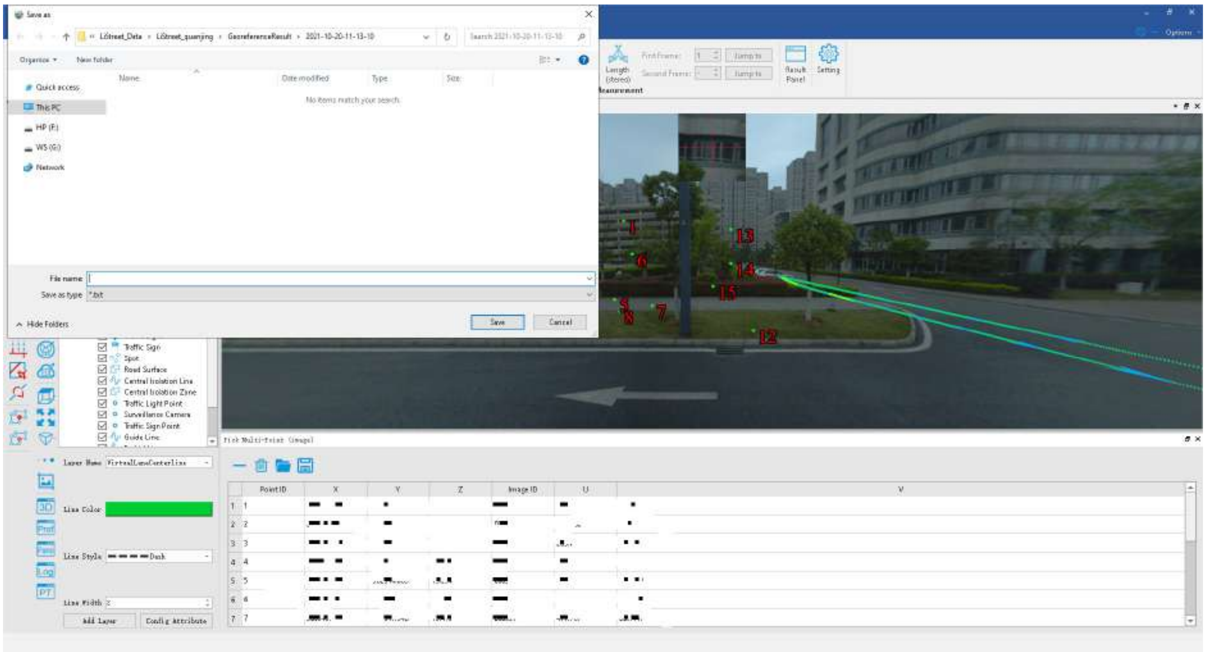


parameter settings

- **X**: The X coordinate value of the point cloud data.
- **Y**: 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.

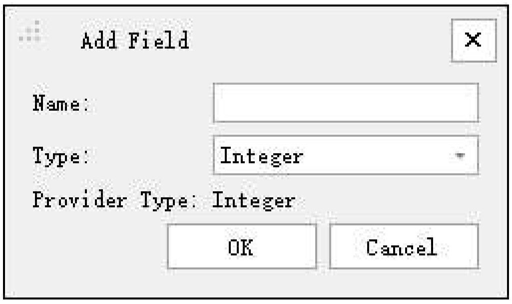
3.The selected point set is exported in *.txt* format. Click the button to save the coordinates and other attribute information as a *.txt* file.



4.If a point has been created, 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 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：This tool is applicable to point cloud data, which calculates the distance between two consecutive points.

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)

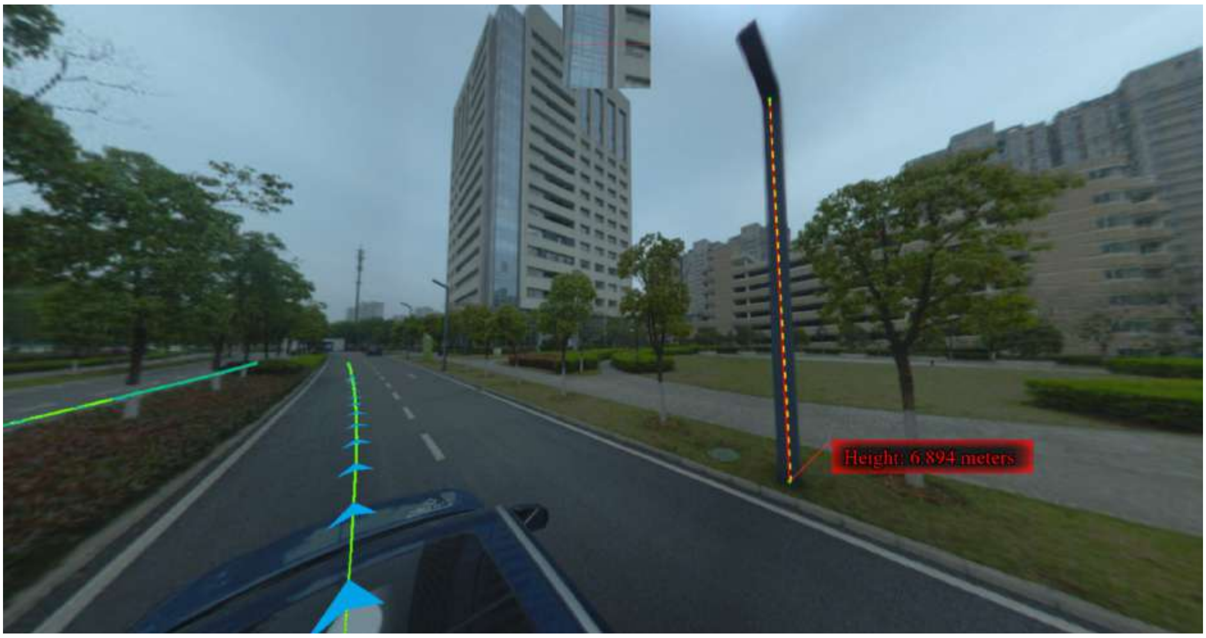
Function Description: This tool is applicable to point cloud data which calculates the relative height difference between two points in panorama window.

Steps

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

2.Select the reference 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)

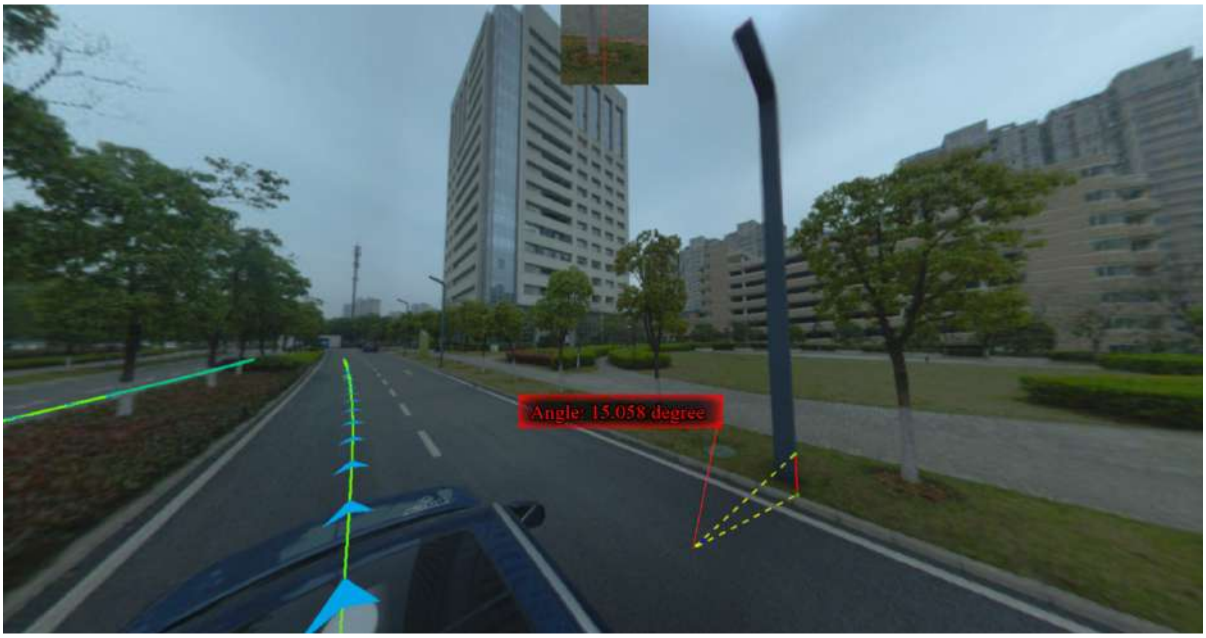
Function Description：This tool is applicable to point cloud data which calculates the angle of pitch between two points in Panorama view.

Steps

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

2.Select the reference point of angle measurement by left-clicking in (Panorama) 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 is displayed in a label as follows.



Panorama Measurements based on Forward Intersection

Using the measuring point to select the tie-points on the two-frame image, and combined 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)

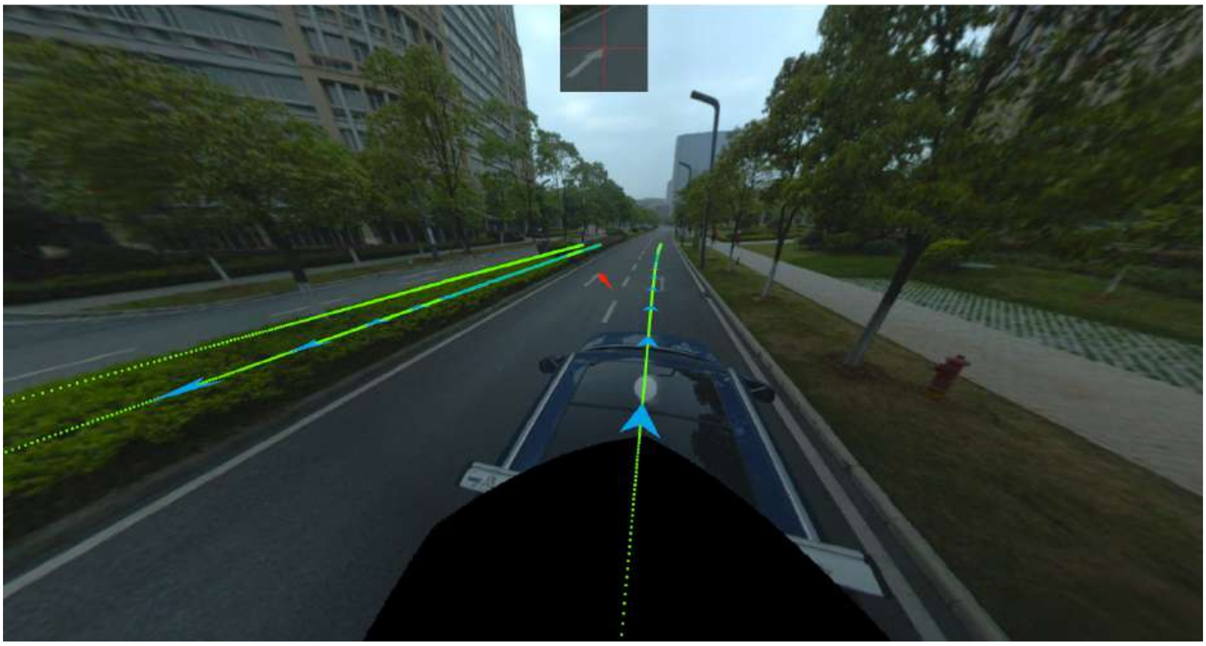
Function Description: This tool is applicable to panoramic data, the attributes that can be queried contain position information.

Steps

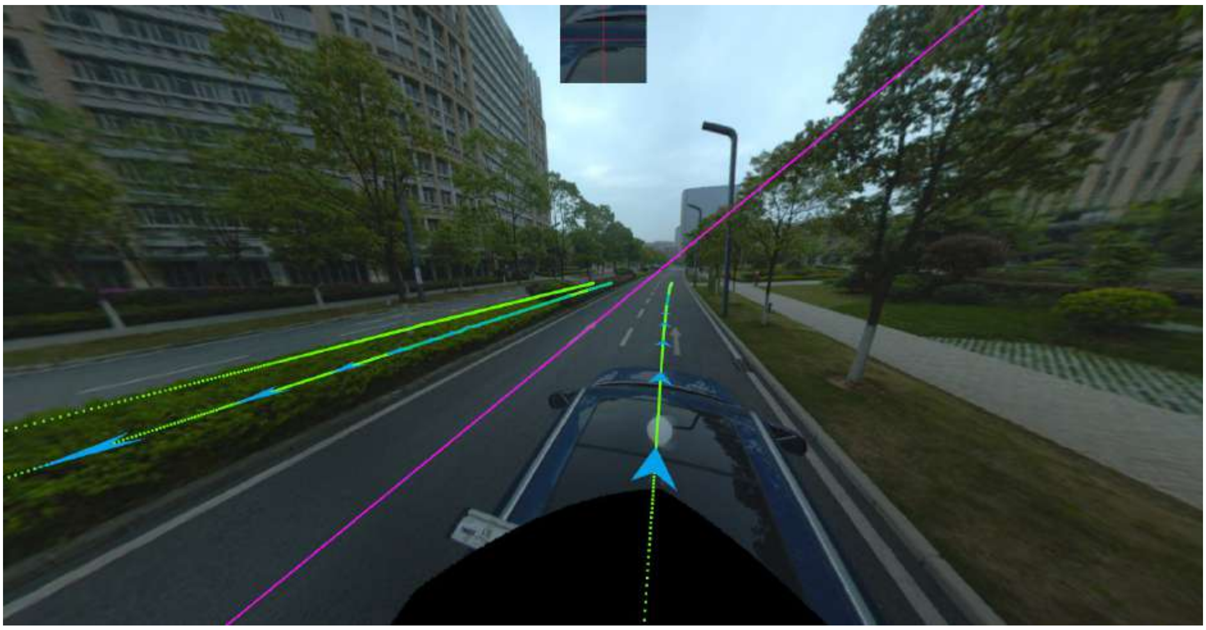
1.Click Pick Point(stereo)  button via Panorama Measurement mode.

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

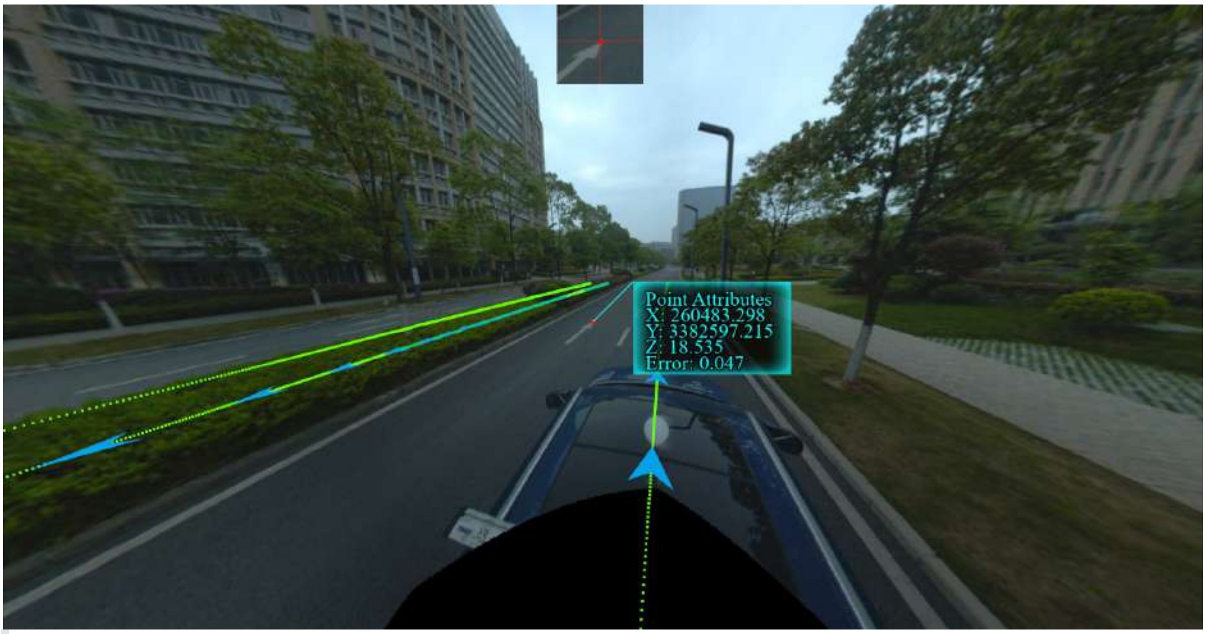


The panorama window will switch to the second frame image automatically and the auxiliary line will display：



3.The image will update automatically in panorama window. Select the second corresponding point 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 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：This tool is applicable to point cloud data, which calculates the distance between two consecutive points.

Steps

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

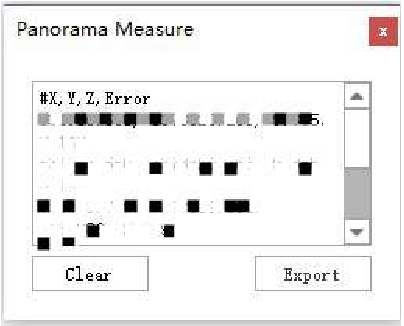
2.Left-click to select the starting point in the scene, the operation is the same as the Pick Point step.

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.

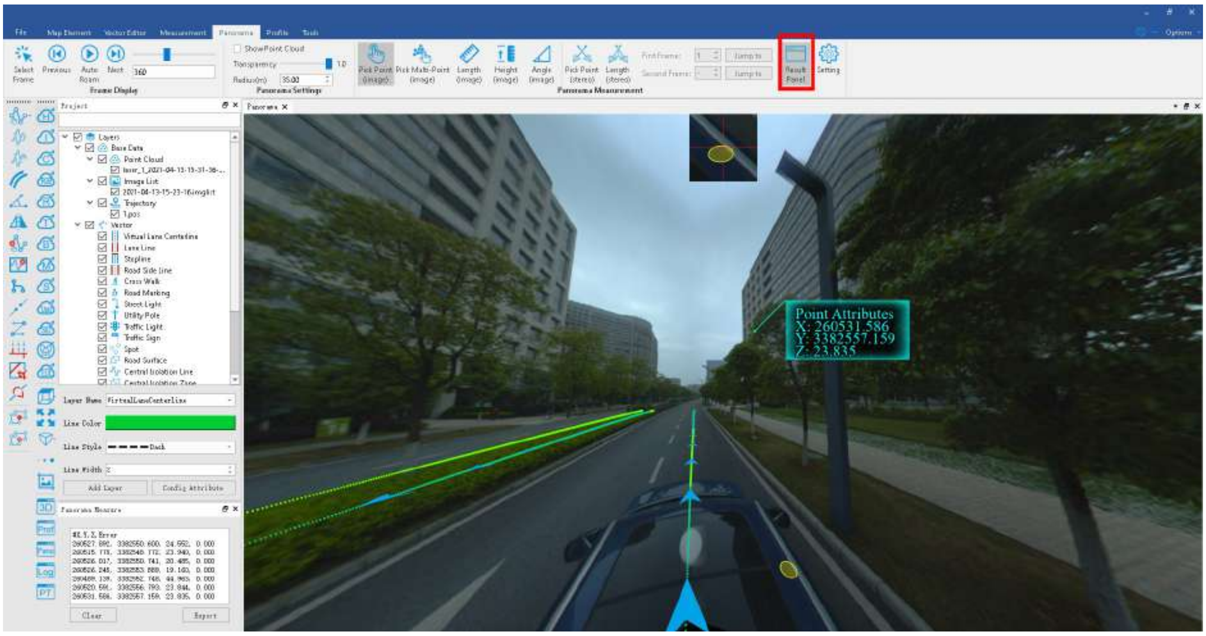
Result Panel

功能描述：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.



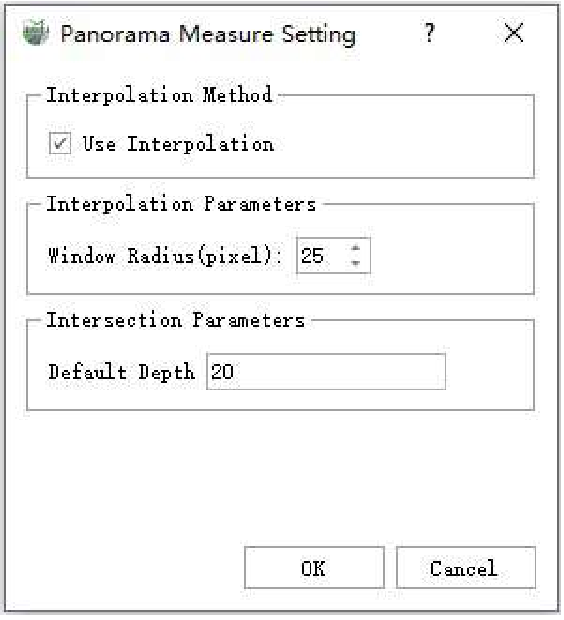
2.Click Clear button to clear Result Panel.

3.Click Export button to export measured points list.

Dialog Setting

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

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



Parameters Introduction

- **Use Interpolation**: 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.
- **Windows Radius**: 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.
- **Default Depth**: It is the assumed depth of measured points. And it is recommended in the second frame image when using the function of Panorama Measurements based on Forward Intersection.

Planar Image

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

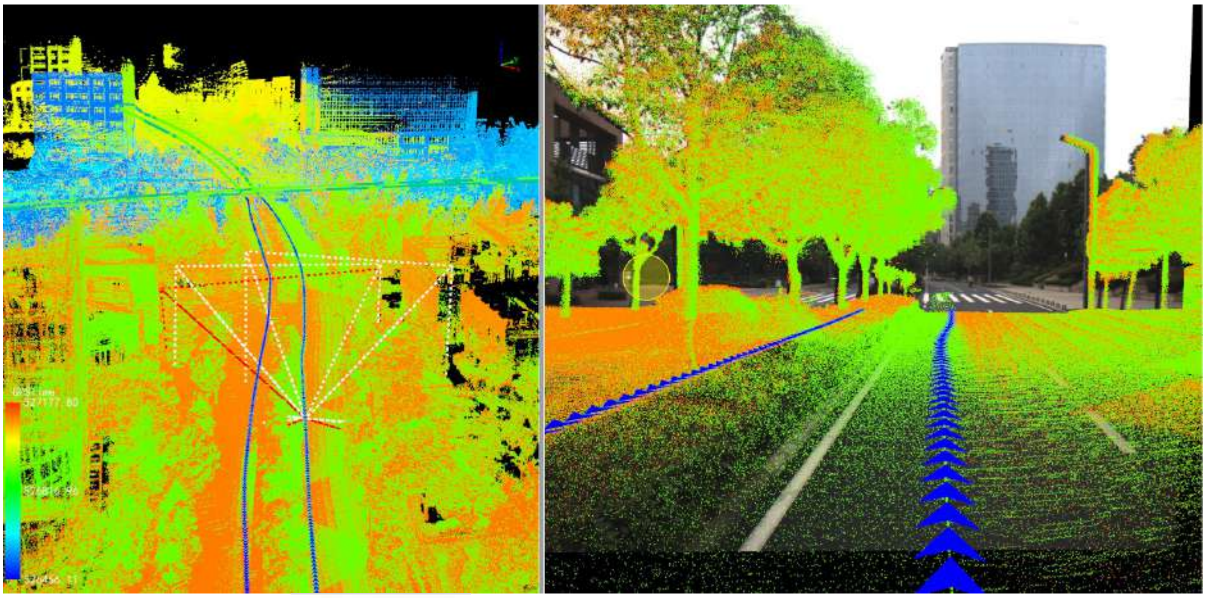
- Point Cloud and Image Roaming
- Image Measurement
- Image Undistorting
- Setting Dialog

Point Cloud and Image Roaming

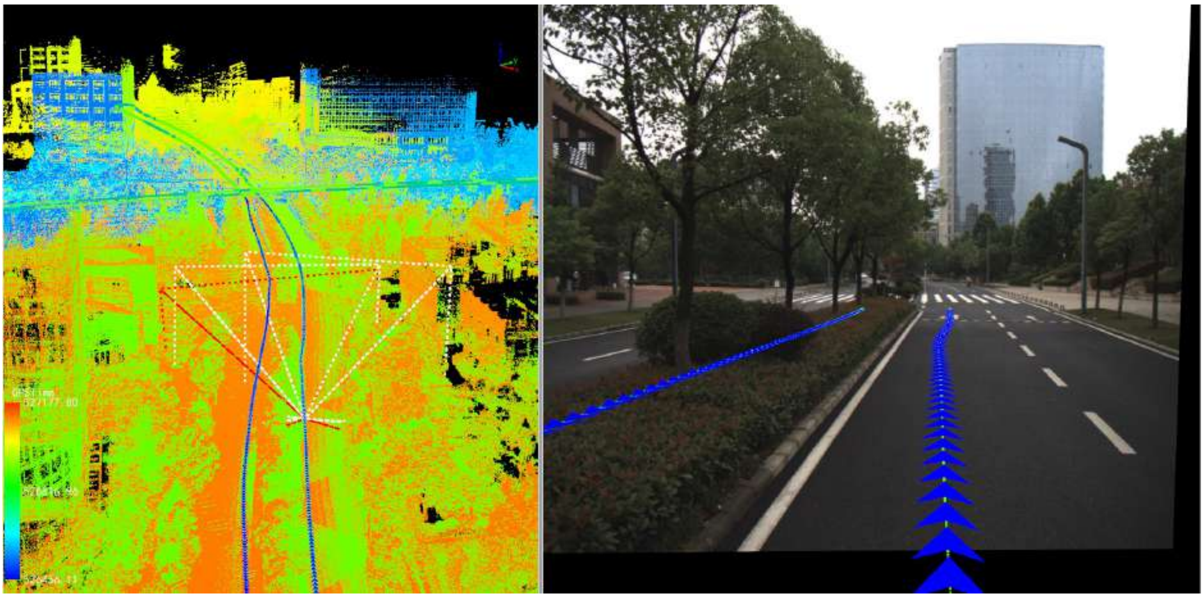
Function description: The software supports point cloud and image roaming. The original image camera file must be added when creating a new project, and the menu bar will be activated.

Steps

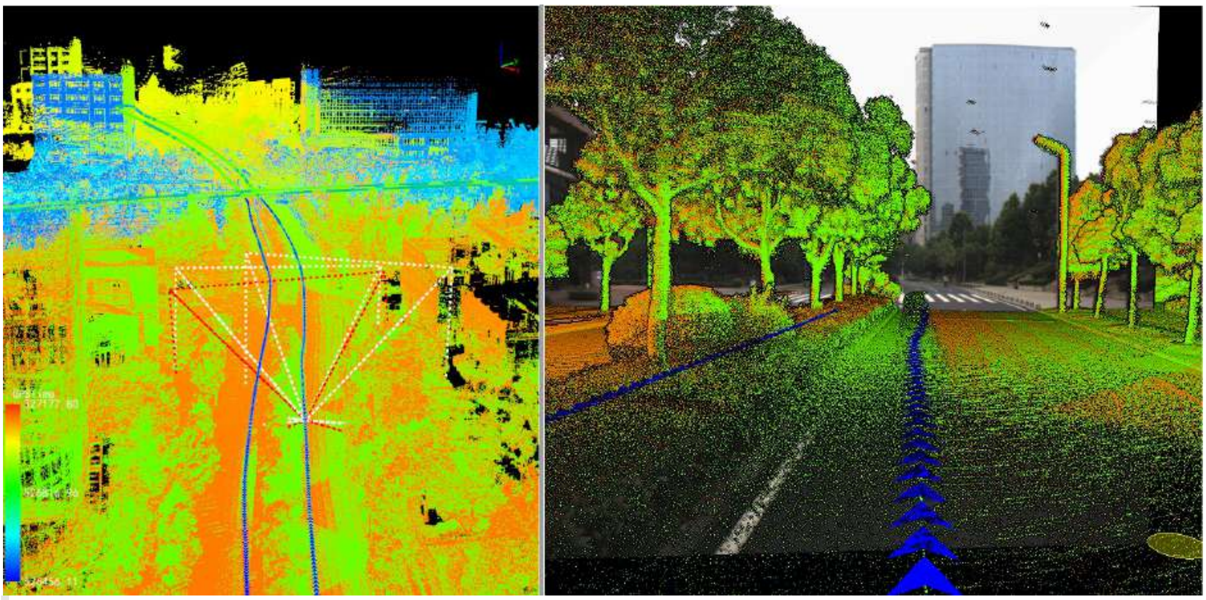
1.After creating a new project, a 3D window and an image window will be opened, the point cloud data will be displayed in both the 3D window and the image window, and the image image will only be displayed in the image window. You can choose to show or hide the corresponding data through the check box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



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



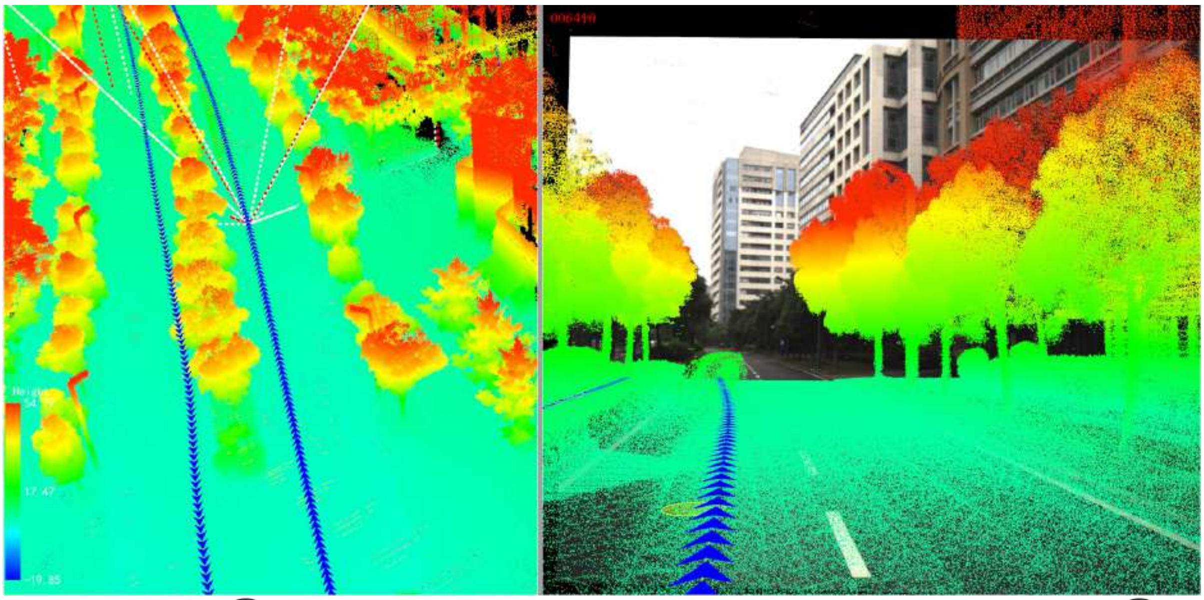
3.Click the color bar tool on the left side of the window to switch the display mode of the point cloud, including display by elevation, intensity, category, RGB, number of echoes, time, mixed display, combined display, display by selected color and EDL display, EDL can be used in conjunction with other 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, you can select the image exposure position on the track in the 3D window or the image window, and jump to the selected image position.

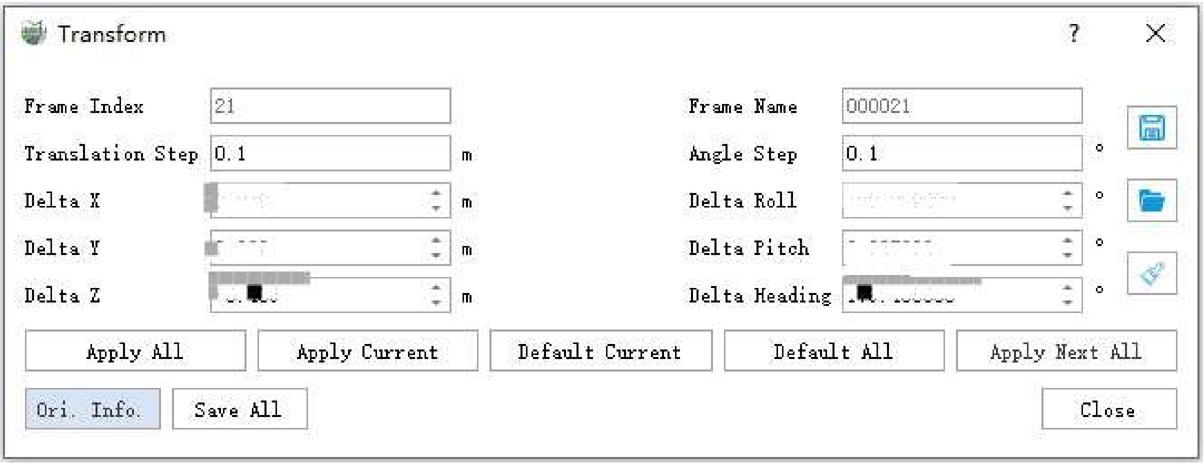


6.Click **Next Frame** button jumps to the next frame of the current image. Click on **previous frame** button jumps to the previous frame of the current image.

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

8.Click **AutoRoam**  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.

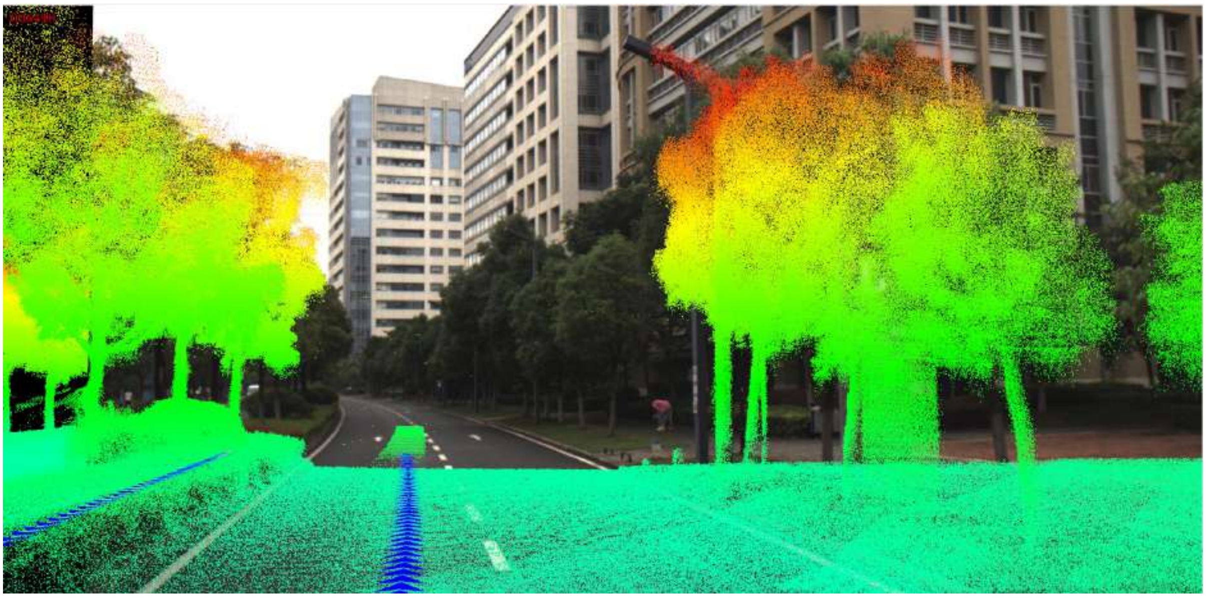
9.Click **Transform**  button, the transformation is mainly to adjust the pose of the image in real time and superimpose the point cloud perfectly.



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.
- **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.

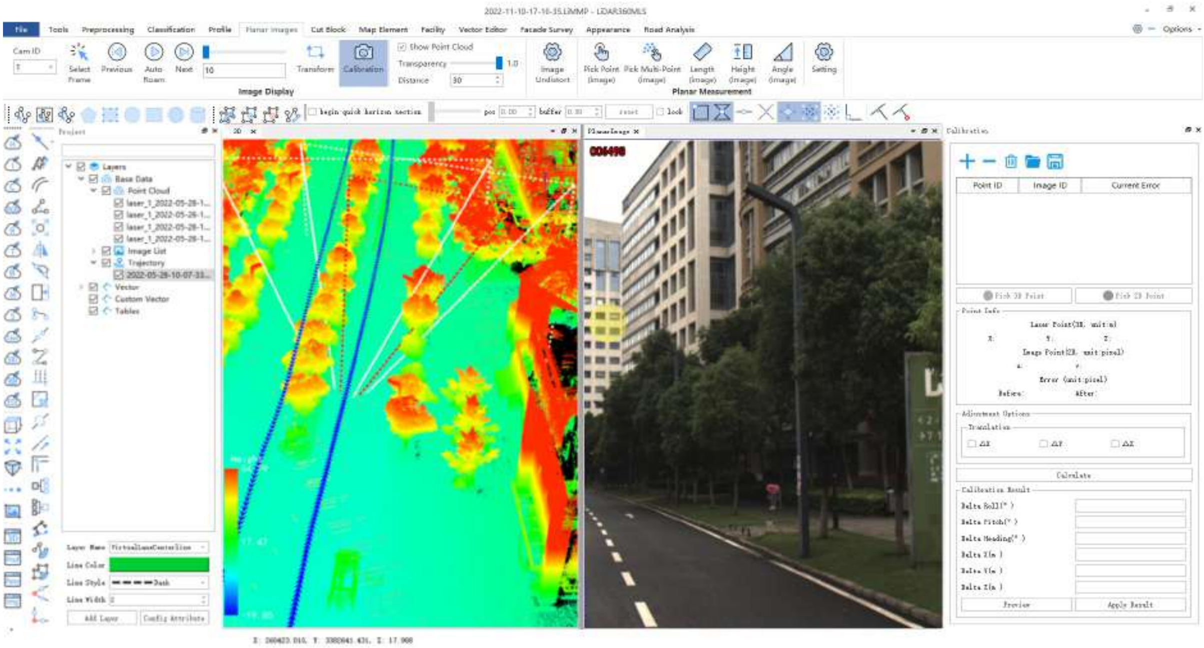
10.Due to the placement error between the image camera and the laser, there is still a certain deviation between the plane image and the point cloud data, which cannot be superimposed perfectly, as shown in the following figure:



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.

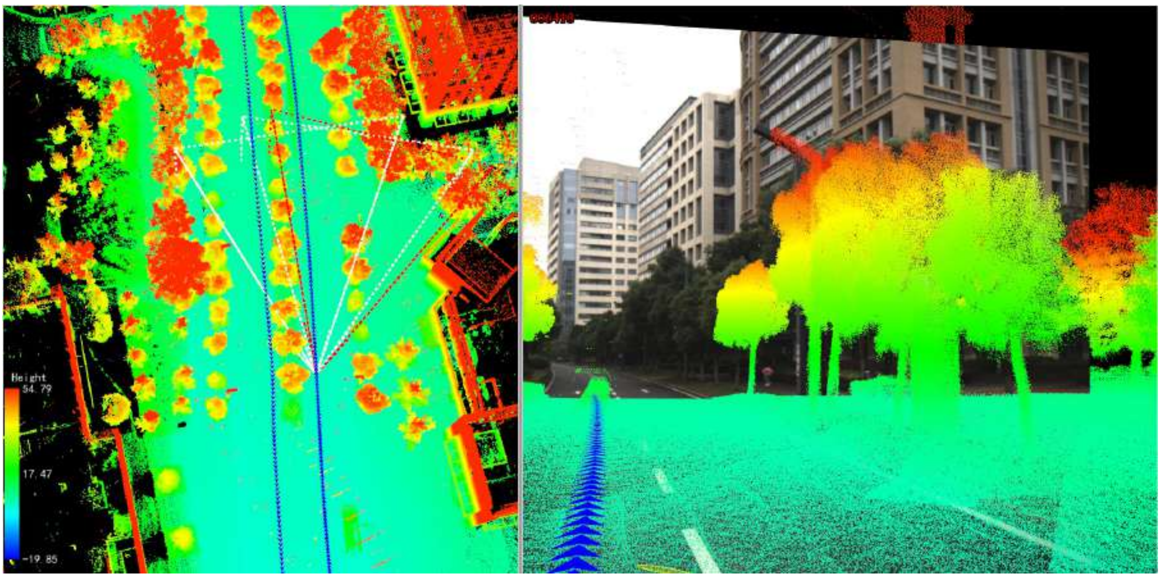
> Note: Set the point cloud display depth by adjusting the menu bar Image->Image Display->Distance, so as to hide the distant point cloud and facilitate the observation of the area of interest. This little trick can also be used in the calibration point selection process below.

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

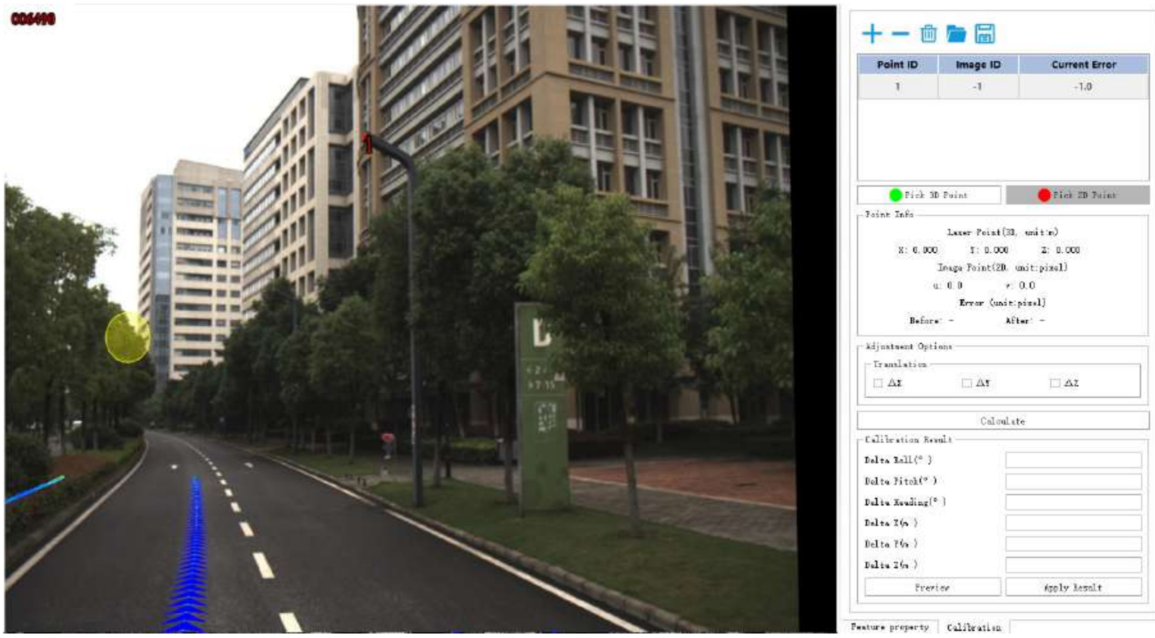


2.Add calibration point. It is recommended to select camera exposure points in multiple directions of the scene, and then select multiple point pairs at each exposure point, at least 4 point pairs need to be added (for example, 4 point pairs are selected at the four exposure moments of the south, south,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

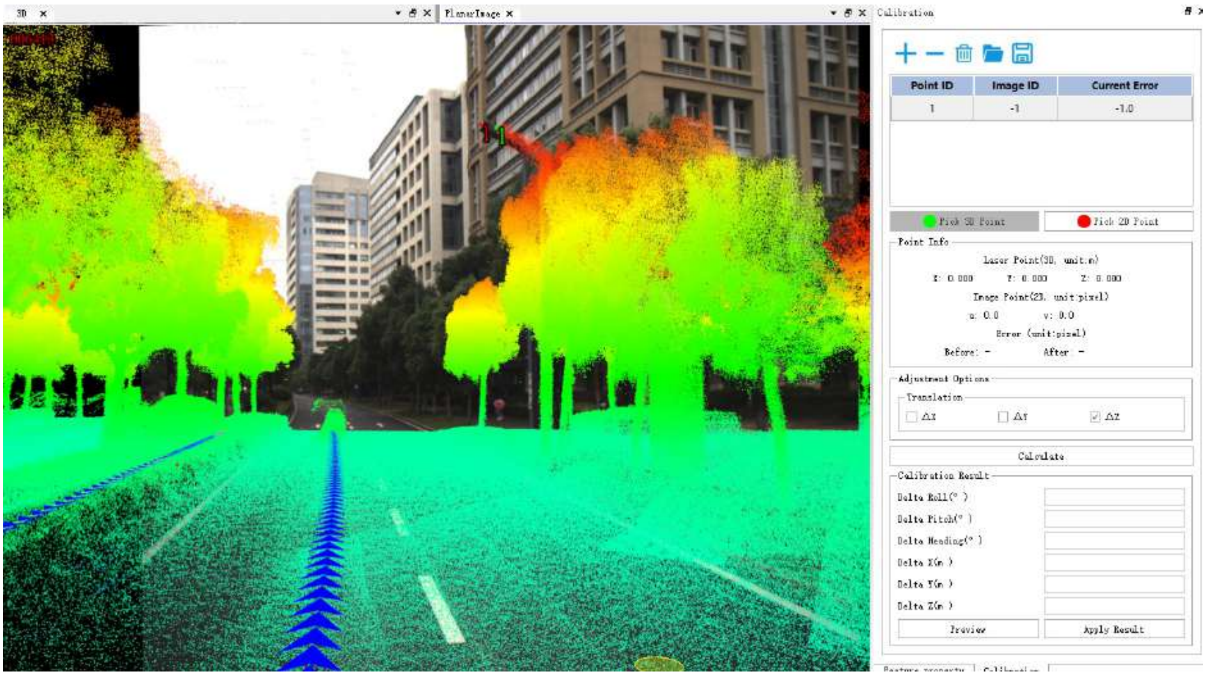


- Click the *Select 3D point* button to select the laser point corresponding to the image point in the image window.
- Repeat the above three steps until enough point pairs are selected on the current image frame (four point pairs are recommended).
- Select the corresponding image frames in the other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

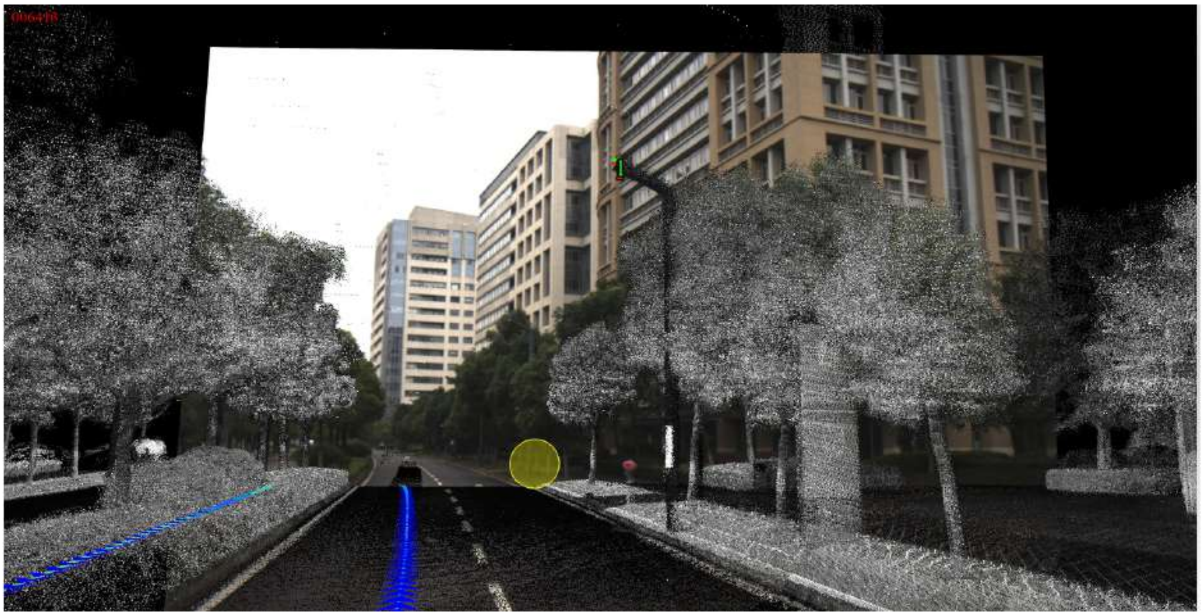
3.Click the *Calculate* button to start the calibration calculation.

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.

Planar Measurement

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

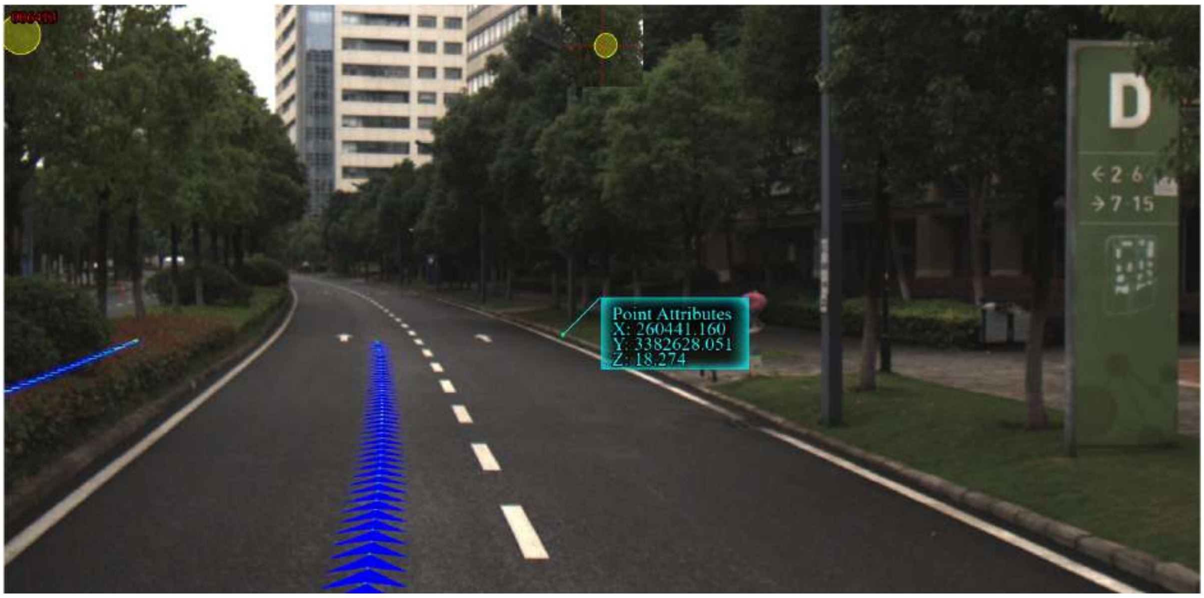
Pick Point (Depth Interpolation)

Function Description：This tool is applicable to panoramic data, 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.Right click mouse button to cancel the selected point.

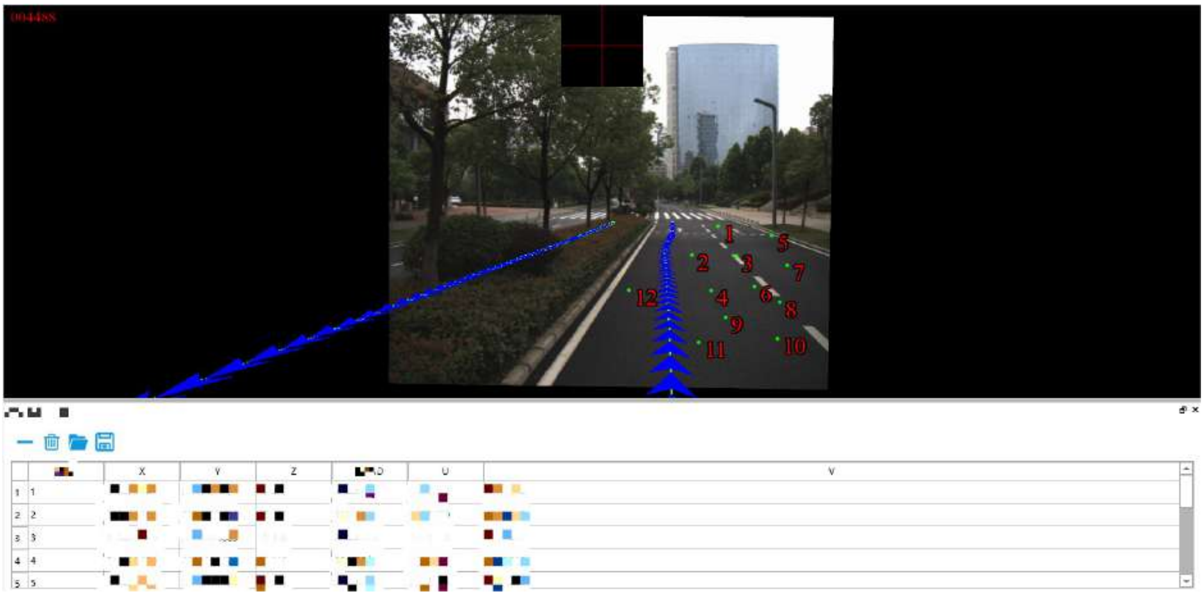
Multi Pick Point (Depth Interpolation)

Function Description: The multi-point selection 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 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 **multi-point selection (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.

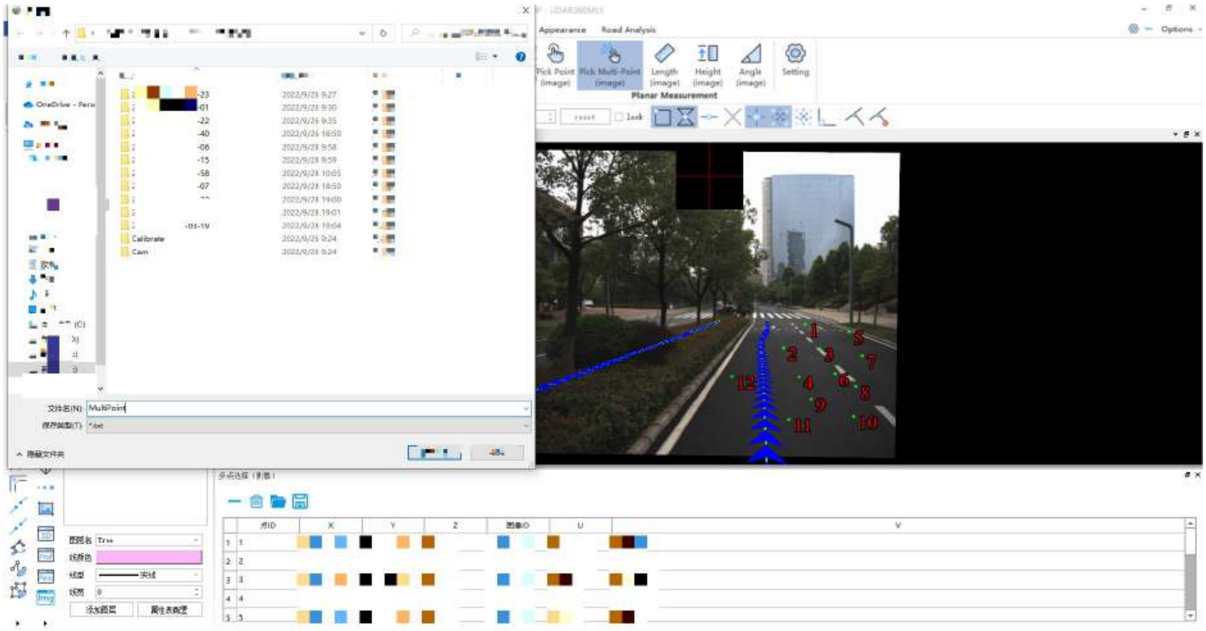


parameter settings

- **X**: The X coordinate value of the point cloud data.
- **Y**: 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, 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 image view window.

Length Measurement (Depth Interpolation)

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

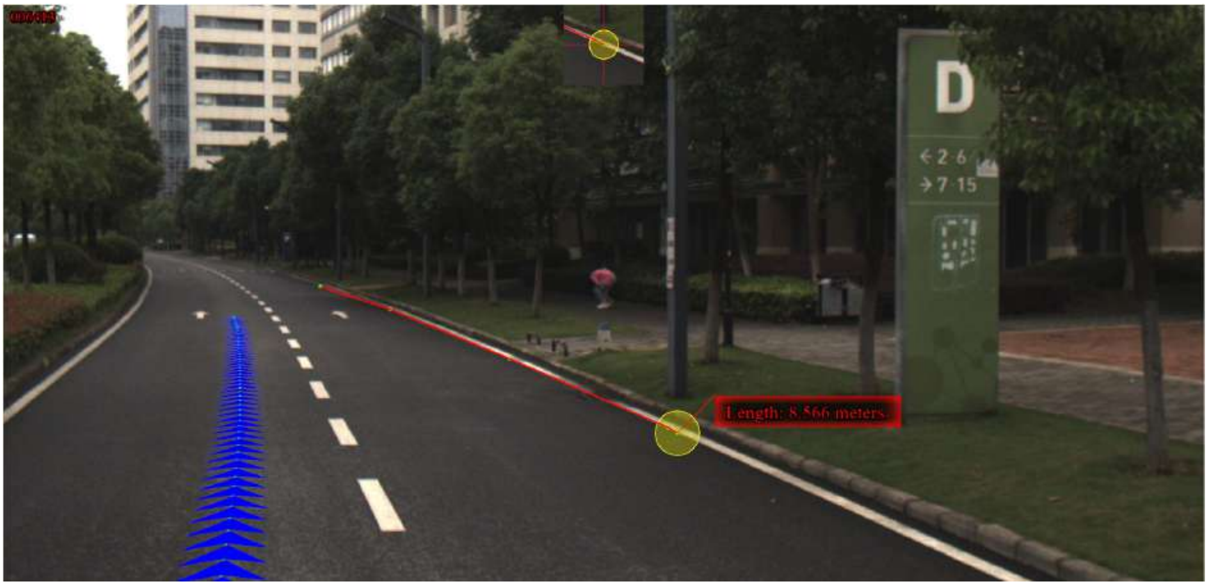
Steps

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

2.Left click in the panoramic display window (*Image*) 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)

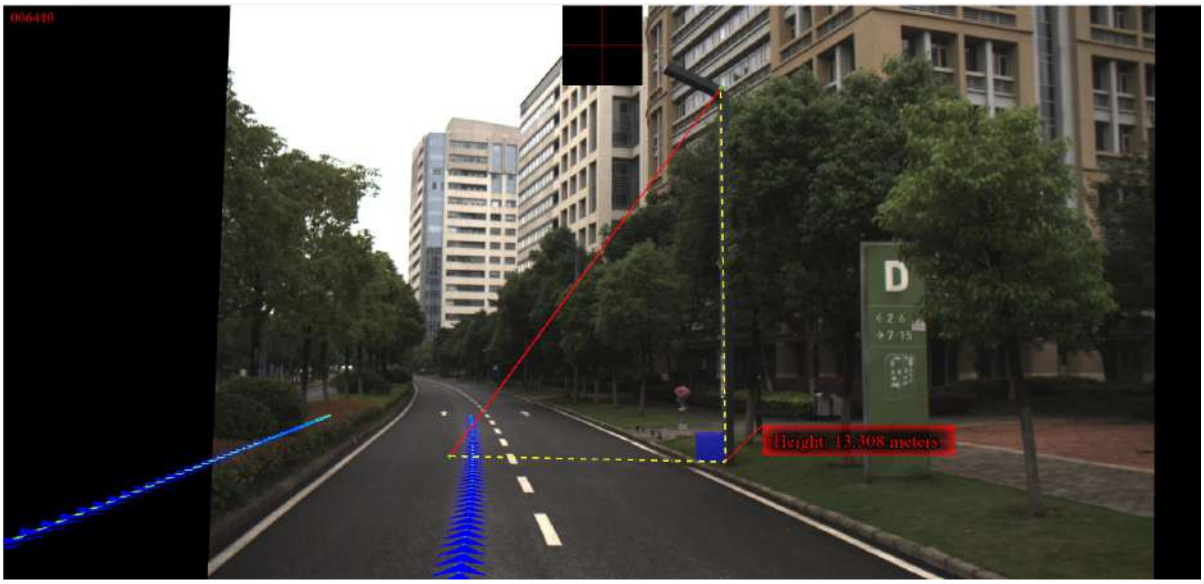
Function Description： This tool is applicable to point cloud data which calculates the relative height difference between two points in panorama window.

Steps

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

2.Select the reference 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

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

Steps

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

2.Select the reference point of angle measurement by left-clicking in (Panorama) 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 is displayed in a label as follows.

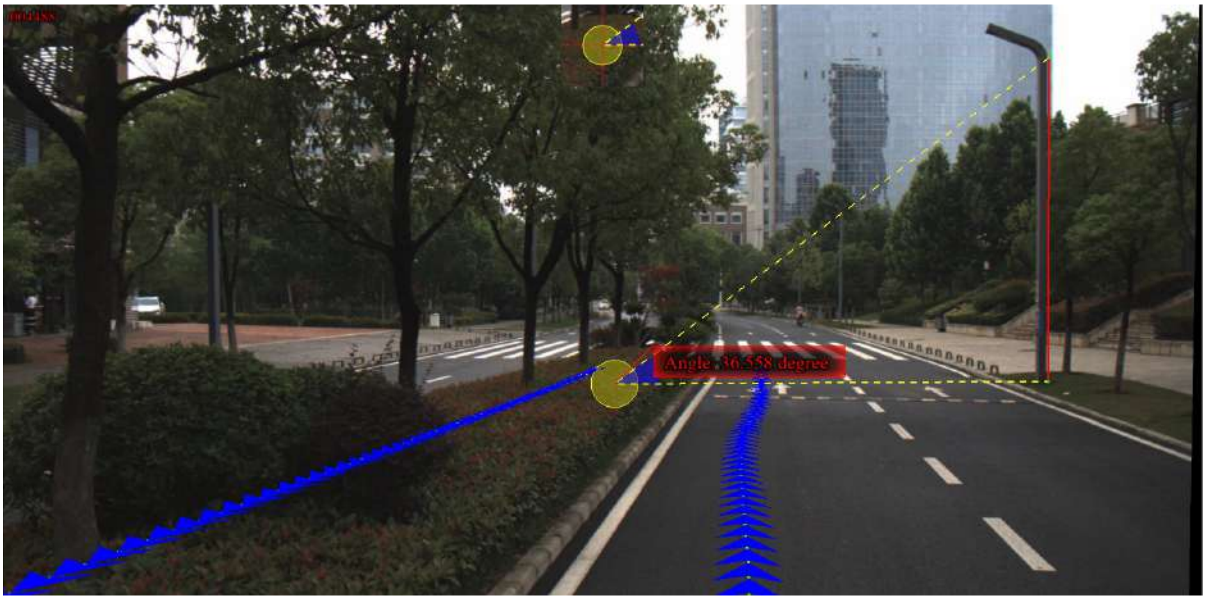


Image Undistorting

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

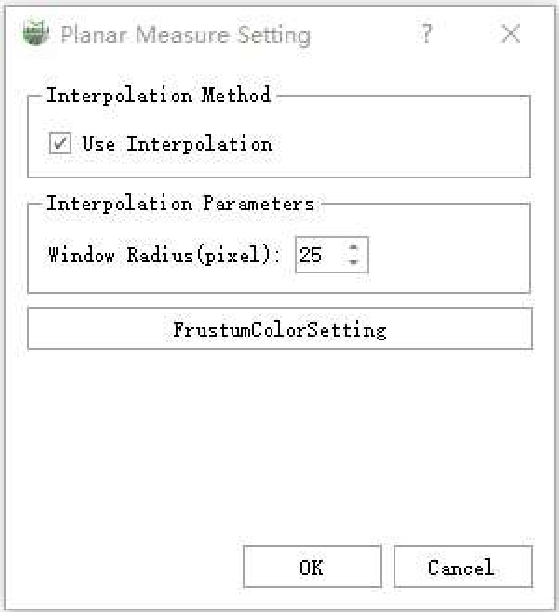
1.Image distortion is due to the lens manufacturing precision and the deviation of the assembly process will introduce distortion, resulting in distortion of the original image. The distortion of the lens is divided into two categories: radial distortion and tangential distortion.

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.
- **Visual cone color settings**: Visual cone color settings.

Preprocessing

After completing the high-precision point cloud solution, the trajectory and point cloud of the result data can be optimized to a certain extent, such as correction, splicing, quality inspection, etc.

Trajectory segmentation

- Segmentation Method
- Trajectory Segment table
- Trajectory Graph
- Trajectory Quality Detection
- Trajectory Burst Repair

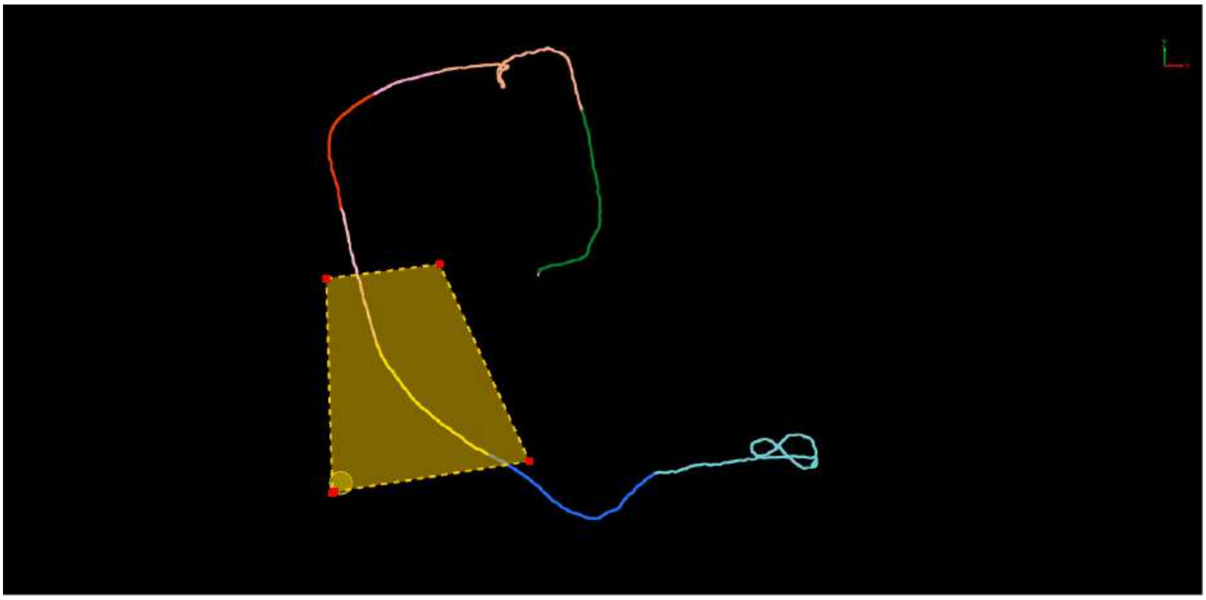
Split method

Split by polygon

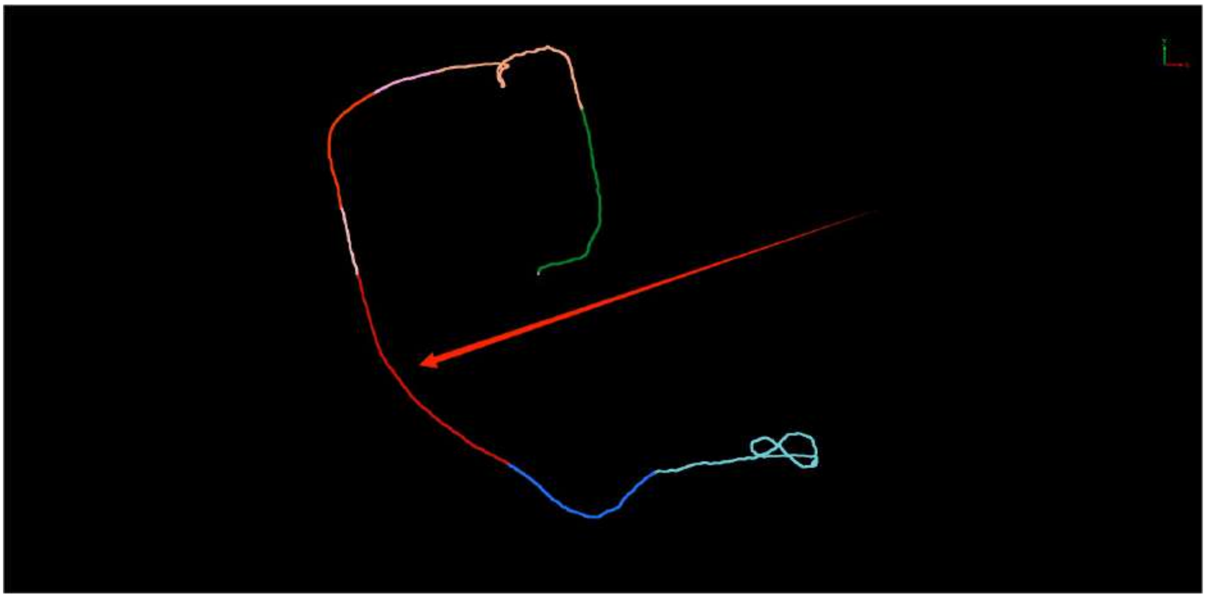
Step

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.

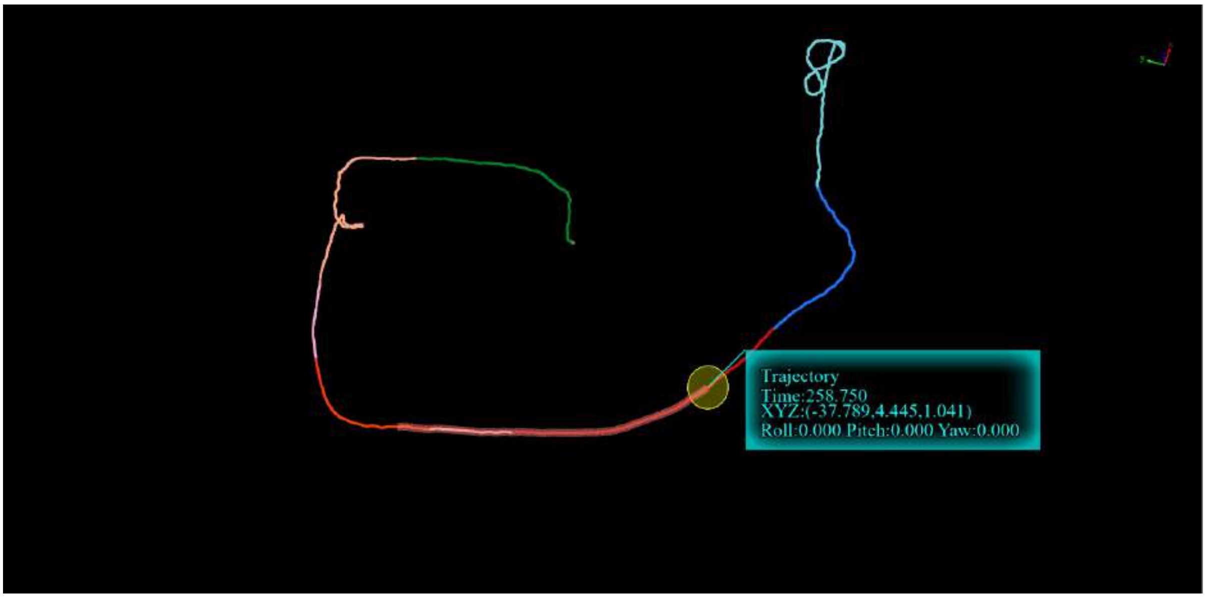


Select on Trajectory

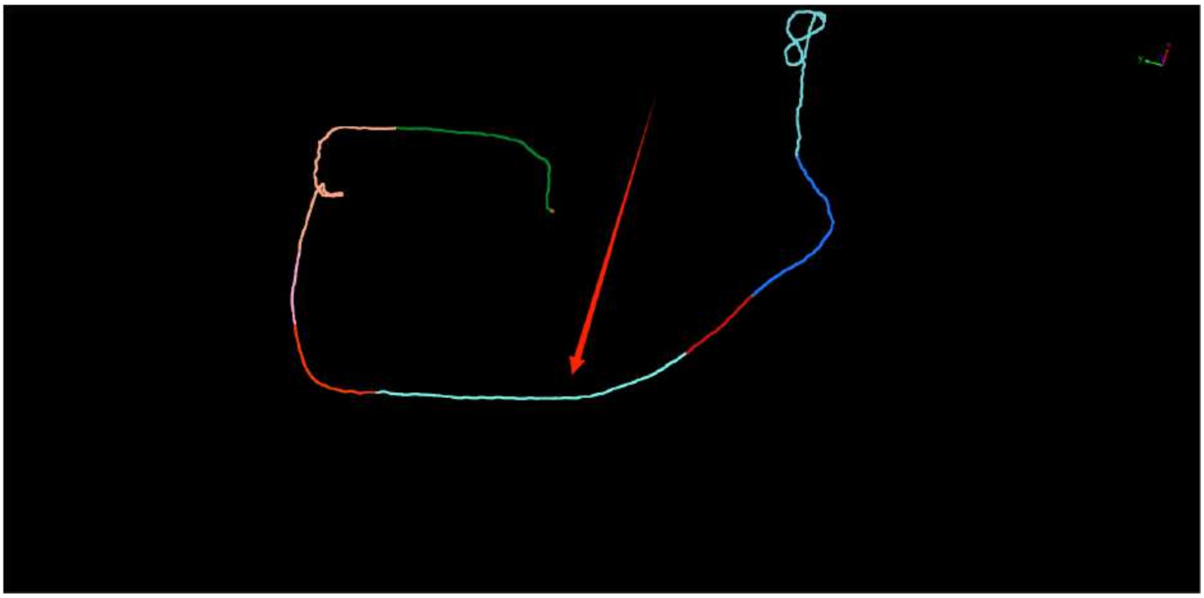
Step

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

2.Click on the result 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 is divided.



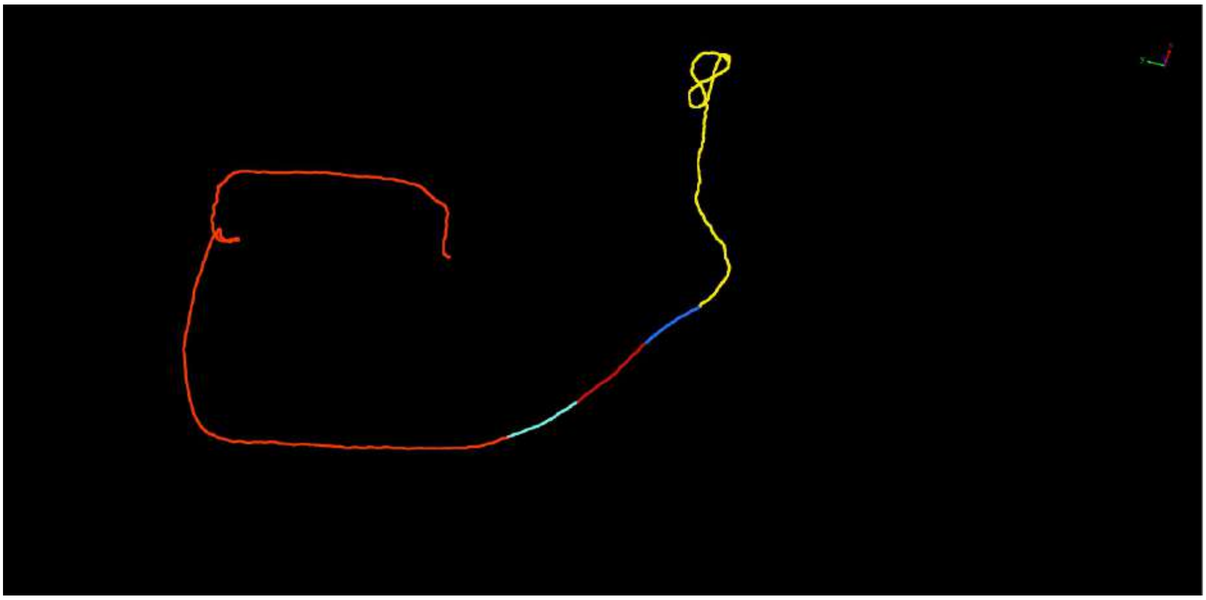
Invert Select on Trajectory

Step

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

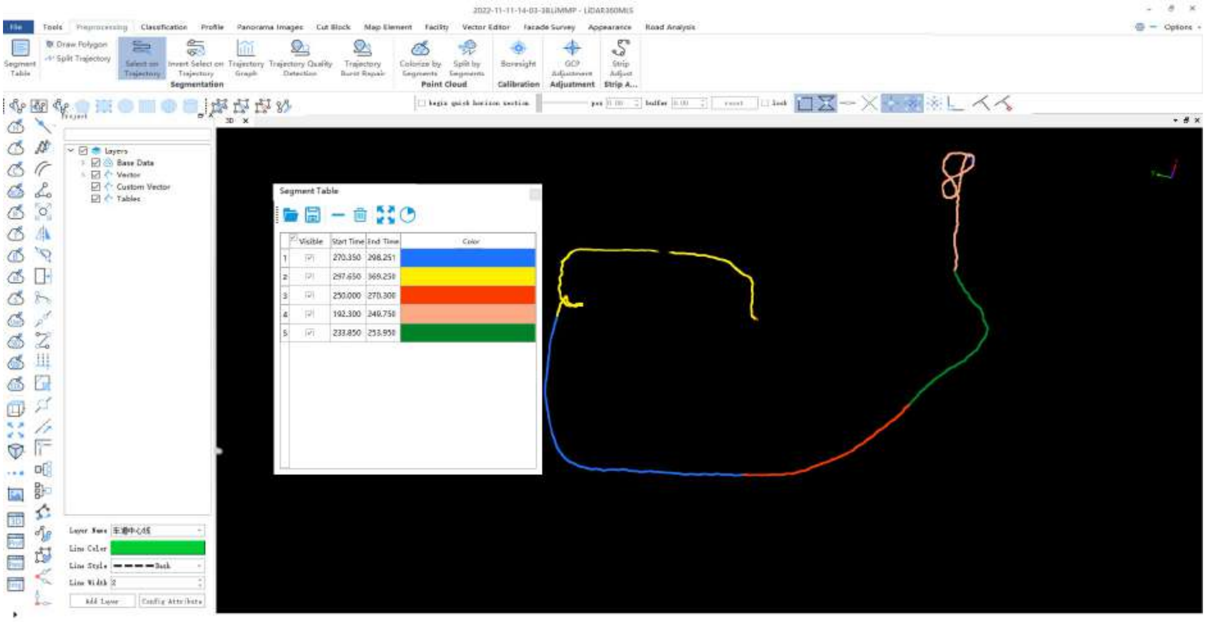
2.Click on the result track in the 3D display window to select the starting point of the segment.

3.Select the end point of the segment along the track, and the 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

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

Open

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

Delete

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

Empty

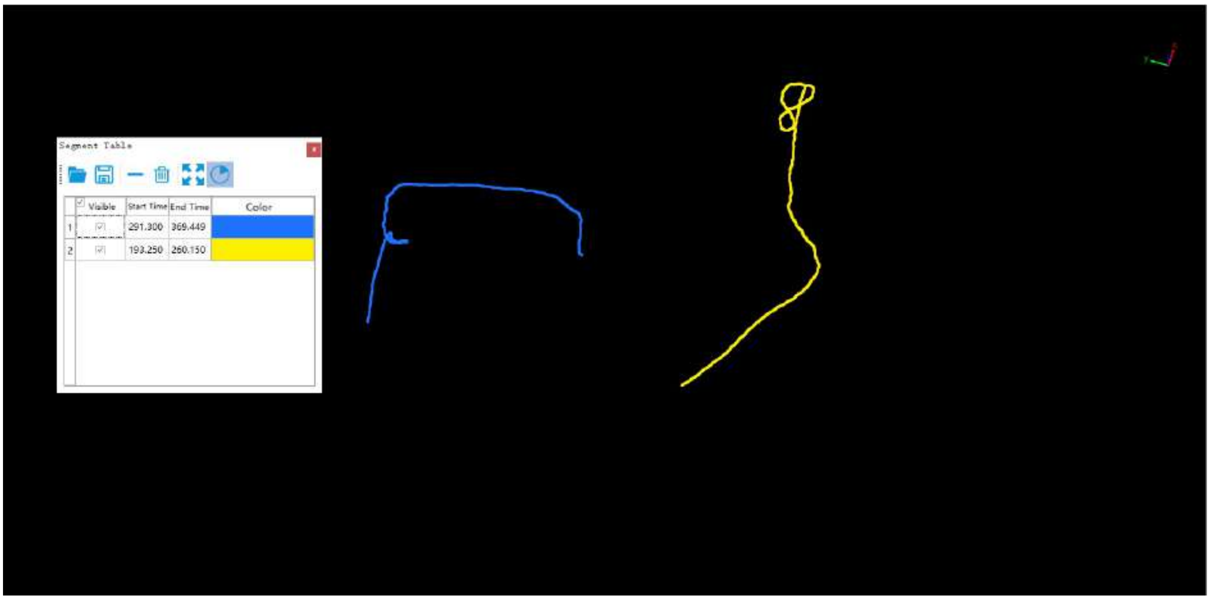
Click the **clear** button  , delete all track segments in the table.

Expand

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

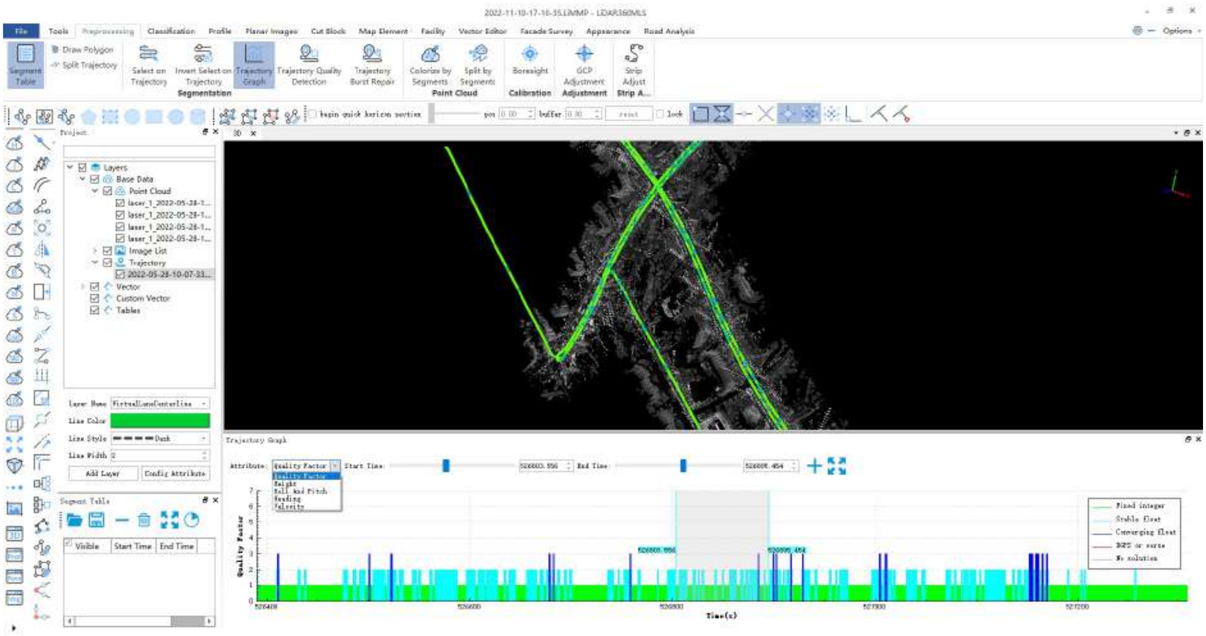
Hide

Click the **hide** 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

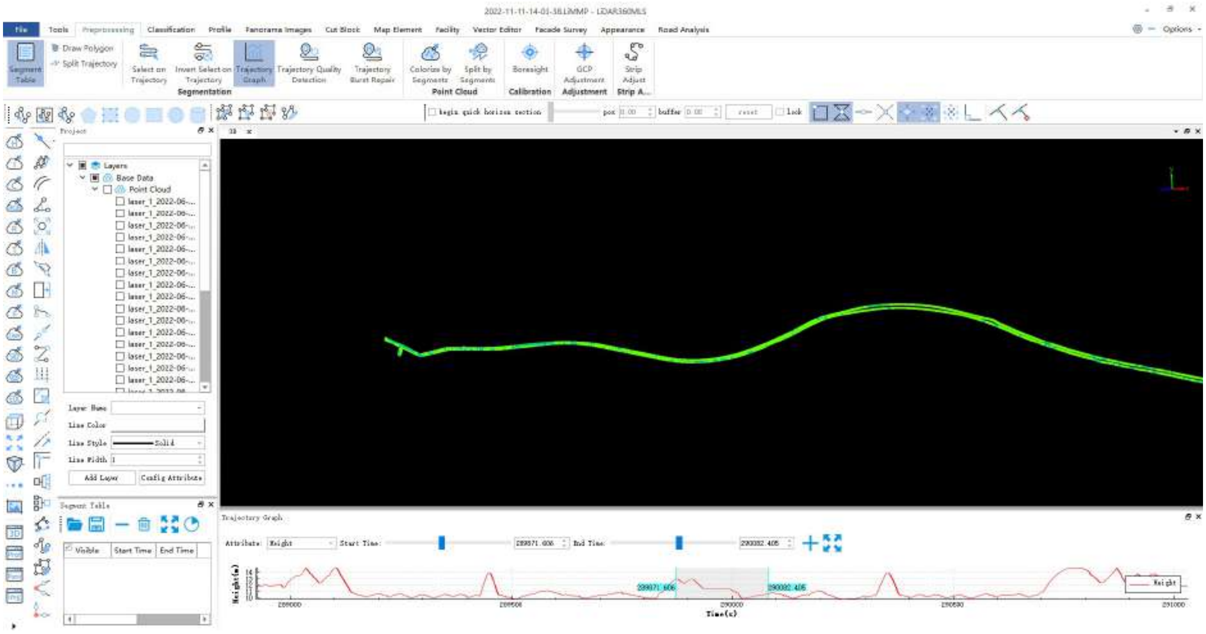
Click the selection box after the **Properties** tab to adjust the information displayed in the segmented status chart. You can choose to display by height, display by quality factor, display by roll angle and pitch angle, display by azimuth, and display by speed.

Add trajectory 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

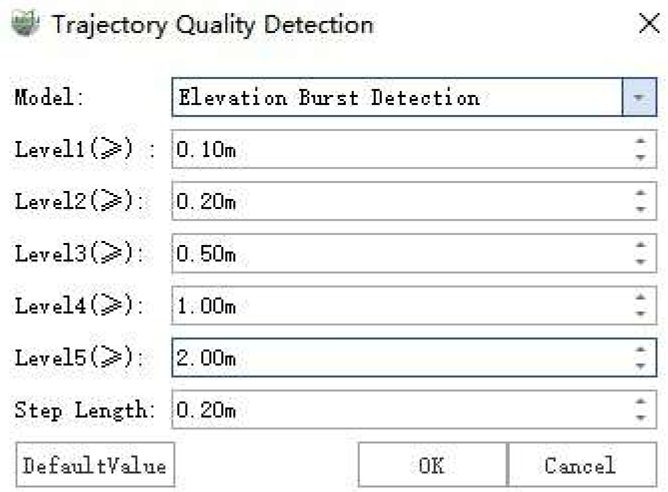
The trajectory graph displays the track segmentation information at all times by default. Use the mouse wheel to slide up and down on the track map to zoom in and out of the display range of the track map, click **Full Extend**  The button restores the default display.

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.
- **Detection step size**: The plane distance between the two points detected, the smaller the distance, the more sensitive it is to the mutation value.

3.**Detection of relative elevation difference in the re-entry area**: 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 Burst Repaire

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

1.Click **TTrajectory Burst Repaire** Button, a pop-up box pops up, 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

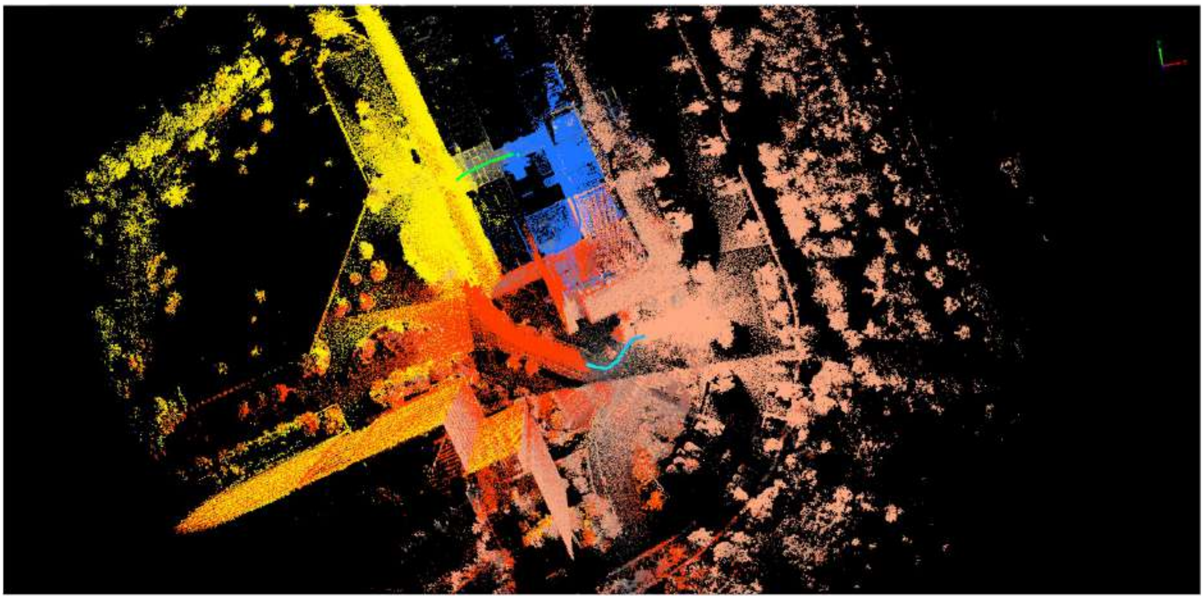
- Colorize By Segments
- Spli By Segments

Colorize by Segments

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

Steps

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.

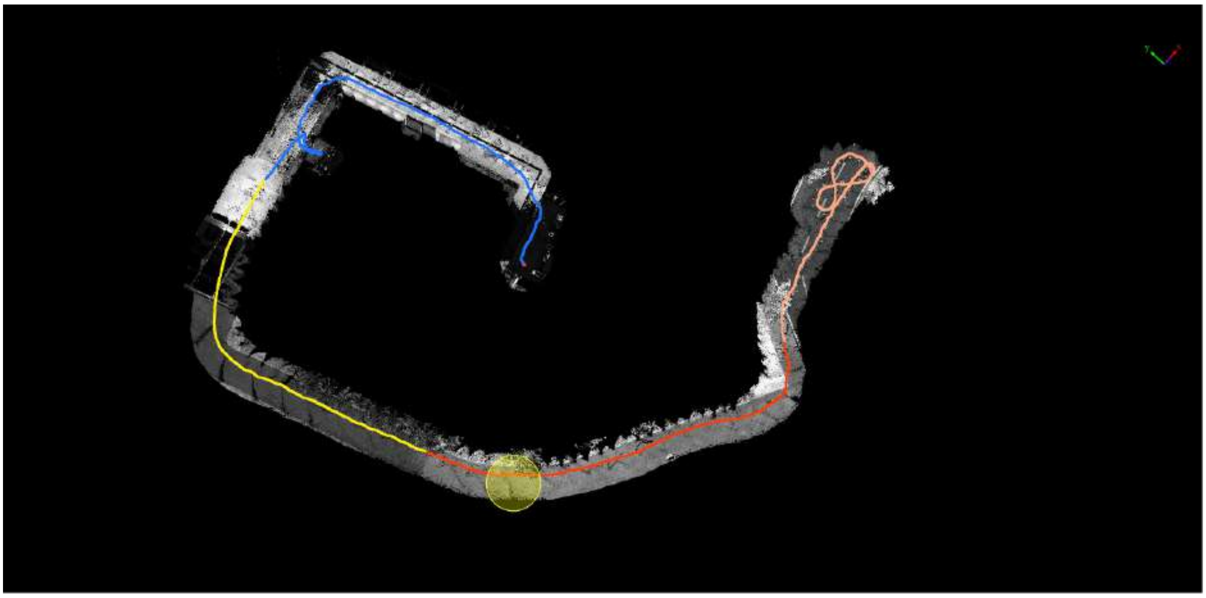


Split by Segments

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

Step

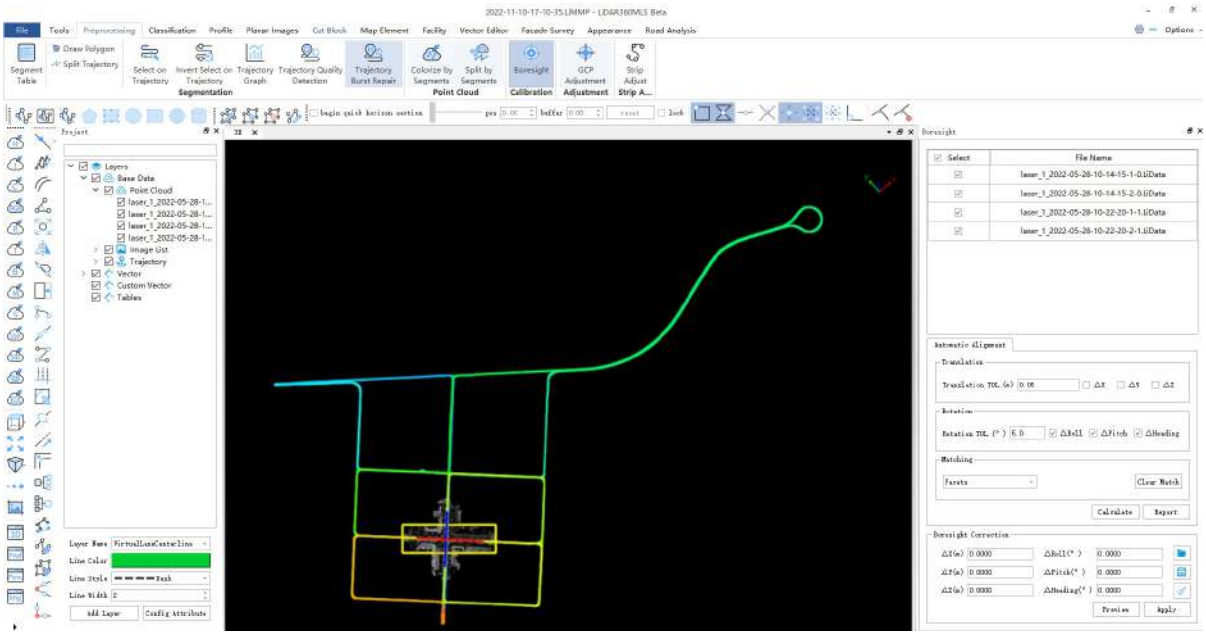
Click **Split by Segments**  button, click the **OK** button in the pop-up window, and the point cloud will be divided into multiple pieces according to the track segment to which it belongs.By checking **Cut by Trajectories'Buffer** and editing the **buffer** value, you can limit the range of segmentation.



Segmented by track fragment

Boresight calibration

1.Clik **Calibration** 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 accordingly 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.

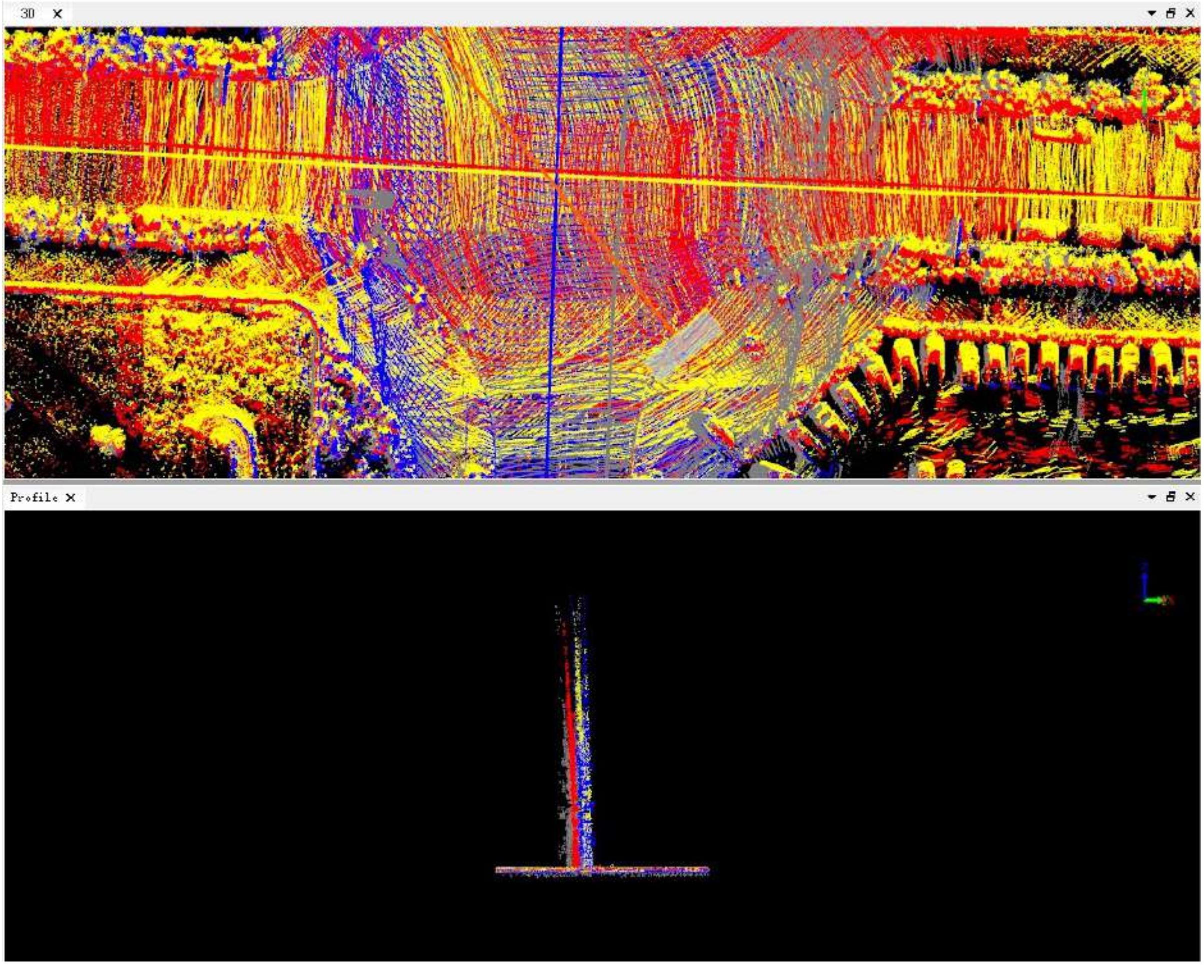


2.Automatic placement error correction：

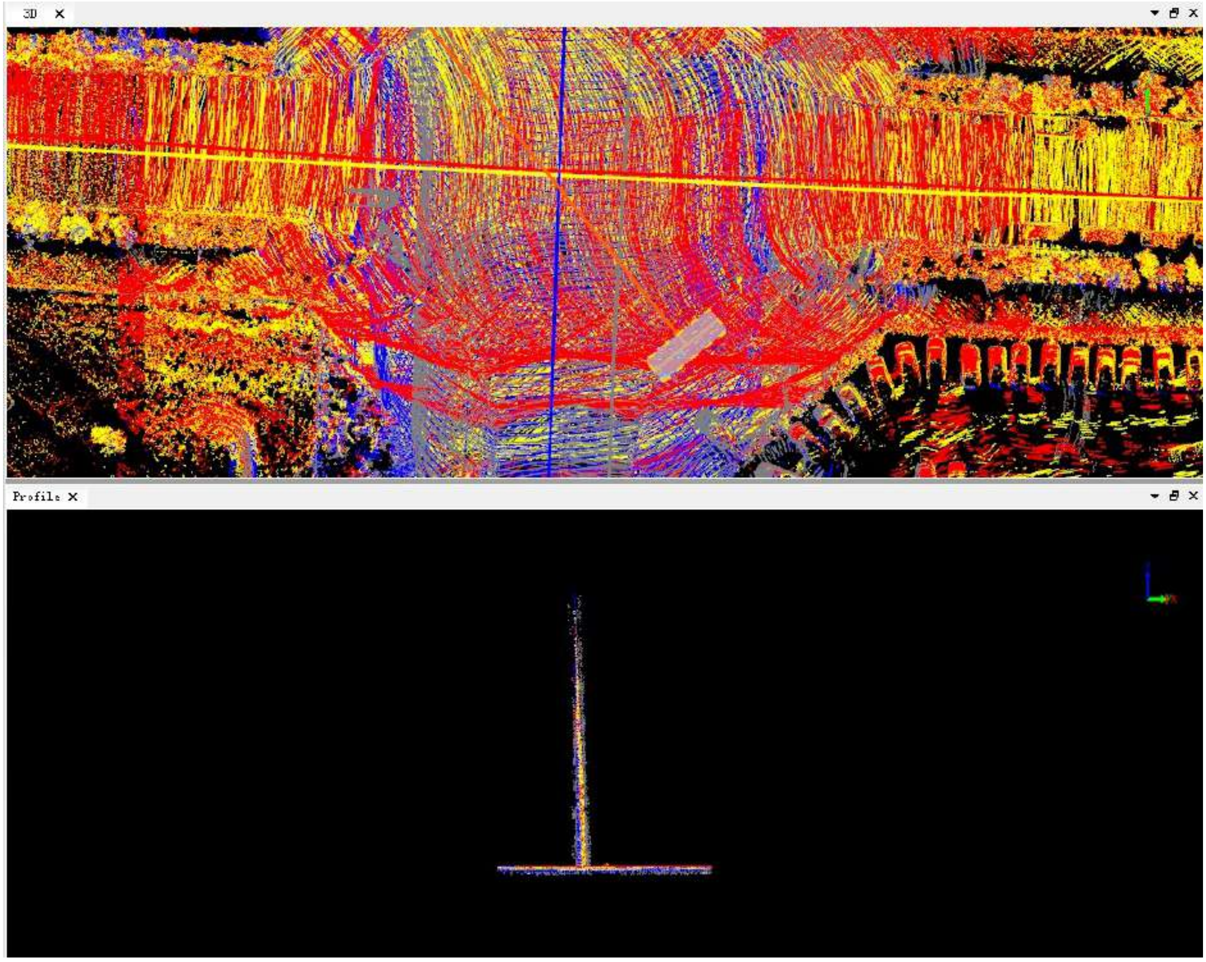
Check the final placement error that needs to be calculated in the placement interface, enter the translation tolerance and rotation tolerance, and Calibration After clicking calculate, the software will automatically calculate the placement error and display the results in the placement error correction 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 placement error correction：

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.

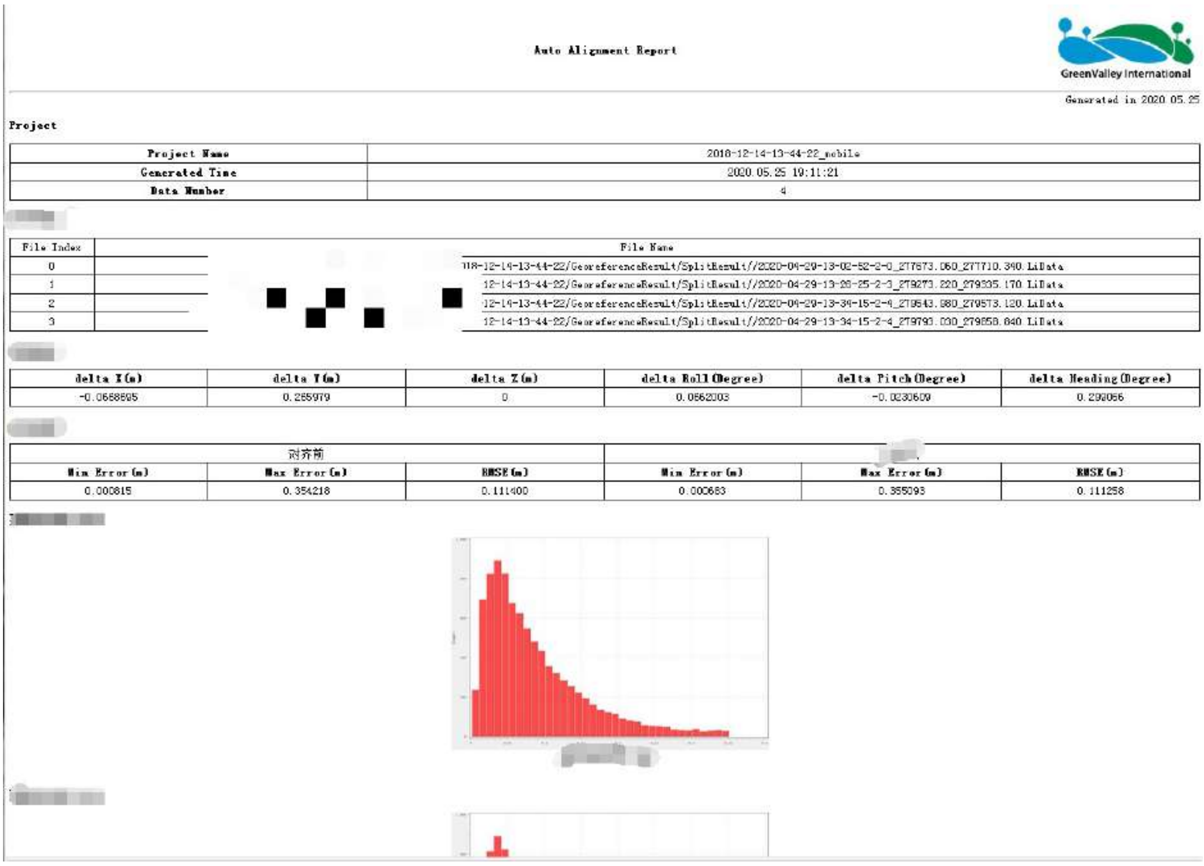


Before correction



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	[redacted]18-12-14-13-44-22/GeoreferenceResult/SplitResult//2020-04-29-13-02-52-2-0_277673.060_277710.340.LiData
1	12-14-13-44-22/GeoreferenceResult/SplitResult//2020-04-29-13-26-25-2-3_279273.220_279335.170.LiData
2	12-14-13-44-22/GeoreferenceResult/SplitResult//2020-04-29-13-34-15-2-4_279543.980_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)
-0.0668695	0.265979	0	0.0662003	-0.0230609	0.292066

对齐前					
Min Error(m)	Max Error(m)	RMSE(m)	Min Error(m)	Max Error(m)	RMSE(m)
0.000815	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: face 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).
    - **Sampling point**: 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

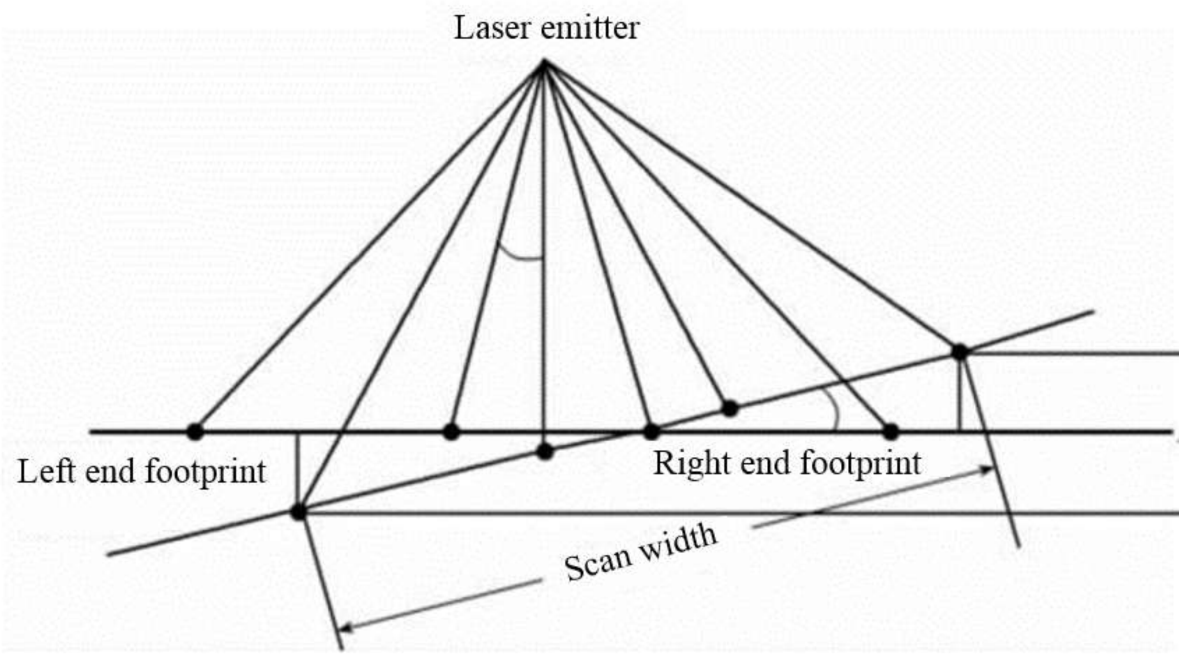
Their principles are described in detail below.

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) Recovered 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 tilt the plane scan line (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.；

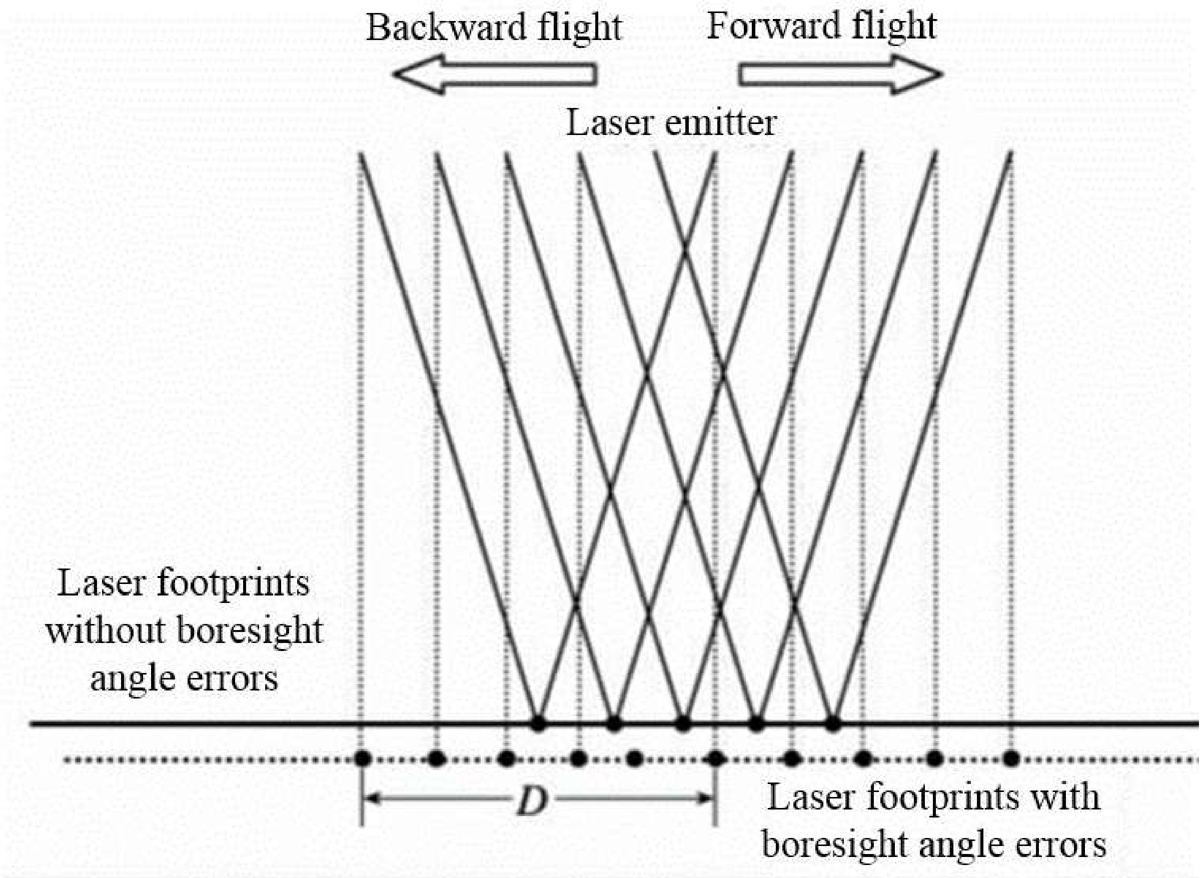
- Measure the horizontal distance r between the approximate object of the same name and the center line of the two air belts in the 2D view.

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 between the center position of the same object along the flight direction D，
- Calculate the average flight altitude H based on the trajectory (try to keep the round-trip flight altitude as

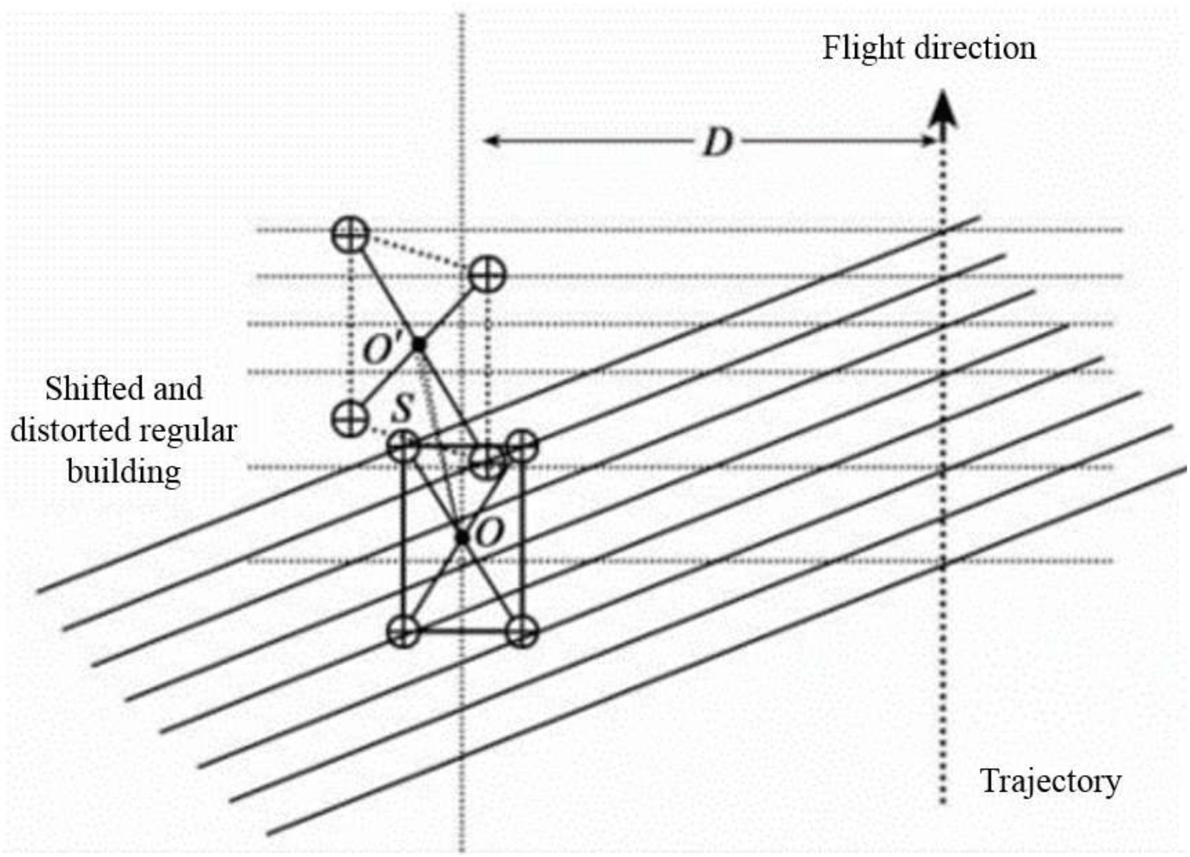
consistent as possible).

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 between the two air belts D

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 operator.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 airband 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},
> }

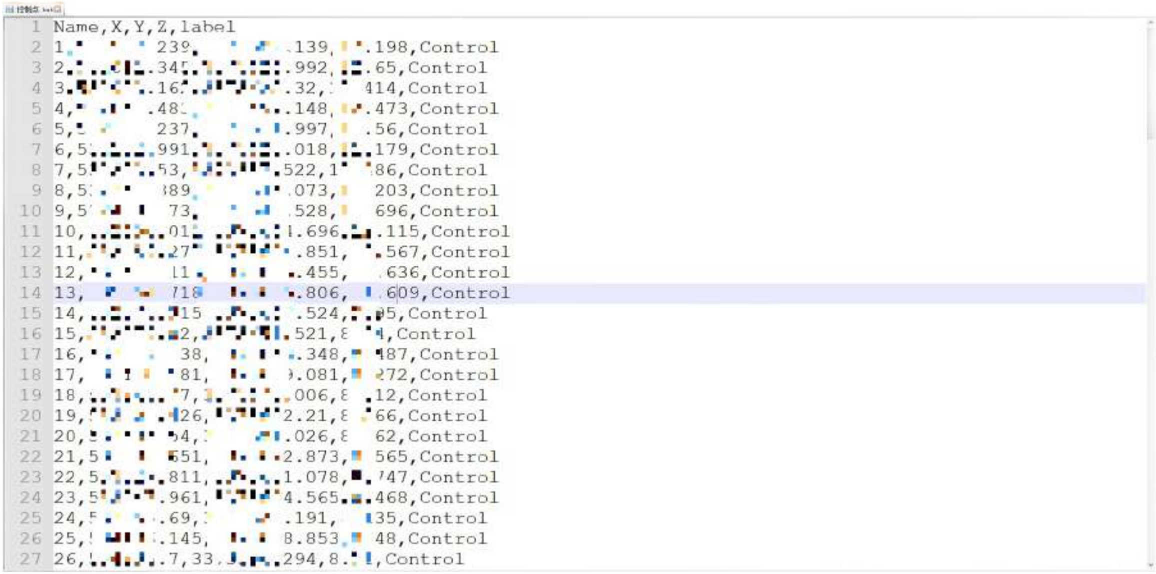
Control Point Alignment

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.

STEP

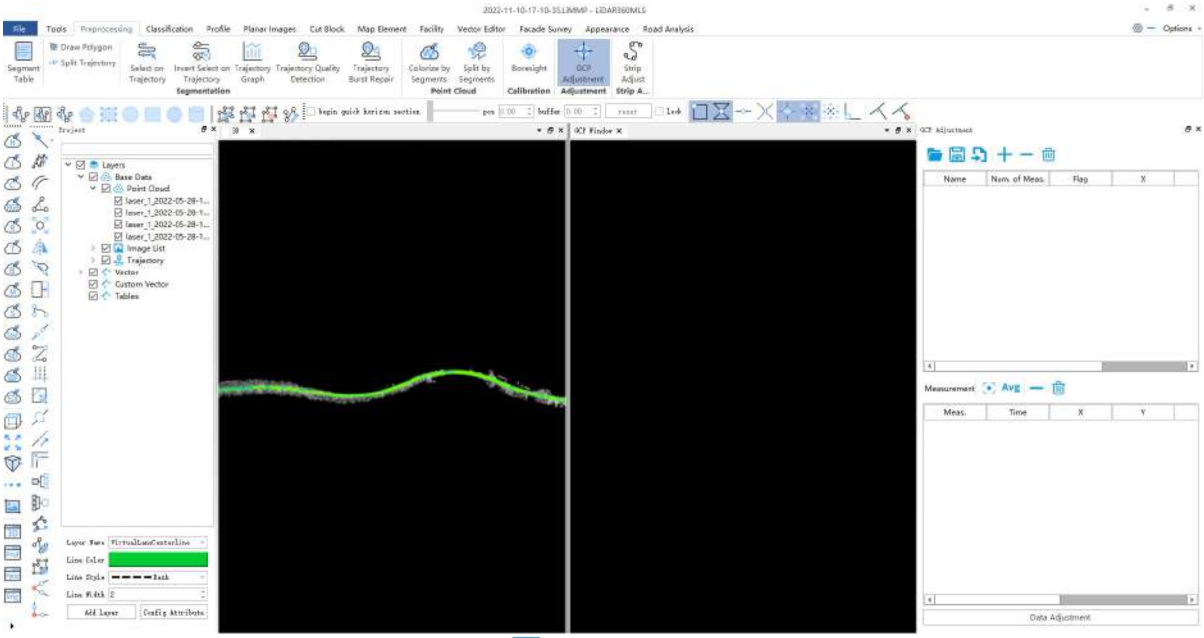
1.Data preparation: Data preparation includes the following two aspects

- 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 project file .ligeo solved by Ligeo, or you can also refer to New Project to configure point cloud, trajectory, Image, 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.

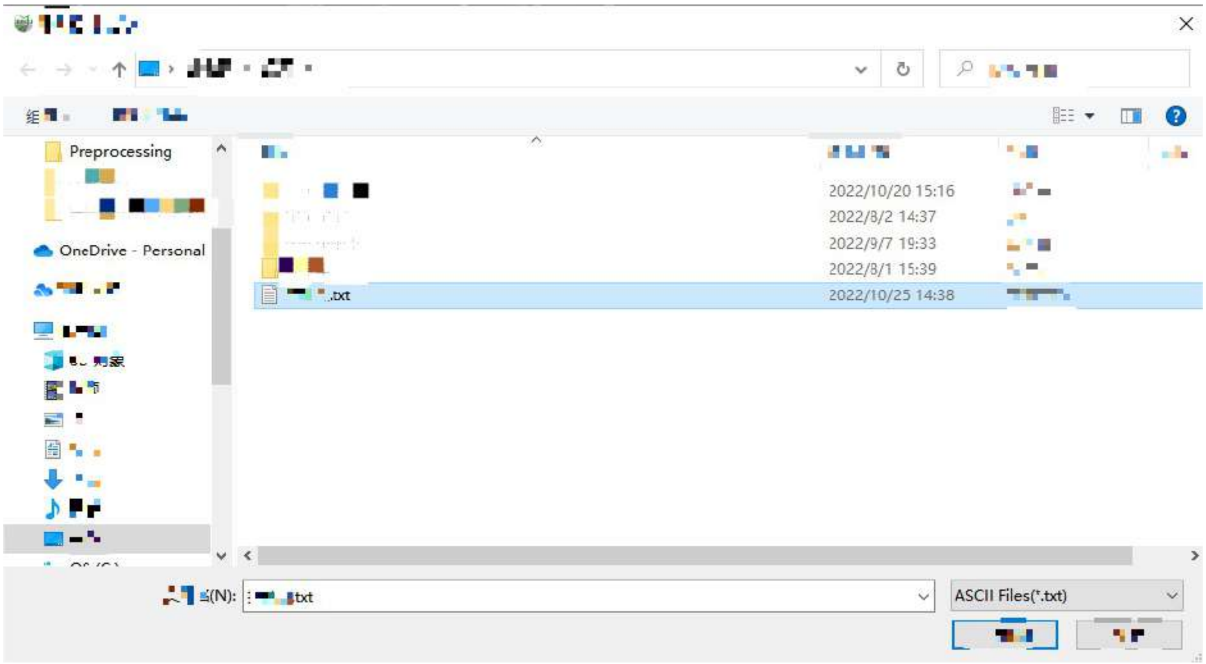


```
Name,X,Y,Z,label
1,[redacted],Control
2,[redacted],Control
3,[redacted],Control
4,[redacted],Control
5,[redacted],Control
6,[redacted],Control
7,[redacted],Control
8,[redacted],Control
9,[redacted],Control
10,[redacted],Control
11,[redacted],Control
12,[redacted],Control
13,[redacted],Control
14,[redacted],Control
15,[redacted],Control
16,[redacted],Control
17,[redacted],Control
18,[redacted],Control
19,[redacted],Control
20,[redacted],Control
21,[redacted],Control
22,[redacted],Control
23,[redacted],Control
24,[redacted],Control
25,[redacted],Control
26,[redacted],Control
```

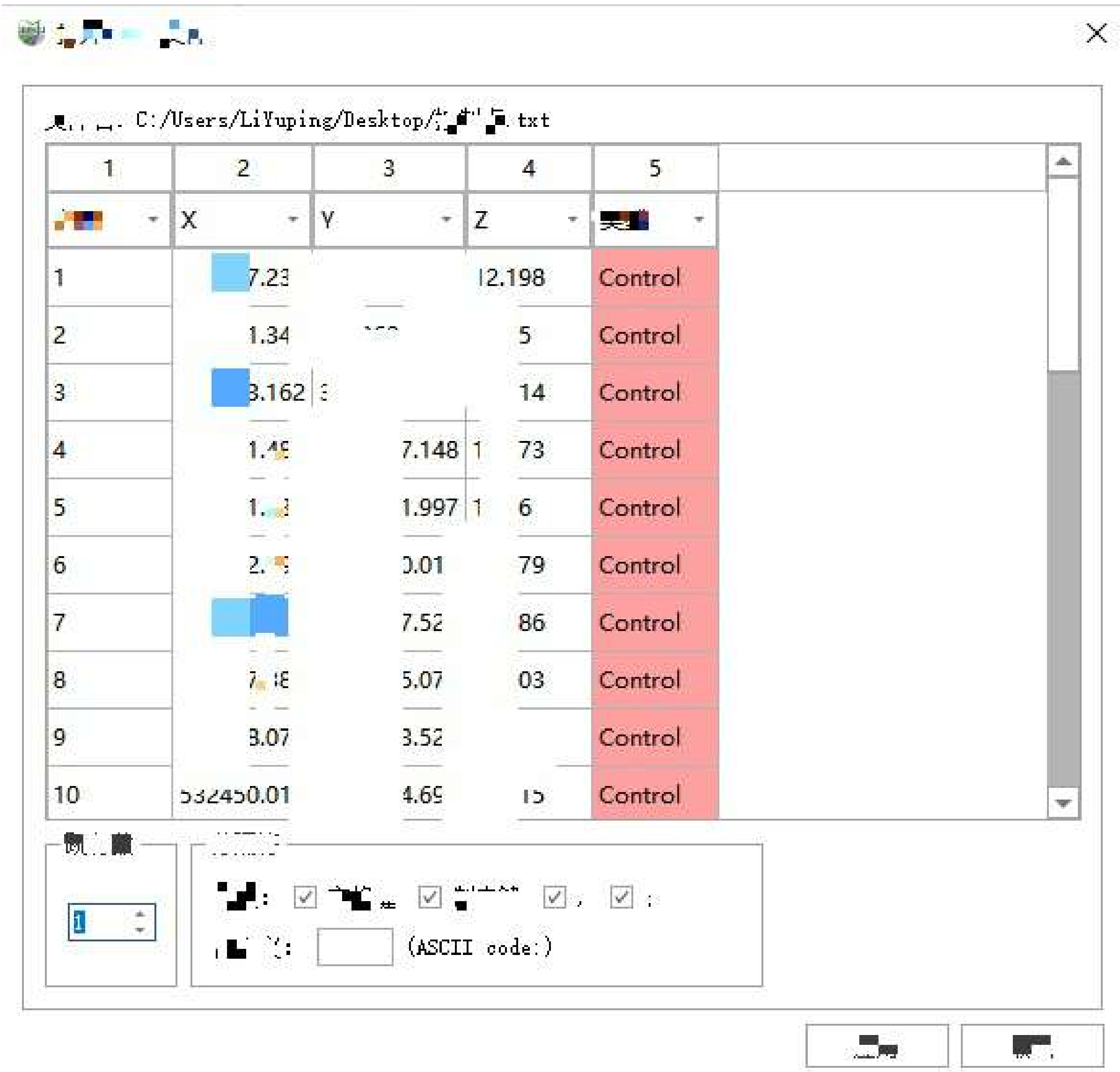
2.Click **Control Point Alignment** button to activate this function, the control point correction dialog box will appear on the right side of the main page, and the GCP window will appear at the same time.



3.Import external control point file: Click **Import**  button to import the control point TXT file prepared in step 1.



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 pop up:

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:

5.1.3D 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 3D window to display the section of the point cloud where the current control point is located.

5.2.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.

5.3.Control point guide dialog:

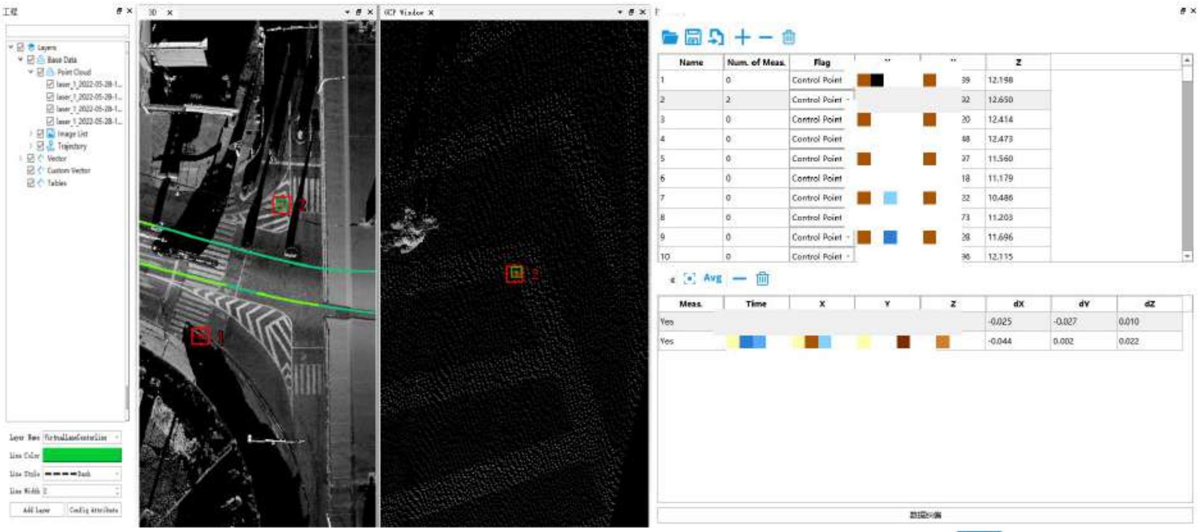
5.3.1.The upper part shows the imported point-by-point information:

- Name: represents the imported control point name
- Num. of Meas.: Indicates that the current control point can control several air strips, that is, the current control point in the lower half has been punctured correspondingly on several air strips
- Flag: Represents 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

5.3.2.The lower part displays the information about each control 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:

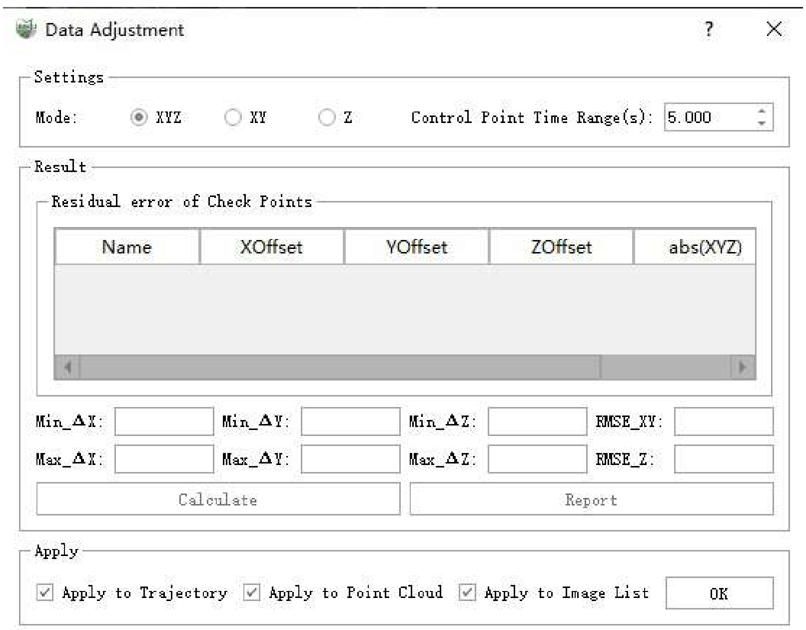
- Meas.: No means that the flight zone is not constrained by the corresponding control point, Yes means the flight zone is constrained by the current control point
- Time: represents the collection time of the current puncture point position

- 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: point cloud elevation value representing the current puncture point position
- dx: represents the offset between the point cloud and the east coordinate of the current control point
- dY: represents the offset of the point cloud from the north coordinate of the current control point
- dZ: represents 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** button to export the GCP control point information, the exported xml file can not only continue to be loaded later Only the current function continues to be modified and used, and this file is also used as the splicing process in the StripAdjust function. control point file information.

7.Click **Data Correction**, and the **Data Correction** 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.



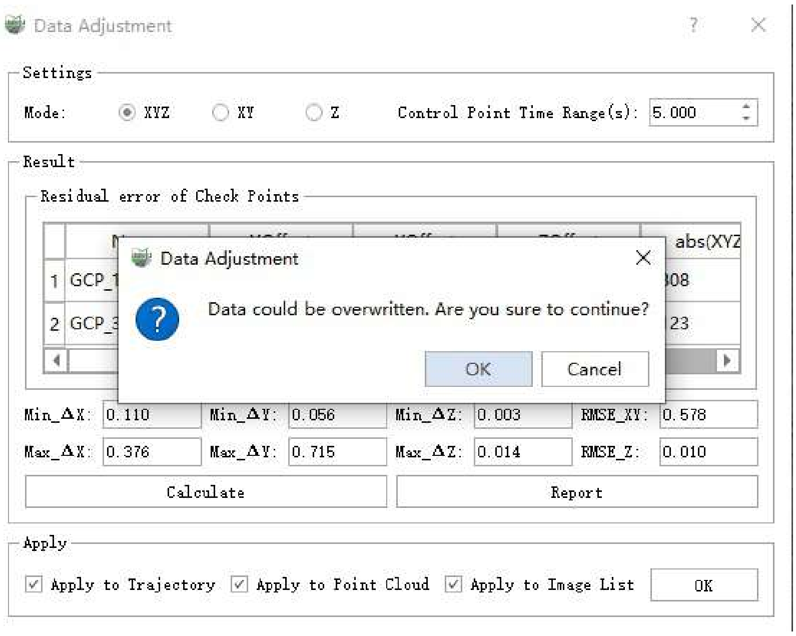
Parameter Description:

- 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: used to calculate the correction parameters of the control point, and check the correction accuracy of the set check point
- Application: 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. It will modify the image information, which may cause the point cloud and the image to be misaligned after the offset correction.

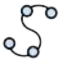
> Note: This step will rewrite the original data, please make sure that there is no problem in the parameter calculation before applying it.



Strip Adjustment

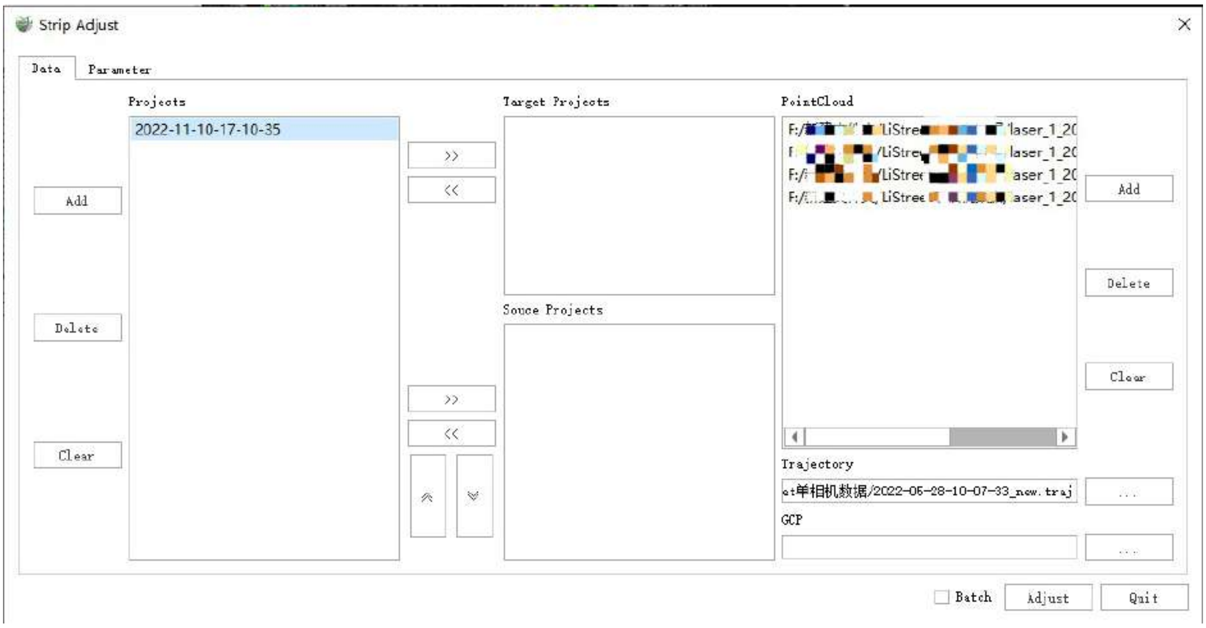
Function description: The strip adjustment supports the adjustment of the point cloud revisit area within the vehicle data of a single project; supports the adjustment calculation of multiple groups of projects, and aligns the registration project point cloud to the Refer to the engineering point cloud 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 interpolation**  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.**Engineering 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 the 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 Down Matching Projects**: 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 the [Control Point Correction] (GCPAdjustment.md) function

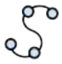
3.**Adjustment parameters**

- **Track Segmentation**:
  - Min Segment Length: The minimum length of the track segment result. It is recommended to set the default value
  - Max Segment Length: 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
- **Segment registration**:
  - Maximum matching distance: matching pairs whose distance is greater than this value do not participate in the adjustment. It is recommended to set the default value.
  - Thinning distance: point cloud thinning distance. It is recommended to set the default value.
  - Minimum height of track from ground point: 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.
  - Number of threads: 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.
- **SLPINE adjustment**:

- 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.
- Maximum position correction amount: The maximum correction amount of SPLINE node position. It is recommended to set the default value.
- Maximum attitude correction amount: 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 the corrected trajectory pose Guaranteed smooth and specific effects. The maximum amount of correction for the attitude of SPLINE nodes. It is recommended to set the default value.

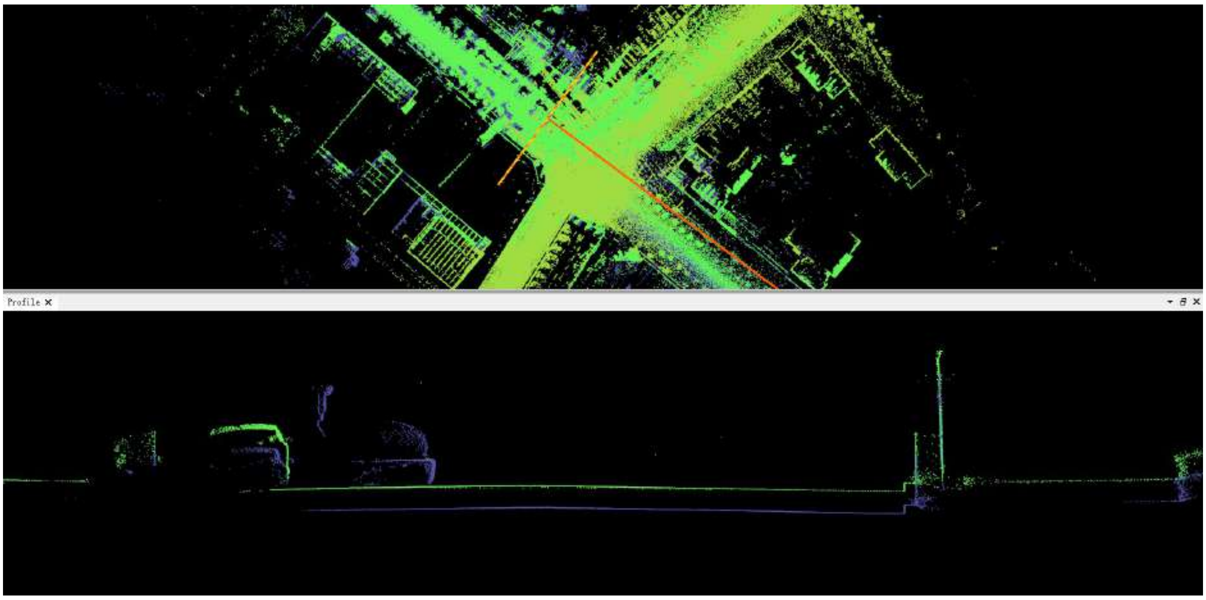
Single project Strip Adjust

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.

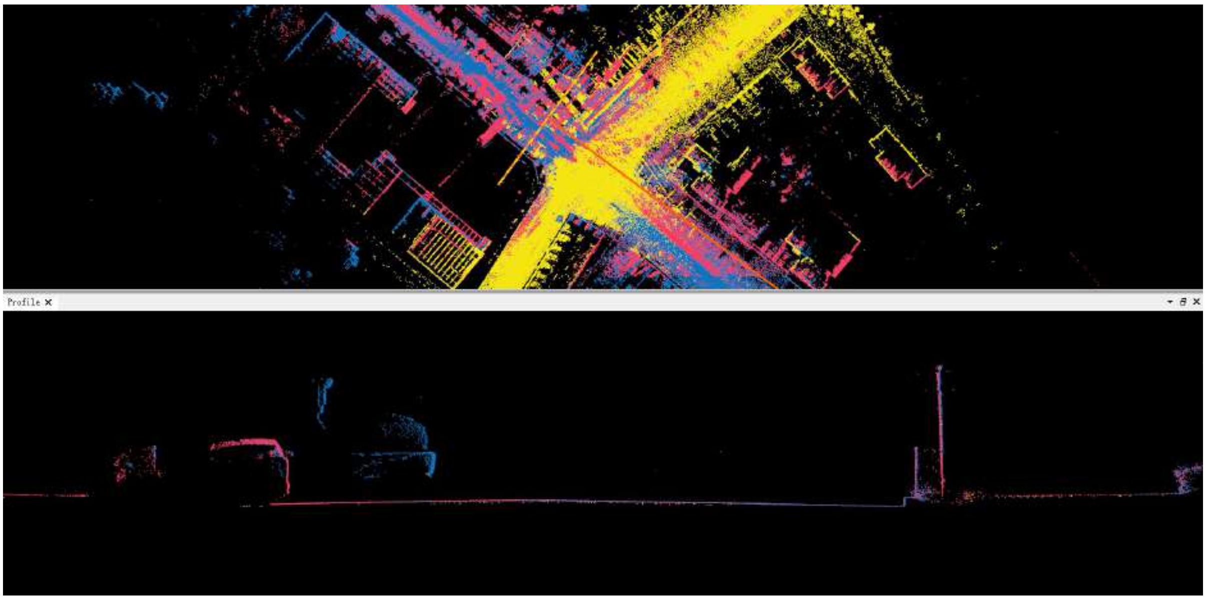
2.Click the  function button, In the setting of the air belt splicing project, move the current project to the matching project. If internal splicing of control point parameters is required, you need to pass Control Point Correction in advance to puncture the control point and save it as an .xml file,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

3.Click the Splicing button to start the splicing operation. 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 processing at the top and bottom. Check box, 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 splicing a single project air belt

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 The 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.

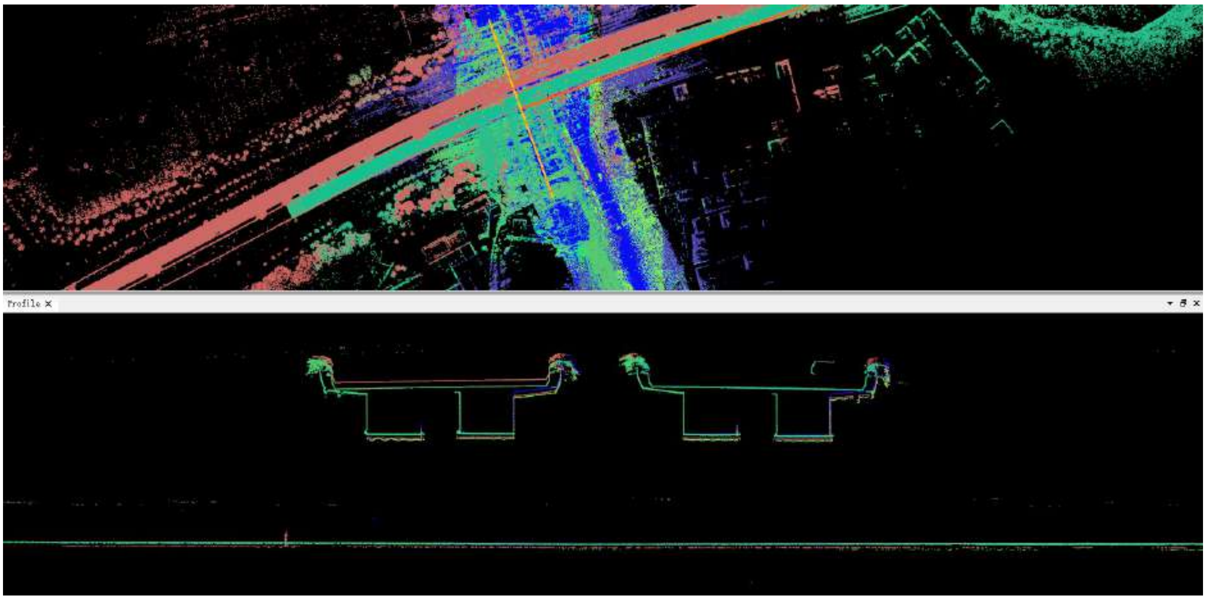
2.Open any group of projects, assuming project A is opened here, click **air belt stitching** button, on the left side of the project list, click the "Add" button to add the BCD project to the project list.

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. in 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.

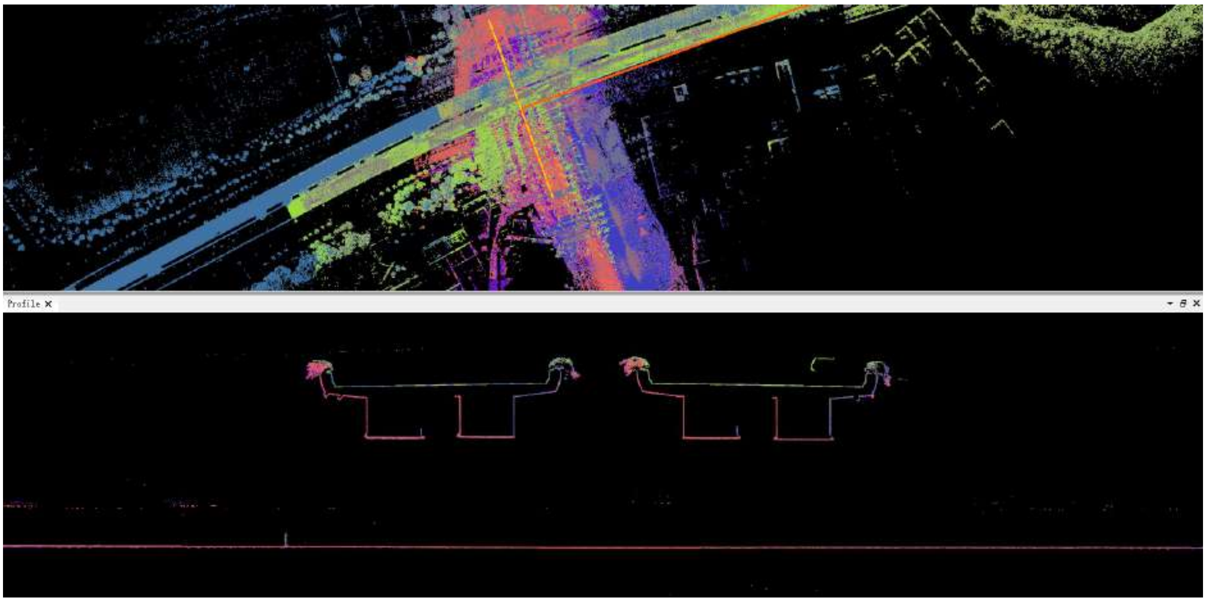
4.Click on each project to view and modify the point cloud, trajectory and control point data of the current project on the right side of the air belt splicing interface. If the control point data is set in a single project, the control points will participate in the adjustment during the process of 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 file



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
- Cut 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 frame.

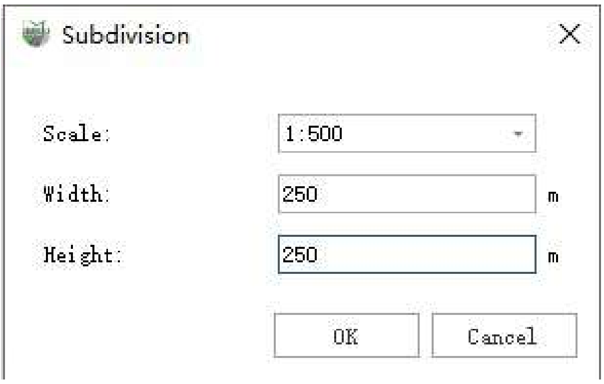
- Point Cloud Subdivision
- Point Cloud Cut Block
- Create Along Trajectory
- Create Along CenterLine

Point Cloud Subdivision

Function description: LiDAR360MLS uses scale-based point cloud segmentation, and can perform point cloud data segmentation 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

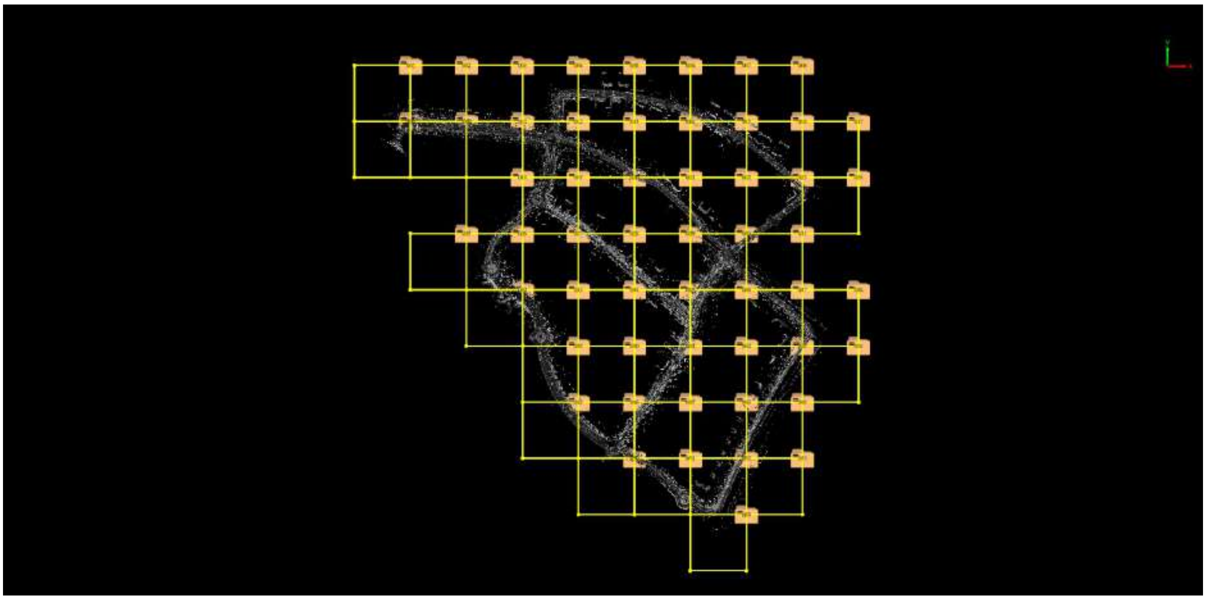
1.Click **Point Cloud Subdivision** button to pop up the following interface



Parameter Description

- **Accuracy**: 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:

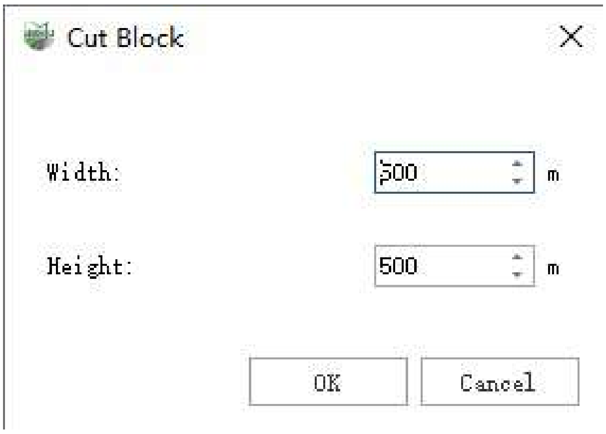


Point cloud Cut Block

Function description: Supports point cloud cutting into blocks mode with user-defined rectangle length and width.

Steps

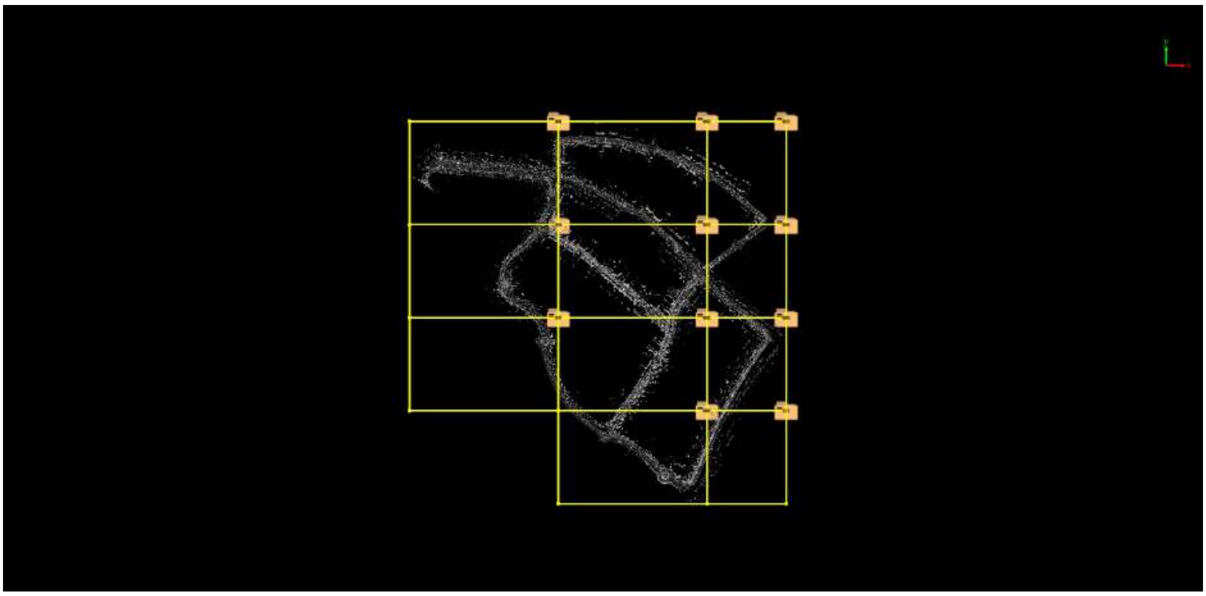
1. Click **Point Cloud Cut Block**, the following interface pops up



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:



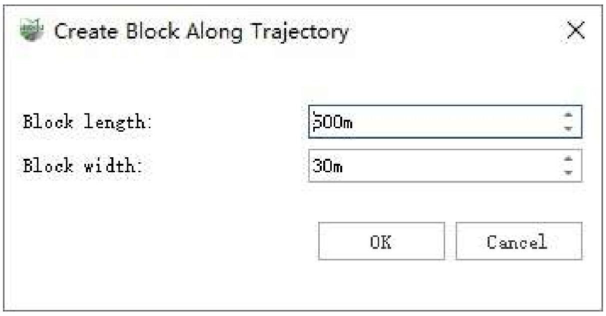
Framing according to trajectory

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

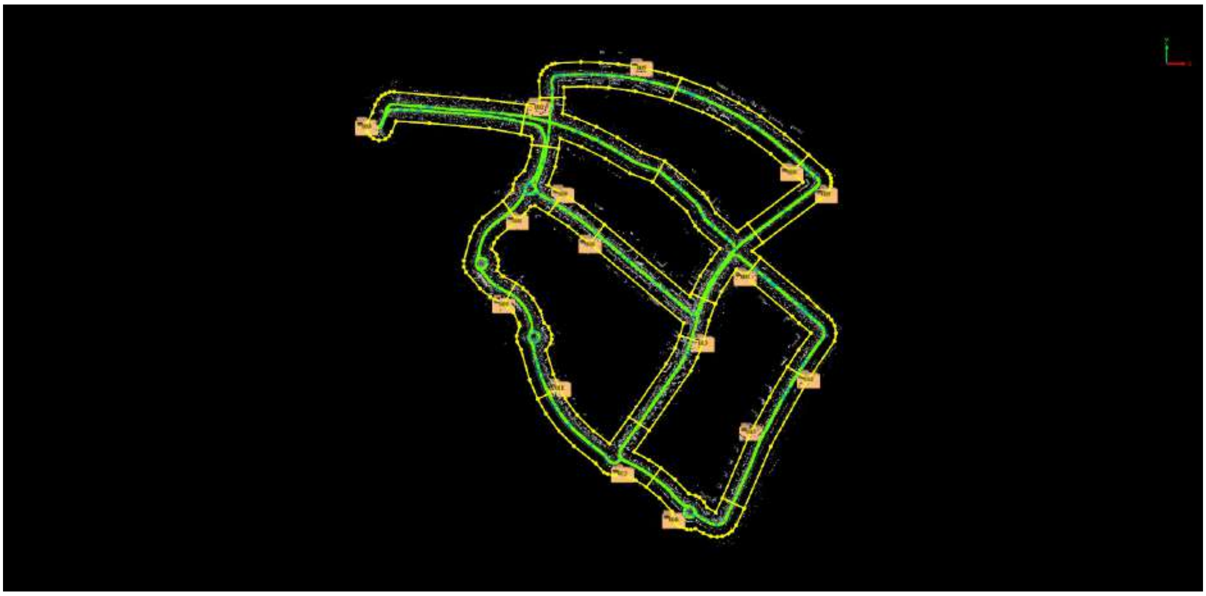
1.Click **SubdivisionByTrajectory**  , the following interface pops up



Parameter Description

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

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

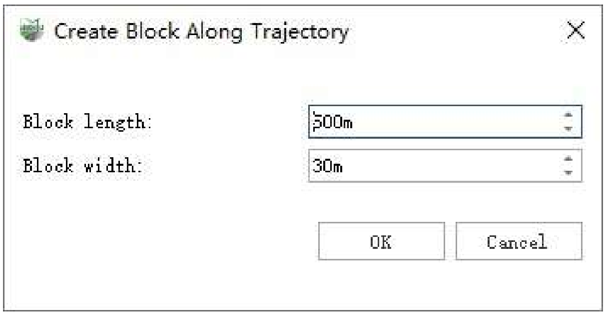


Create Along CenterLine

Function description: This function only divides according to the vector line, and does not divide 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.

Steps

1.Click **Order according to centerline**  , the following interface will pop up



Parameter Description

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

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

Cut Block

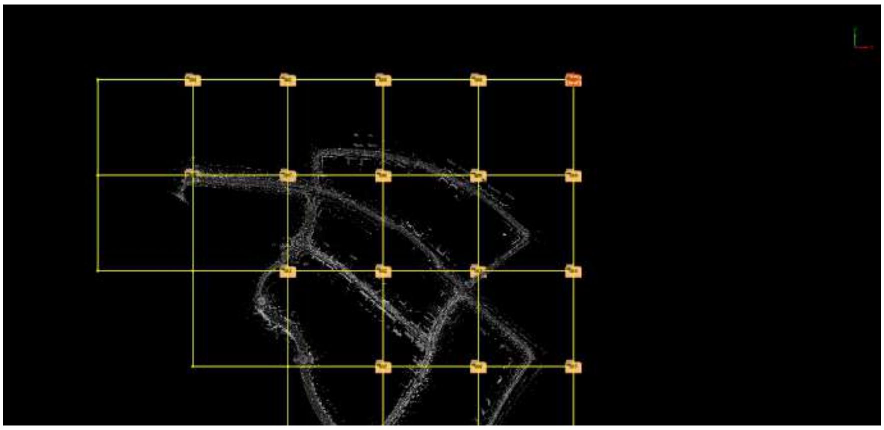
- Cut Block Edit
- Split Point Cloud

Cut Block Edit

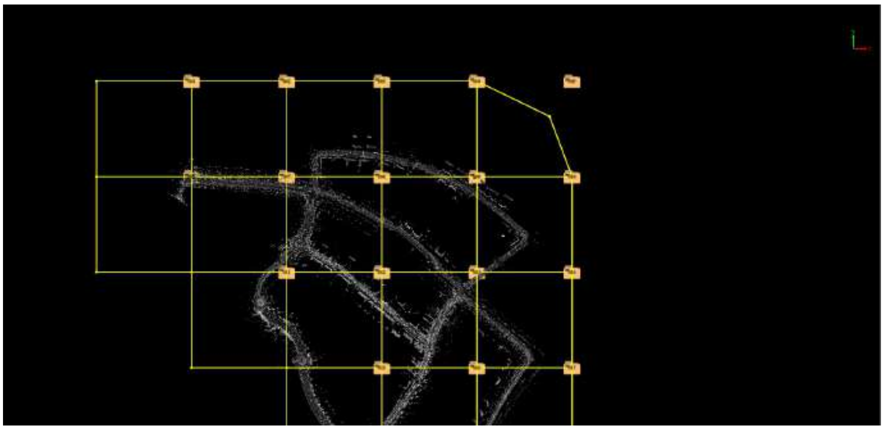
Function description: This function module is used to edit the area block after framing based on the track.

Steps

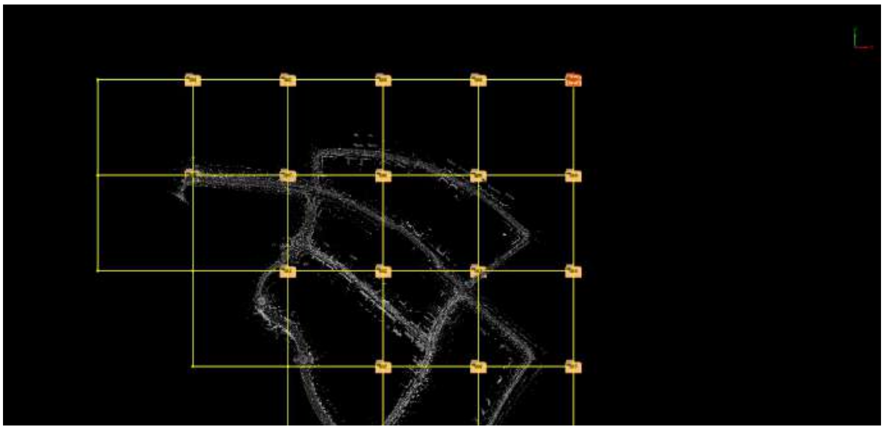
- **Edit Block Node** : Click the button After that, move the mouse to the node of the frame, when the color of the mouse turns red, it means that the current node is captured. Press and hold the left mouse button and drag the current node to a new position, as shown in the figure below, before editing:



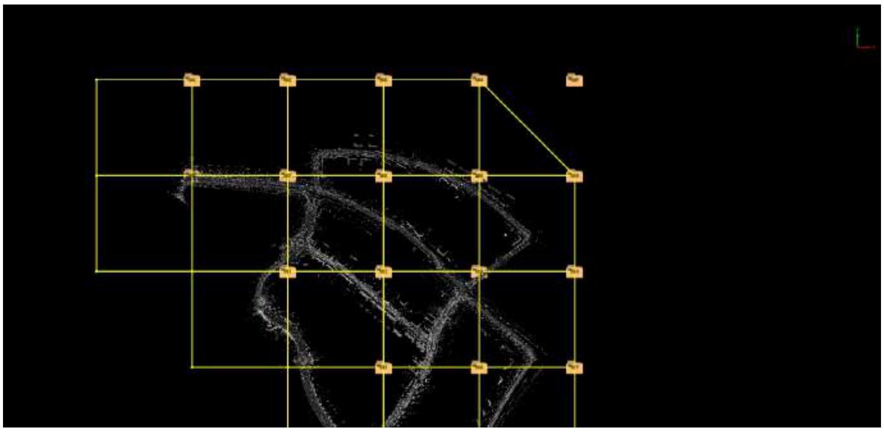
After editing:



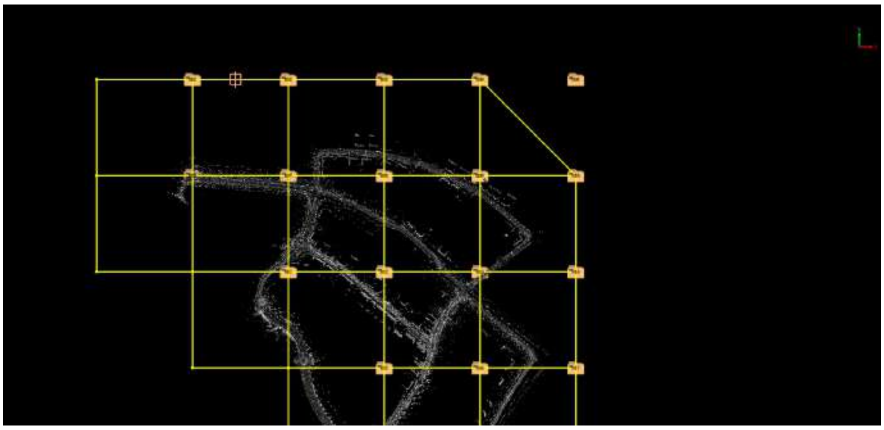
- **Remove Block Node** : Click After pressing the button, move the mouse to the node of the frame. When the color of the mouse changes to red, it means that the current node is captured, and the left mouse button can be held down. As shown below, before editing:



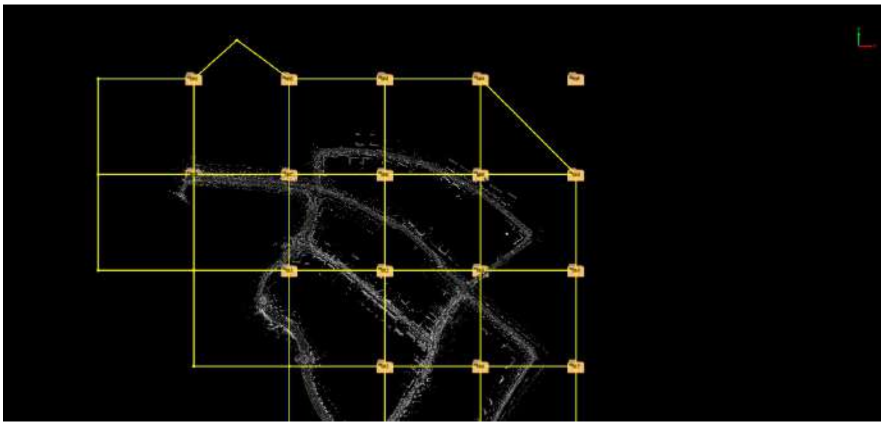
After removing the node:



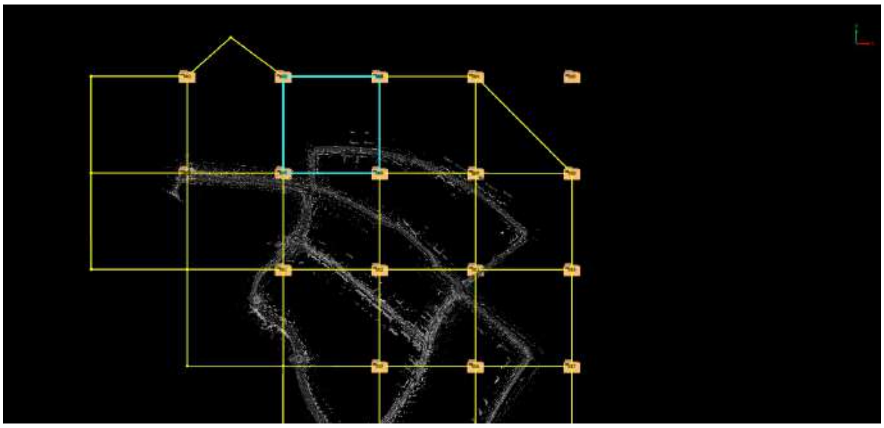
- **Insert Block Node** : Click the  button Then, move the mouse to the position where the node needs to be inserted in the frame, and click the left button of the mouse to insert a node at this position. As shown below, before inserting a node:



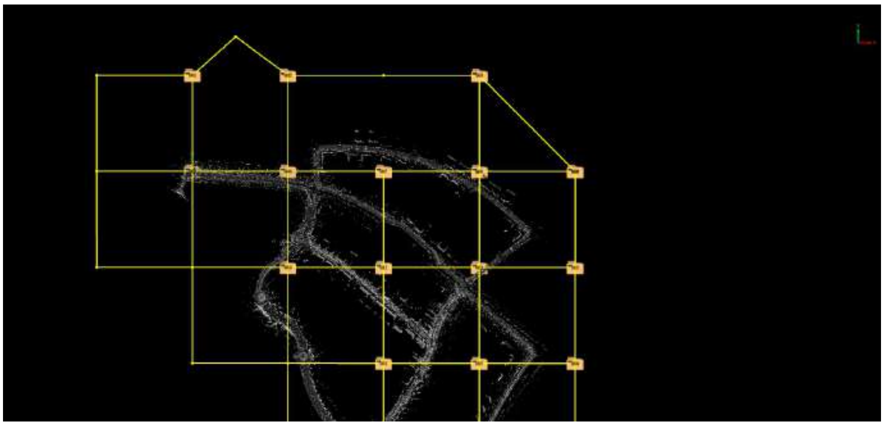
After inserting the node and moving it:



- **Merge Blocks** : Click After pressing the 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:

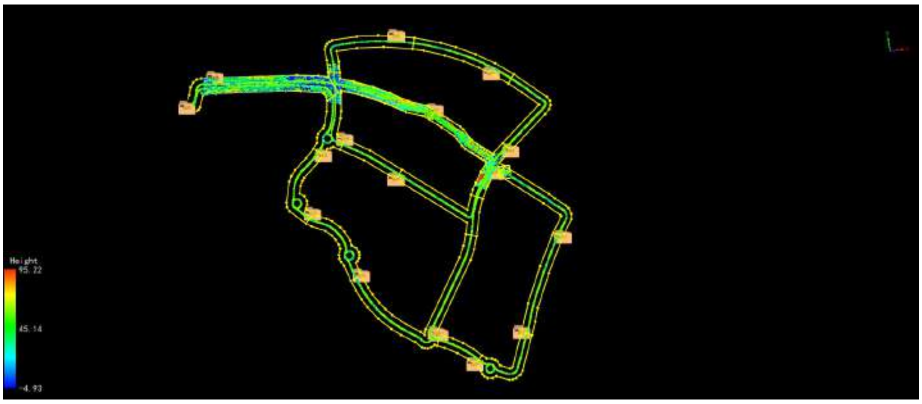


Split Point Cloud

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 will pop up:

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

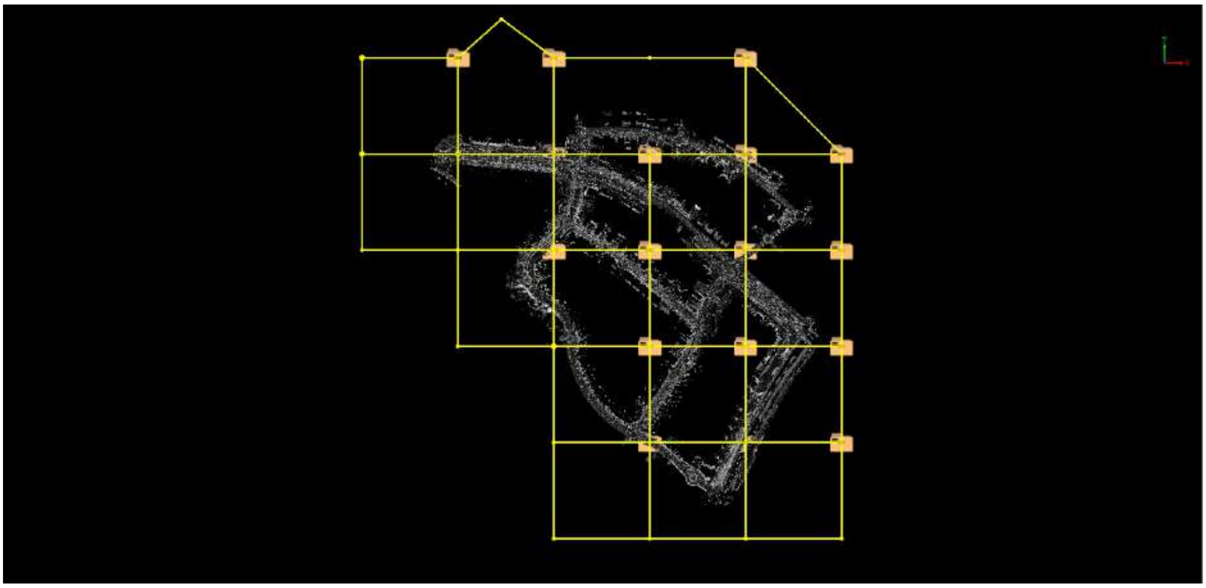
Cut Block Display

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

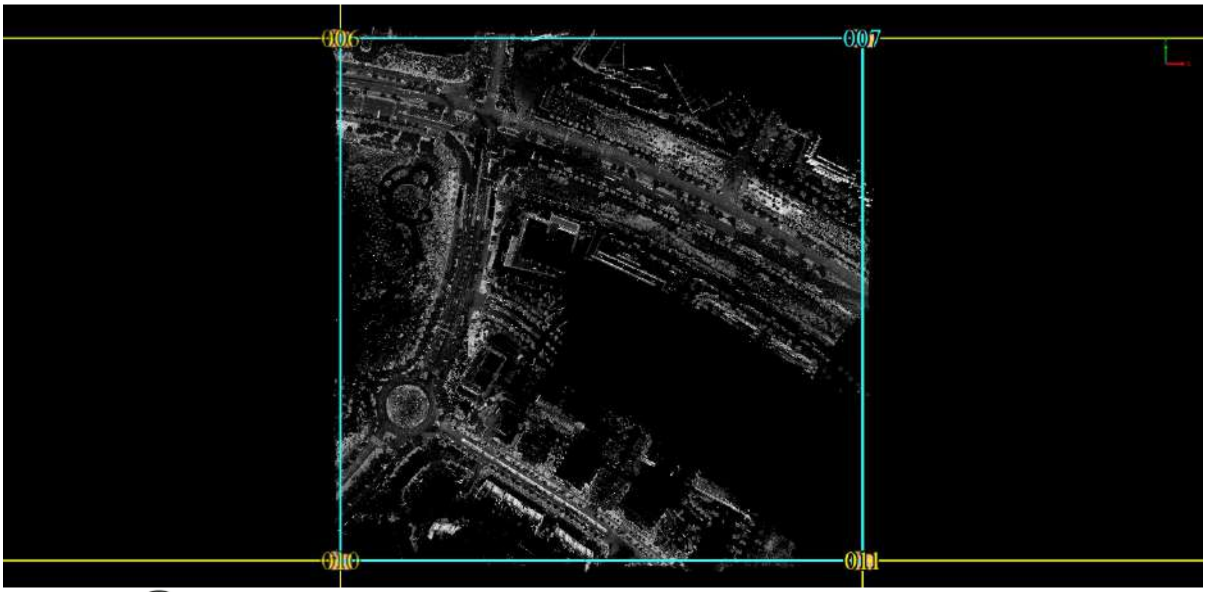
Steps

1.After the point cloud segmentation is completed, in the area block display, click **icon** , 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.

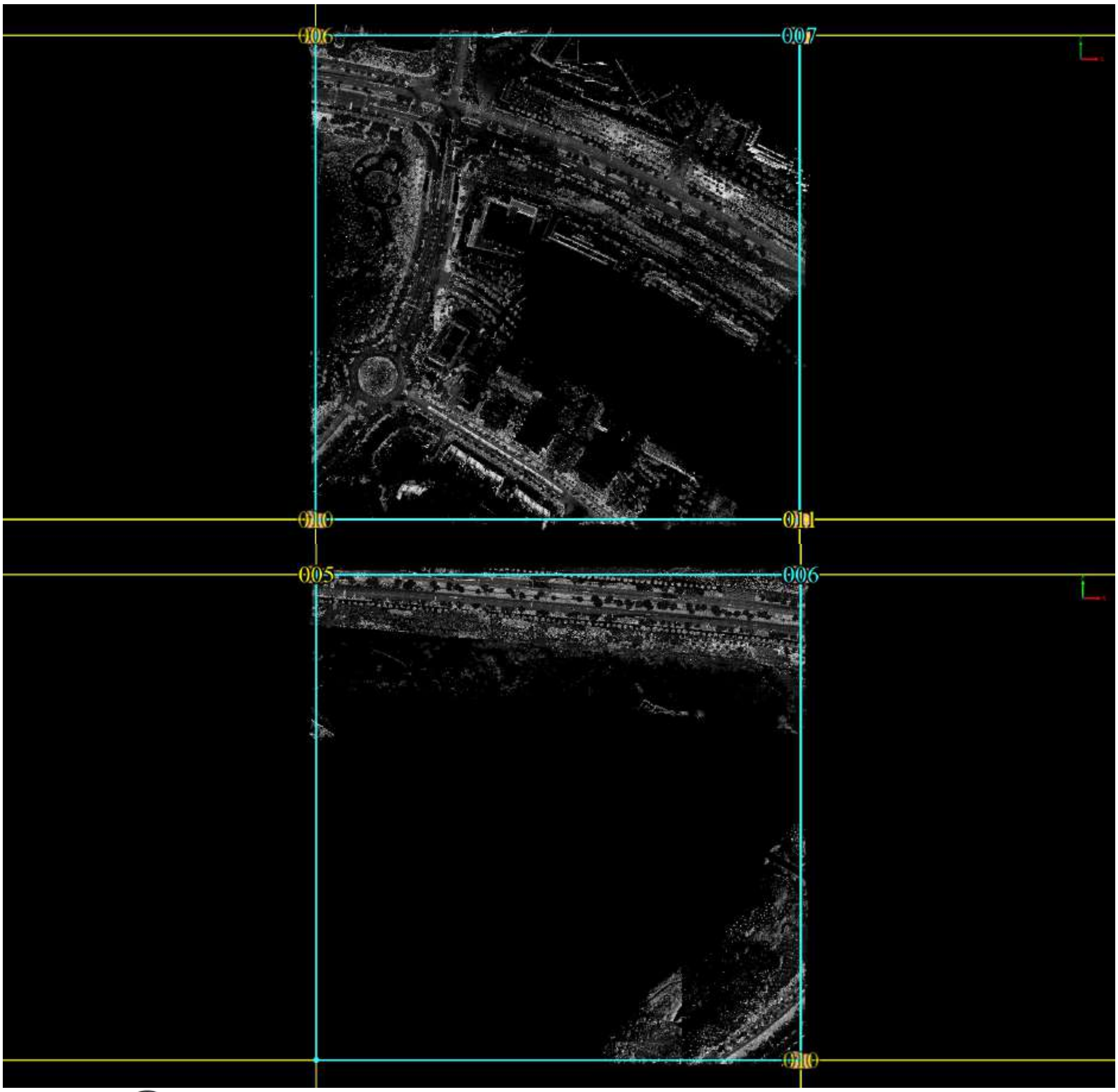
Before frame selection:



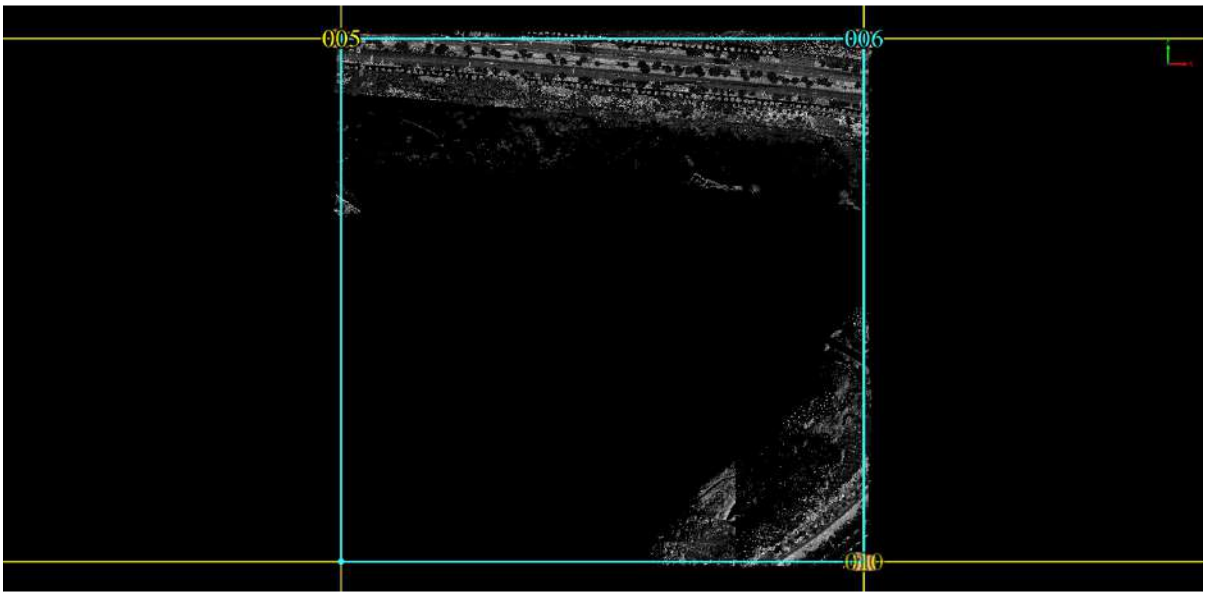
After selecting a frame:

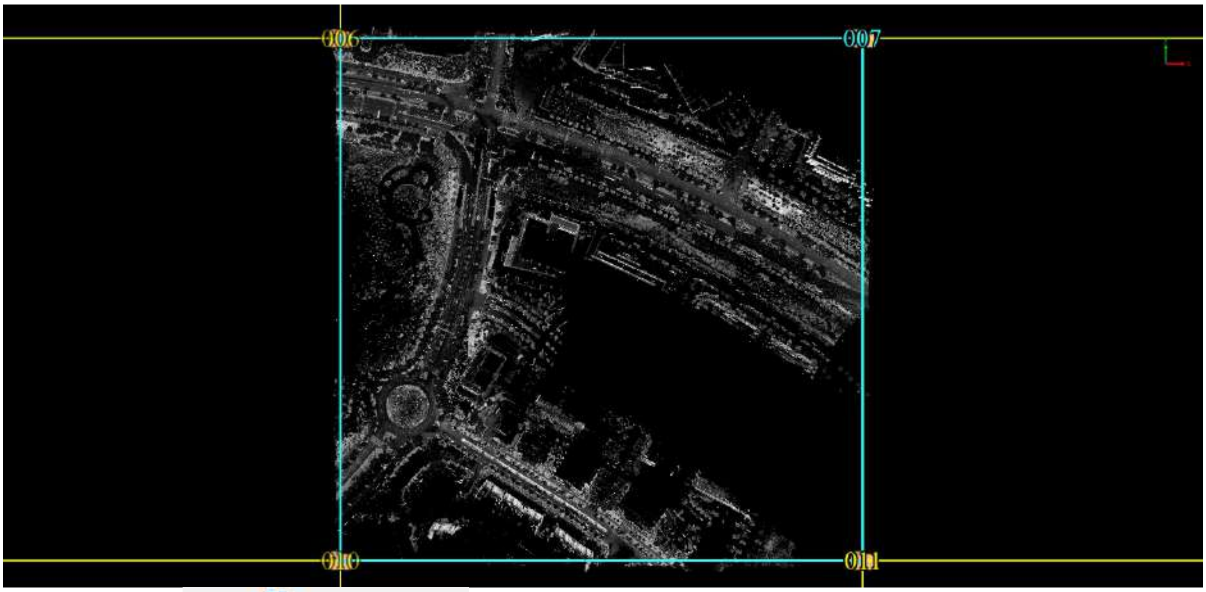


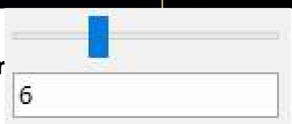
2.Click **icon** to jump To the previous area block of the current area block number.



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





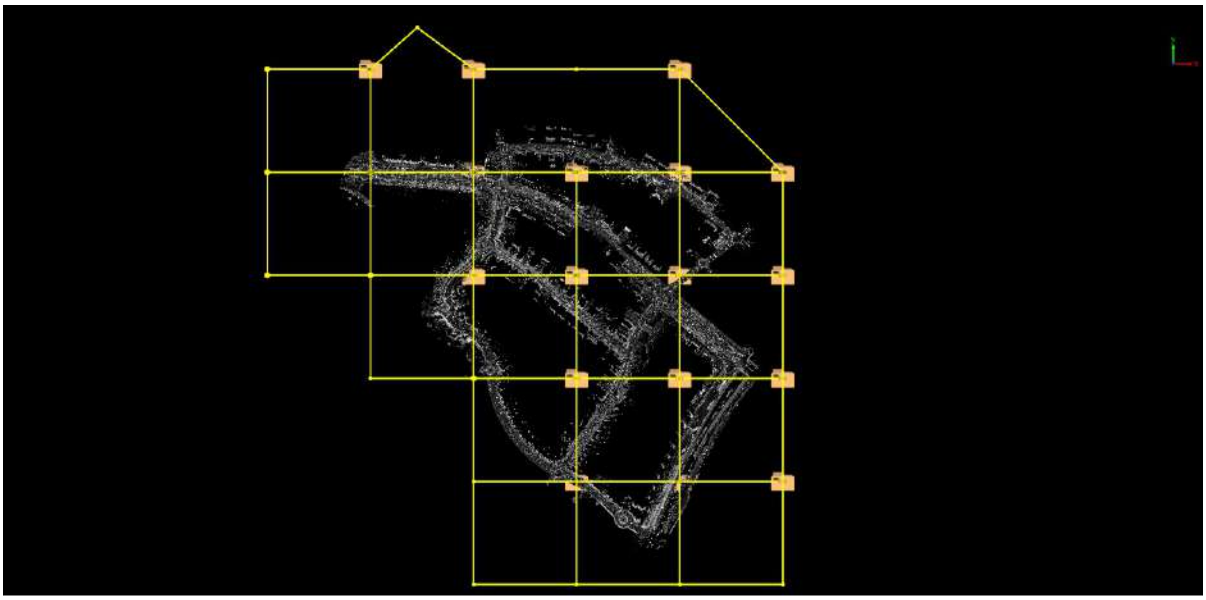
4.Drag the **slider** 

, switch different area blocks to select and highlight ;

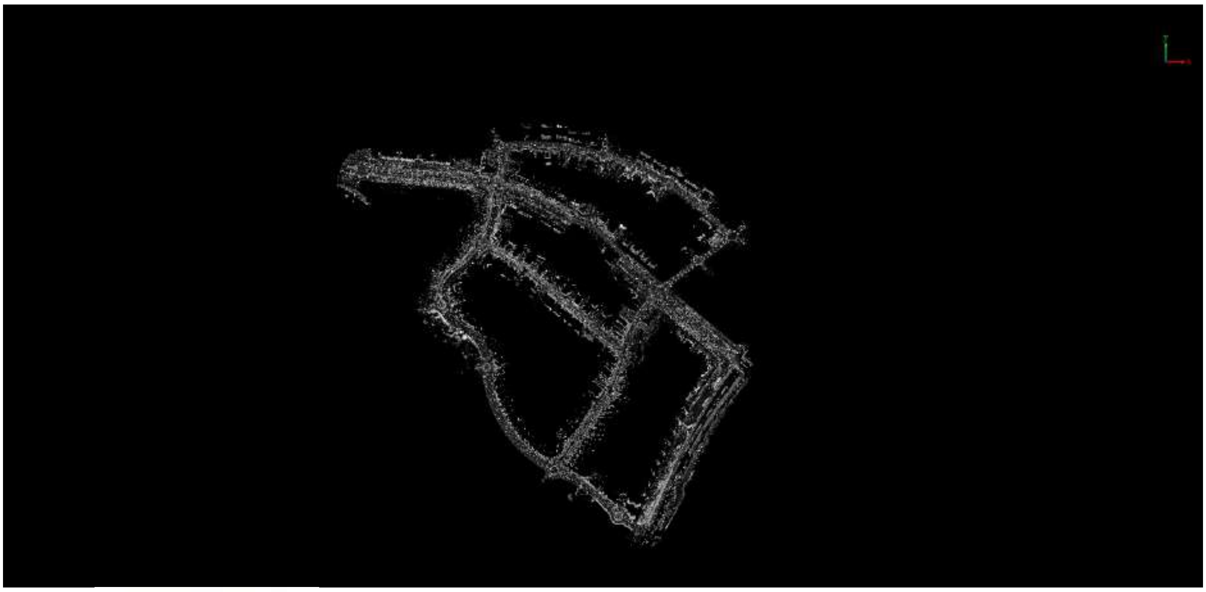
5.Select 

in the menu bar, you can show or hide area blocks and labels;

Display area block:

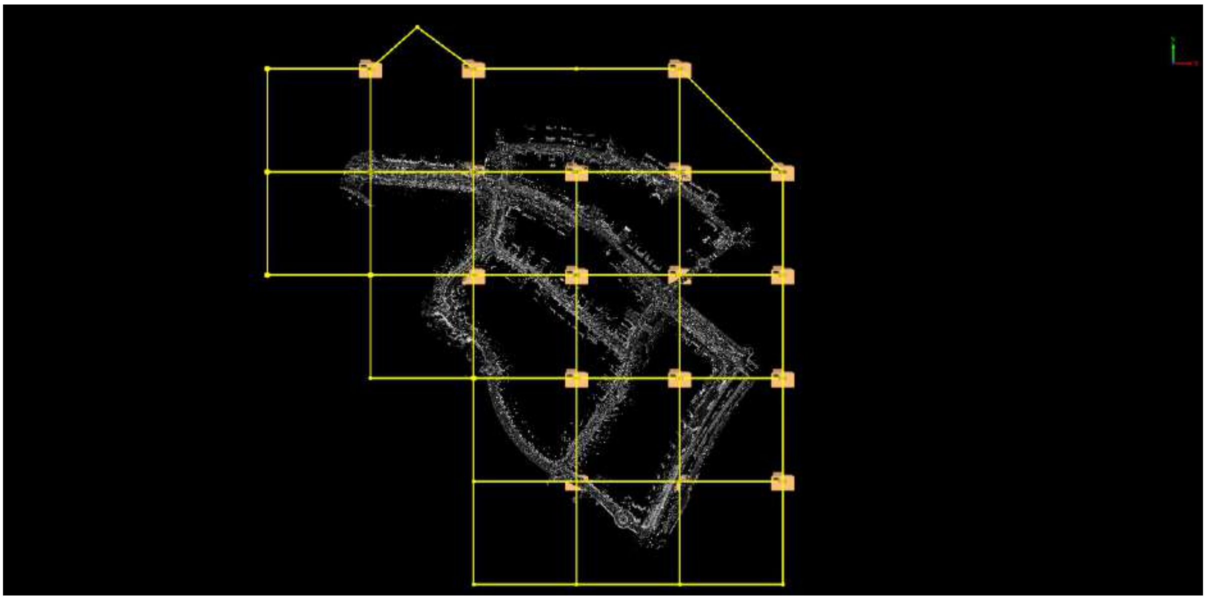


Hidden area block:

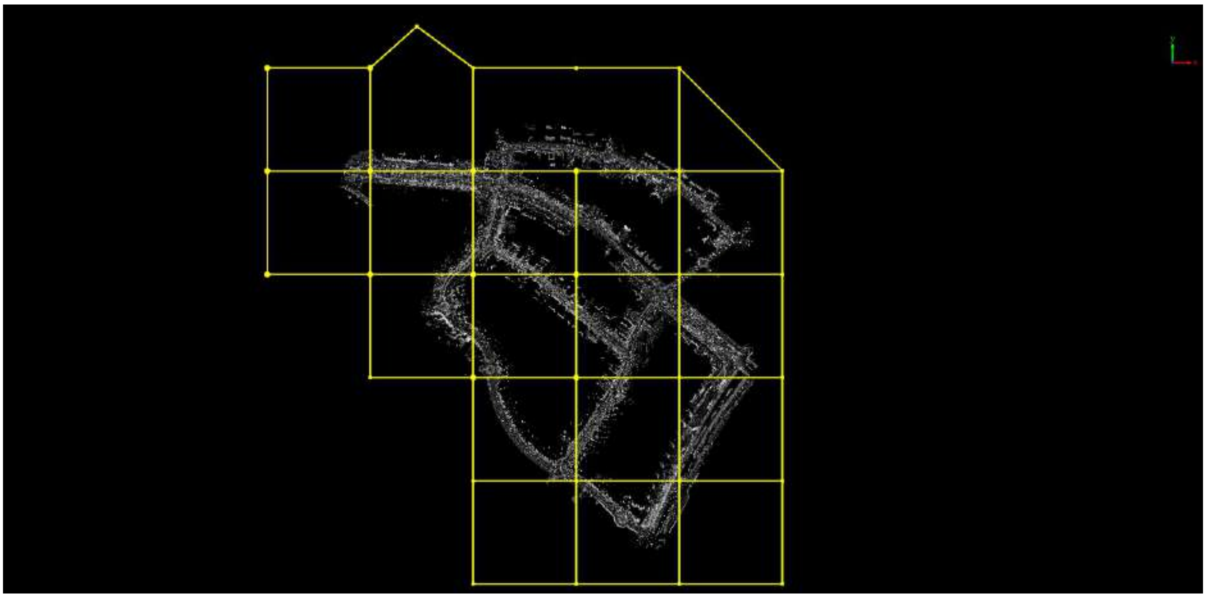


6.Select  in the menu bar to display or hide the corresponding label of the area.

Show labels:



Hide tabs:



Map Elements

The map element module is mainly 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 generation method** can be divided into **semi-automatic** and **manual**.

Manual Function

- Lane Centerline
- Road Lane line
- Road Dashed line
- Stop Line
- Road Curb

Semi-Automatic Functions

- Lane solid line detection
- Lane dotted line detection
- Road Curb Detector
- Generate Road Center line

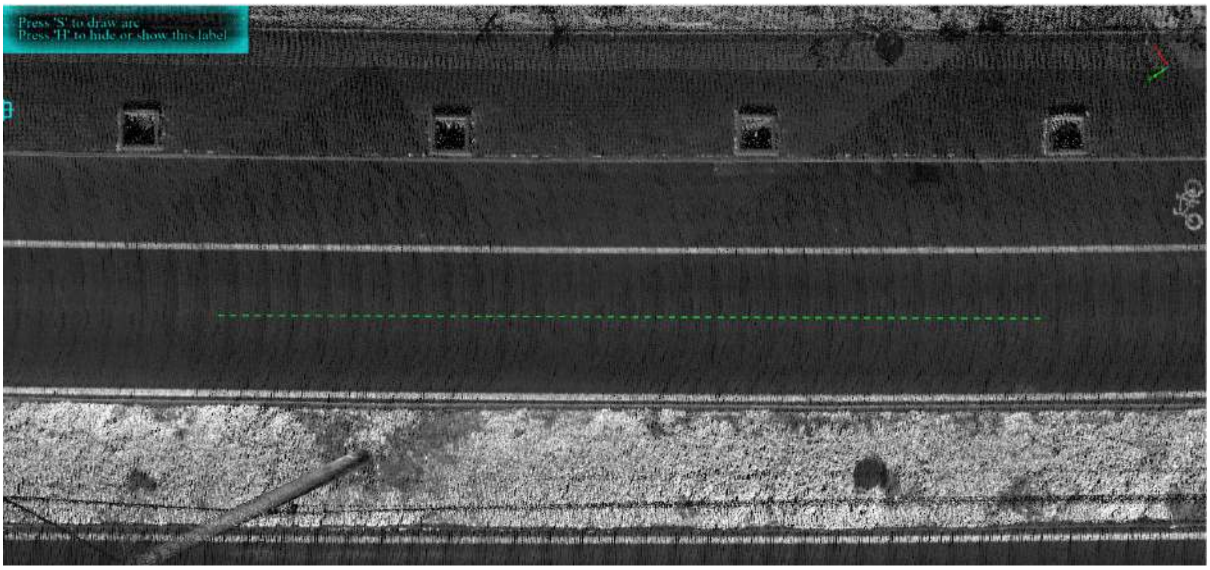
Lane center line

Function Description: The lane center line is located at the center of the two lane lines and is a virtual line. The software uses a dotted line to express the lane center line. This function is used to manually select the nodes of the line to generate the centerline of the lane.

Steps

1. Click the **Lane Center line** button in the Road Shape panel
2. Left click to select the starting point of the line
3. Left click the selected line node until the end point, and quickly double-click the left mouse button to determine a lane centerline

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



Drawing road 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 lane solid line function with the right mouse button and shortcut key

Lane Line

Function Description: This tool is used to manually select the nodes of the line, multi-point or double-point to generate the solid 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 **Lane Line**  button

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

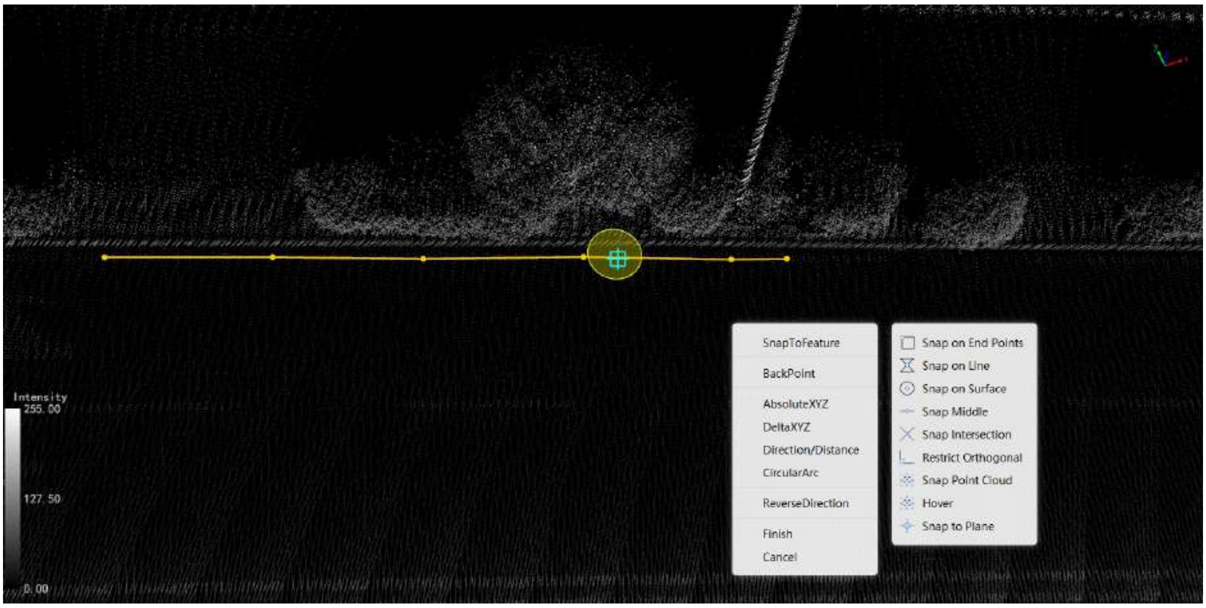
3.Move the mouse, and **left-click** in turn to select the node of the lane line

> - When the point cloud is outside the current window, you can drag the point cloud by **press and hold the right mouse button and hold it down**
> - **Press and hold the left mouse button** to adjust the viewing angle

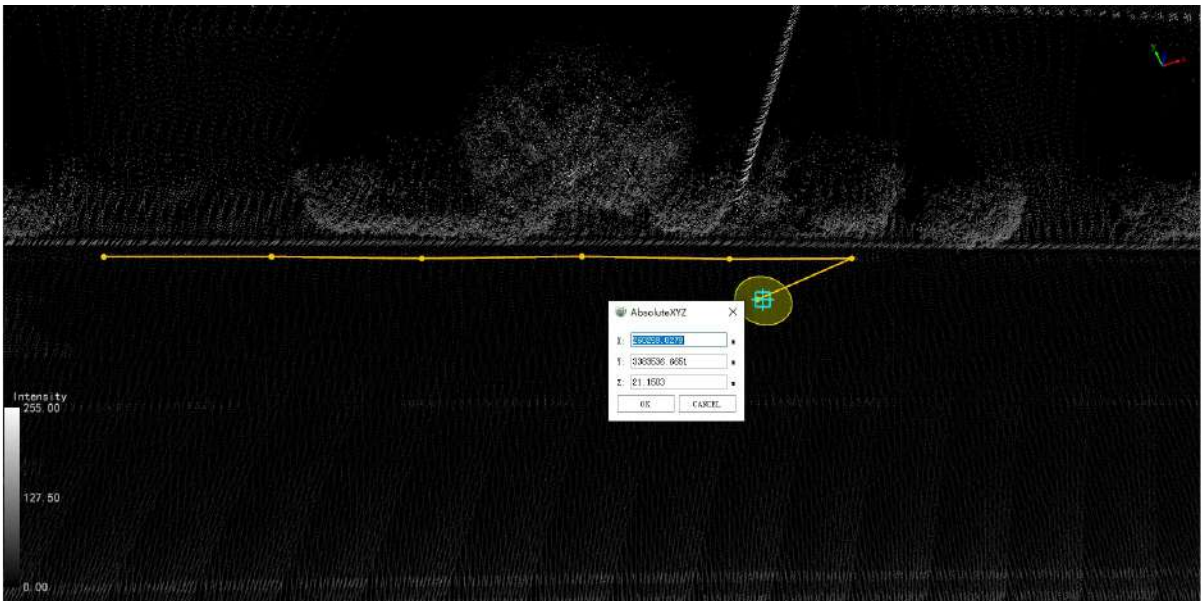
4.switch between different drawing modes through shortcut keys during drawing

function	Shortcut key	describe
Line, polygon	S/s	Briefly 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 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

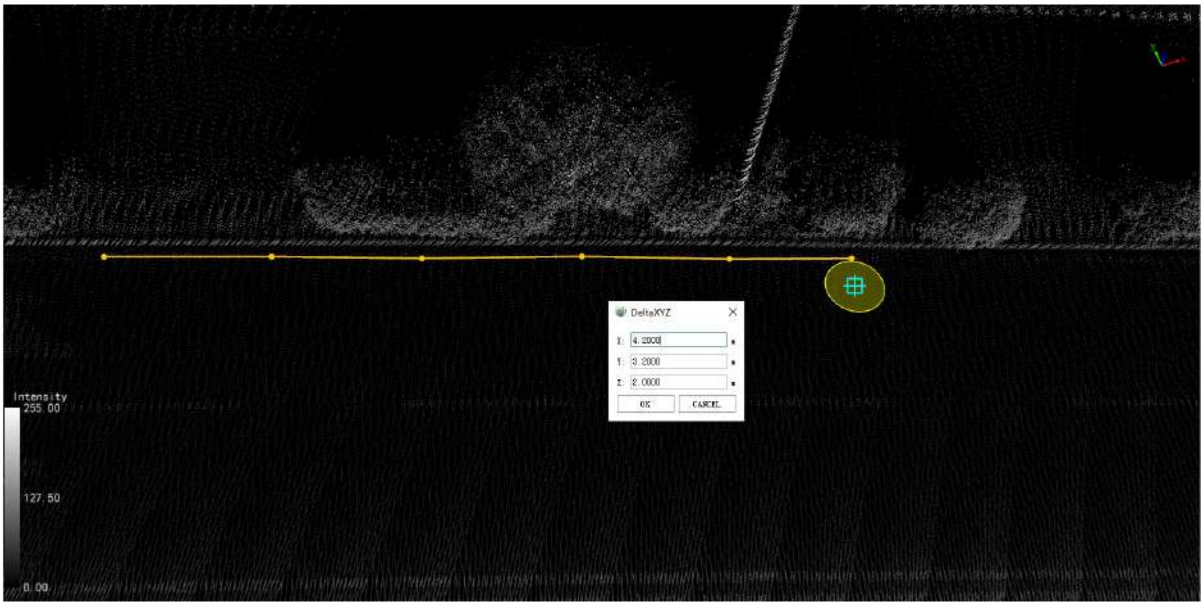
5.The right mouse button contains the following operations:



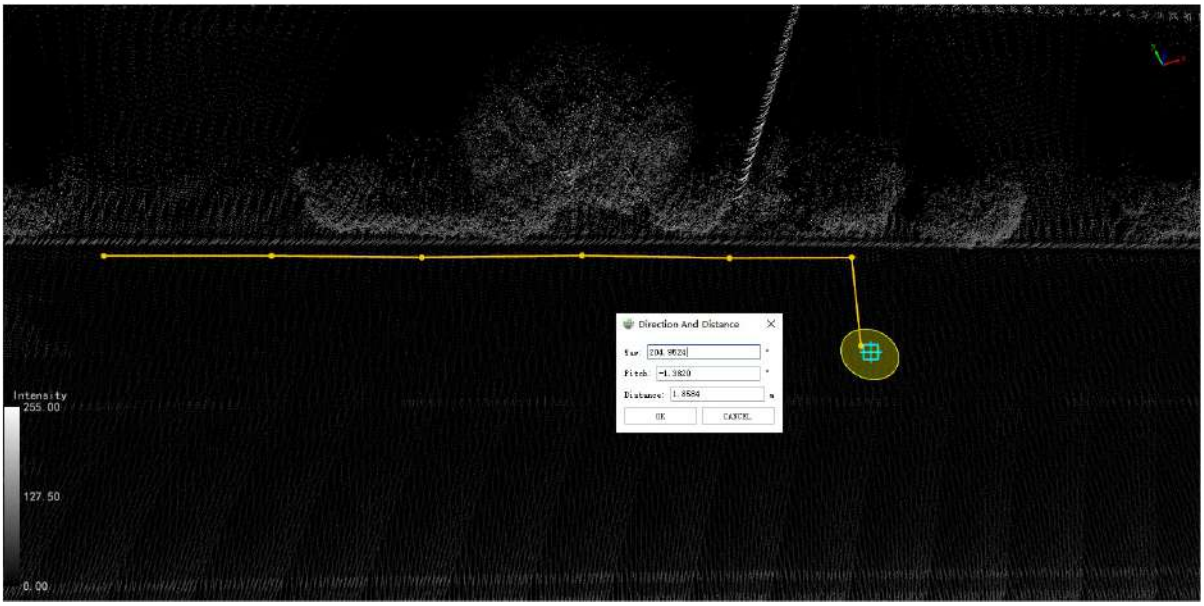
- SnapToFeature: various snap modes can be switched at any time in the snap mode menu.
- Back Point: click a back point to back out nodes in order. (The first node cannot be rolled back.)
- Absolute XYZ: With the absolute XYZ function, you can specify the absolute XYZ coordinates during the drawing process. The coordinate value entered in the pop-up box is the position drawn by the node.



- Delta XYZ: With the relative XYZ 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.



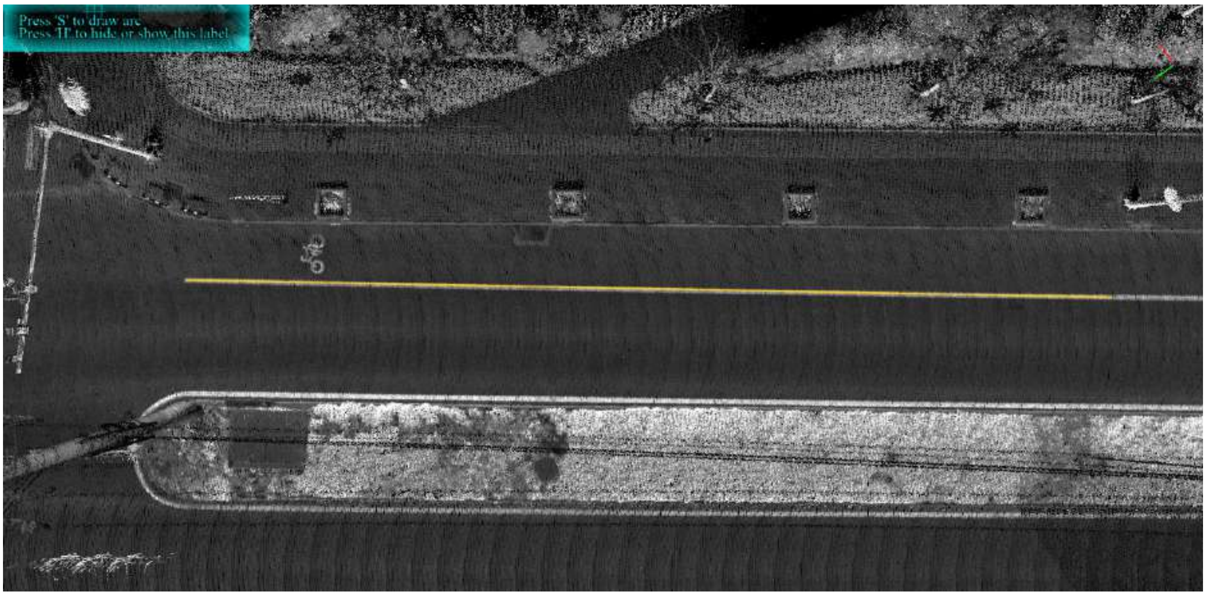
- Direction Distance: With the direction distance function, you can specify the direction distance of the next node relative to the previous node in the drawing process. The deflection angle is the included angle with the Y axis of the coordinate system, the pitch angle is the included angle with the 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.



- CircularArc: with the arc calculation function, an arc with specified direction, size and position can be generated during drawing. As shown in the figure below, the arc direction refers to the angle of the arc relative to the Y axis of the coordinate system; Arc pitch refers to the direction relative to the XY plane of the coordinate axis; Arc tumbling 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; The arc length refers to the side length of the generated arc; Edge 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, you can double-click the last node with the left mouse button to finish the drawing, or you can click the Finish button with the right mouse button to finish the drawing of the current object.
- Cancel: In the drawing process, you can click Cancel with the right mouse button to exit the current drawing.

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



Lane solid line drawing

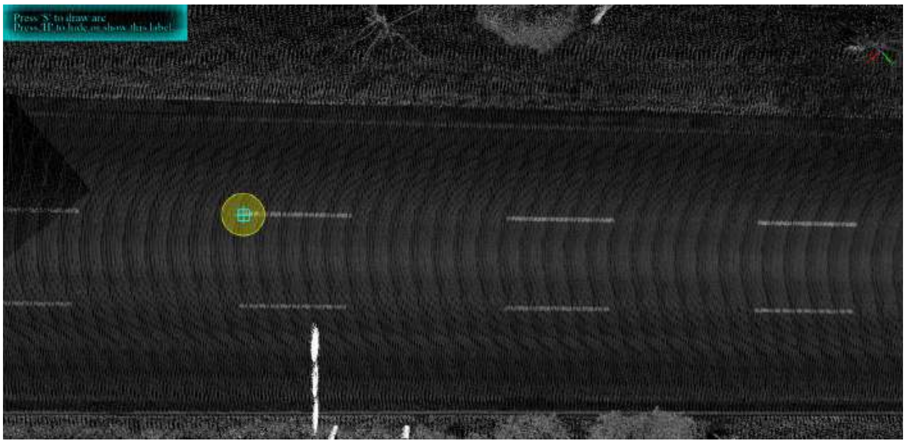
> Note: In the process of clicking the node, you can use the mouse **wheel, right button** to respectively zoom and move the point cloud.

Lane Dashed Line

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

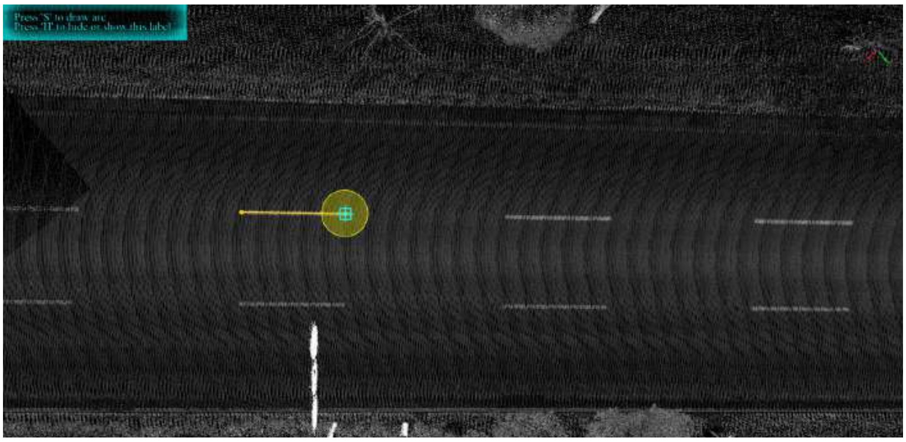
Steps

1. In the Road Shape panel, click the drop-down mark on the right side of the lane line, and select the **lane dashed line** button
2. Mouse **left click** click **start point as the first key point**, **determine the starting point of the dashed line**



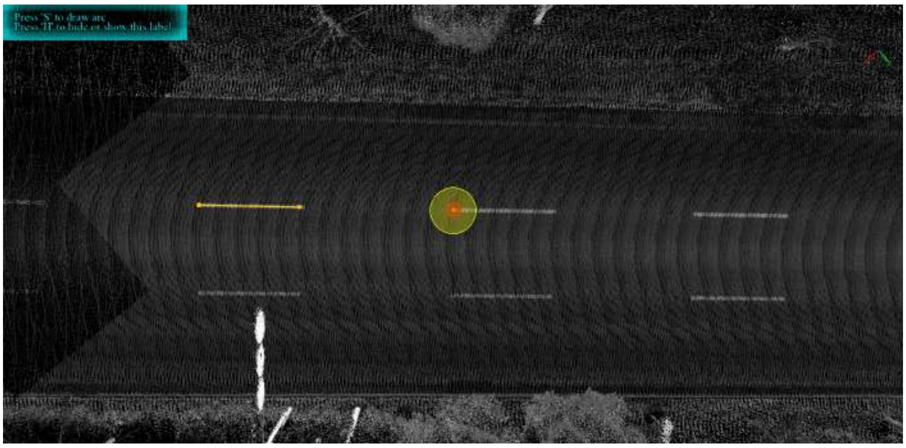
The first key point

3.Move the mouse, **left click** 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

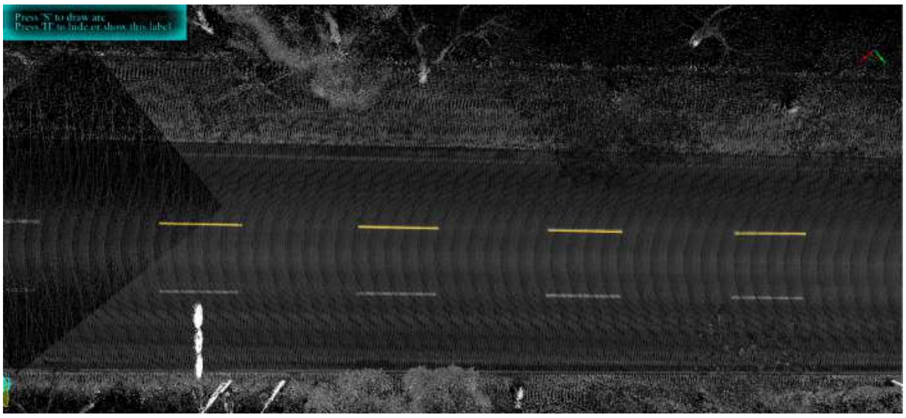
4.Move the mouse, **left click** click **the third key point**, the details are shown in the figure, **used to determine the length between the dashed segments**



The third key point

5.**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, you can drag the point cloud by **press and hold the right mouse button and hold it down**
> - **Press and hold the left mouse button** to adjust the viewing angle



Draw dash Line

> Note:
>
> - In the process of clicking on a node, you 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 solid lane function

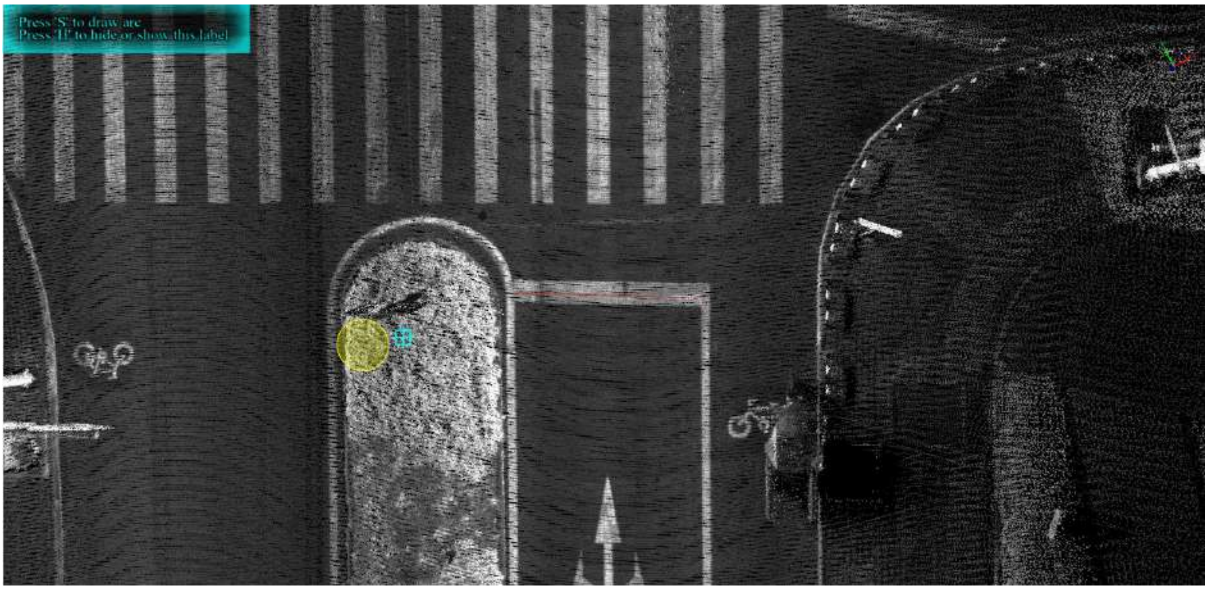
Stop Line

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

Steps

1. Click the **Stop Line**  button in the Road Shape panel
2. Mouse **left button** in the point cloud **click** to select the starting point of the line
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, you can drag the point cloud by **press and hold the right mouse button and hold it down**



Drawing stop line

> Note:
>
> - In the process of clicking on a node, you 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 Solid Line function

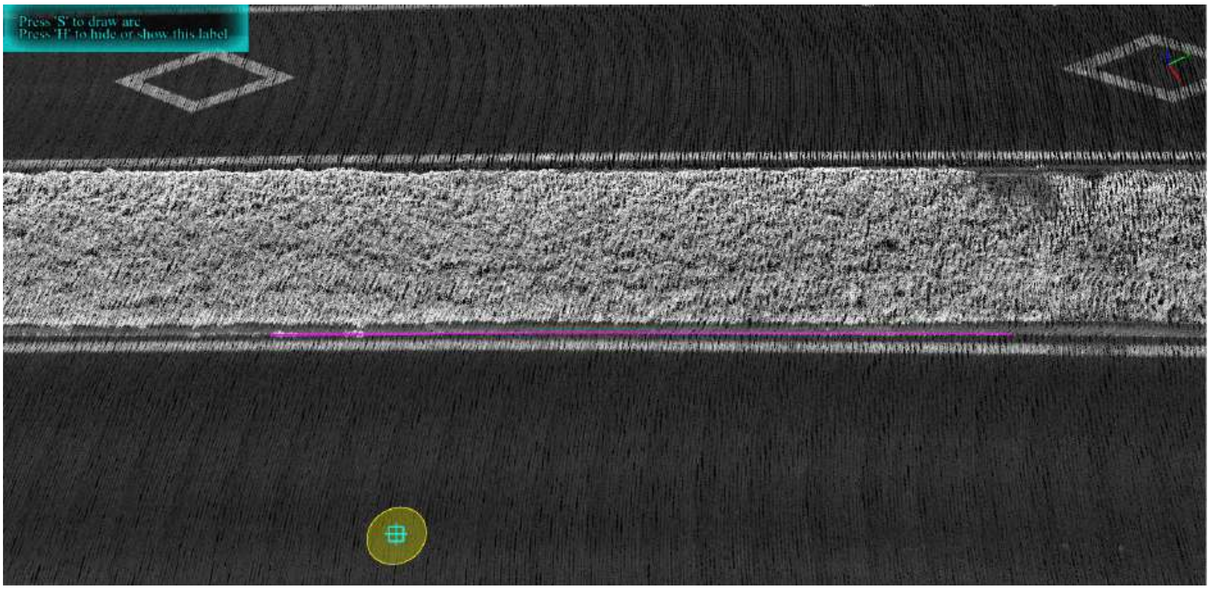
Road Curb

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 **Road Curb** 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 curb

> When the point cloud is outside the current window, you 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



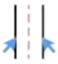
Road curb drawing

> 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.
> - The right mouse button operation and shortcut button of this function are the same as the Lane Solid Line function

Generate Road Center line

Function Description： It is 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 Road 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, you can drag the point cloud by **press and hold the right mouse button and hold it down**
   > - **Press and hold the left mouse button** to adjust the viewing angle

Before generating

After generating

> Note: In the process of clicking, you can use the mouse **wheel, right button, and left button** to respectively zoom and move the point cloud, adjust the angle of view

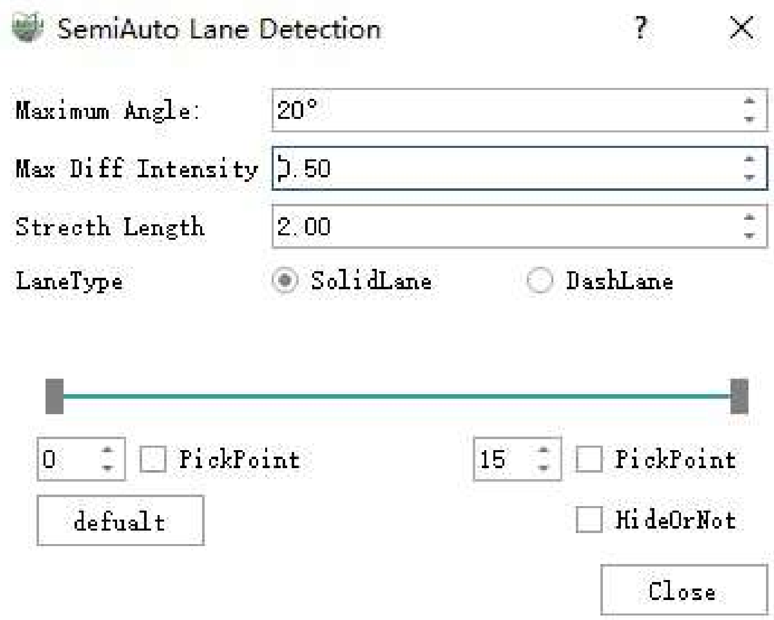
Solid Lane Detecter

Function Description: 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 **Lane Line Detection button**  in Road Shape panel.

2.The **Detection Settings dialog box** will pop up.



Solid line 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, **suggested below 30 degrees**
- **Maximum intensity deviation**：The default value is 0.5, which indicates the tolerance value of the intensity difference between the front and rear segments of the solid line. If the intensity difference is greater than the set value, the growth will stop.**The default value is recommended**
- **Extendable distance**： 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**
- **Lane Type**: The default type is solid line, **selects solid line**
- **Intensity adjustment**: For stretching the intensity by manually selecting the maximum and minimum values where the strength is not easily distinguished

- **Click**：The left is the minimum intensity and the right is the maximum intensity. The maximum and minimum intensity can be determined by clicking the point cloud with the left mouse button after checking the dot selection box.

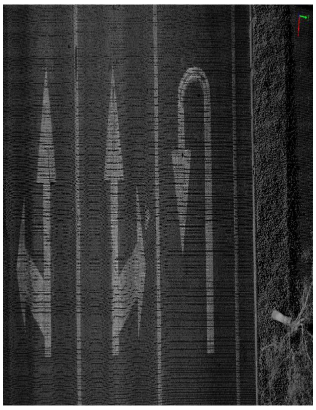
  > The scroll bar, value box and point selection box are linked together, and the intensity can be adjusted in any operation mode. Meanwhile, the point cloud display will also be linked in real time (intensity stretching of the whole point cloud).

- **Default**：Restore the original maximum and minimum intensity, and the point cloud display will also be linked in real time

- **Hide the point cloud outside the range**：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.**Left mouse click**Click on a point on the lane line point cloud as the**start point**, and then**left mouse click**to select another point on the lane line as the**growth direction**

4.Automatic lane detection starts 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 adjustment, dragging, and zooming** through **left mouse button, right mouse button, and scroll wheel**.

5.Use shortcut keys for **real-time shape adjustment** :

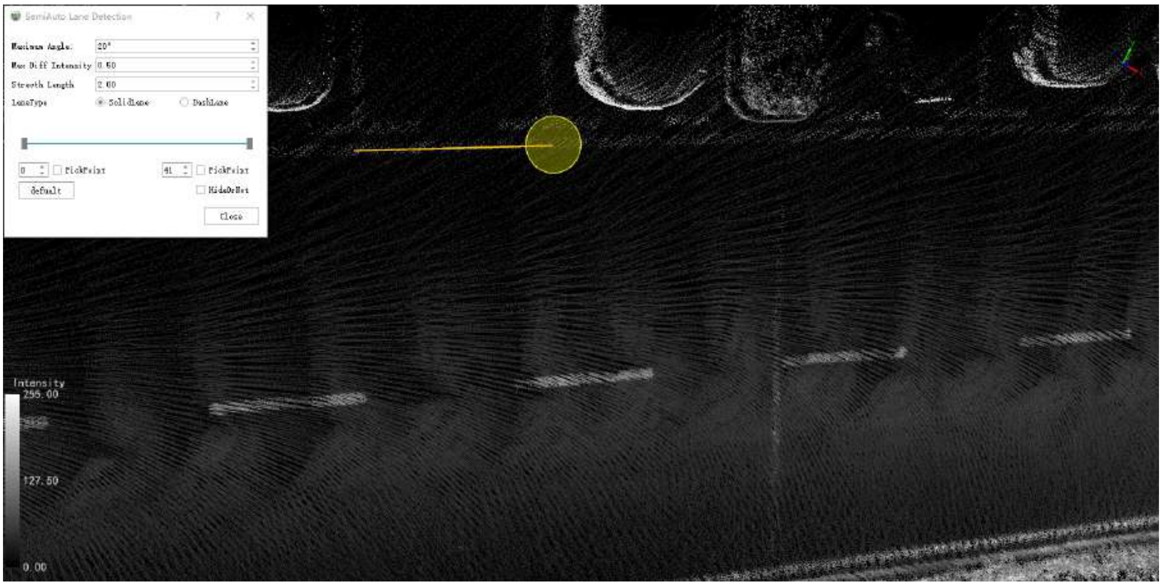
Shortcut name	Effect
Space	Interrupt the growth
Enter	Confirm growth results
Right mouse button	Rewind a dashed segment (only valid after interruption, invalid during growth)

- The program is automatically **interrupted** due to the reasons such as low curb height, actual termination, actual interruption, etc.; if the automatic detection does not grow correctly, you can use the **space bar** to manually **interrupt**
- After **interruption**, you can **right click** to go back to adjust the shape
- After **interruption**, operations 3 and 4 can be repeated, followed by automatic detection

6.When the detection 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, direction point**



Click the starting point, direction point

- **Growth results**

Solid line growth results

Dashed Lane Detecter

Function Description: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.

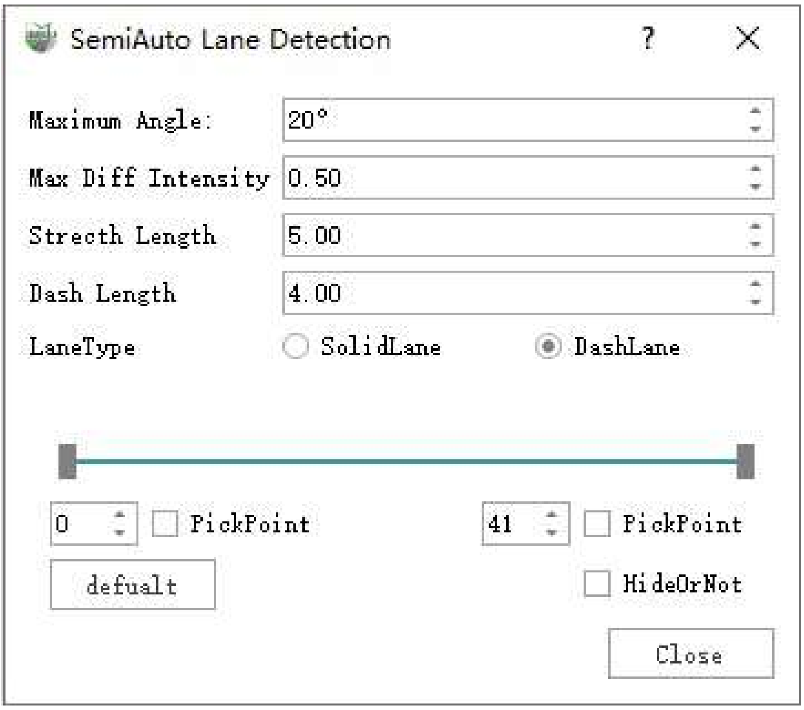
> Note: This function is only effective for areas where the lane line and road surface have a high degree of recognition in point cloud intensity

Steps

1.Click **Lane Line Detection button** in Road Shape panel.

2.The **Detection Settings dialog box** will pop up.

Dashed Lane Dialog



Dashed 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, **suggested below 30 degrees**
- **Maximum intensity deviation**: The default value is 0.5, which indicates the tolerance value of the intensity difference between the front and rear segments of the solid line. If the intensity difference is greater than the set value, the growth will stop.**The default value is recommended** the interval between two dashed line segments.**Smaller intervals should be selected if the dashed line has different intervals**
- **Extendable distance**:The default value is 5 meters, which indicates the interval between two imaginary line segments.**If the dotted line has different intervals, smaller intervals should be selected**

- **Dotted line segment length**： The default is 4 meters, which means the length of a single **dashed line segment**
- **Lane line type**：line type, select dashed line
- **Intensity adjustment**：

  > Intensity adjustment, please refer to see Lane Solid Line intensity adjustment module

3.**Left mouse click**Click on a point on the lane line point cloud as the**start point**, and then**left mouse click**to select another point on the lane line as the**growth direction**

4.Automatic lane detection starts 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 adjustment, dragging, and zooming** through **left mouse button, right mouse button, and scroll wheel**.

5.Use shortcut keys for **real-time shape adjustment** :

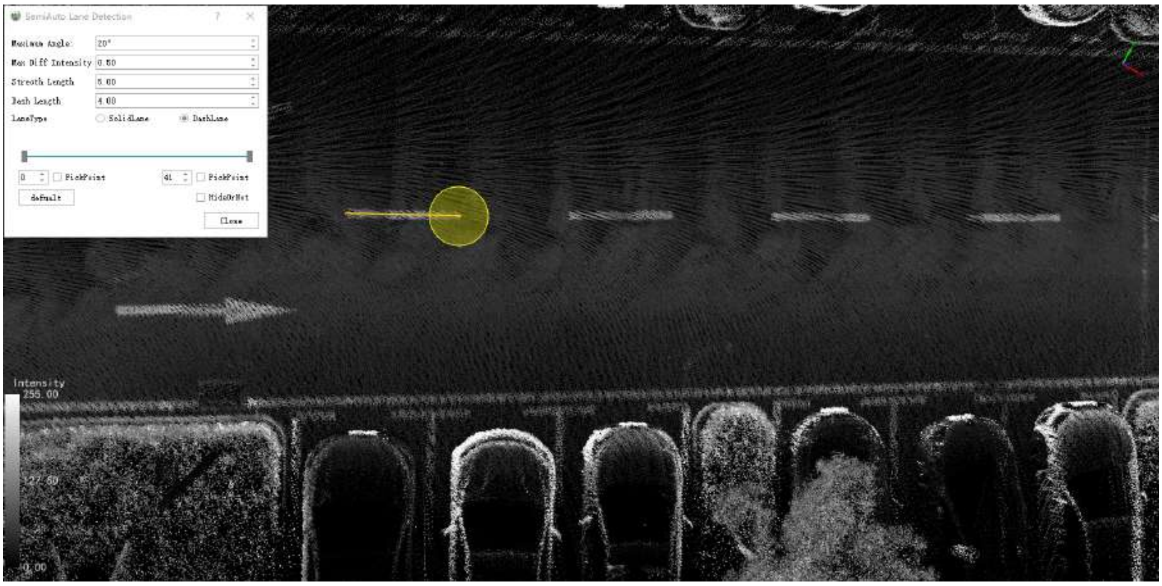
Shortcut name	Effect
Space	Interrupt the growth
Enter	Confirm growth results
Right mouse button	Rewind a dashed segment (only valid after interruption, invalid during growth)

- The program is automatically **interrupted**due to drastic changes in intensity, actual termination, actual interruption, etc.; You can use the **space** to manually **interrupt** if the automatic growth is not correct growth
- After **interruption**, you can use **right mouse button** to go back (go back a dashed line) to adjust the shape
- After **interruption**, you can repeat operations 3 and 4, followed by automatic growth

6.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, you can continue to right-click to go back and adjust the threshold again for growth.

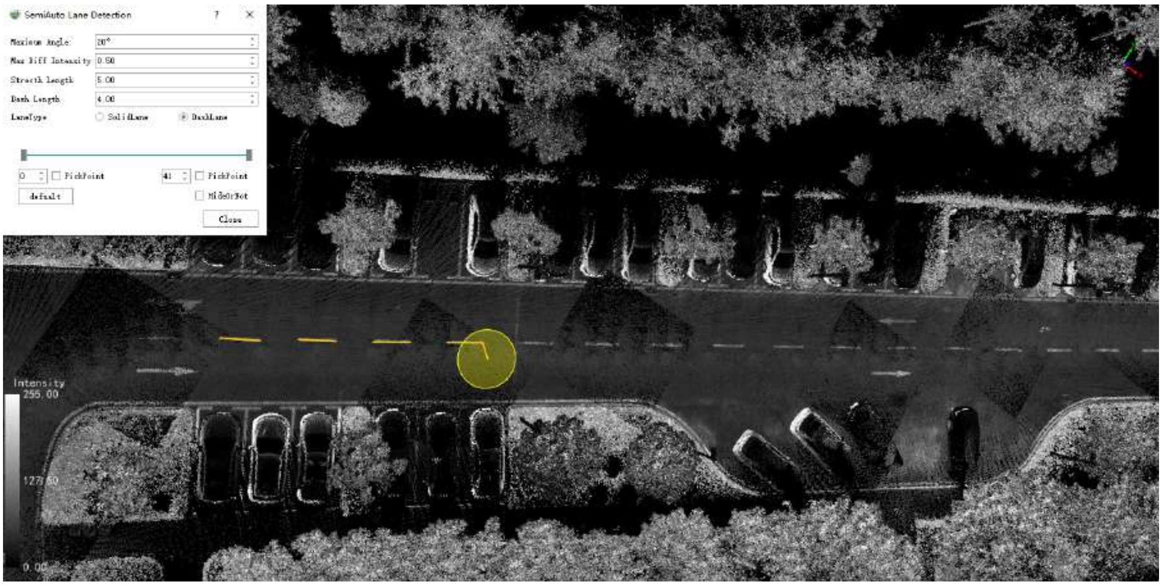
Rendering

- **Click the starting point, direction point**



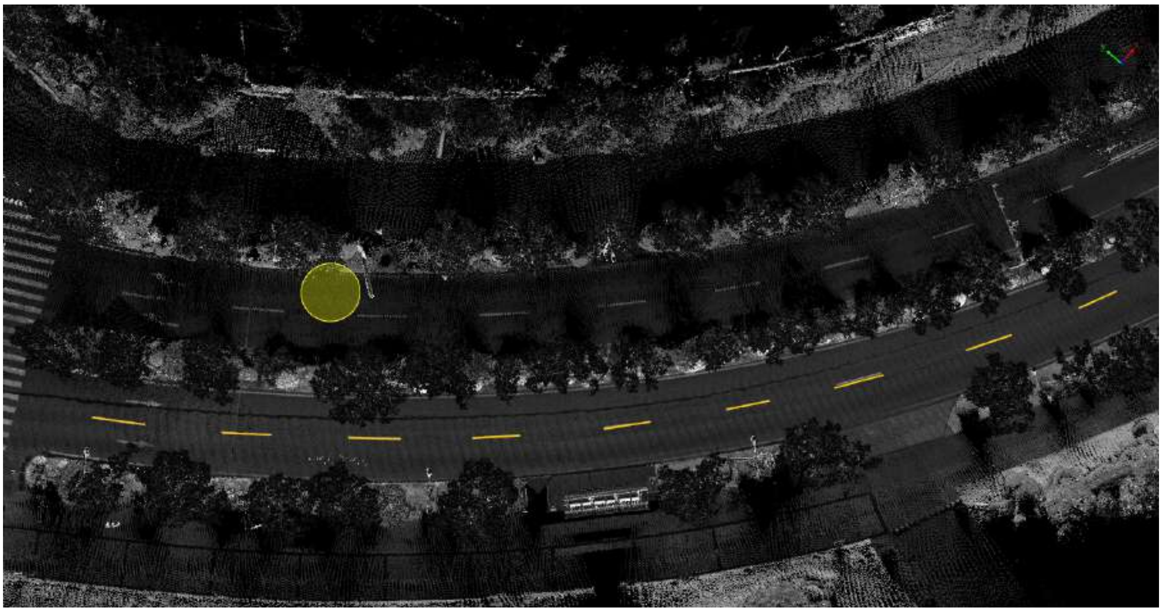
Click the starting point, direction point

- **Automatic or manual interruption during growth**



Dotted line growth results

- **Growth results**


Dashed line growth results

Road Edge Detection

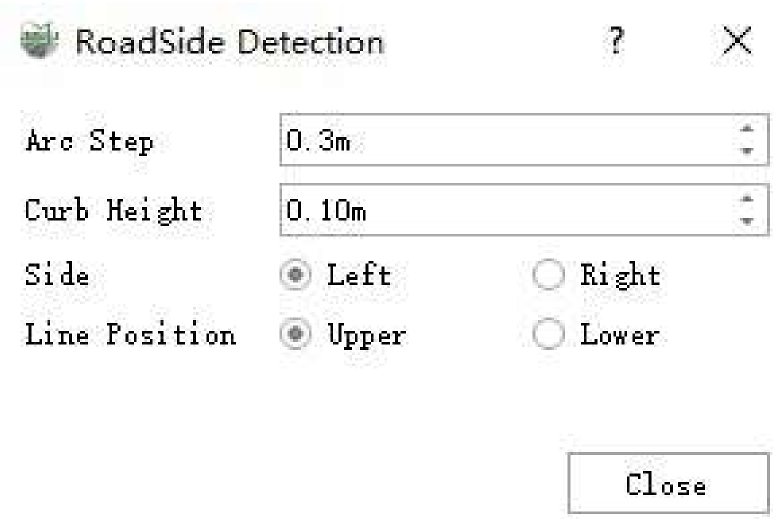
Function Description：Left click to select two points to obtain the starting point and direction point. Along the selected direction, automatic roadside detection will be carried out from the starting point.

> Note: This function is only effective for areas with clear curbs and the curb height is above 10 cm

Steps

1.Click the **Road Edge Detection**  button in the Road Shape panel

2.Pop up **Detection Settings dialog box**



Pop up Detection Settings dialog box

Parameter Description:

- **Arc step length**: 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
- **Minimum 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.
- **Curve position**: The default is the left side, which indicates which side of the straight line formed by the starting point and direction points of the actual curb point cloud is located, which is selected according to the actual situation
- **Line Position**: The default is not the upper edge, 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.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 initial 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

4.Along the selected direction, automatic roadside detection starts from the starting point

> During the detection process, the center of the screen is automatically adjusted to the point currently detecting; 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.

5.Use shortcut keys for **real-time road shape adjustment**:

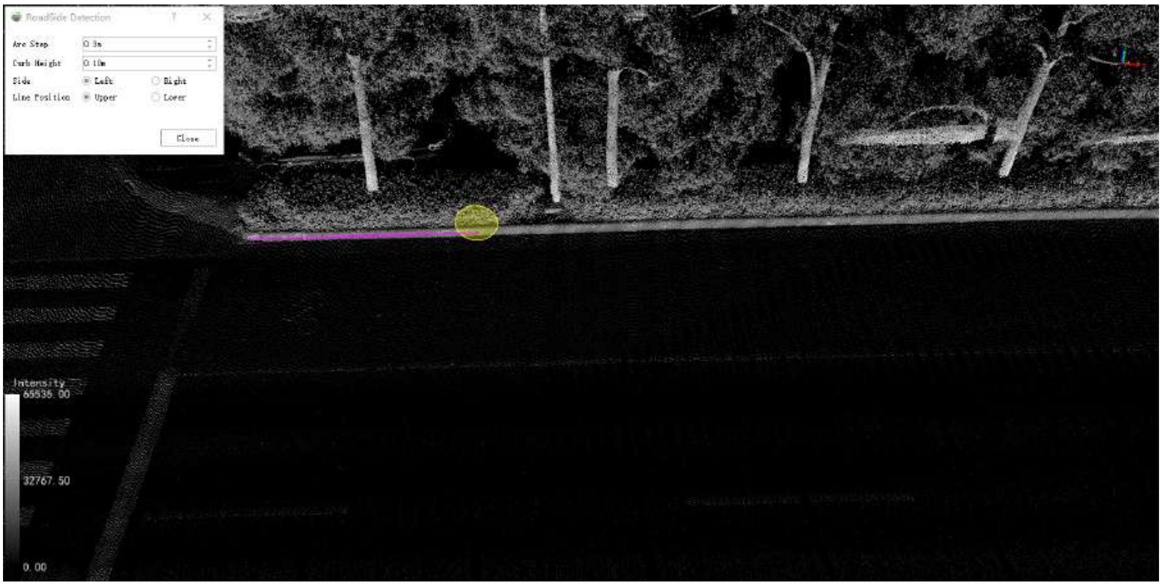
Hotkey	Function
Space	Interrupt growth
Enter	Confirm growth results
Right click	Go back a little (only after line break, invalid during growth)

- The program is automatically **interrupted** due to the reasons such as low curb height, actual termination, actual interruption, etc.; if the automatic detection does not grow correctly, you can use the **space bar** to manually **interrupt**
- After **interruption**, you can **right click** to go back to adjust the shape
- After **interruption**, operations 3 and 4 can be repeated, followed by automatic detection

6.When the detection 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

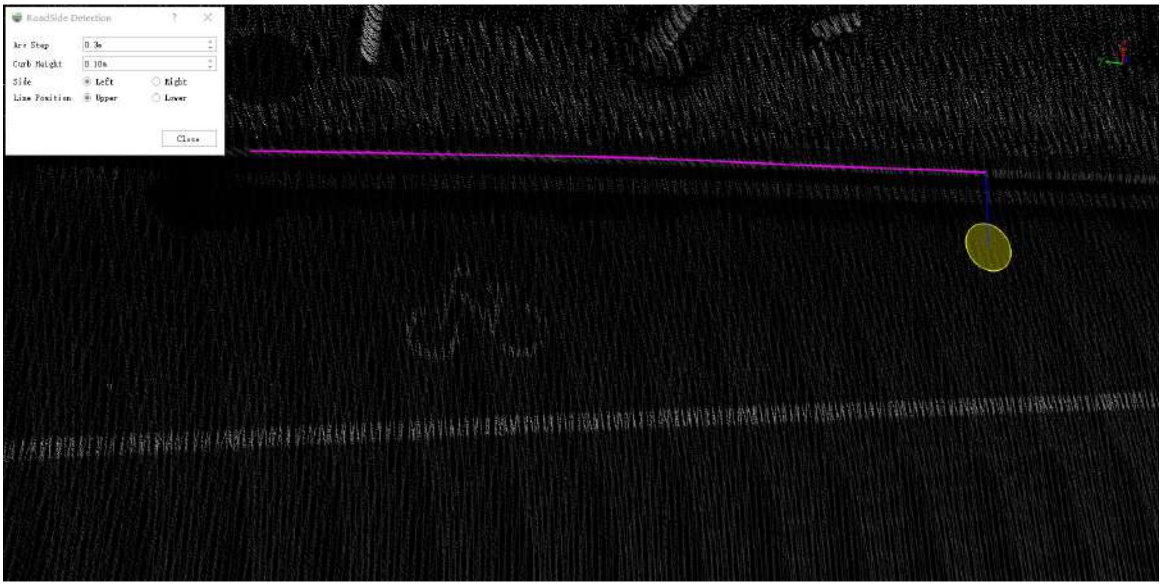
Straight Road Curb

- **Click the starting point, direction point**



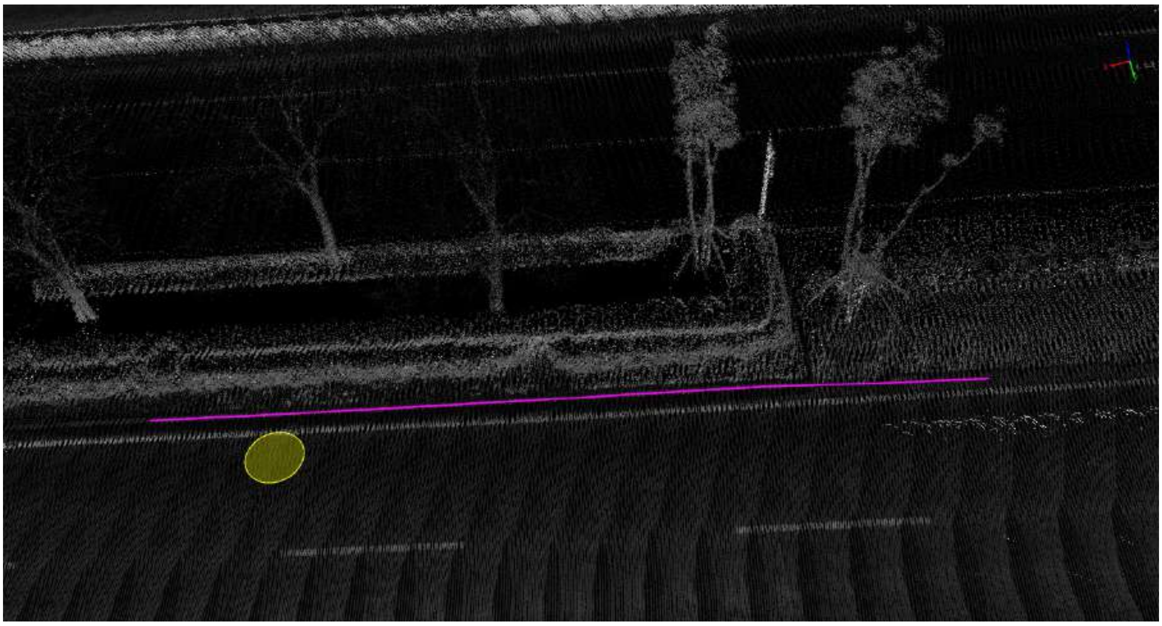
Click the initial point and direction point

- **Interrupt**, and click again



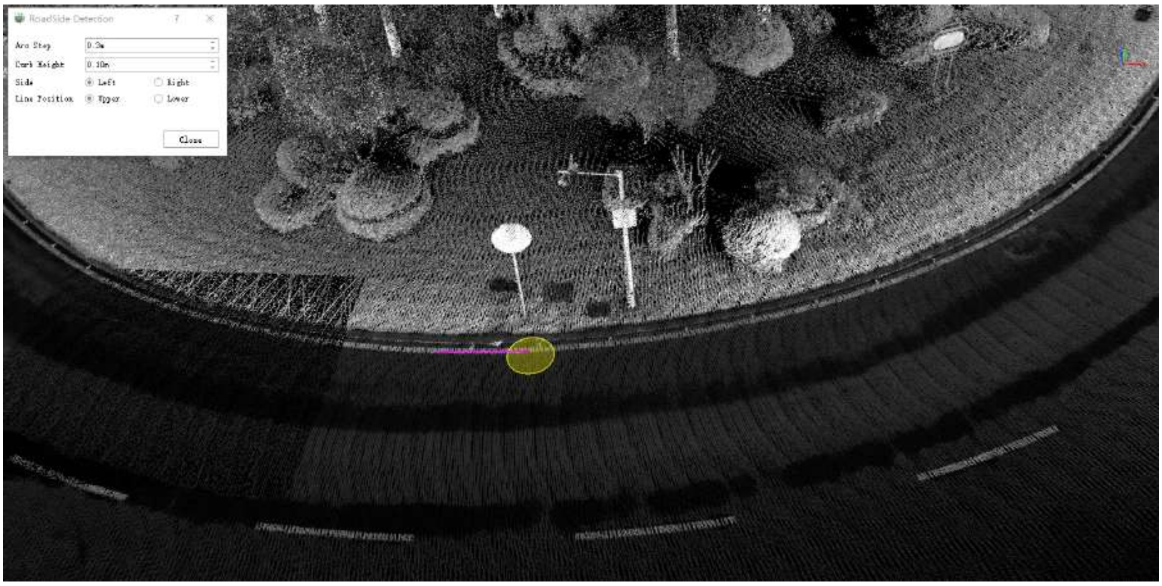
Interrupt

- **Curb generation result**


Curb generation result

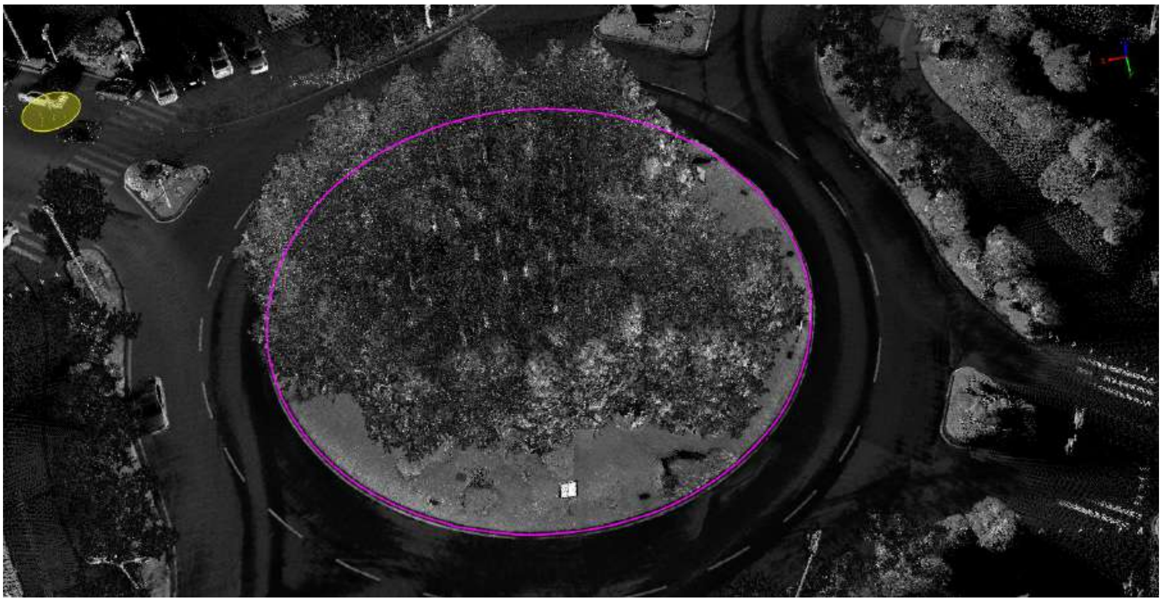
Closed Round Curb

- **Click the starting point, direction point**



Click the initial point and direction point

- **Round curb generation result**


Round curb generation 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
    - Straight ahead
    - turn left
    - turn right
    - turn left/right
    - Straight/Left Turn
    - Straight/Right
    - Straight/left turn/right turn
    - TurnAround
    - Straight/TurnAround
    - Left turn/Turnaround
    - Merge Left
    - Merge Right
    - RightTurn/Turnaround
    - Straight/RightTurn/Turnaround
    - Left turn/RightTurn/Turnaround
- Horizontal marking
    - Crosswalks
    - Longitudinal deceleration markings
- Pavement graphic markings
    - Crosswalk preview
    - SpeedReduceMark
    - No-ParkingArea
    - Non-motorized Vehicle Marking
    - No U-turn sign
    - Add text
- TemplateMatching
    - TemplateMatchingSidebarParametersDetail
    - TemplateMatchingCustomTables
    - TemplateMatchingAutoMatch
    - TemplateMatchingManual
    - TemplateMatchingDatabaseFile
    - New Template
- GroundMarkingBatchProcess

Straight Arrow

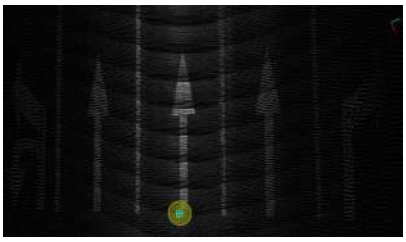
Function Description：The straight arrow means that the current lane can only go straight. This function is used to manually select the point cloud to vectorize the straight arrow.

Steps

1.In the Road Surface panel, click the **Straight**  button

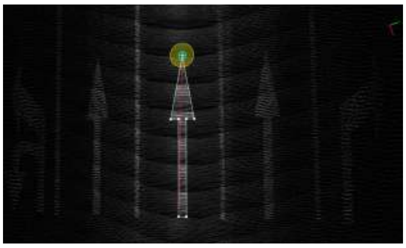
> - The key points in the arrow icons need to be aligned with the actual point cloud
> - The red point is the first key point and the green point is the second key point

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



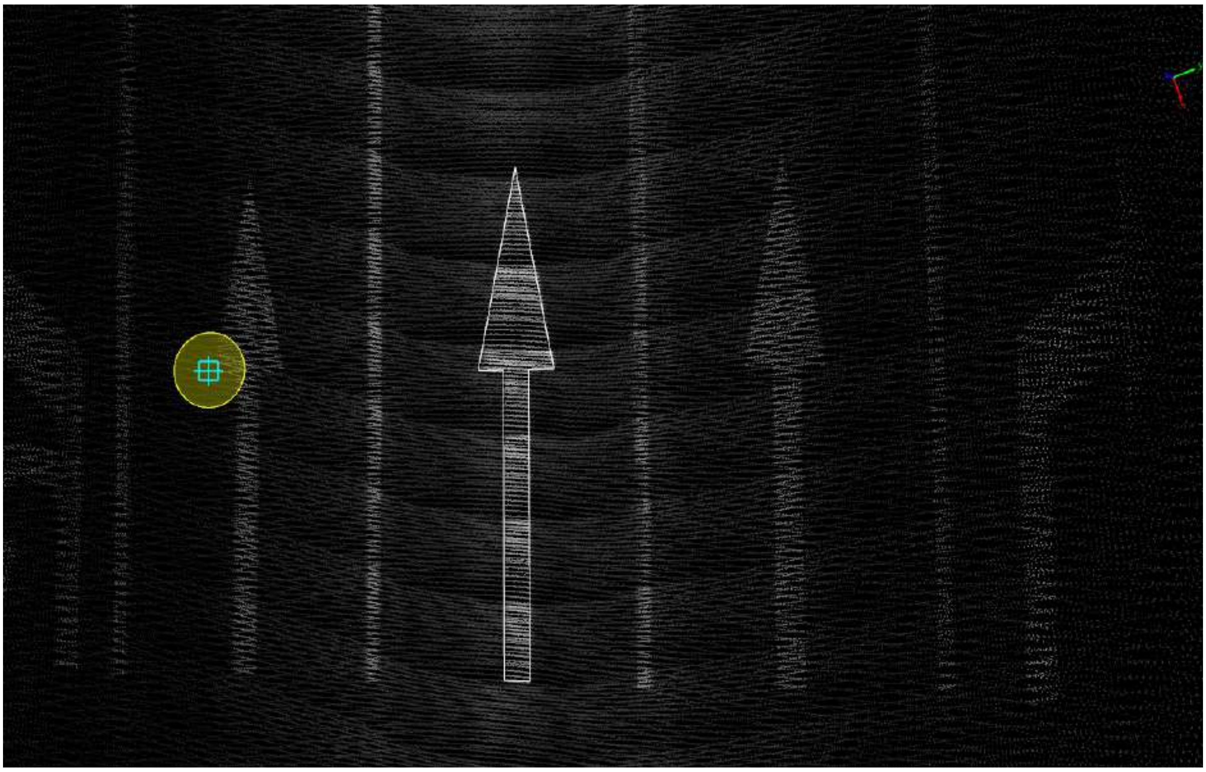
First key point

3.Move the mouse to see **template** and **actual point cloud** fit, **left mouse click** tap **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 generation

Turn left

Function Description：The left turn arrow indicates that the current lane can only be turned left, this function is used to manually tap the point cloud for vectorisation of the left turn arrow.

Steps

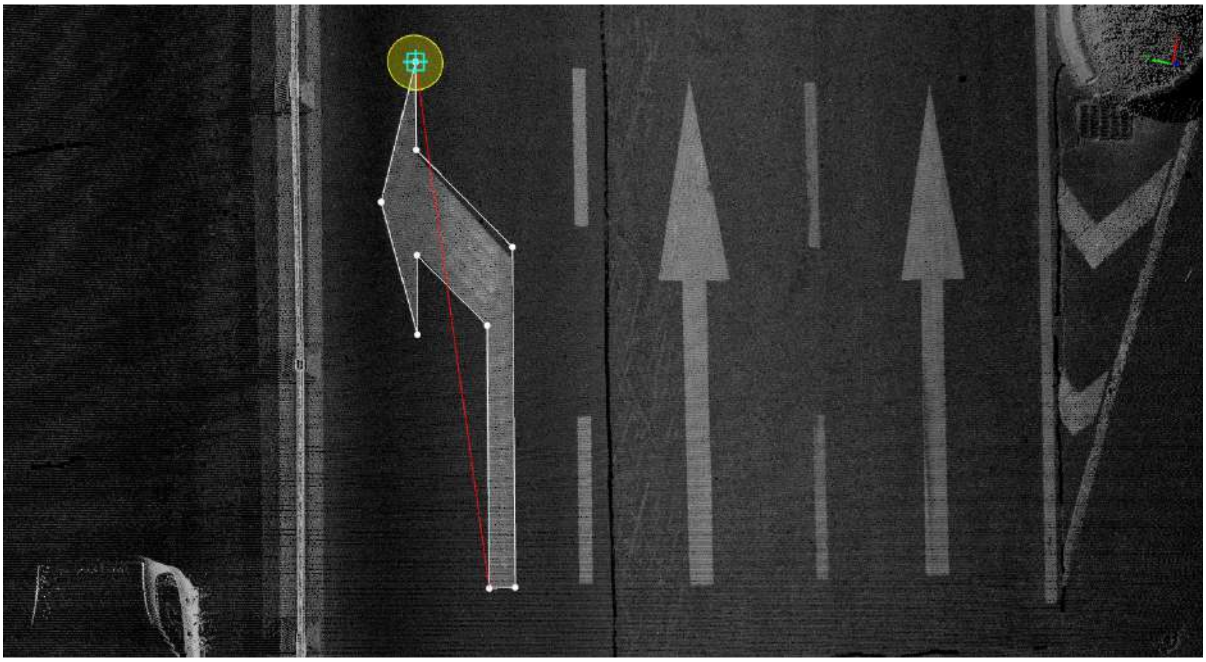
1.Click the **Turn Left**  button in the Road Surface panel

> - 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

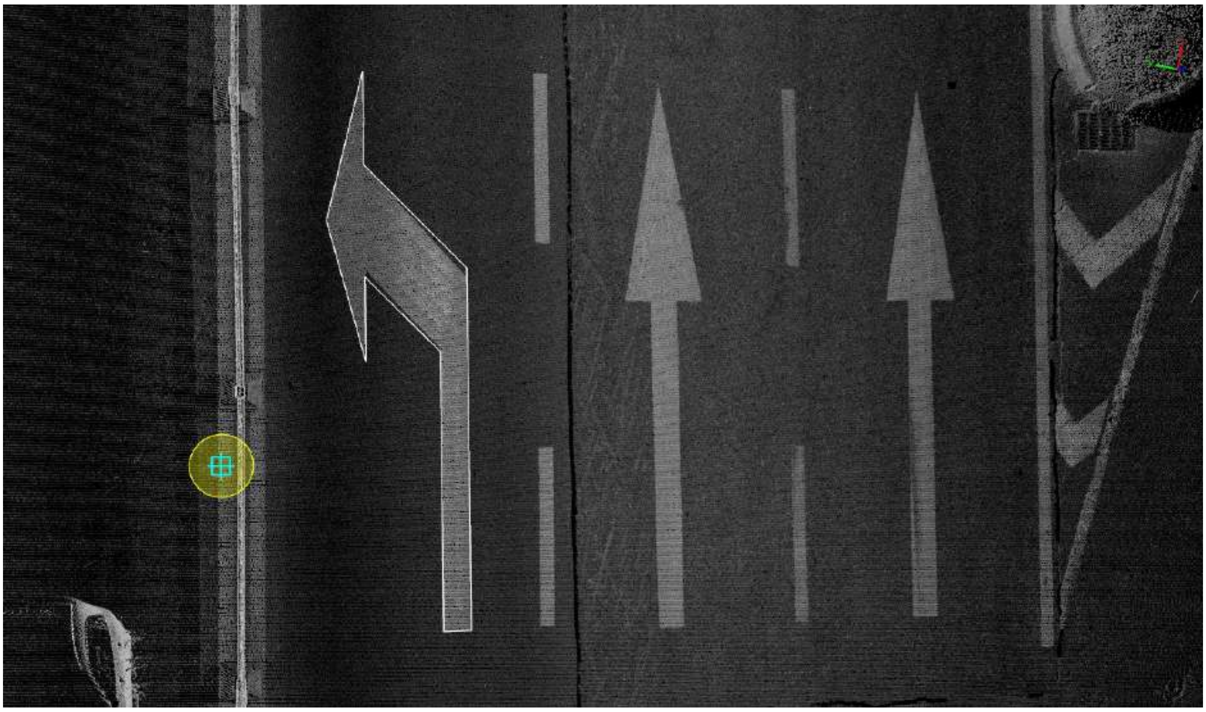
2.**Left click to select the key point** of the left arrow point cloud, the details are as follows, **the first key point** in the icon

First key point

3.Move the mouse to check the fit between the **template** and the **actual point cloud**, left 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



Left arrow generation

Turn right

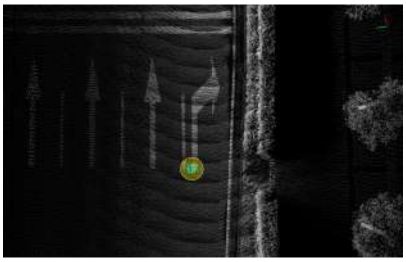
Function Description：The right turn arrow indicates that the current lane can only be turned right, this function is used to manually tap the point cloud for vectorisation of the right turn arrow.

Steps

1.Click the **Turn 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 dot is the first key point and the green dot is the second key point

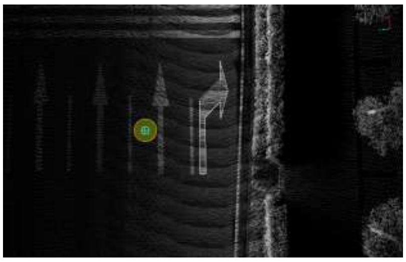
2.**Left mouse click** to select the right turn arrow point cloud key point with the following details and the **first key point** in the icon



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 right turn arrow production, details are as follows, icon in **second key point**

Second key point



Right turn arrow drawing

Left and Right Arrow

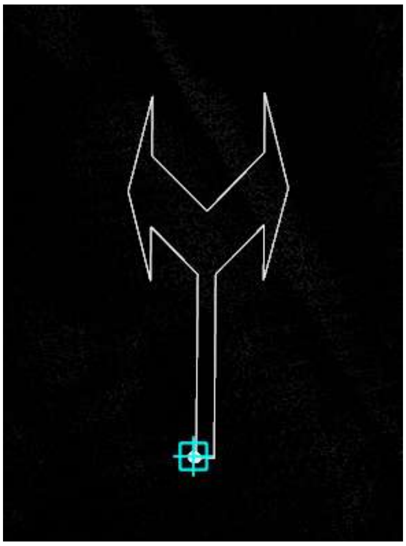
Function Description: The left/right arrow indicates that the current lane can only turn left or right. This function is used to manually click on the point cloud to vectorize the left/right arrow.

Steps

1.Click the **Turn Left/Turn Right**  button in the Road Surface panel

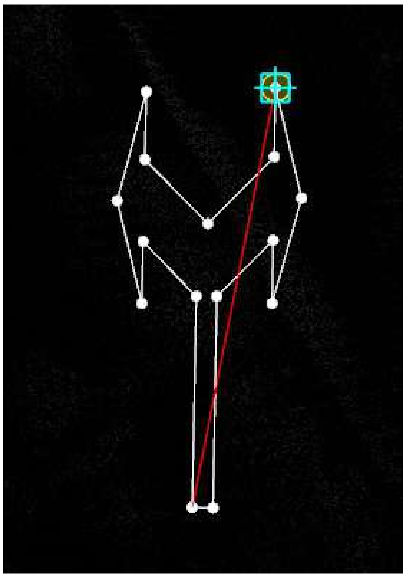
> - The key points in the arrow icons need to be aligned with the actual point cloud
> - The red point is the first key point and the green point is the second key point

2.**Left 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

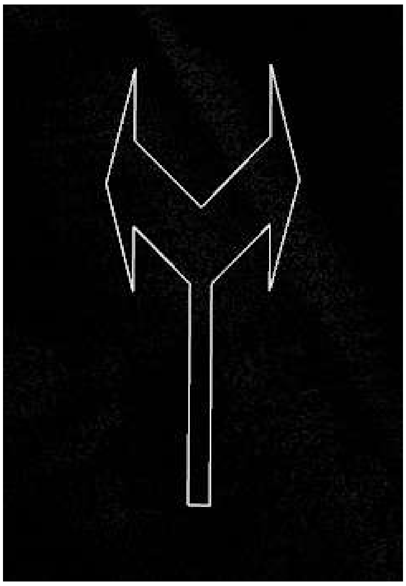


First key point

3.Move the mouse to check the fit between the **template** and the **actual point cloud**, left 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



Left and right arrow generation

Straight and Left Arrow

Function Description: The straight/left turn arrow indicates that the current lane can turn left or go straight. This function is used to manually click on the point cloud to vectorize the straight/left arrow.

Steps

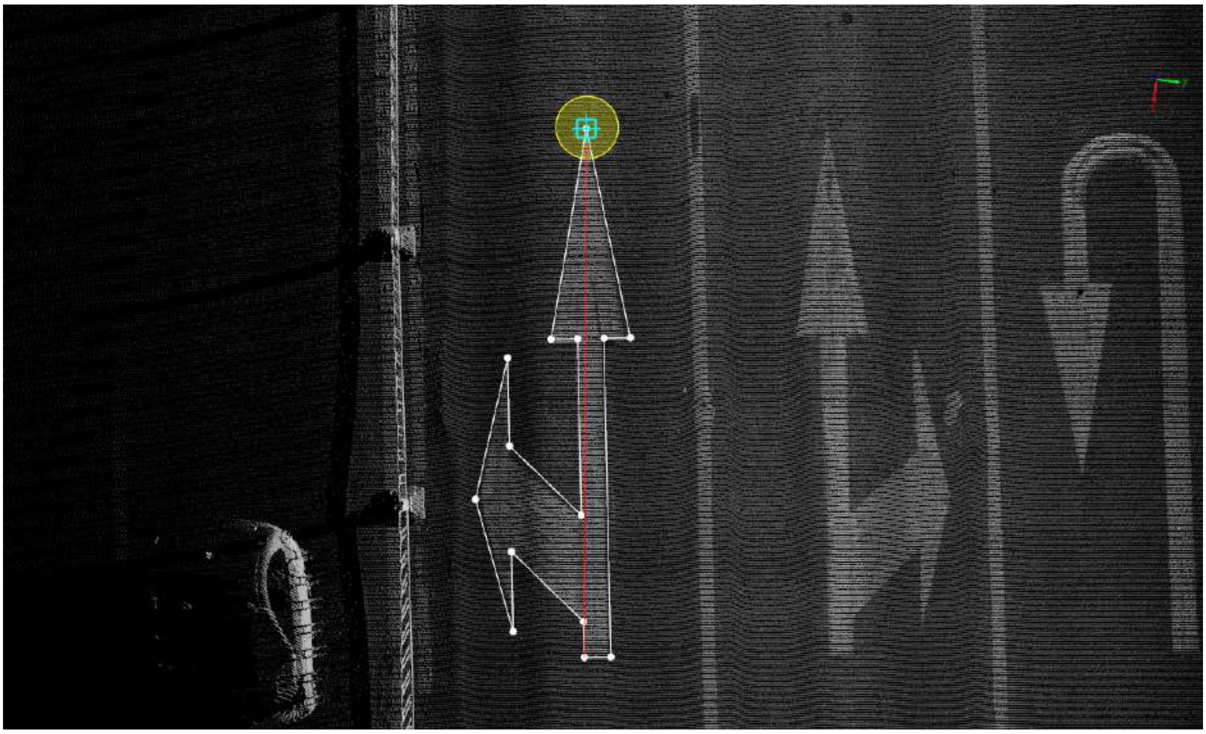
1.Click the **Straight/Turn left**  button in the Road Surface panel

> - 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

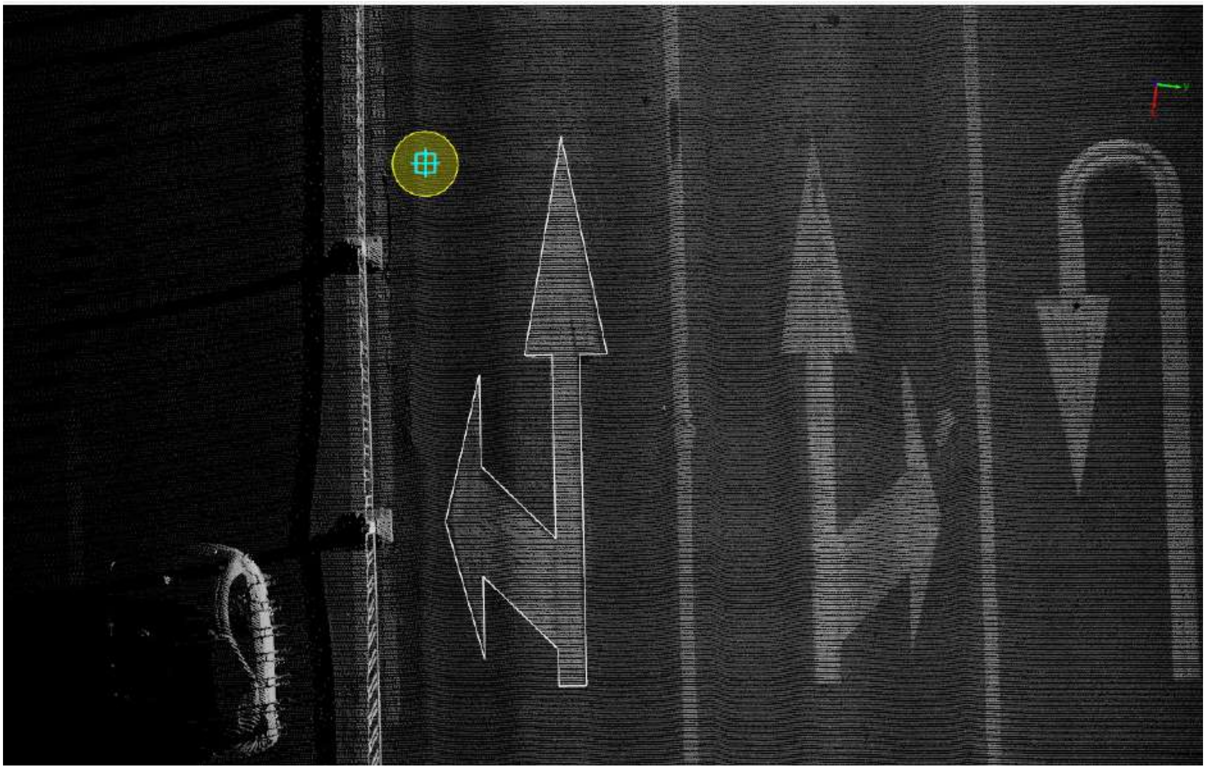
2.**Left mouse click** to select the straight/left turn arrow point cloud key point, the details are as follows, the icon in the **first key point**

First key point

3.Move the mouse to view **template** and **actual point cloud** fit, **left mouse click** tap **second key point** to complete the straight/left turn arrow production, details are as follows, icon in **second key point**



Second key point



Straight and left arrow generation

Straight and Right Arrow

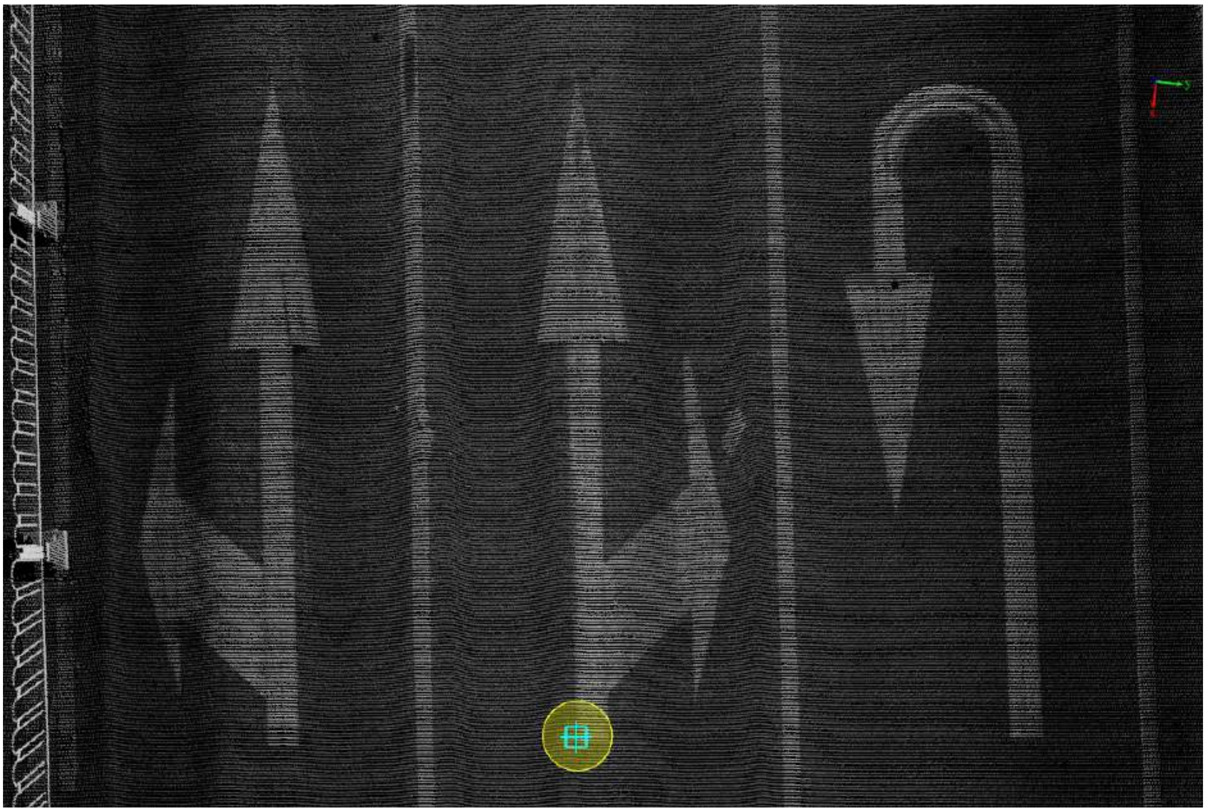
Function Description：The straight/Right turn arrow indicates that the current lane can turn Right or go straight. This function is used to manually click on the point cloud to vectorize the straight/Right arrow.

Steps

1.Click the **Straight/Turn Right**  button in the Road Surface panel.

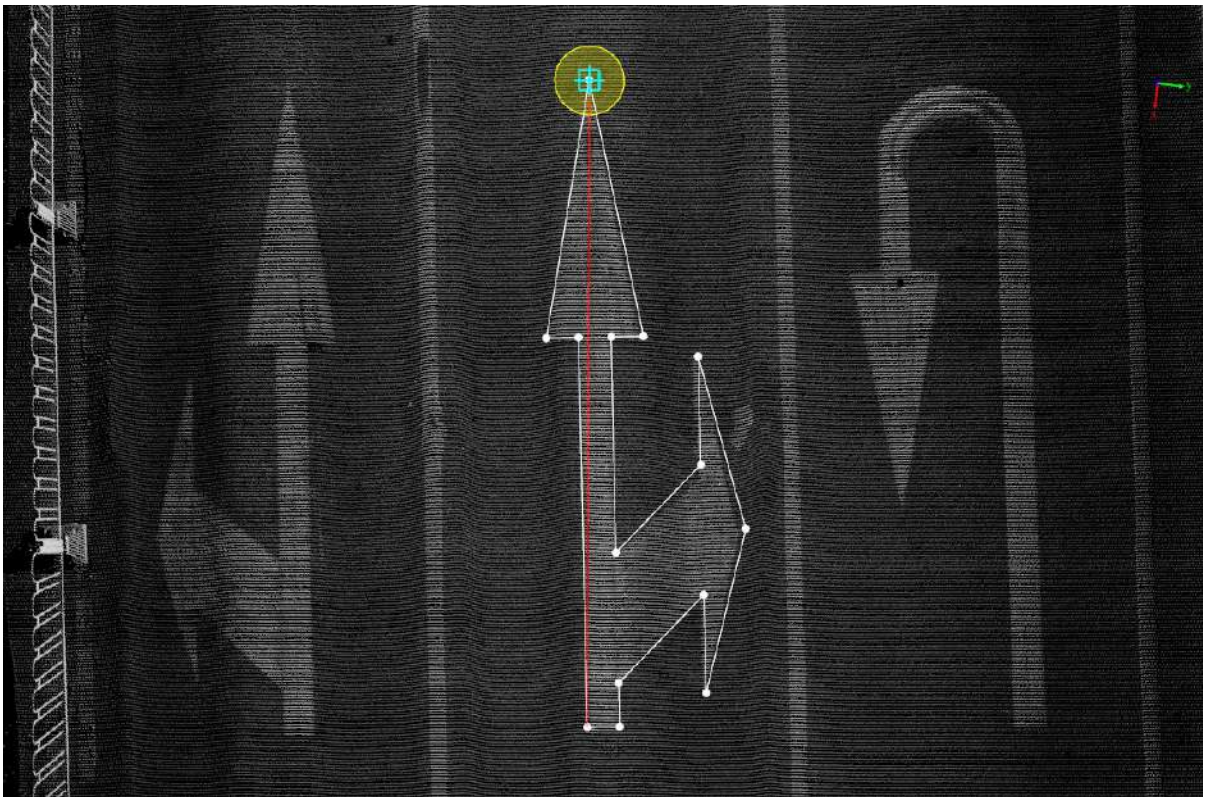
> - The key points in the arrow icons need to be aligned with the actual point cloud
> - The red point is the first key point and the green point is the second key point

2.**Left mouse click** to select the straight/right turn arrow point cloud key point, the details are as follows, the icon in the **first key point**。

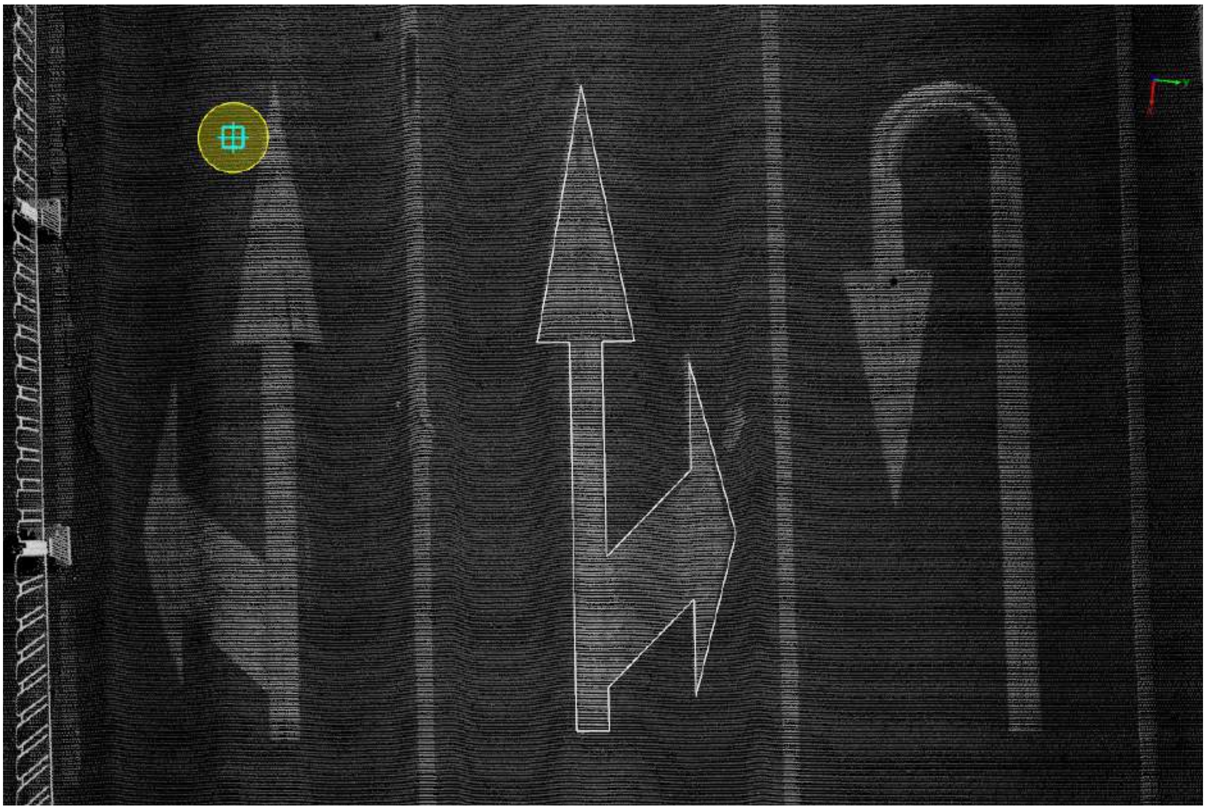


First key point

3.Move the mouse to see **template** and **actual point cloud** fit, **left mouse click** tap **second key point** to complete the straight/right turn arrow production, details are as follows, icon in **second key point**



Second key point



Straight and right arrow generation

Straight,Left and Right Arrow

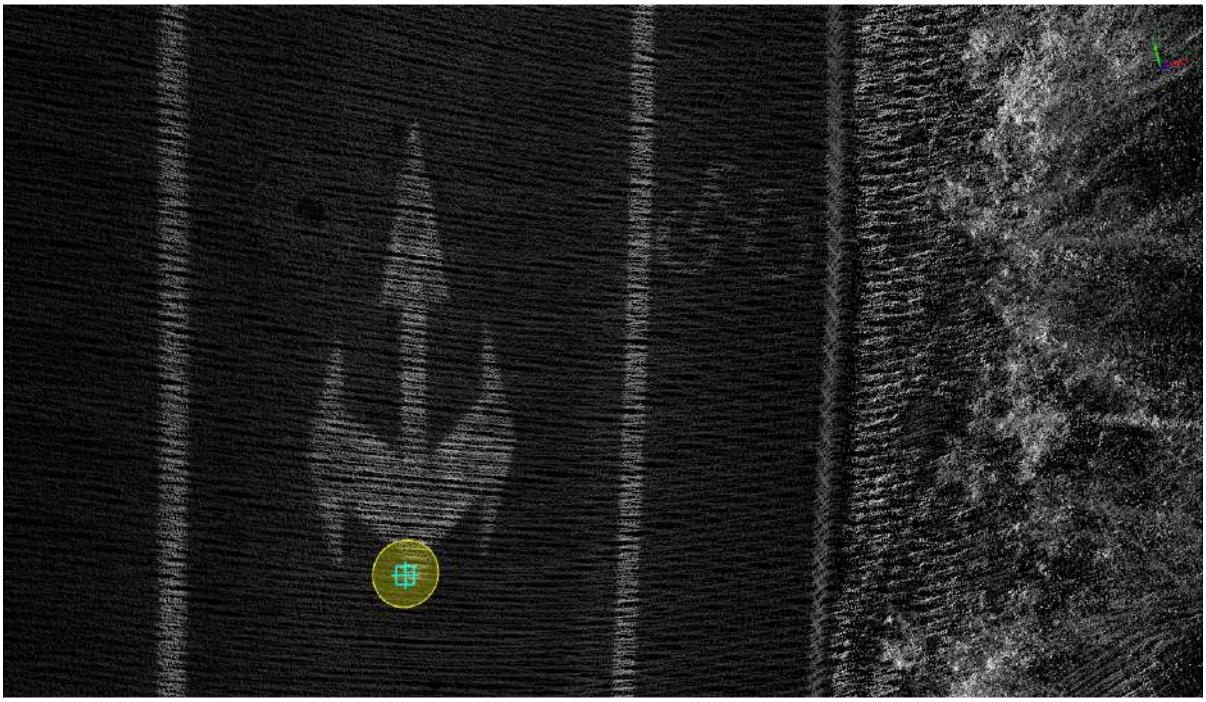
Function Description：The straight/left/right turn arrows indicate that the current lane can go straight, turn left, or turn right. This function is used to manually tap the point cloud for vectorization of straight/left/right turn arrows.

Steps

1.Click the **Straight,Left and Right**  button in the Road Surface panel

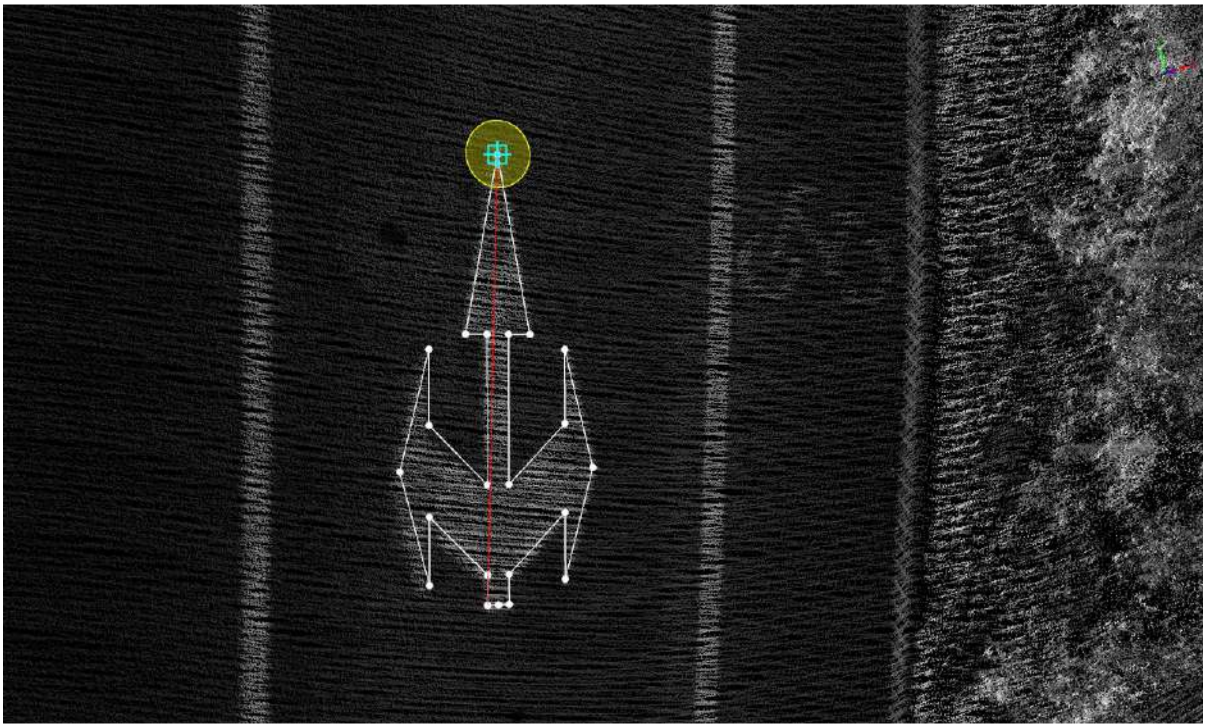
> - The key points in the arrow icons need to be aligned with the actual point cloud
> - The red point is the first key point and the green point is the second key point

2.**Left mouse click** to select the straight/left/right arrow point cloud key point, the details are as follows, the icon in the **first key point**

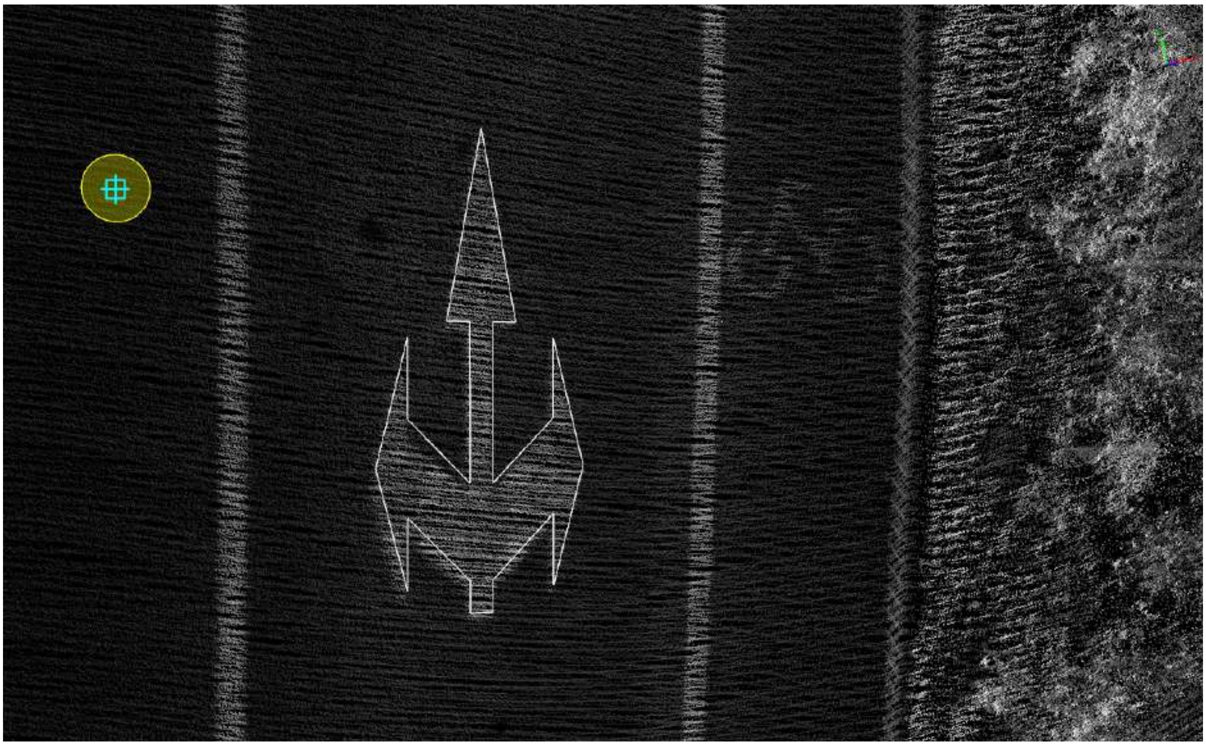


First key point

3.Move the mouse to see **template** and **actual point cloud** fit, **left mouse click** tap **second key point** to complete the straight/left/right arrow production, details are as follows, icon in **second key point**



Second key point



Straight and left arrow generation

U-Turn Arrow

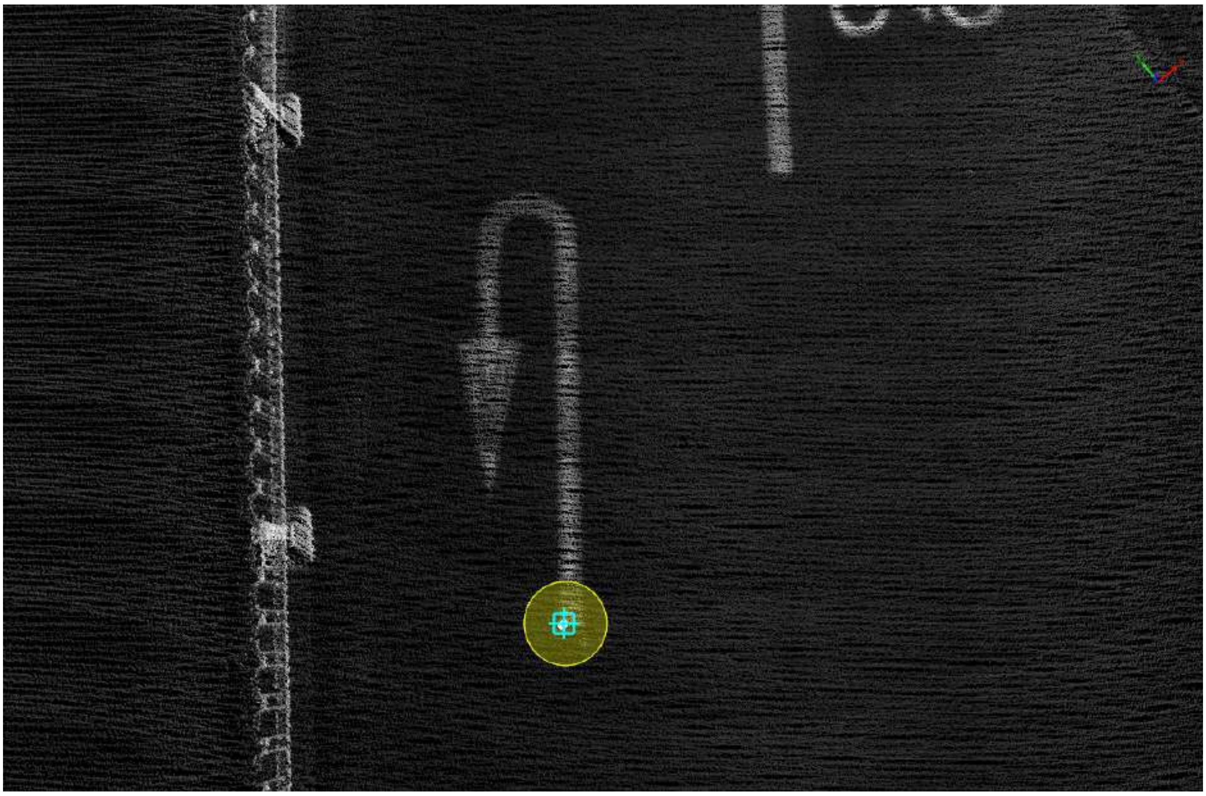
Function Description：U-turn arrow indicates that the current lane will be a U-Turn. This function is used to manually click on the point cloud to vectorize the U-turn arrow.

Steps

1.Click the **U-turn**  button in the Road Surface panel.

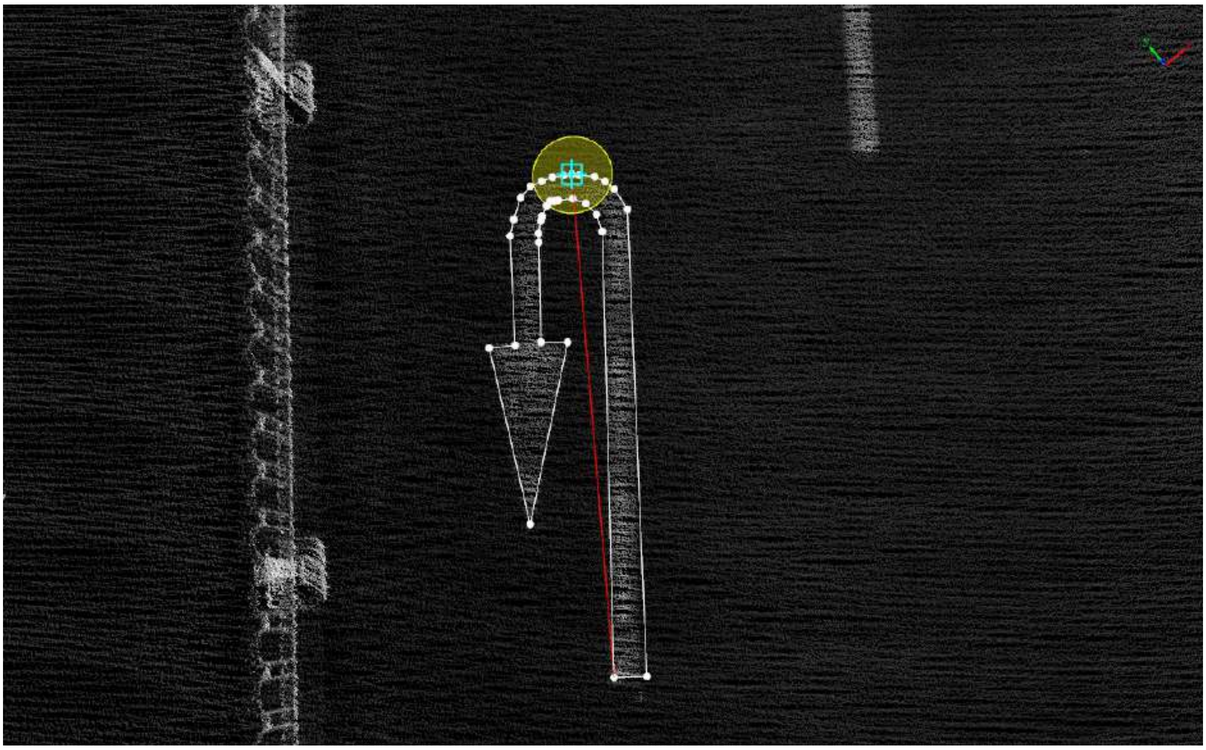
> - The key points in the arrow icons need to be aligned with the actual point cloud.
> - The red point is the first key point and the green point is the second key point.

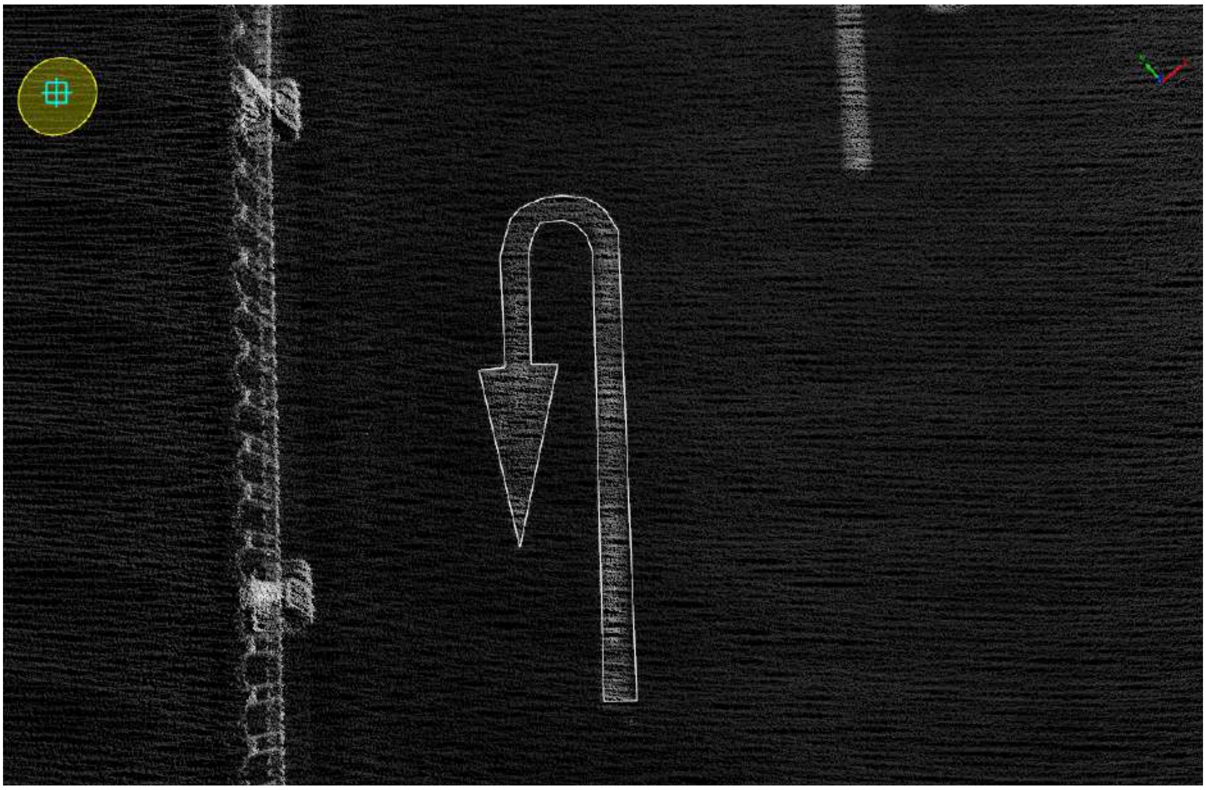
2.**Left mouse click** to select the turnaround arrow point cloud key point, the details are as follows, the icon in the **first key point**.



First key point

3.Move the mouse to see **template** and **actual point cloud** fit, **left mouse button click** click **second key point** to complete the production of the turnaround arrow, the details are as follows, the icon in **second key point**.


Second key point


U-Turn arrow generation

Straight and U-Turn Arrow

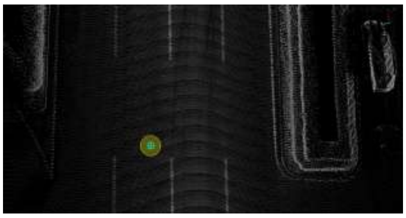
Function Description：Straight/U-turn arrow indicates that the current lane can be U-turn or go straight. This function is used to manually click on the point cloud to vectorize the straight/U-turn arrow.

Steps

1.Click the **Go straight/U-turn**  button in the Road Surface panel

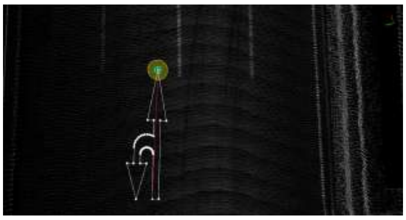
> - The key points in the arrow icons need to be aligned with the actual point cloud
> - The red point is the first key point and the green point is the second key point

2.**Left mouse button click** to select the straight/turning arrow point cloud key point, the details are as follows, the icon in the **first key point**

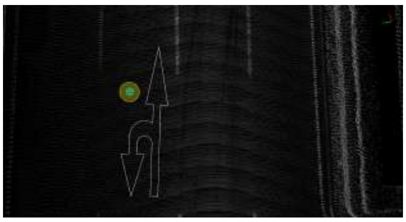


First key point

3.Move the mouse to view the **template** and **actual point cloud** fit, **left mouse click** tap **second key point** to complete the production of straight line/turnaround arrow, details are as follows, icon in **second key point**



Second key point



Straight and U-Turn arrow generation

Left turn/Turn Around

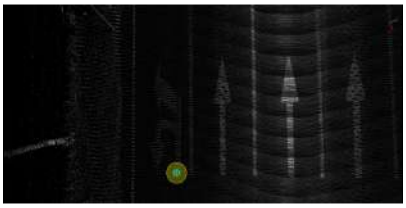
Function Description: Left-turn/turnaround arrows indicate that the current lane can turn left or turn around, this function is used to manually tap the point cloud for vectorization of left-turn/turnaround arrows.

Steps

1.Click the **Turn Left/U-Turn**  button in the Road Surface panel

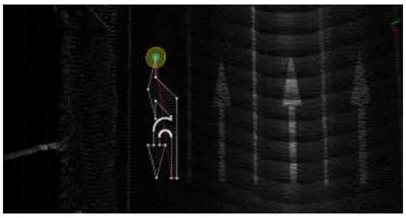
> - The key points in the arrow icons need to be aligned with the actual point cloud point cloud
> - The red point is the first key point and the green point is the second key point

2.**Left mouse click** to select the left turn/turnaround arrow point cloud key point, the details are as follows, the icon in **the first key point**

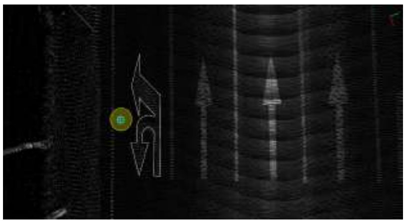


First key point

3.Move the mouse to view **template** and **actual point cloud** fit, **left mouse click** tap **second key point** to complete the left turn/turnaround arrow production, details are as follows, icon in **second key point**



Second key point



Left turn/turnaround arrow drawing

Left Merge Arrow

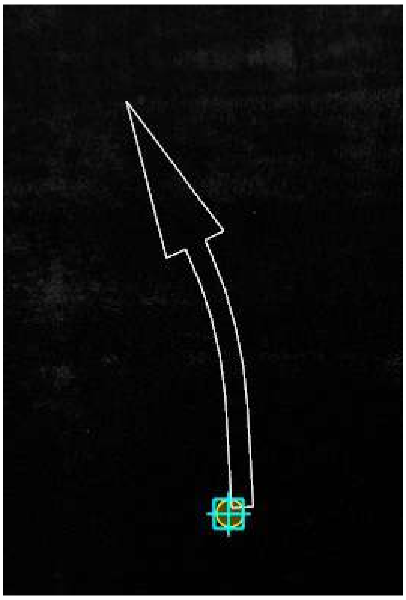
Function Description: The left merging arrow indicates that the current lane merges to the left. This function is used to manually select the point cloud to vectorize the left merging arrow.

Steps

1.Click the **Left Merge**  button in the Road Surface panel

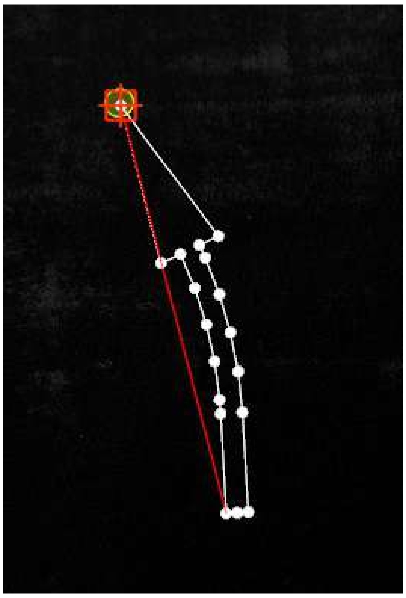
> - The key points in the arrow icons need to be aligned with the actual point cloud point cloud
> - The red point is the first key point and the green point is the second key point

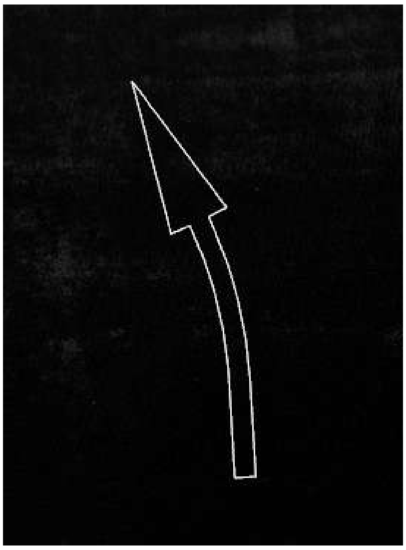
2.**Left click to select the key point** of the left converging arrow point cloud, the details are as follows, **the first key point** in the icon



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 converging arrow. The details are as follows, **the second key point** in the icon


Second key point


Left merge arrow generation

Merge Right

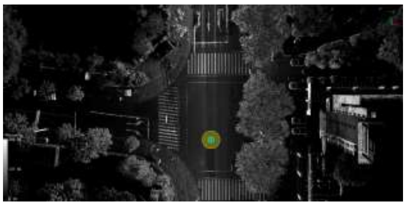
Function Description： The right merge arrow indicates that the current lane is merging to the right, this function is used to manually tap the point cloud to vectorise the right merge arrow.

Steps

1.Click the **Right Merge**  button in the Road Surface panel

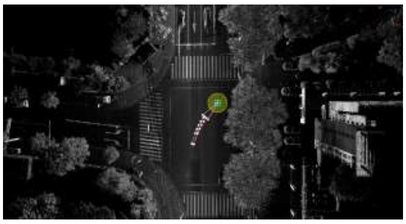
> - The key points in the arrow icons need to be aligned with the actual point cloud
> - The red dot is the first key point and the green dot is the second key point

2.**Left mouse click** to select the rightward merging 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

Drawing of rightward merging arrow

Right Turn/U-Turn

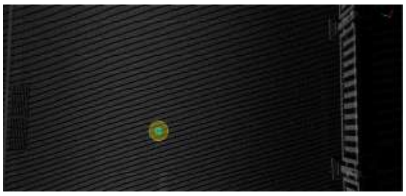
Function Description：The right turn/u-turn arrow indicates that the current lane allows a turn right or u-turn. This function is used to manually click on the point cloud to vectorize the right turn/u-turn arrows.

Steps

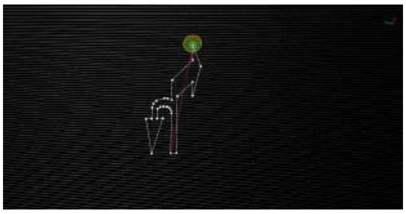
1.Click the **Right/U-Turn**  button in the Road Surface panel

> - 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



The first key point



The second key point

Right turn/turnaround arrow drawing

Straight/Right Turn/U-turn Arrow

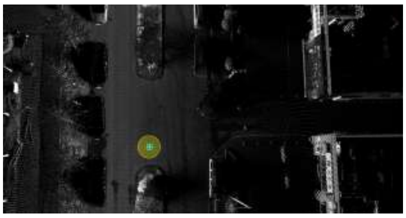
Function Description： Straight/right turn/U-turn arrows indicate that the current lane allows going straight, right-turn, and U-turn. This function is used to manually click on the point cloud to vectorize the straight/right-turn/turn arrows.

Steps

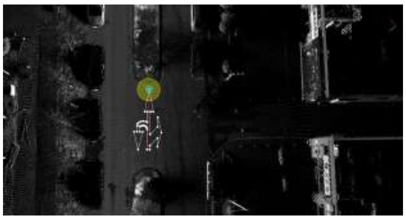
1.Click the **Straight/Right Turn/U-Turn**  button in the Road Surface panel.

> - 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**,**second key point**



The first key point



The second key point

Straight/right turn/turnaround arrow drawing

Left turn/RightTurn/Turnaround

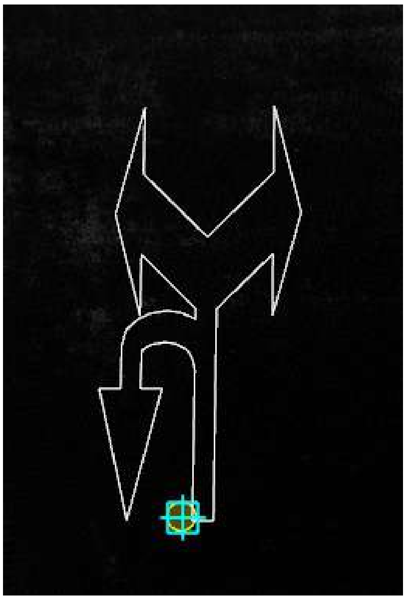
Function Description：Left turn/right turn/turnaround arrows indicate that the current lane can turn left, right, or turn around, this function is used to manually tap the point cloud for vectorization of left turn/right turn/turnaround arrows

Step

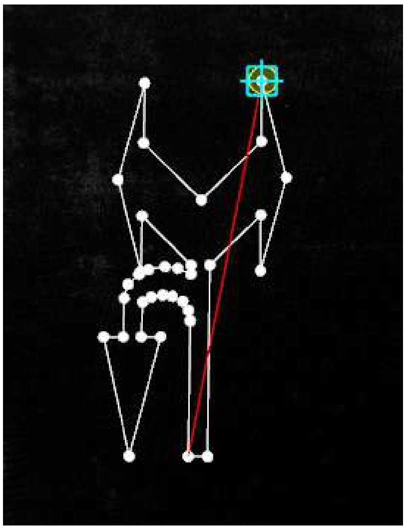
1.Click the **Left turn/RightTurn/Turnaround**  button in the Road Surface panel

> - 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

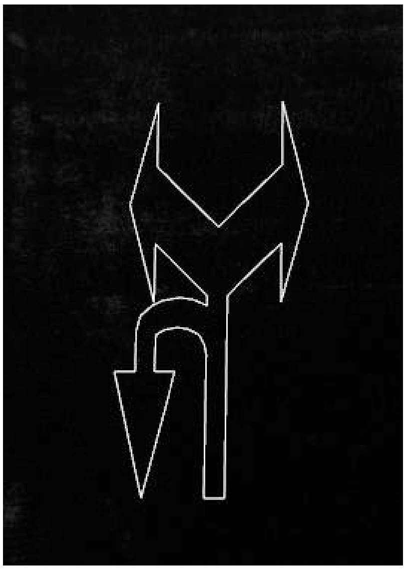
2.**Left mouse click** to select the point cloud key points, the details are as follows, **the first key point**,**the second key point** in the icon



first key point



second key point


Left turn / right turn / turnaround arrow drawing

Crosswalk

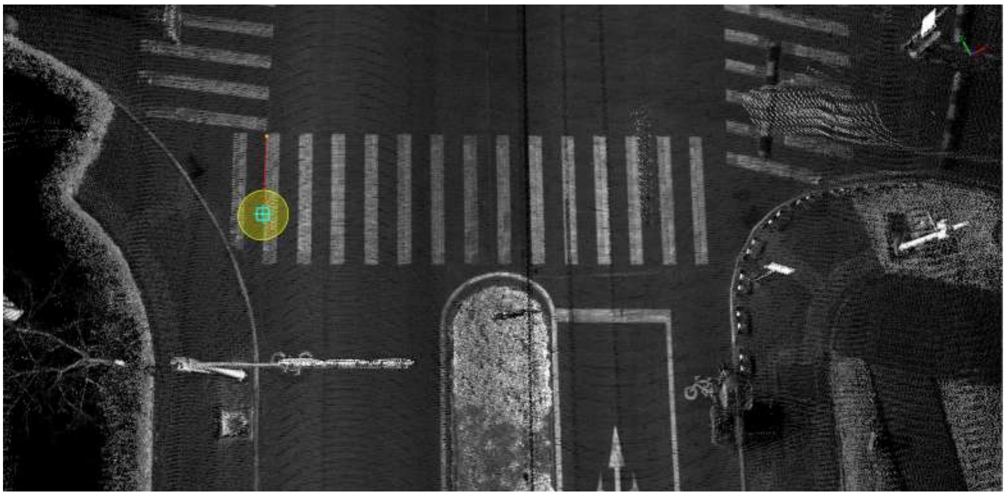
Function Description：Use three key points, manually click the point cloud, and make the vectorization of the crosswalk.

Steps9

1.In the drop-down logo in the Road Surface panel, click the **crosswalk** button

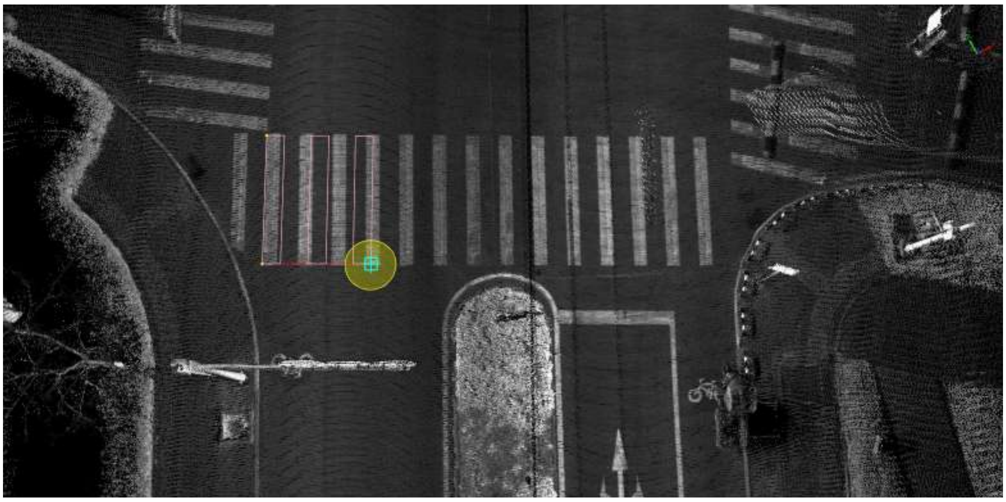
> - The **yellow point** in the figure below is the key point, and the key point needs to be aligned with the actual point cloud
> - **Yellow dot** position is the **position of the left click**

2.**Left click** to select the **first key point** of the crosswalk point cloud, the details are as follows, the **yellow dot** position is the **first key point** in the icon



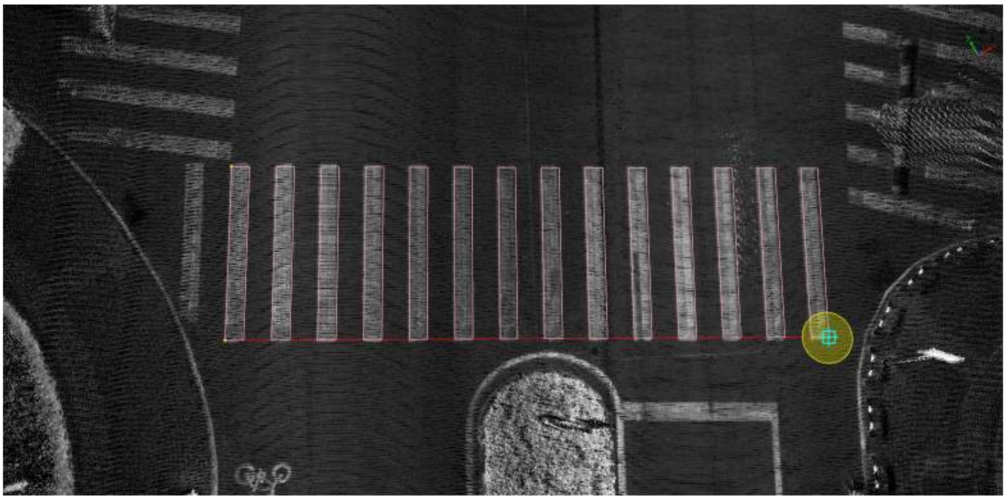
First key point

3.Move the mouse to check the fit between the **template** and the **actual point cloud**, left click **the second key point**, and determine **crosswalk width**, The details are as follows, the **yellow dot** is the **second key point in the icon**

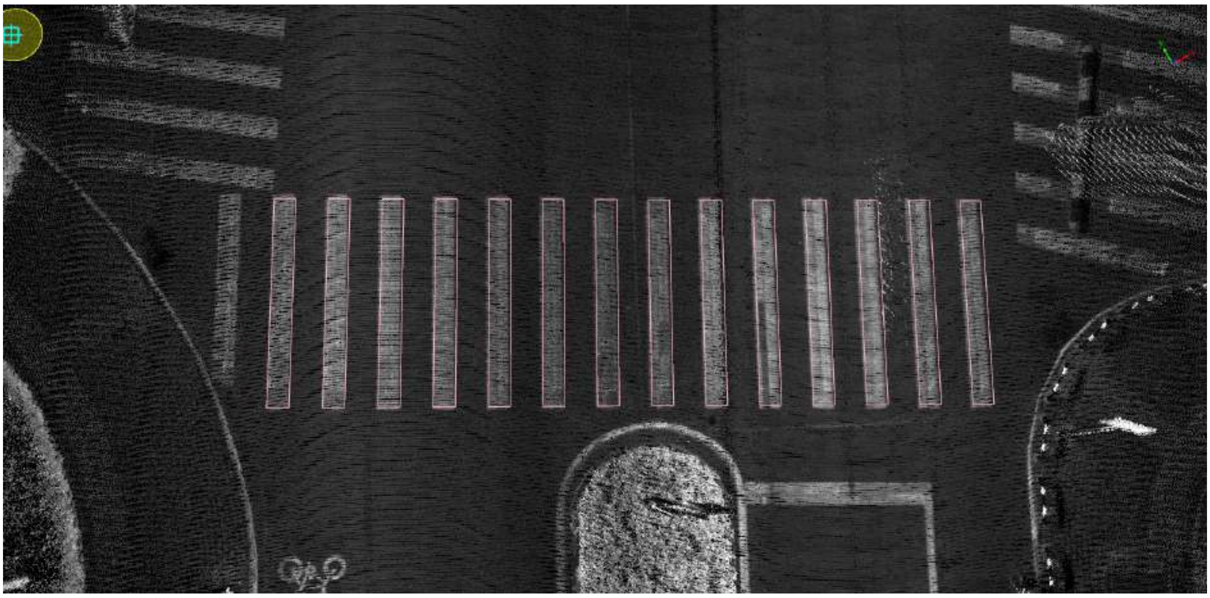


Second key point

4.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 dot** is the **third key point in the icon**



Third key point



Crosswalk generation

Longitudinal Deceleration Markings

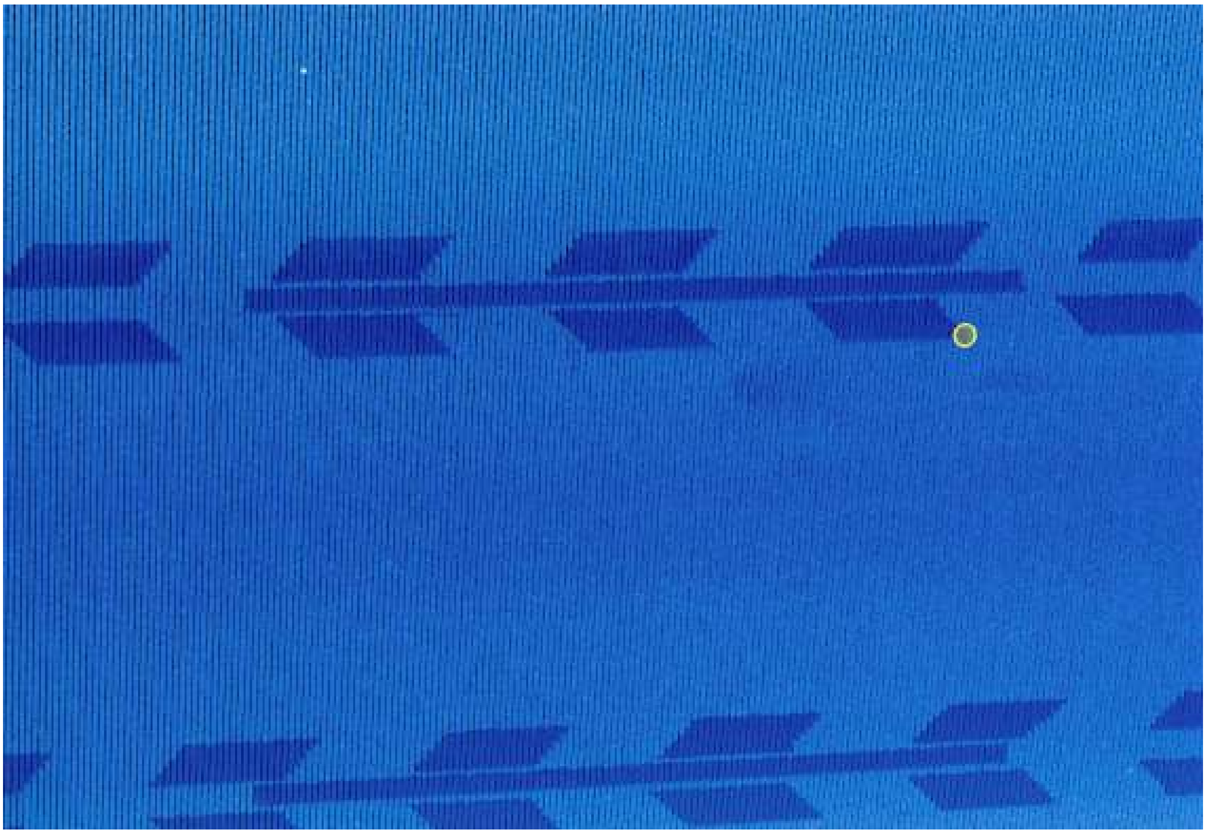
Function Description：For three key points, manual point selection point cloud for vectorization of longitudinal deceleration markers.

Steps

1.In the Road Surface panel, click the drop-down icon on the right, and select the **Longitudinal Deceleration Markings**  button

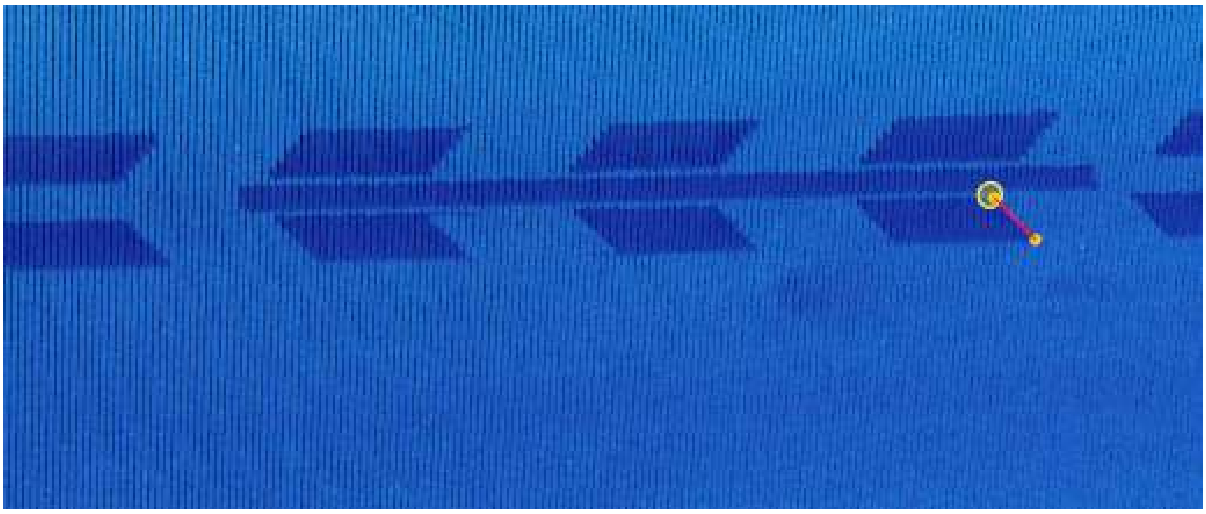
> - The **yellow point** in the figure below is the key point, which needs to be aligned with the actual point cloud
> - **Yellow dot** position is **left mouse click** position button click**

2.**Left mouse click** to select the **first key point** of the vertical deceleration marker point cloud, the details are as follows, **yellow dot** position is the **first key point** in the icon



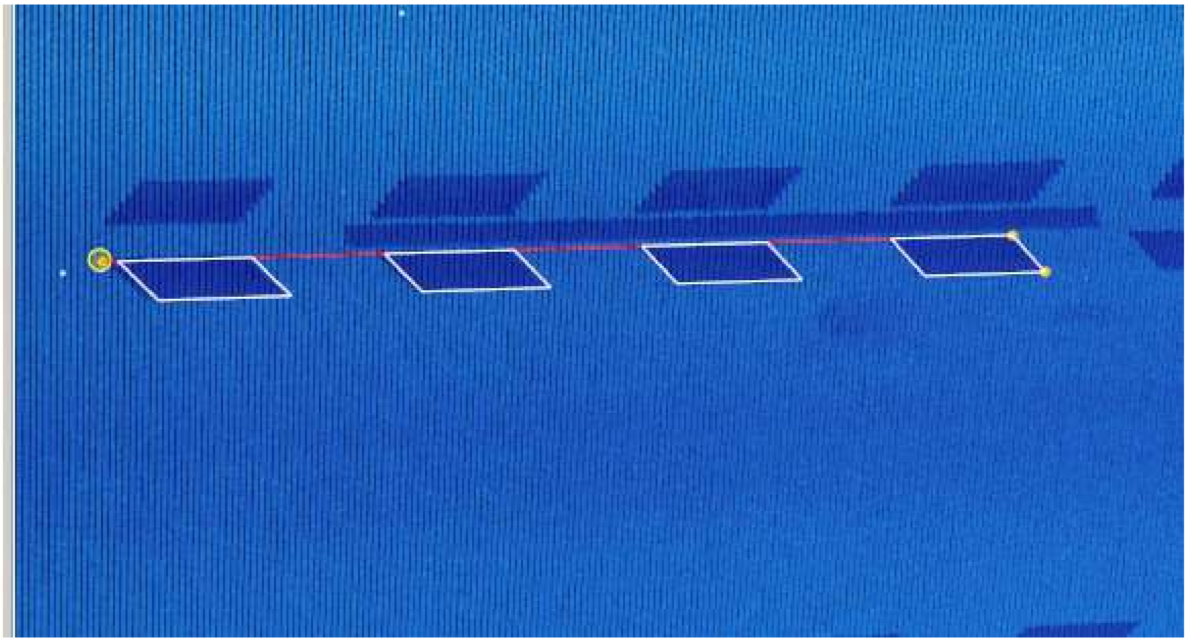
First key point

3.Move the mouse to view **template** and **actual point cloud** fit, **left mouse button click** click **second key point**, determine **width**, details are as follows, **yellow point** is the icon in **second key point**

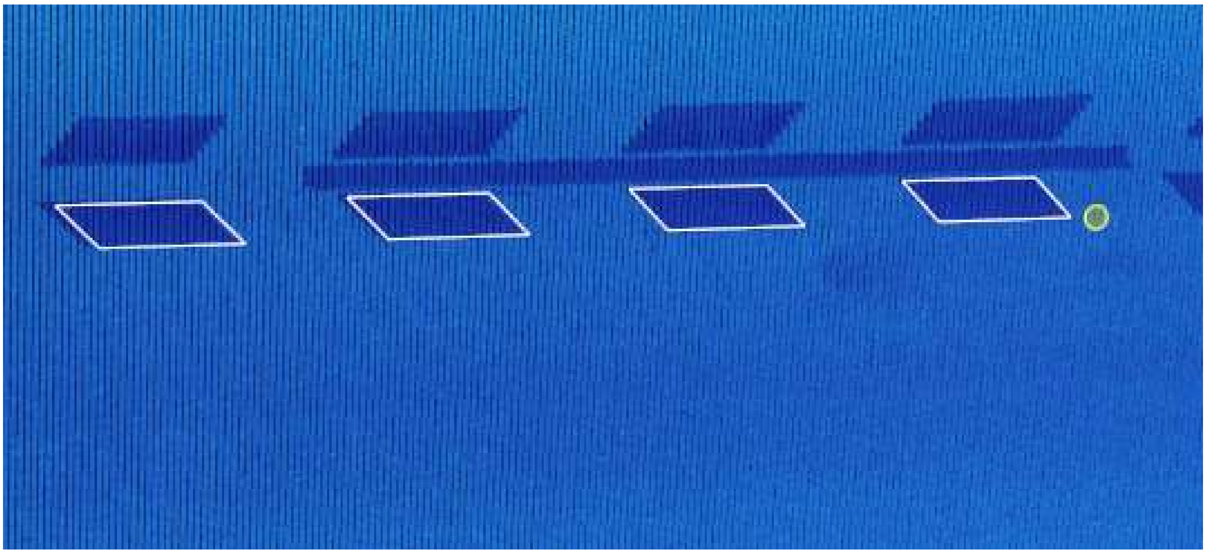


Second key point

4.Move the mouse to view **template** and **actual point cloud** fit, **left mouse button click** click **third key point**, determine **length**, you can complete the production, the details are as follows, **yellow point** that is, the icon in **third key point**



Third key point



Longitudinal deceleration marker drawing

Reduce Speed Mark

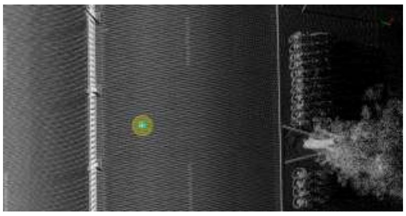
Function Description：Used to use two key points, manually click on the point cloud, and make the vectorized production of the reduce speed mark.

Steps

1.In the Road Surface panel, click the drop-down icon on the right, and select the **Reduce Speed**  button

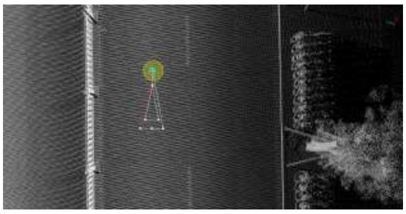
> - The key points in the arrow icons need to be aligned with the actual point cloud
> - The red point is the first key point and the green point is the second key point

2.**Left mouse click** to select the **first key point** of the deceleration let line marker point cloud, the details are as follows, the icon in the **first key point**

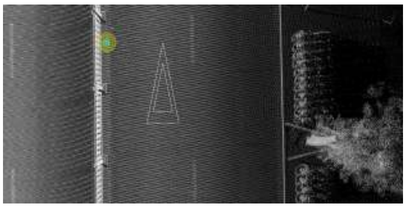


first key point

3.Move the mouse to view **template** and **actual point cloud** fit, **left mouse button click** click **second key point**, determine **reduce the speed to let the line logo size**, you can complete the production, the details are as follows, the icon in **second key point**



Second key point



Reduce speed mark generation

Crosswalk Warning

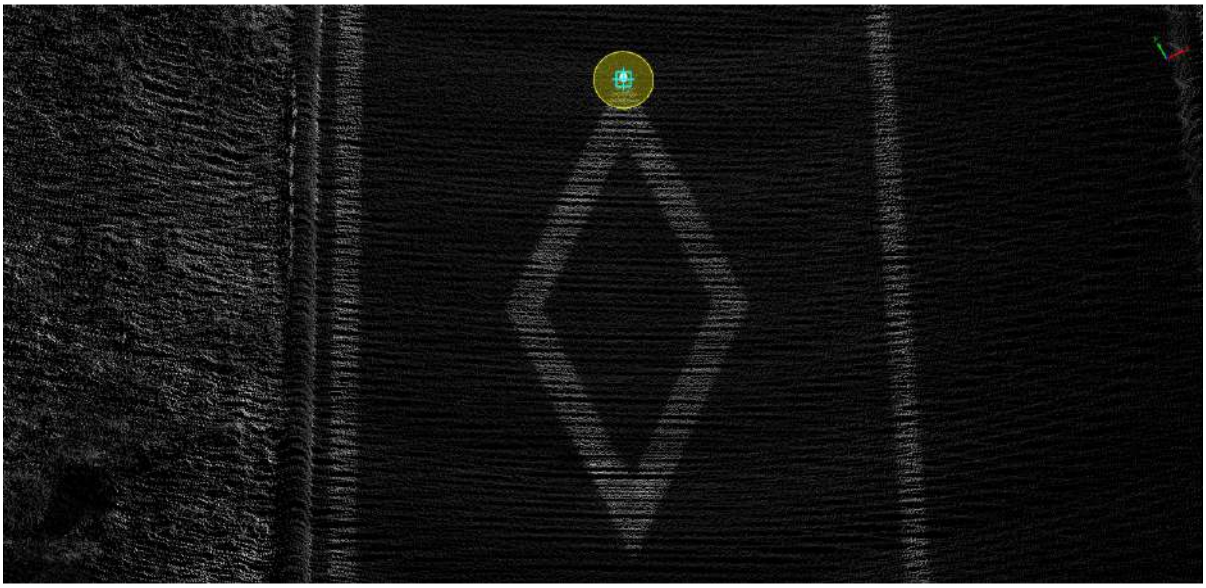
Function Description：The crosswalk warning sign is used to predict the crosswalk appearing ahead. This function is used for two key points. Manually click on the point cloud to vectorize the crosswalk warning.

Steps

1.Click the drop-down arrow on the right in the Road Surface panel, and select the **Crosswalk Warning**  button

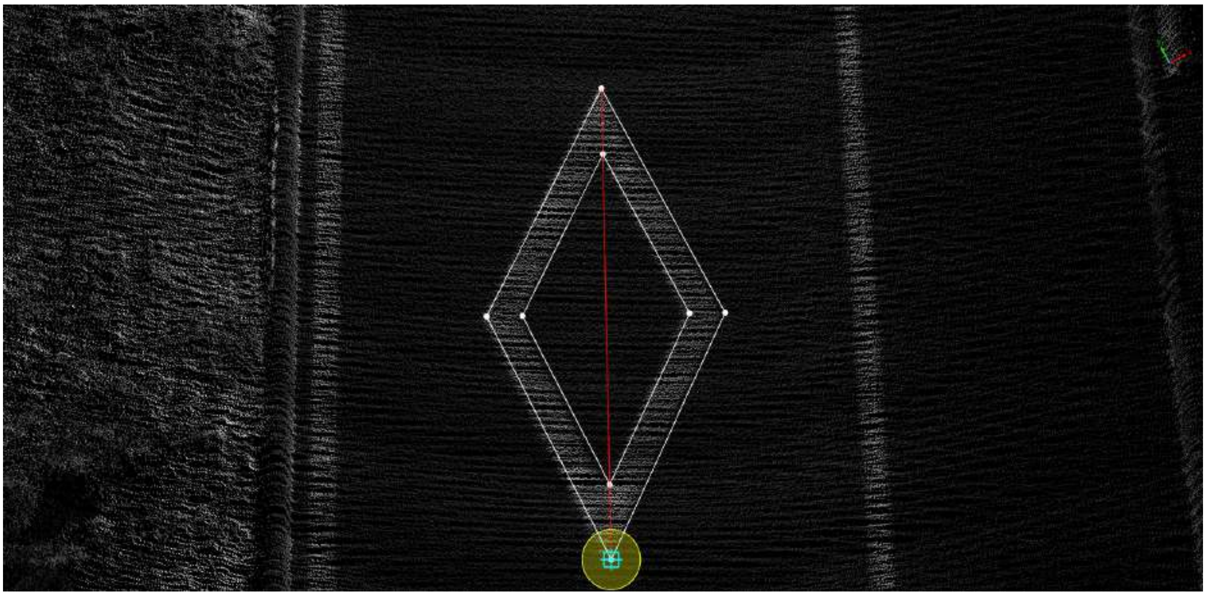
> - The key points in the arrow icons need to be aligned with the actual point cloud
> - The red dot is the first key point and the green dot is the second key point

2.**Left click** to select the **first key point** of the crosswalk notice point cloud, the details are as follows, the **yellow dot** position is the **first key point** in the icon

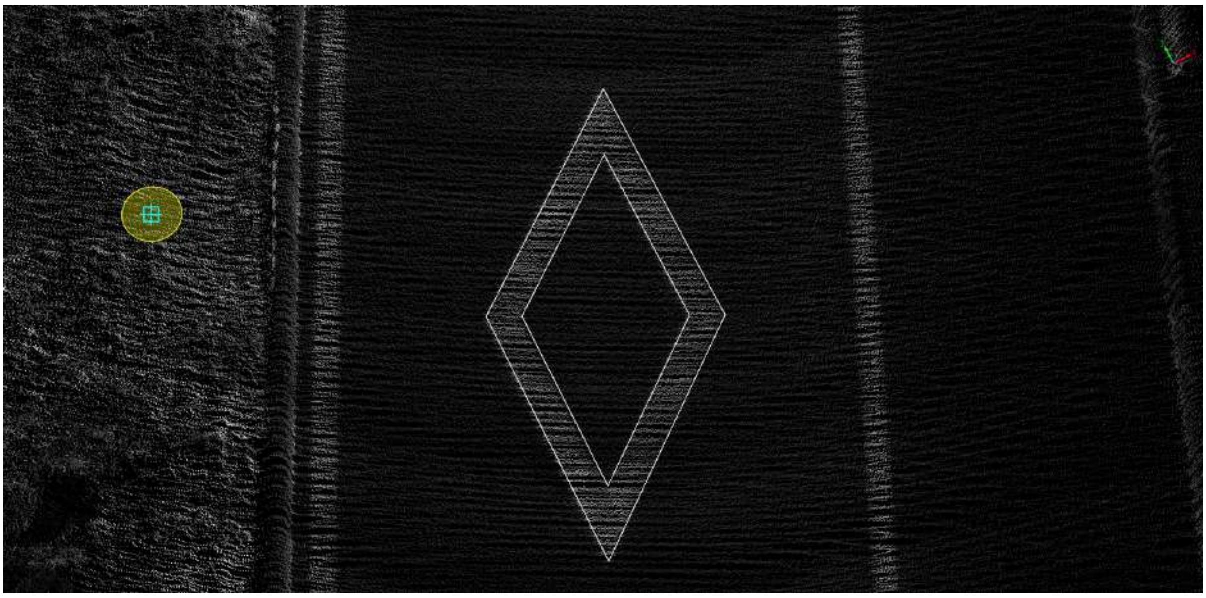


First key point

3.Move the mouse to check the fit between the **template** and the **actual point cloud**, Left click **the second key point** to determine the size of the **crosswalk warning** , You can complete the production, the details are as follows, the **yellow dot** is the second key point in the icon



Second key point



Crosswalk warning generation

No U-Turn Sign

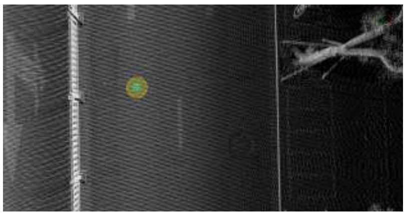
Function Description：The U-turn prohibition sign indicates that the current lane does not allow U-Turns. This function is used to manually click on the point cloud to make the vectorized production of the no U-turn sign.

Steps

1.Click the **No U-turn**  button in the Road Surface panel

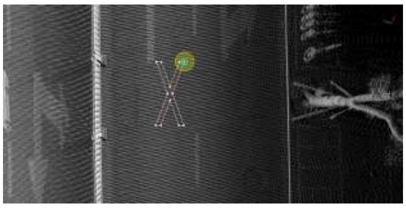
> - The **red dots** in the arrow icons are key points, and you need to align the key points with the actual point cloud
> - **Red dot** position is **left mouse button click** position

2.**Left mouse click** to select the key point of the No U-turn marker point cloud, details are as follows, **yellow dot** position is the **first key point** in the icon

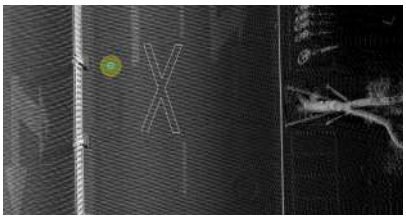


First key point

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 dot** is the **second key point** in the icon



Second key point



No U-Turn sign drawing

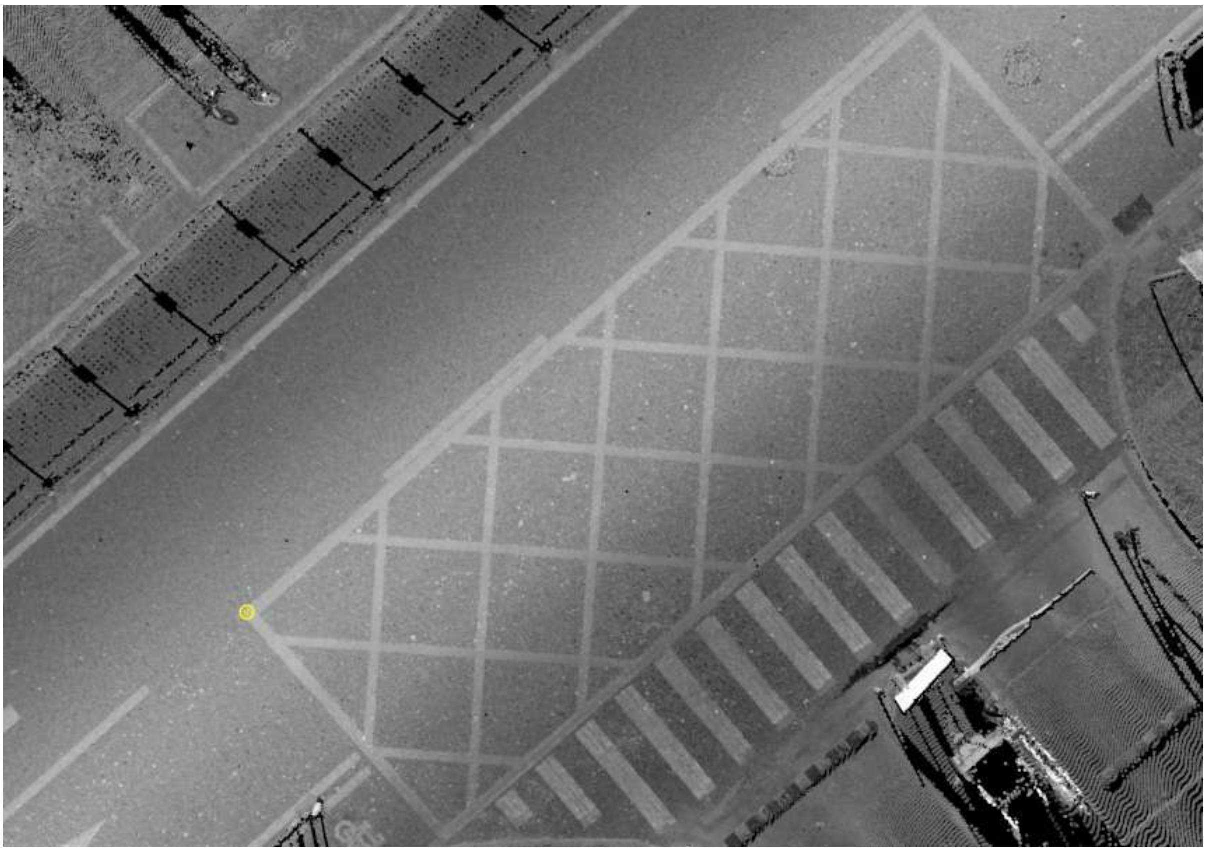
No-ParkingArea

Function Description：Used for 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 forbidden line.

Steps

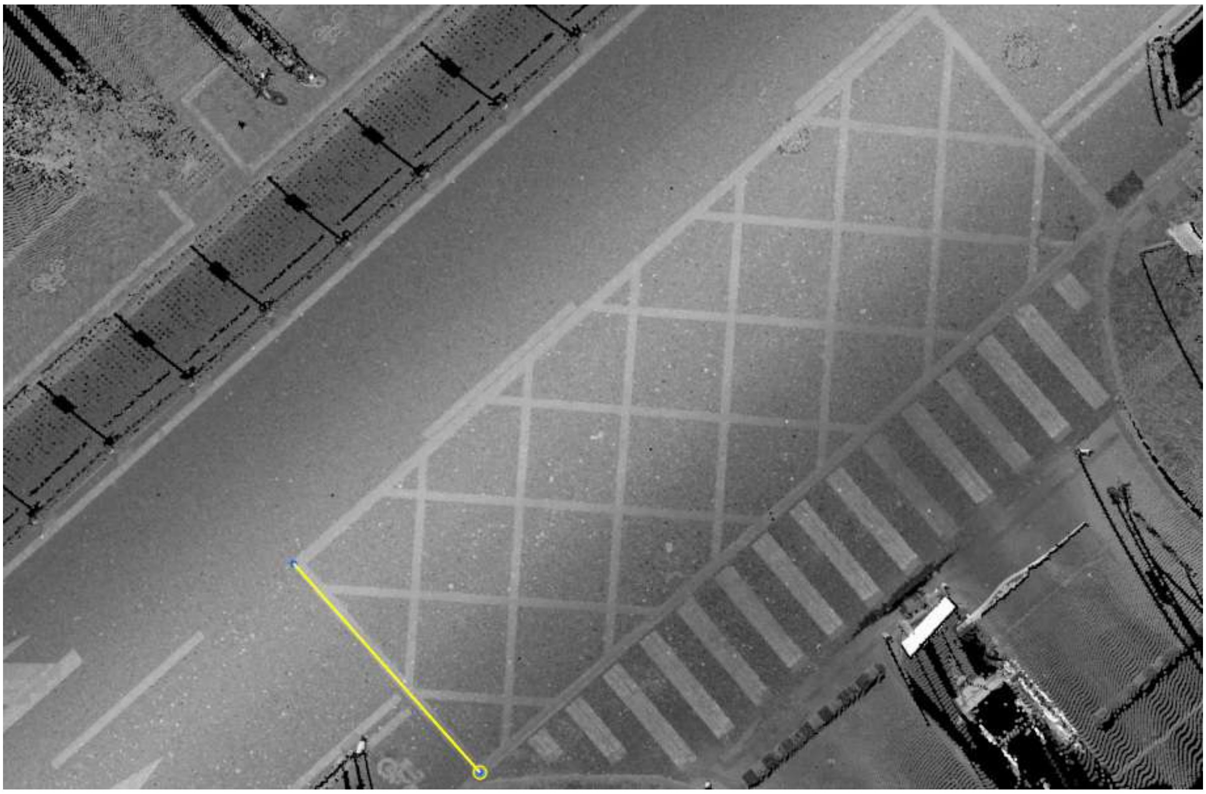
1.In the Road Surface panel, click the drop-down icon on the right, and select the **No-ParkingArea**  button

2.**Left mouse click** to select the **first key point** of the no-stop line point cloud, the details are as follows, **yellow dot** position is the **first key point** of the icon

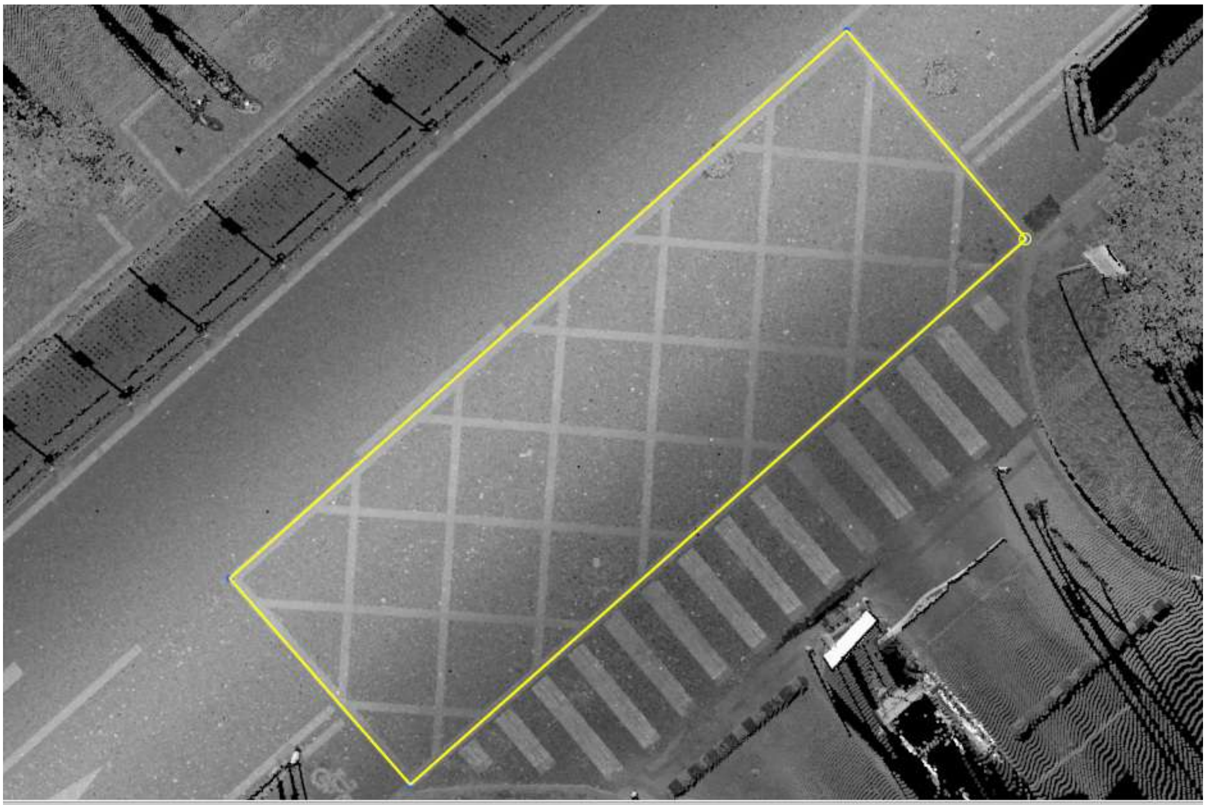


First key point

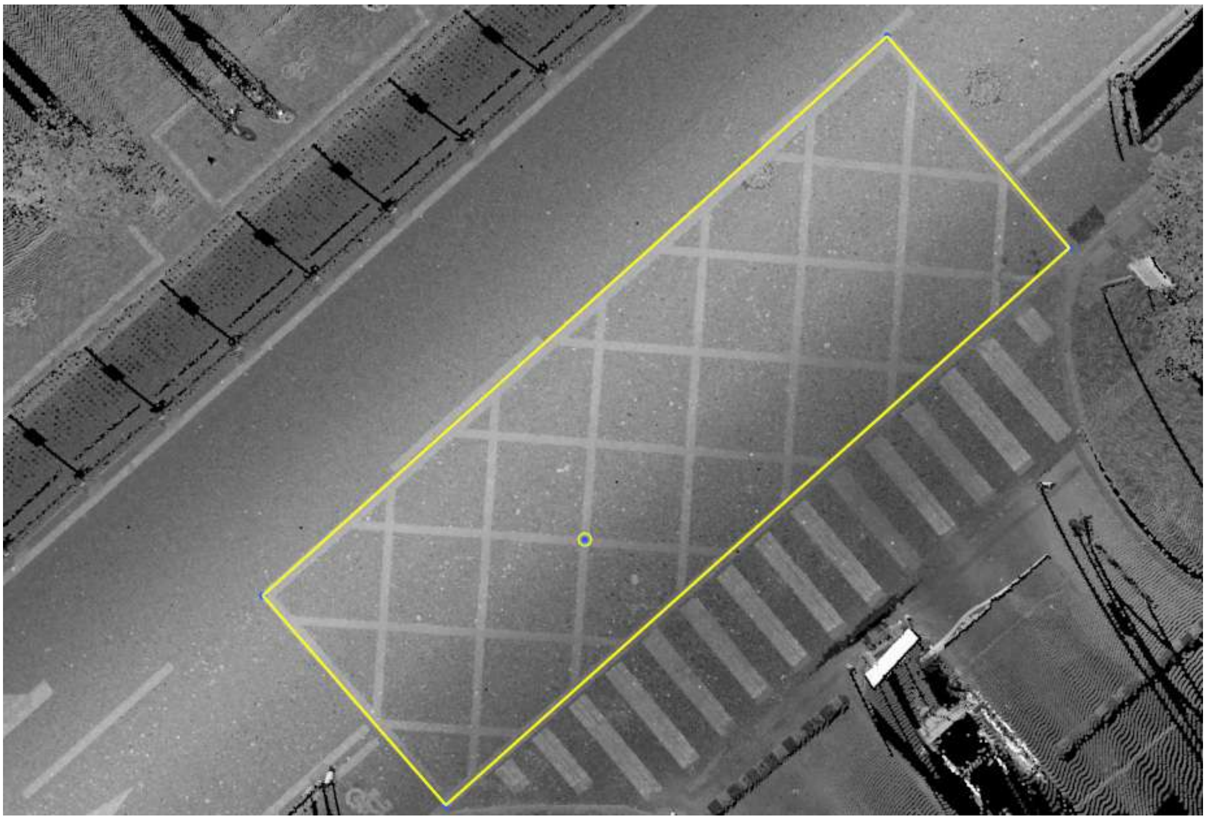
3.Move the mouse to view **template** and **actual point cloud** fit, **left mouse button click** click **second key point**, determine **external rectangle width**, details are as follows, **yellow point** that is, the icon in **second key point**


Second key point

4.Move the mouse to view the **template** and **actual point cloud** fit, **left mouse button click** 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**

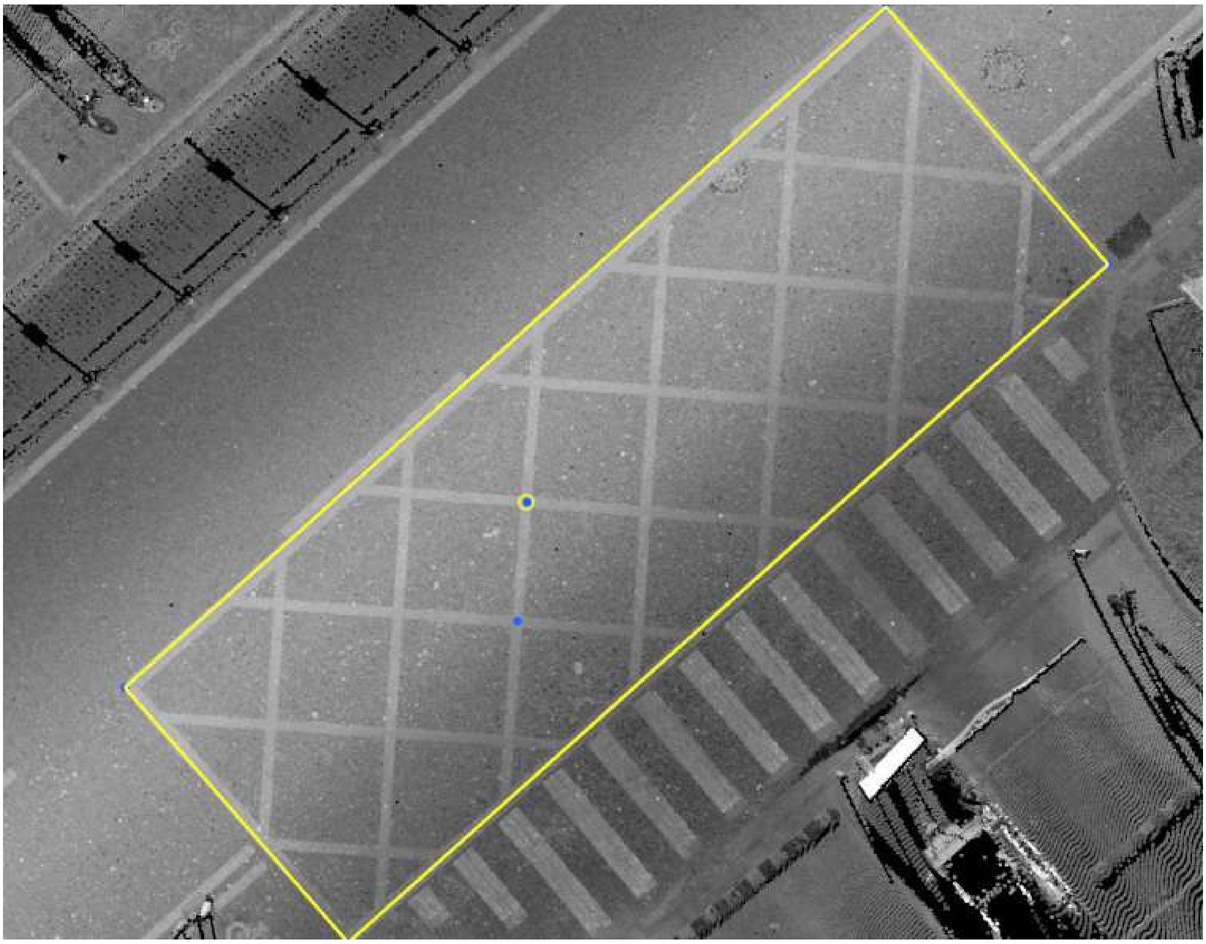

Third key point

5.Move the mouse to view the **template** and **actual point cloud** fit, **left mouse button click** click **the fourth key point**, the details are as follows, **yellow dot** is the **fourth key point** in the icon



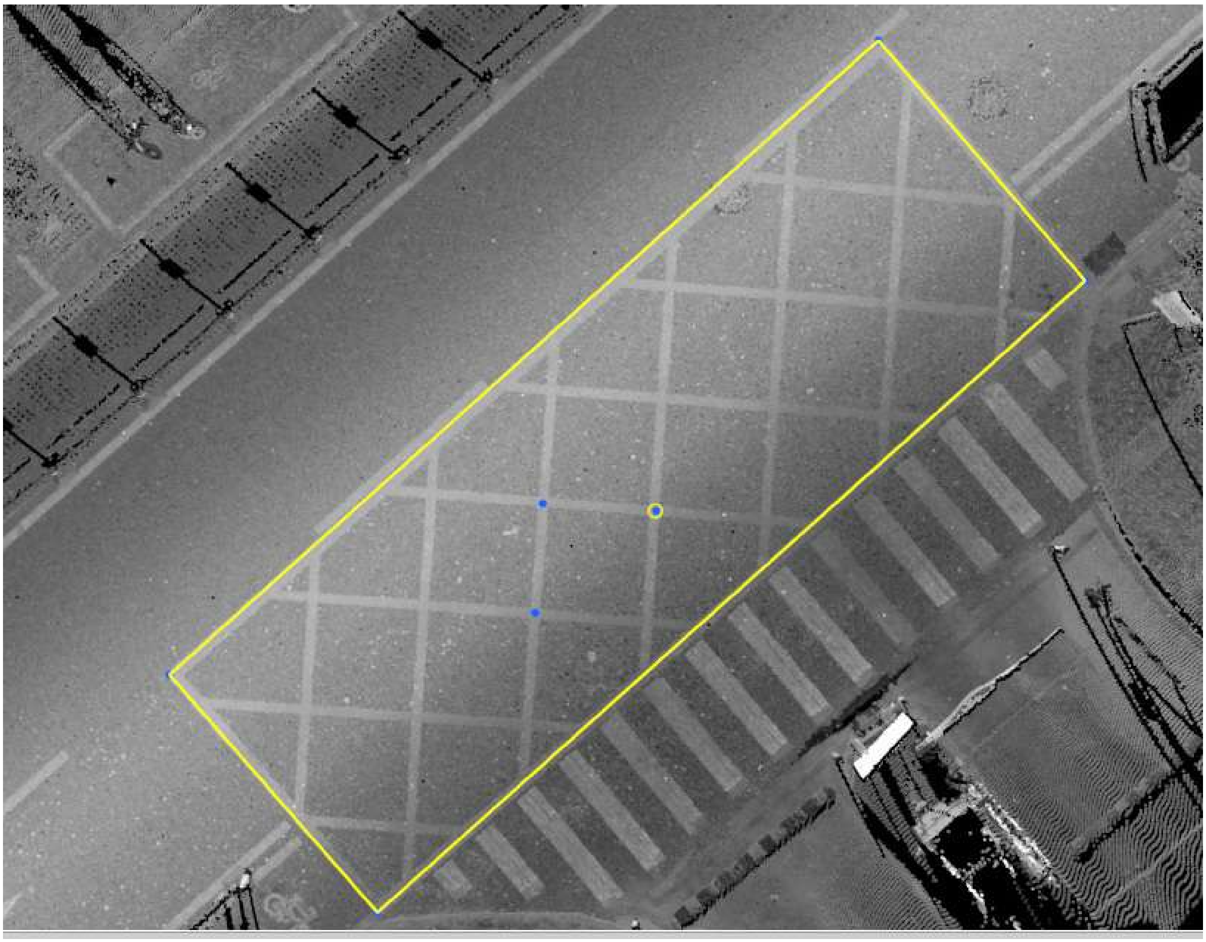
Fourth key point

6.Move the mouse to view the **template** and **actual point cloud** fit, **left mouse button click** click **the fifth key point**, the details are as follows, **yellow dot** that is the **fifth key point** in the icon

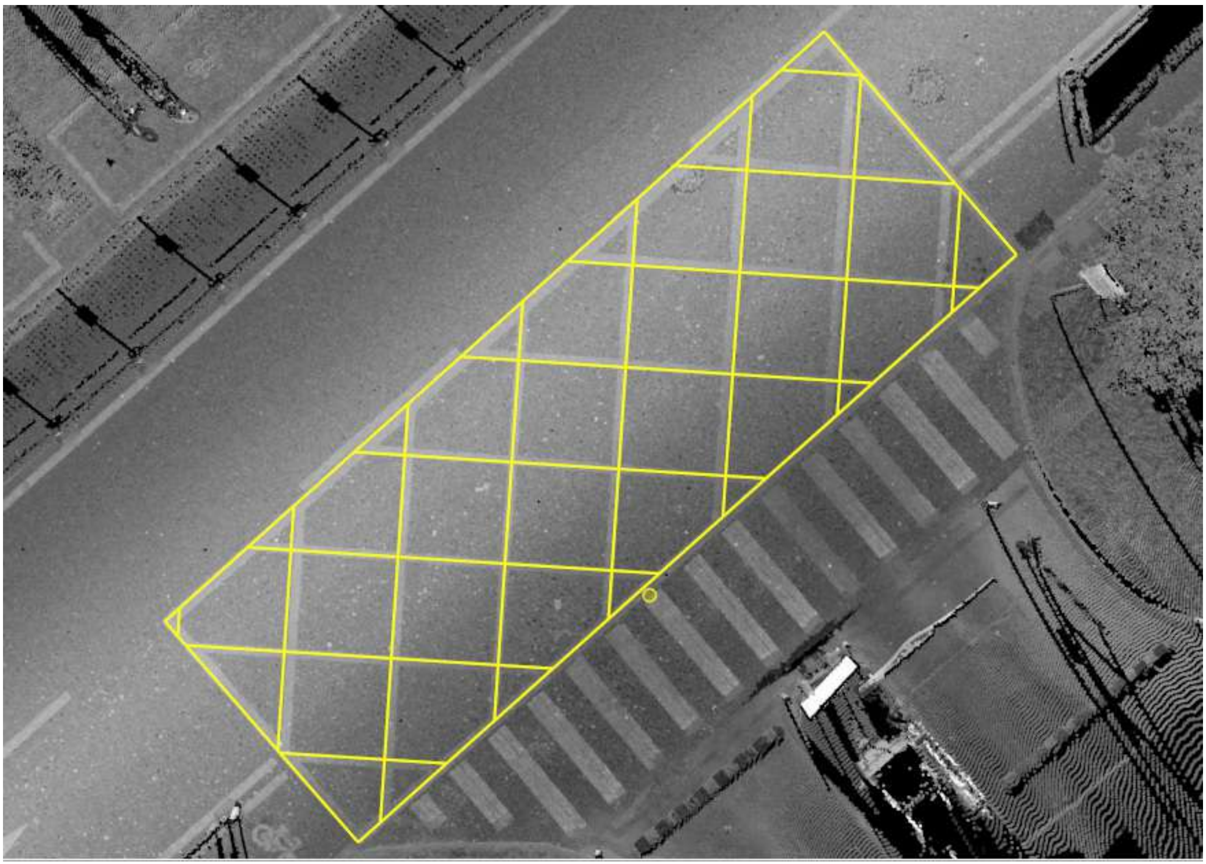


Fifth key point

7.Move the mouse to see **template** and **actual point cloud** fit, **left mouse button click** click **the sixth key point** to complete the production, the details are as follows, **yellow dot** that is, the icon in **the sixth key point**



Sixth key point



No-ParkingArea Drawing

Bike Lane

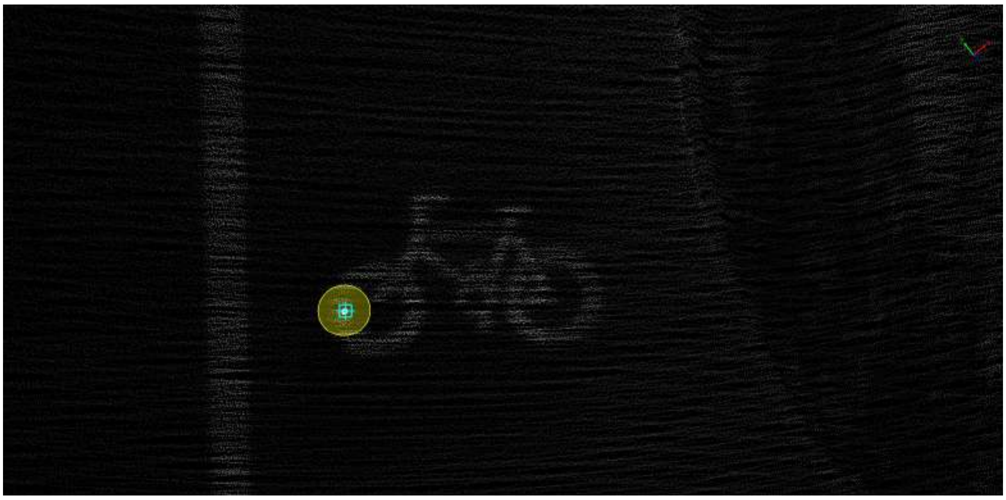
Function Description：The Bicycle/non-motor vehicle sign indicates that the current lane is a non-motor vehicle lane. This function is used to manually click on the point cloud to make vectorized non-motor vehicle signs.

Steps

1.Click the **Bike Lane**  button in the Road Surface panel.

> - The **red dot** in the arrow icon is the key point, which needs to be aligned with the actual point cloud
> - The **red dot** position is the **left click** position

2.**Left 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 dot** is the **second key point** in the icon

Second key point

Bike lane sign drawing

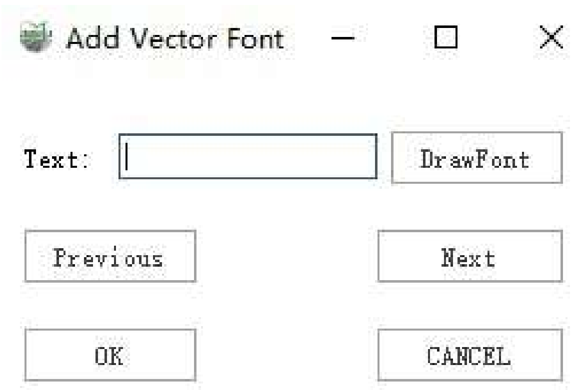
Add Text

Function Description：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

Steps

1.Click the **Add Text**  button in the Road Surface panel to pop up the Add Vector Text dialog box



Add text dialog box

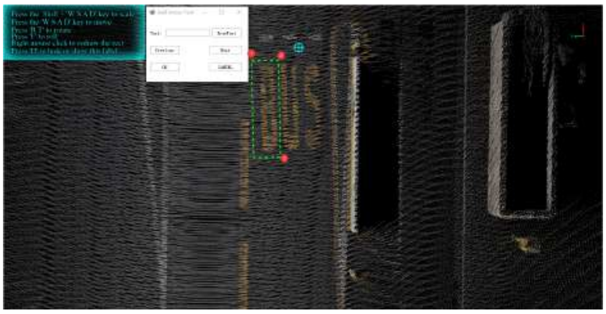
Parameter Descriptions

- **Text**：The text to be entered, you can enter a single letter or a single number.
- **Draw Text**：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, scaling, etc.

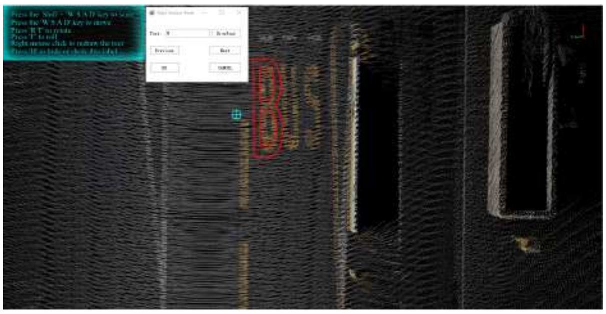
  > Starting from the first text, until clicking OK is a group of operations, the previous and the next refer to the current operation

- **Next**： Save this vector result and start the next operation.
- **Confirm**：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.

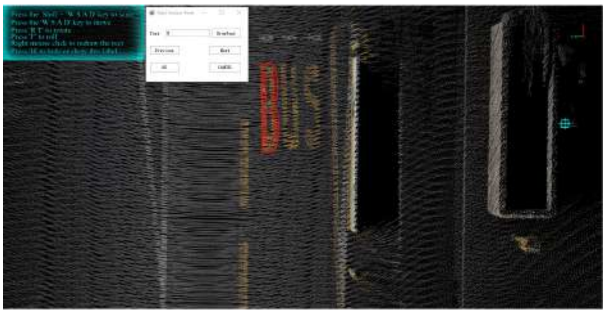
2.**Left click** three points, determine the position of the current text, the direction of the pull box is the initial direction of the text drawing.


Text position

3.Enter text in the dialog box, such as in the figure: B, and click the draw text button to actually display it on the point cloud.


Show actual vector

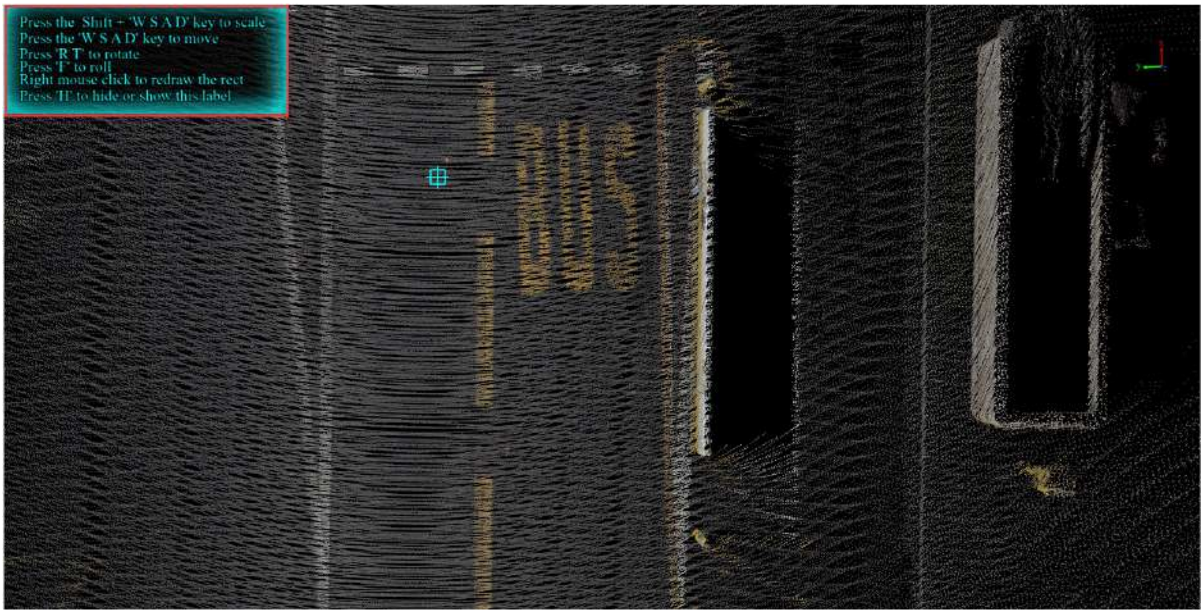
4.Use shortcut keys to perform operations such as rotation, translation, scaling, etc., making the vector fit in to the point cloud as smooth as possible.


Editing

Shortcut name	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

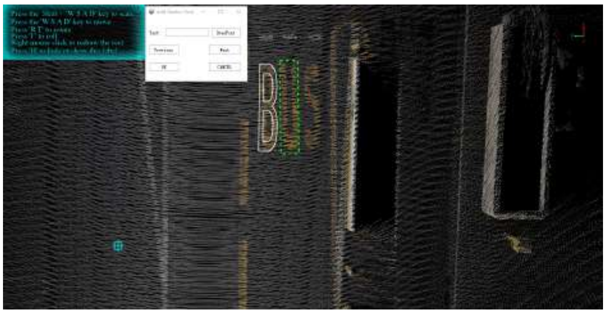
F	Flip the current text left and right

> **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 current object has the last text, click **OK** to automatically save the current object

Complete drawing process

Template matching

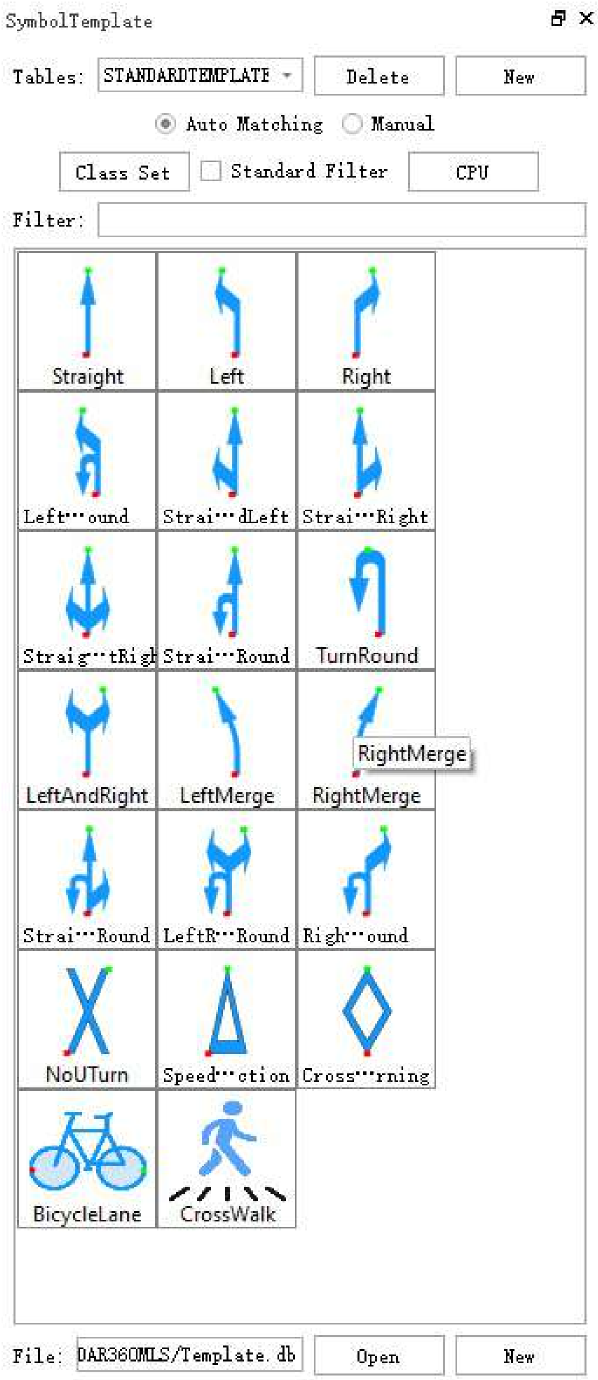
Function description:
1. Automatic matching: built-in arrow template of Chinese national standard size, as well as ground signs such as crosswalk warning, deceleration sign, and U-turn prohibition of Chinese national standard size, three keys can be used Click the box to select the point cloud to perform automatic template matching, and directly obtain 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 Match** sidebar pops up on the right side of the software, as shown in the image below, for button meanings see Detailed explanation of template matching sidebar parameters



Sidebar preview

3.**Custom Tables** (optional): You can add or delete new templates by yourself, see Template Matching Custom Tables

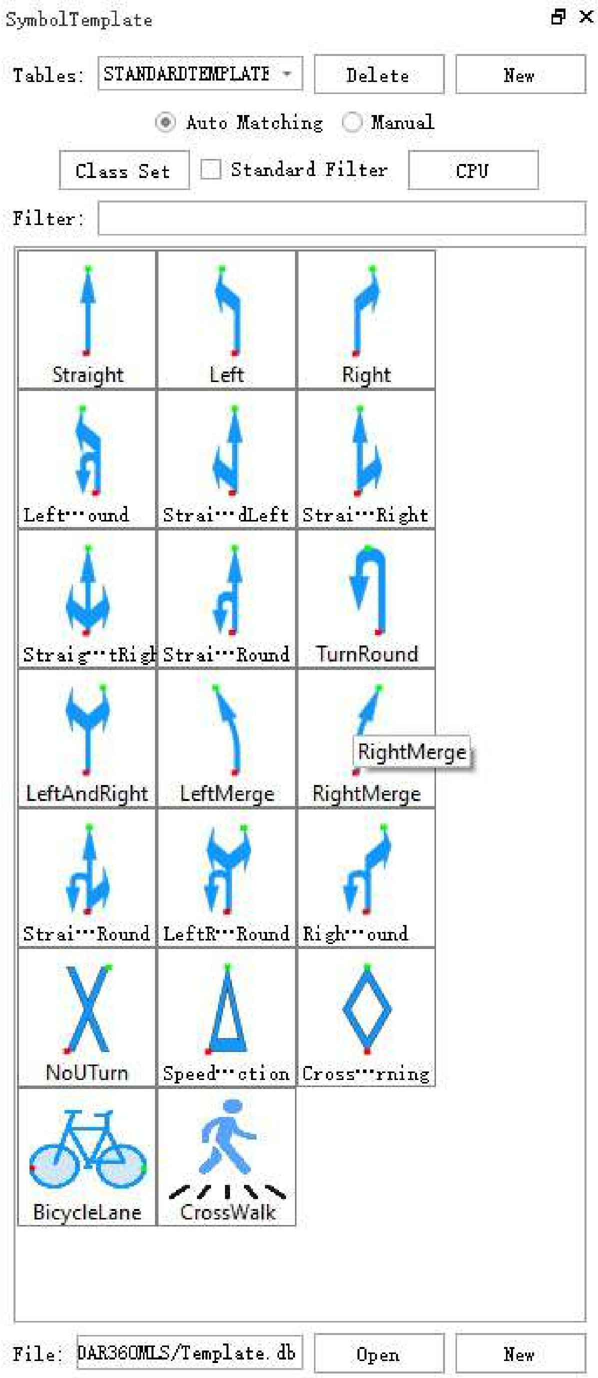
4.**Auto-matching**: Use the currently activated form and use the three-key point method to perform automatic template matching, see Template Automatic Matching

5.**Manual making**: Use the currently activated form to make manual vector, see Template Manual Making

Template Matching Para

description: Describes the template matching sidebar parameters in detail.

Sidebar



Sidebar

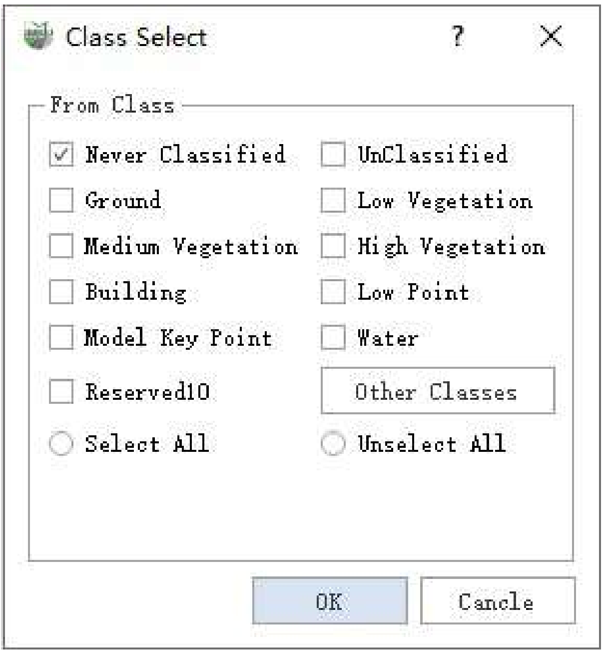
Parameter Description

- **Table**: The template matching module uses a database to store various template tables, as well as user-defined tables.

- Custom tables, you can add and delete new templates by yourself, 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 form 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
- **Category setting**: 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.



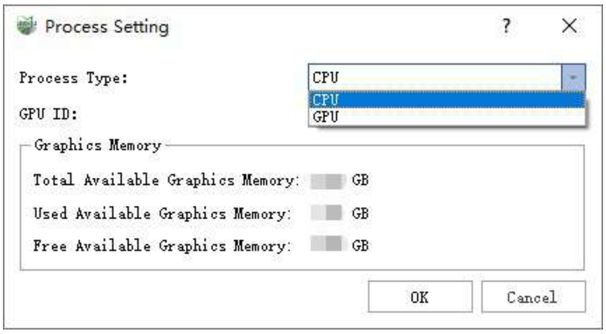
Category Settings

- **Standard filter**: Unchecked by default, if checked, the recognition results will be filtered according to the 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 screening is enabled, results with a length of less than 3m will be filtered out

- **Mode: CPU, Mode: GPU**: Set the operation mode, you 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 category setting, standard filtering, 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

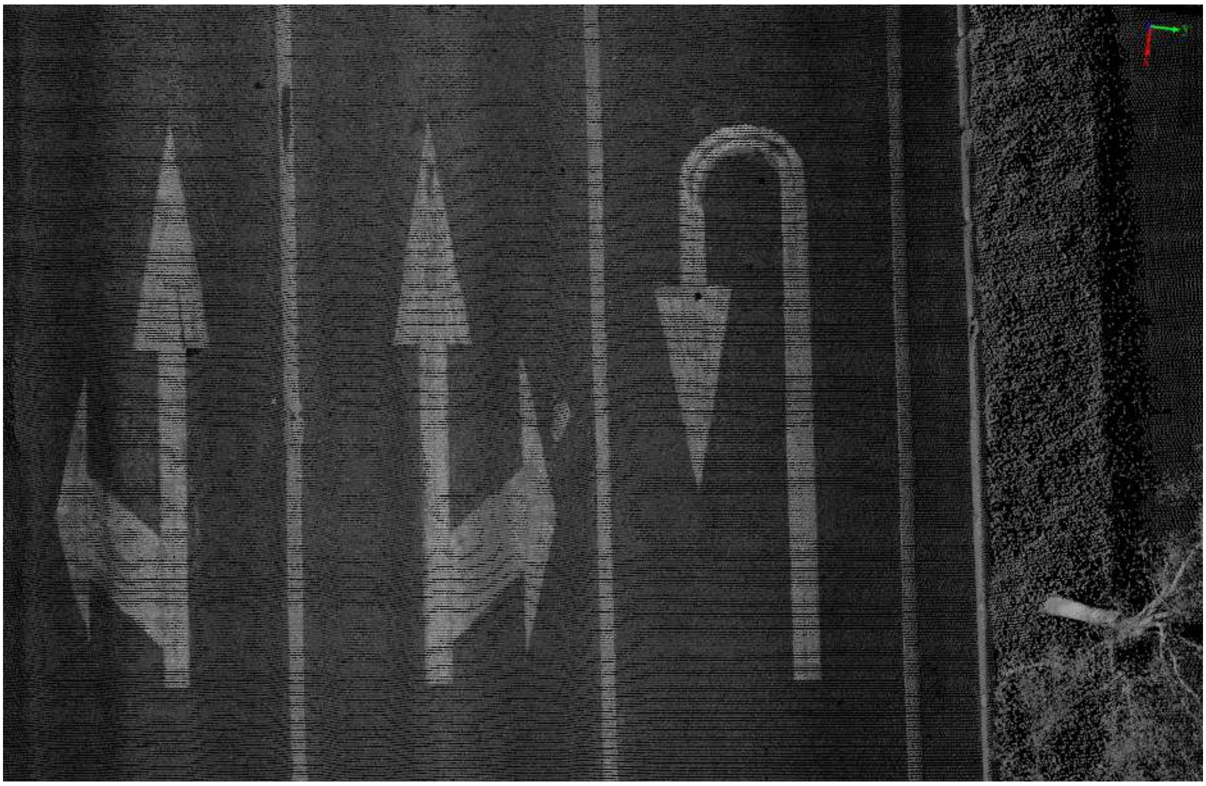
Dialog

- **Click**: The left is the minimum intensity, and the right is the maximum intensity. After checking the box, you can click the point cloud with the left mouse button to determine the maximum and minimum intensity respectively

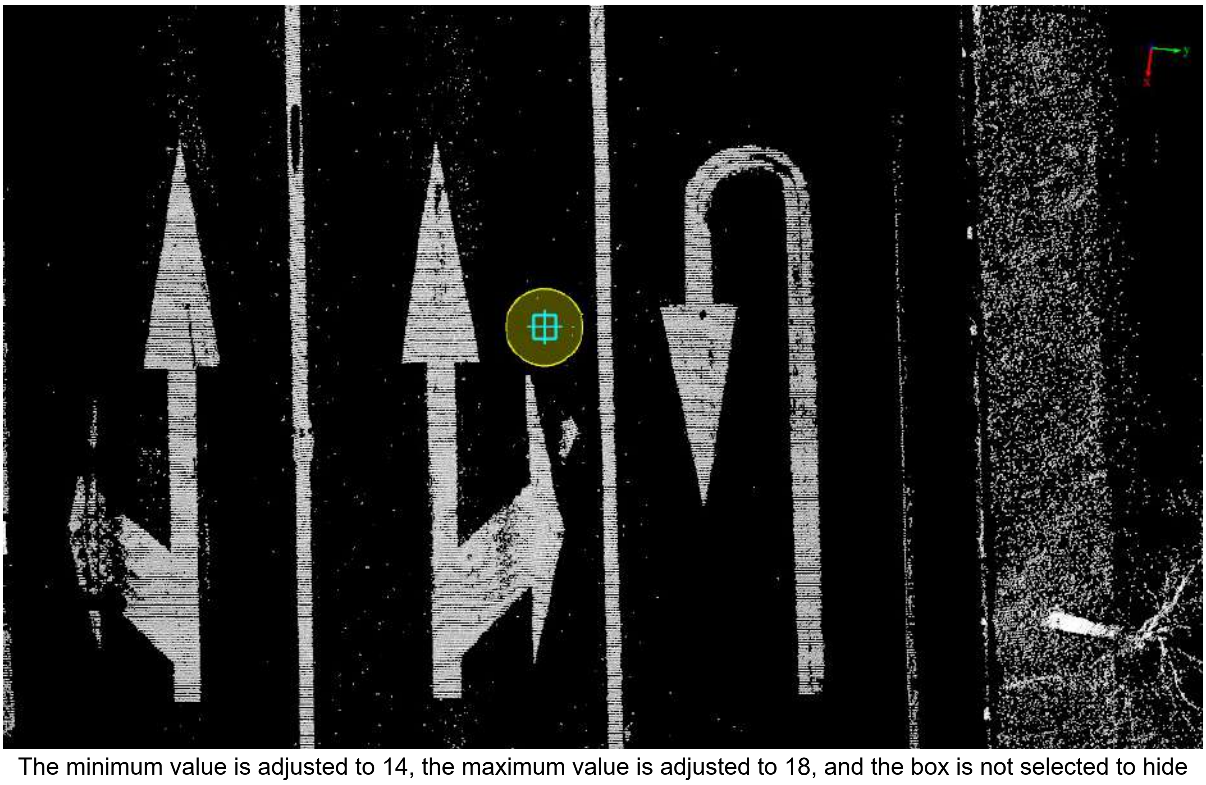
  > Scroll bar, value box, click box, the three are linked together, you can choose one of the operation methods to adjust the strength, and the point cloud display will also be linked in real time (stretch the strength of the overall point cloud)

- **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

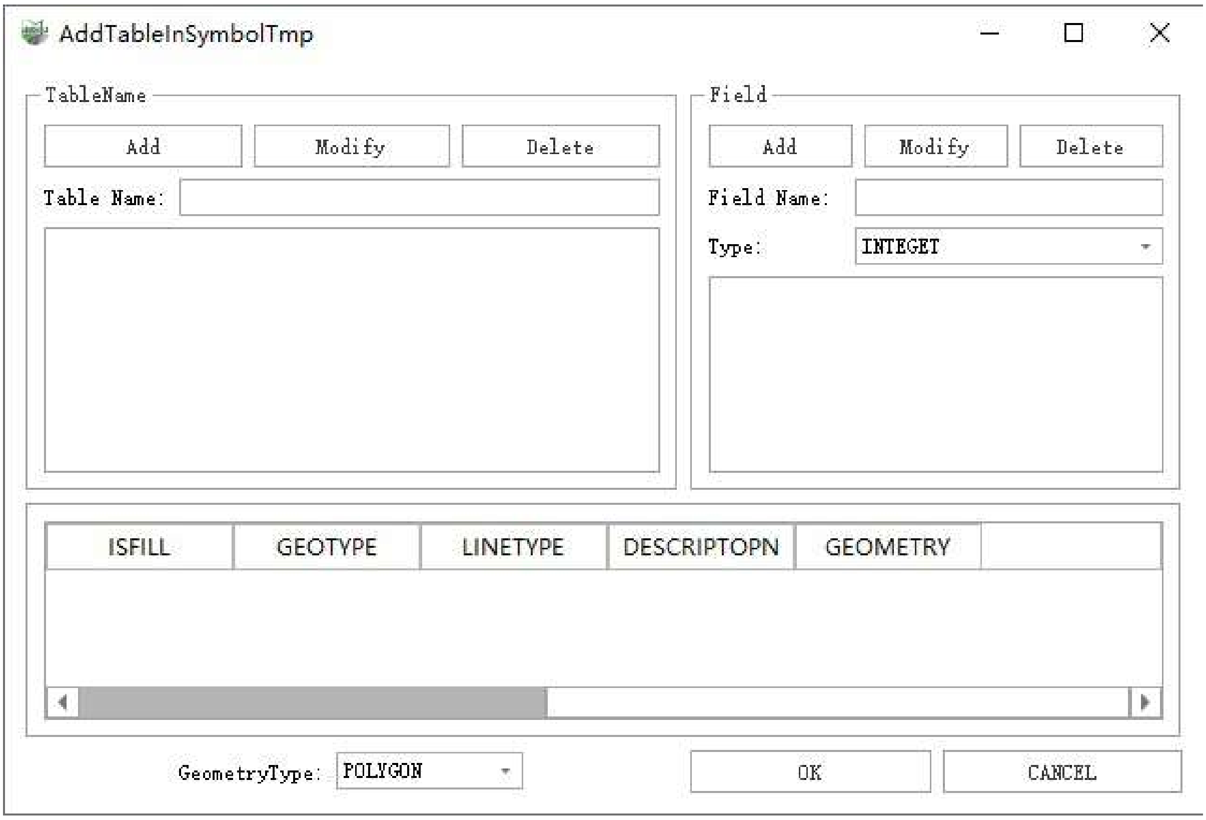


The minimum value is adjusted to 14, the maximum value is adjusted to 18, and the box is not selected to hide

Template Matching Tables

Function Description: Create a custom form, create a custom template.

Dialog



Dialog

Parameter Description

- **Table Name**
  - **Table Name**: Enter the table name
  - **Add**: Confirm to add form
  - **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)
  - MUTILPOLYGON: Multi-contour, Chinese characters, English and other texts or other ground signs with multiple contours, use this type (commonly used)
  - TEXT: Text type (only for backup)

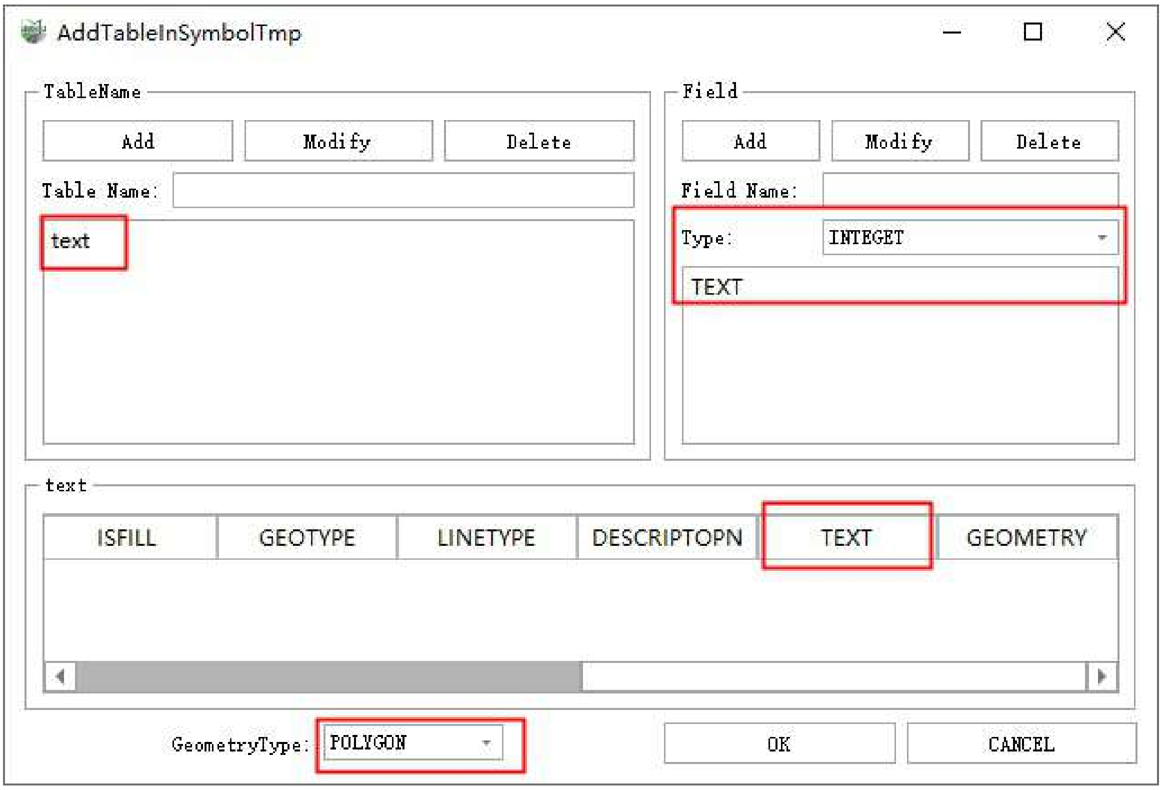
steps

1.Click the **Delete** button to delete the current table

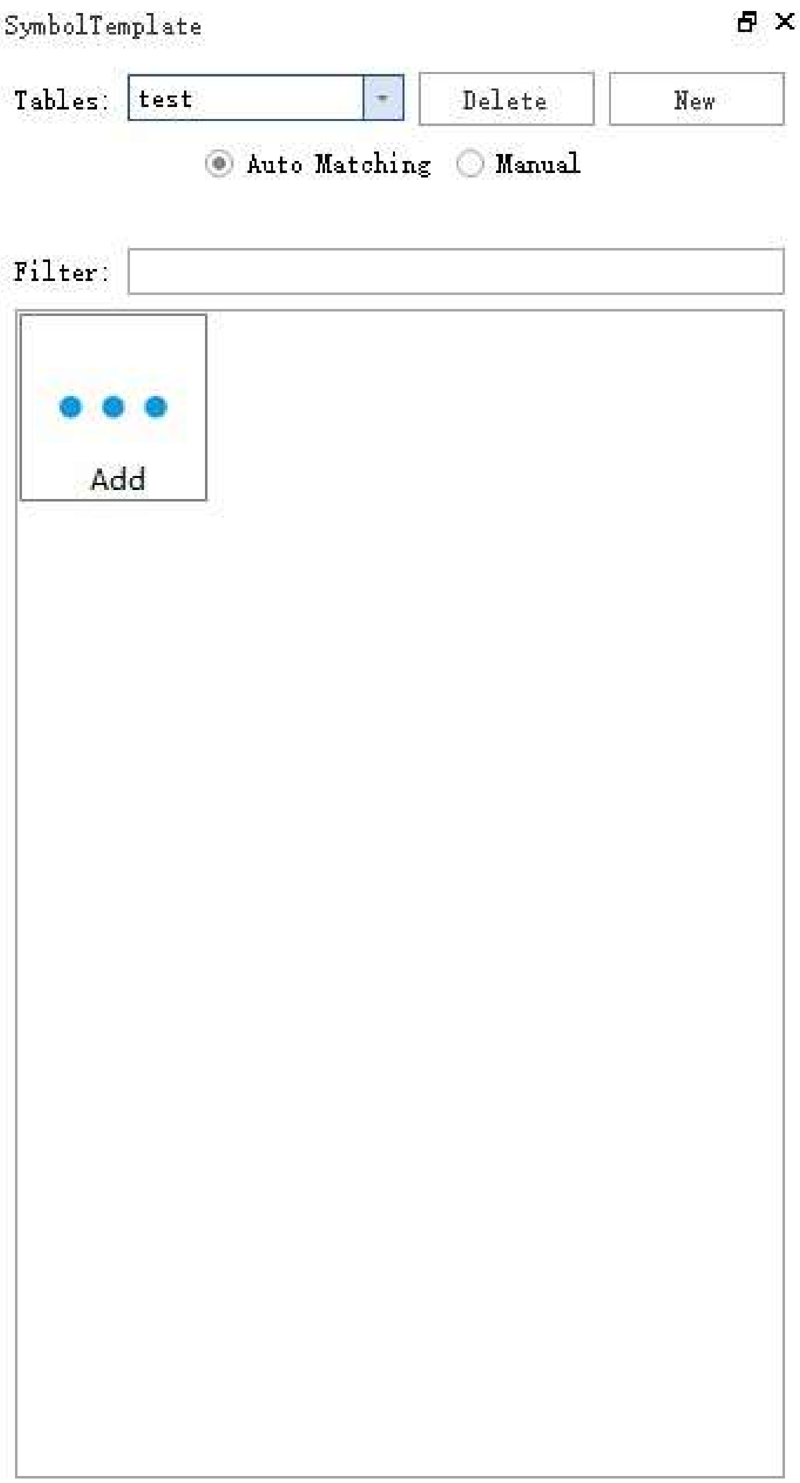
> Built-in table, cannot be deleted

2.Create a new form:

- 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 form type (required)
- Click the OK button



New form



Add results

3.Create a new template, see Template Matching Custom Template

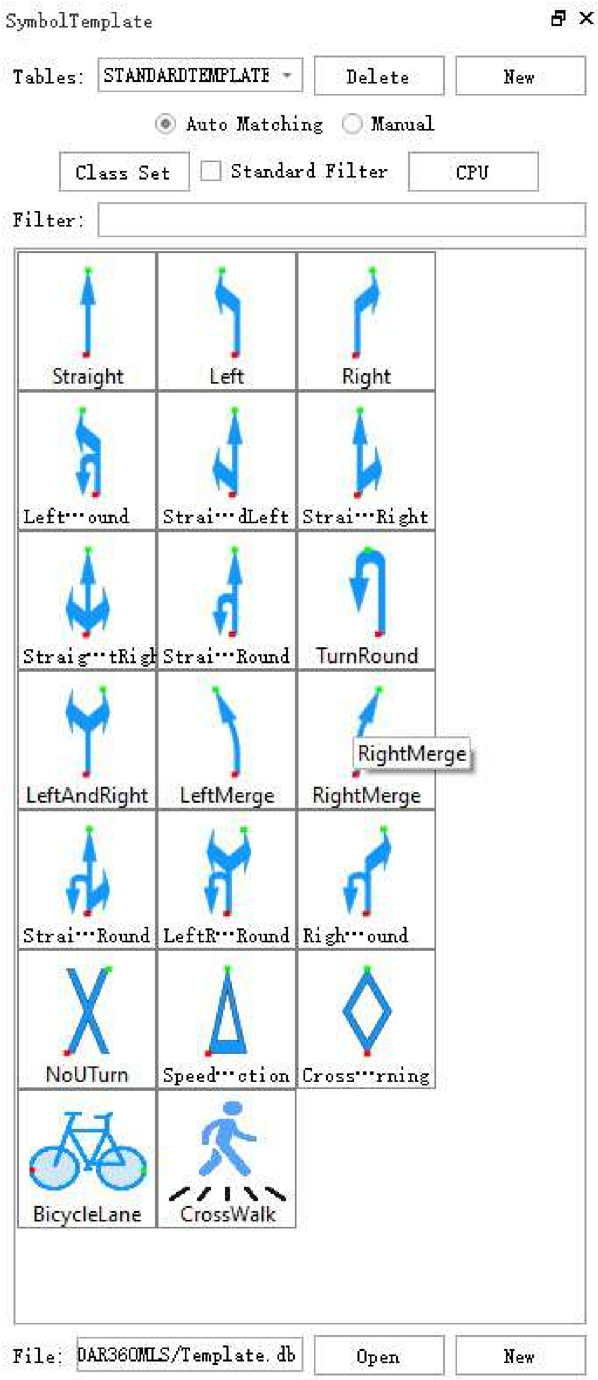
> Built-in forms, cannot add new templates

Template Matching Auto Match

Function description: 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:

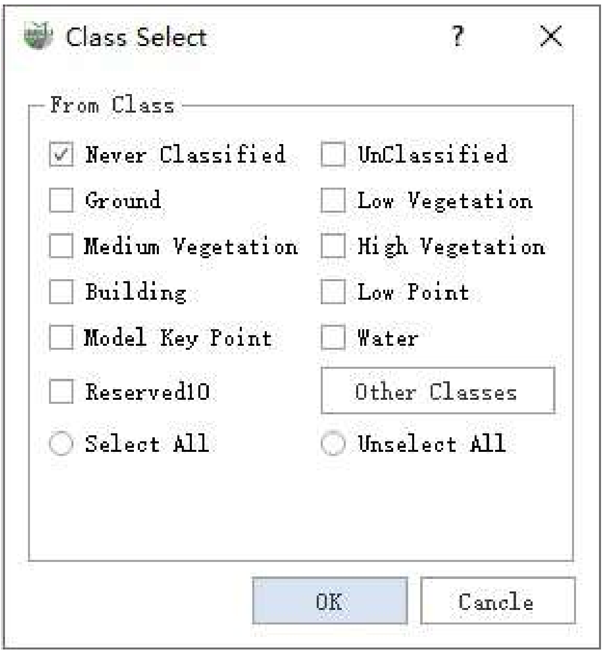
- **Table**: used to select the template table to be used, the main form below displays the content of the currently selected template table in the form of graphics + name

  > There are only built-in standard templates by default, you can view the supported types of built-in templates in the main form below

- **Delete**: delete the currently selected table, among which, the built-in template table cannot be deleted
- **New**: Used to create a custom template table
- **Category setting**: Appears if and only when the currently used template table is a built-in template table and the mode is auto-matching, it is used to select the category for calculation, multiple selections are possible

> **strongly recommended**: only use the category of the ground, if the point cloud has not been classified, **strongly recommended**: when selecting the point cloud, click on the ground point

  - **Other categories**: used to display and select more categories
  - **Select All**: Check all categories
  - **Uncheck All**: Uncheck all categories
  - **OK**: Complete the category settings
  - **Cancel**: Cancel category settings



- **Standard Filter**: Appears if and only when 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 you are 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

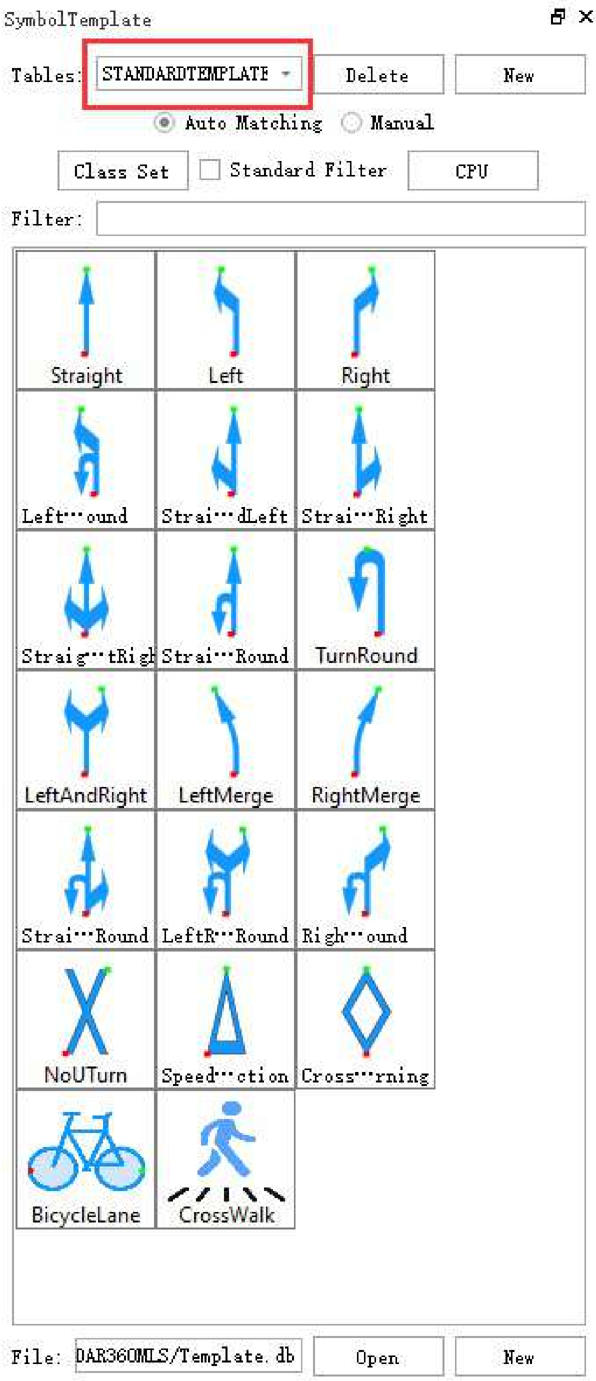
- **Mode: CPU, Mode: GPU**: Appears if and only if the currently used template table is the built-in template table and the mode is Automatch, and is used to select the operation mode of the function

  > The default is CPU mode. When the computer has a GPU environment, you 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

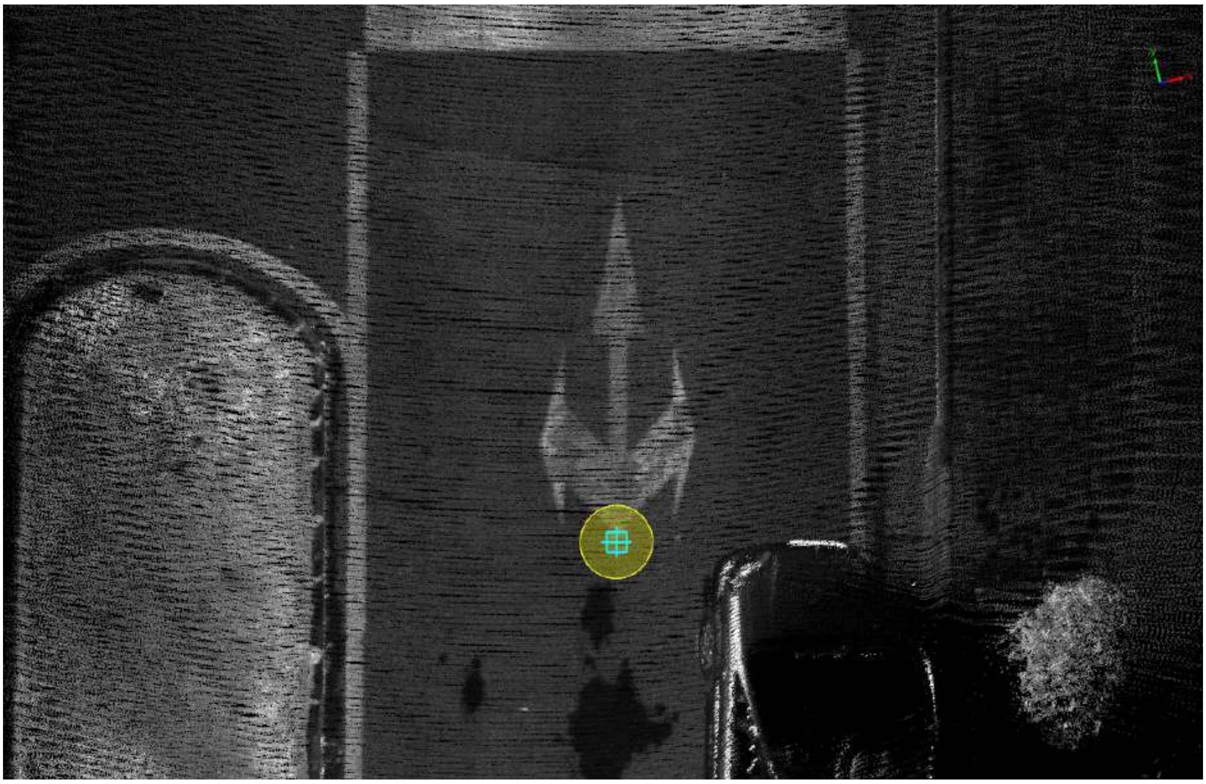
1.Select the built-in standard template form



Select form

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 category setting, set the input category to the category of the ground point If the point cloud data has no classification information, or use all categories as input categories, **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



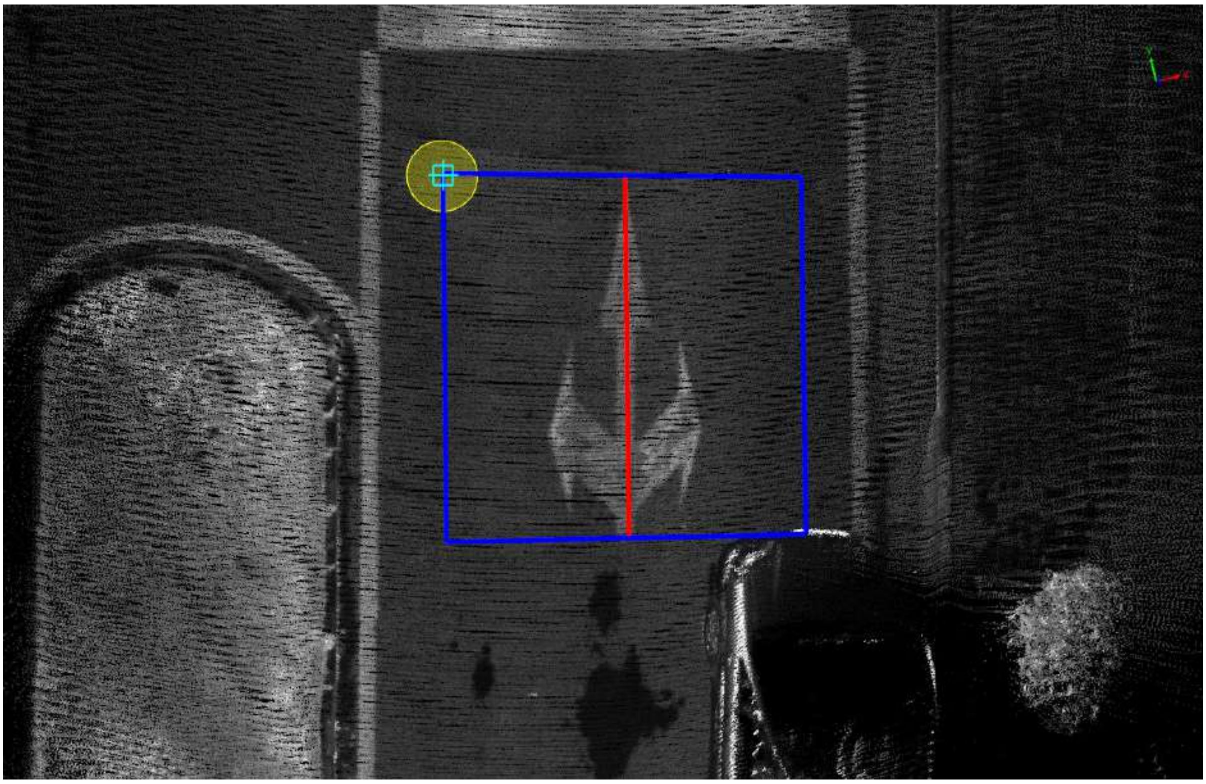
Key point drawing

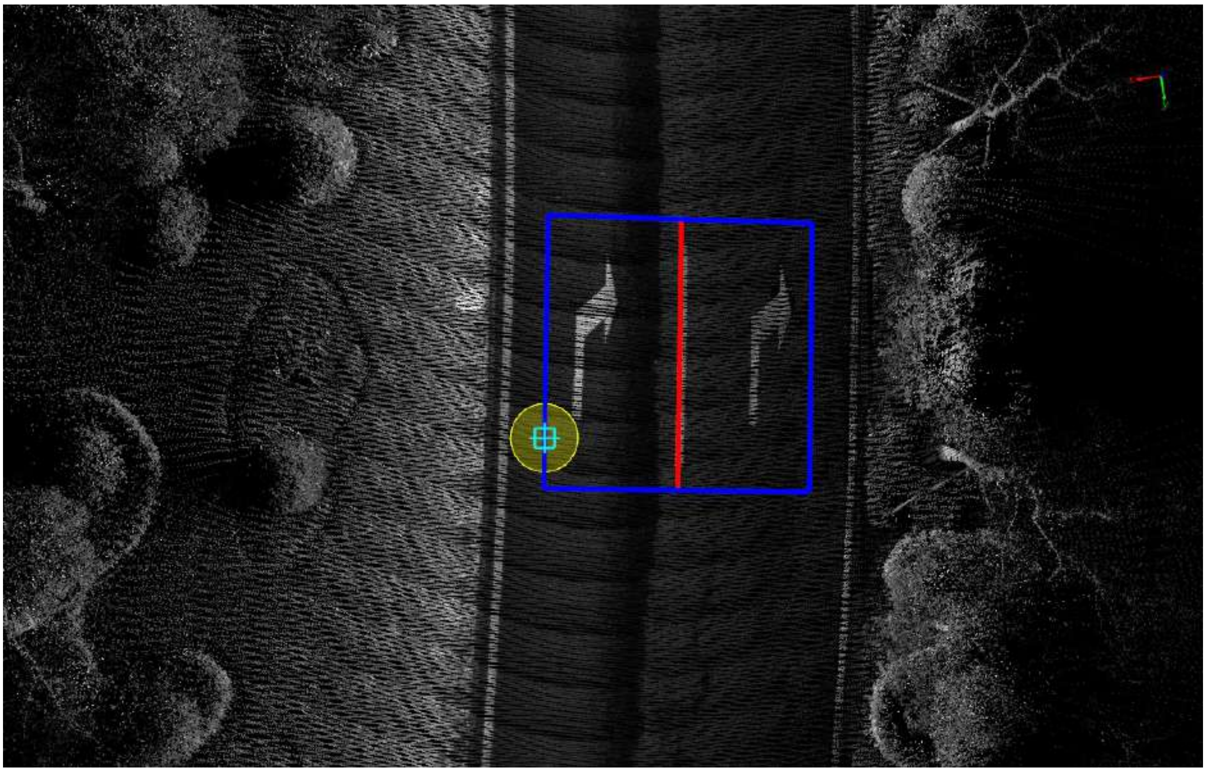
3.Left mouse button, click the second key point

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

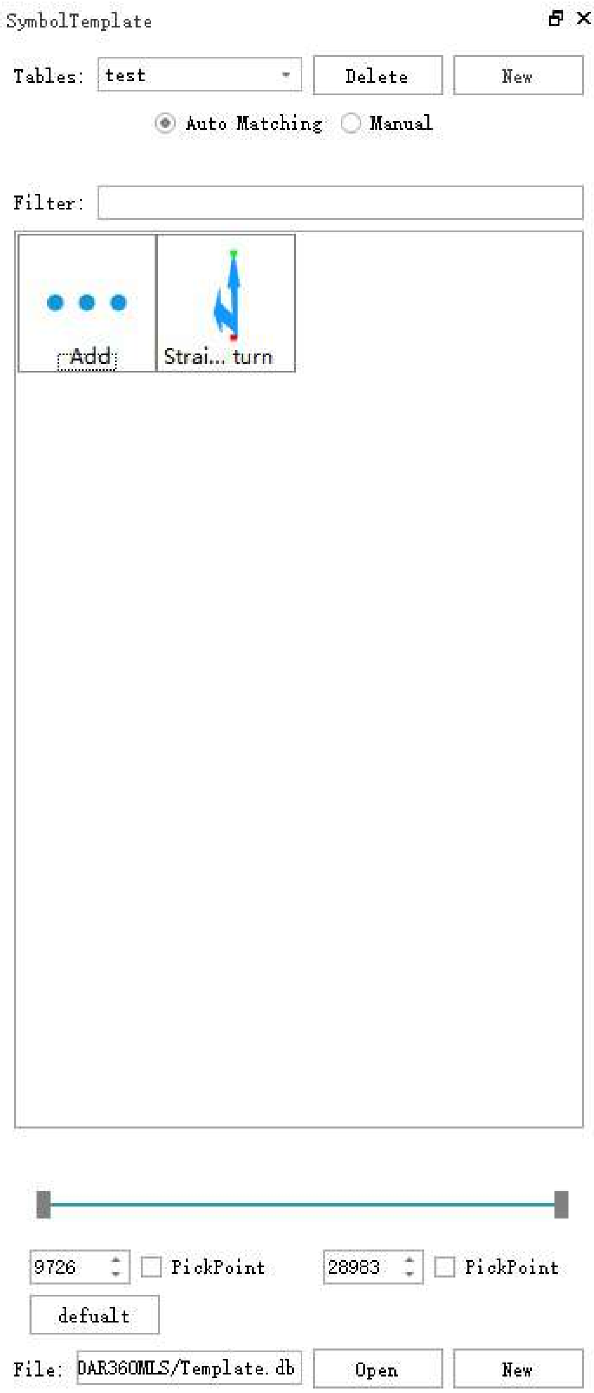

Recommended size of standard template table pull box


Detection results

Custom template automatic matching steps

The custom template can only be selected once, and one target can be detected

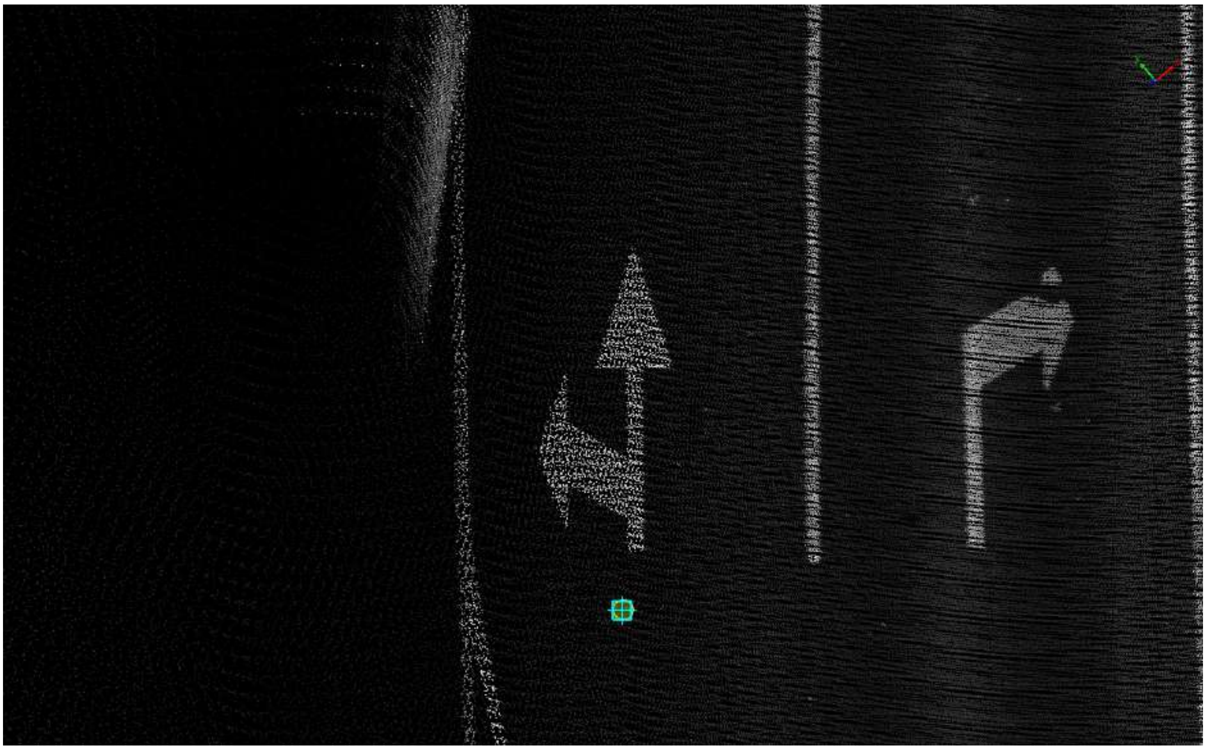
1.Select custom template form



Select form

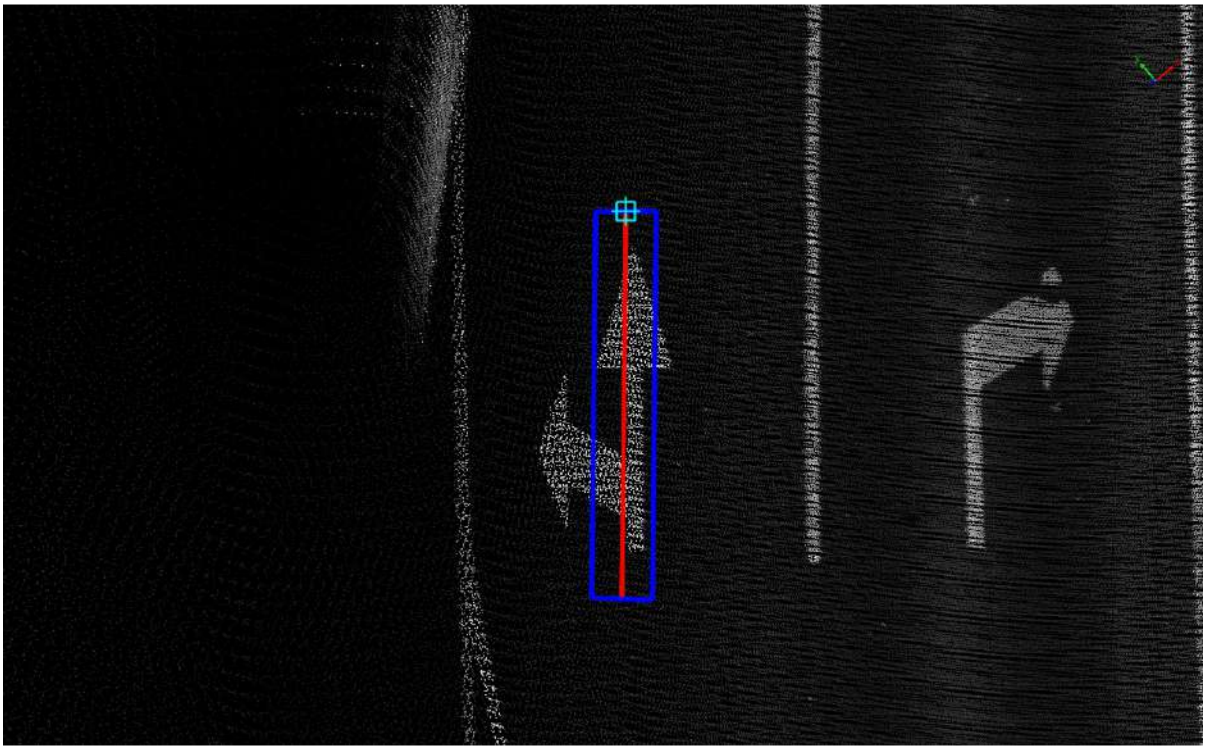
2.Left mouse button, click on the first key point of the target area

> If the point cloud data has classification information, it is strongly recommended that in the category setting, set the input category to the category of the ground point
>
> If the point cloud data has no classification information, or use all categories as input categories, **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

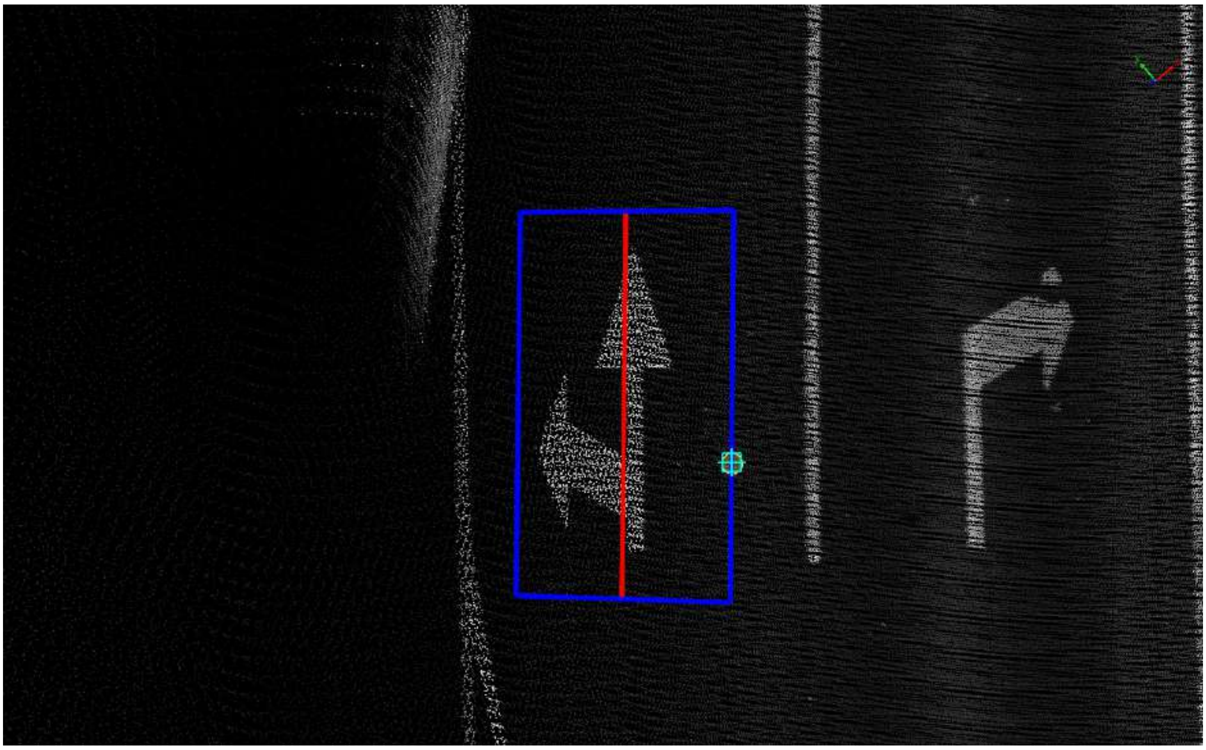
3.Left mouse button, click the second key point



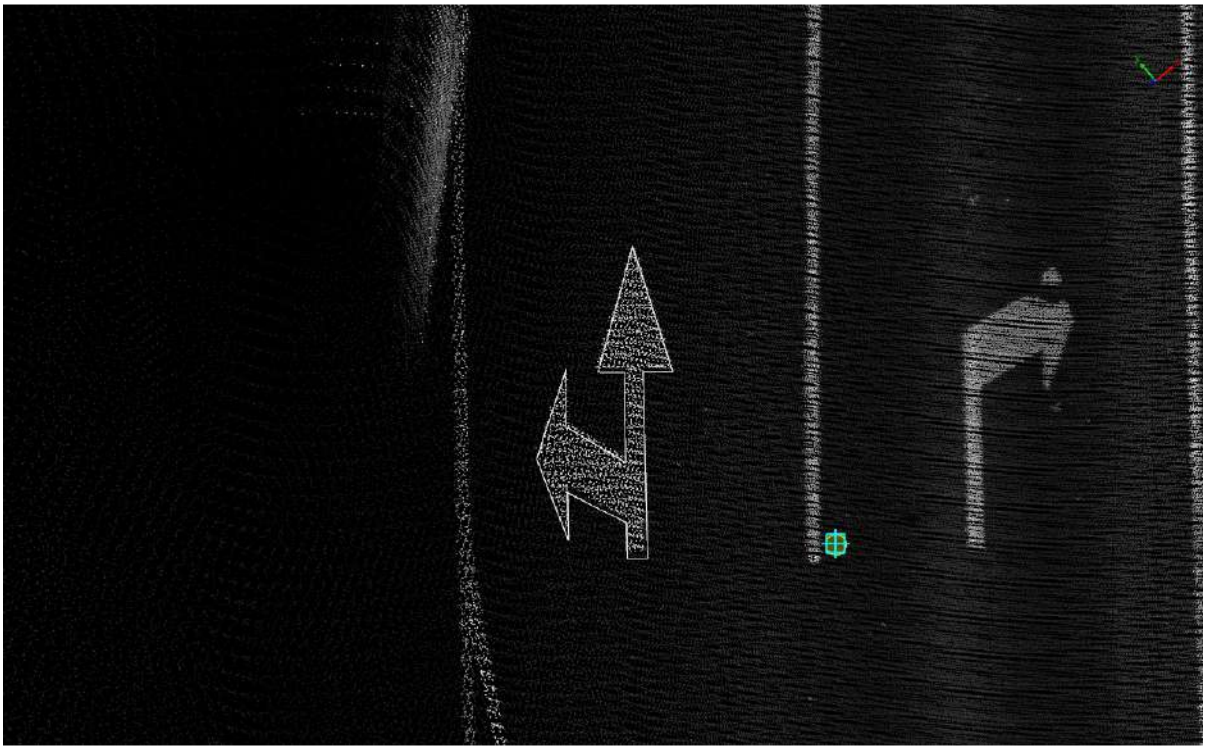
Key point drawing

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

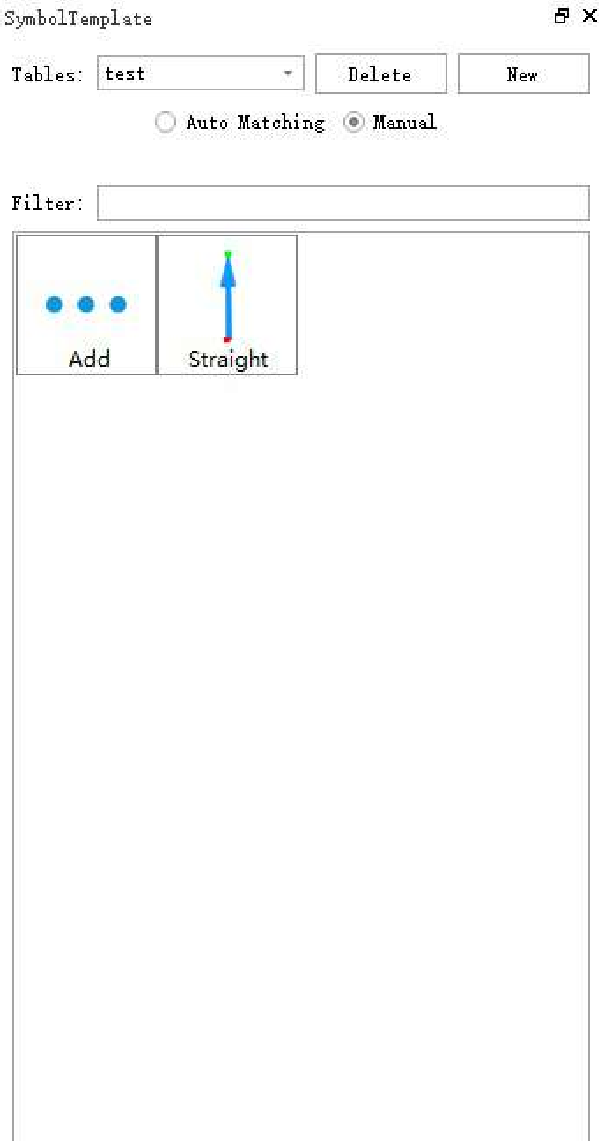
Feature Description: Make a new template, or manually add vector results from an existing template.

New template creation

To make a new template, see Template Matching--New Template

Template based, manually drawn

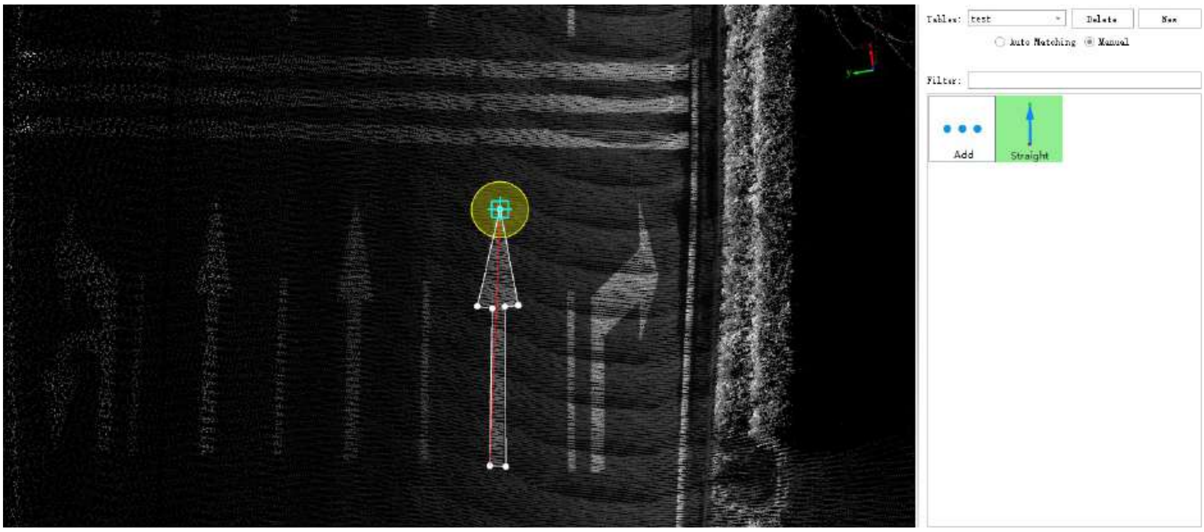
1.Left mouse button in the preview window, select the template to be added manually



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

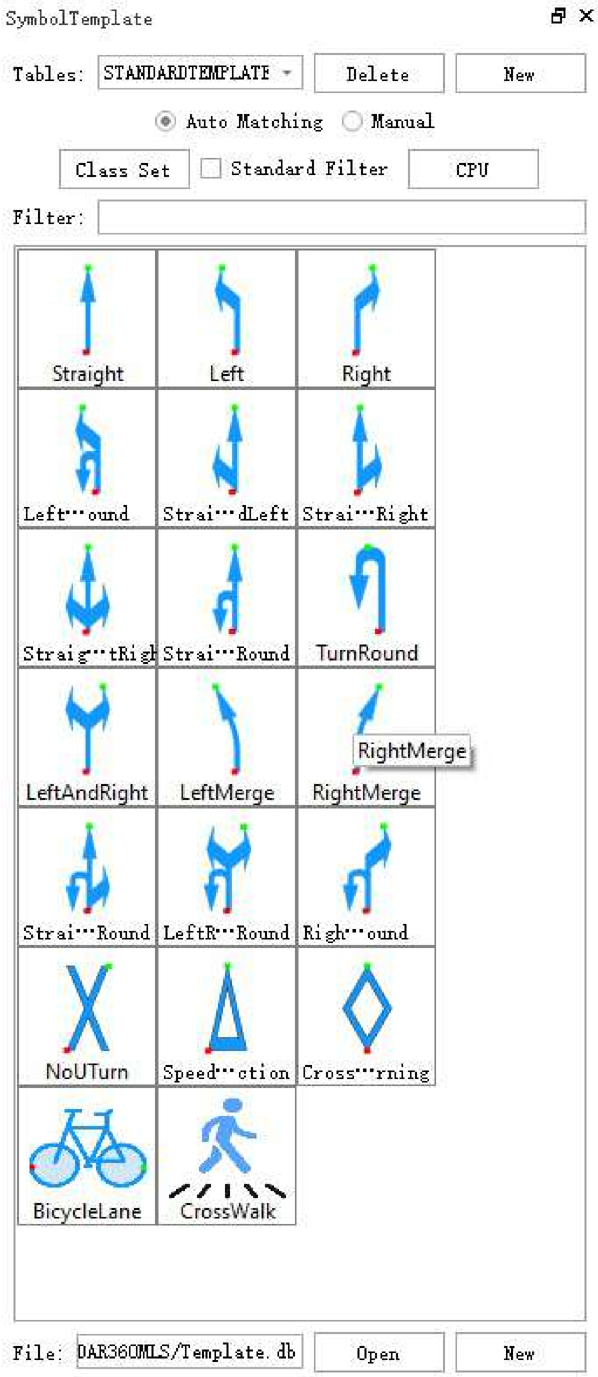

Tables: test
Delete
New
Auto Matching
Manual
Filter:
Add
Straight
Rendering

Template matching database file

Function description: Create and open database files.

> Open built-in database file by default

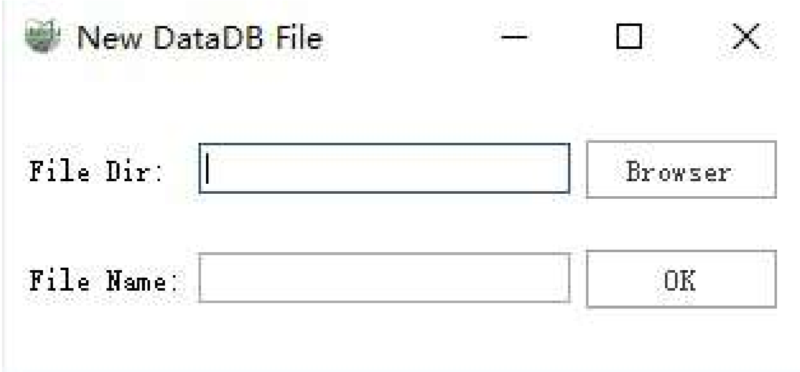
steps



Shortcut hints

1.Click the Open button to pop up the Select File dialog box to select the database file.

2.Click the New button to pop up the New Data File dialog box.



3.Click the browse 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.

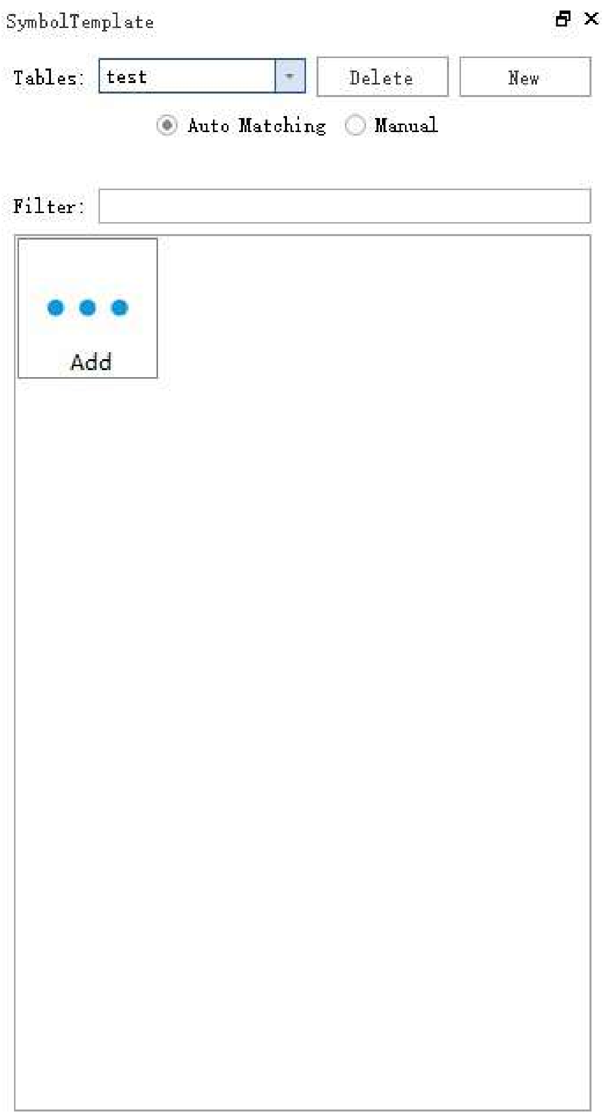
Template Matching New template

Function description: In the custom form, create a new custom template.

> New templates can only be created in custom forms

steps

1.Select a custom form

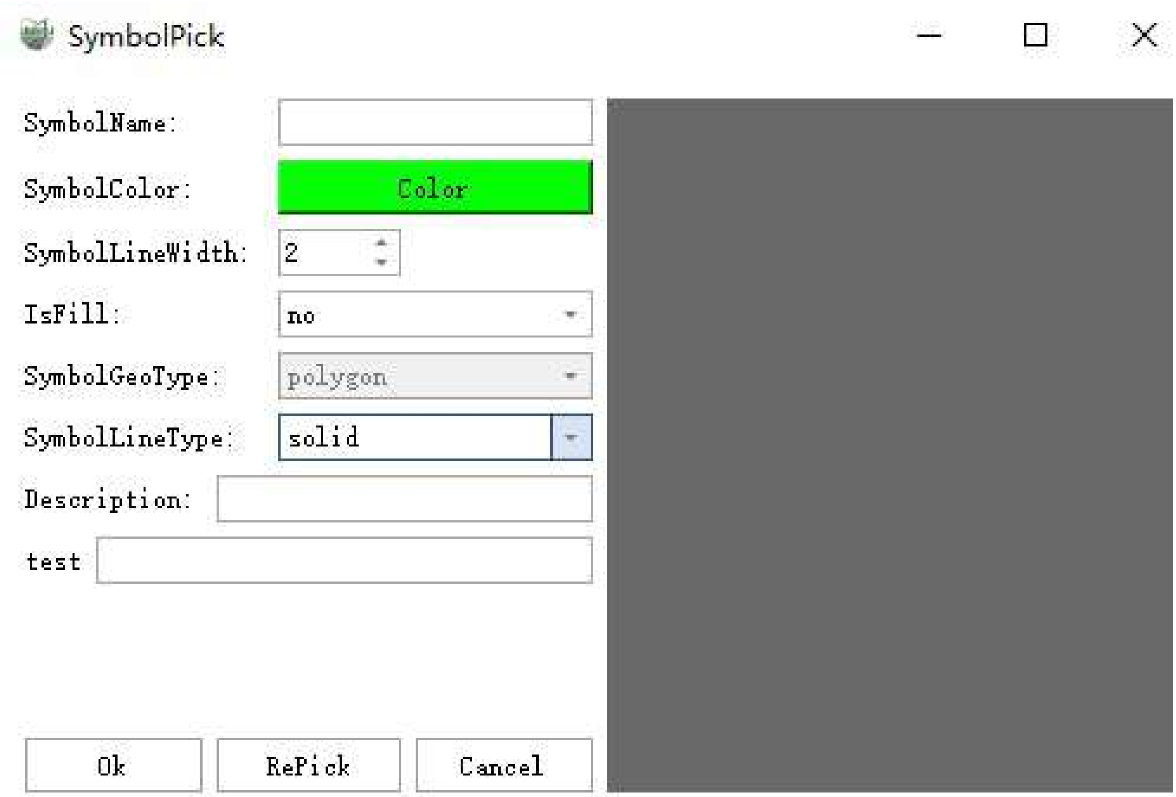


Custom form

2.Click the **Manual creation** checkbox

3.Click the **Add** icon in the preview

4.Pop up **New Template Creation dialog**



Dialog

Parameter Description

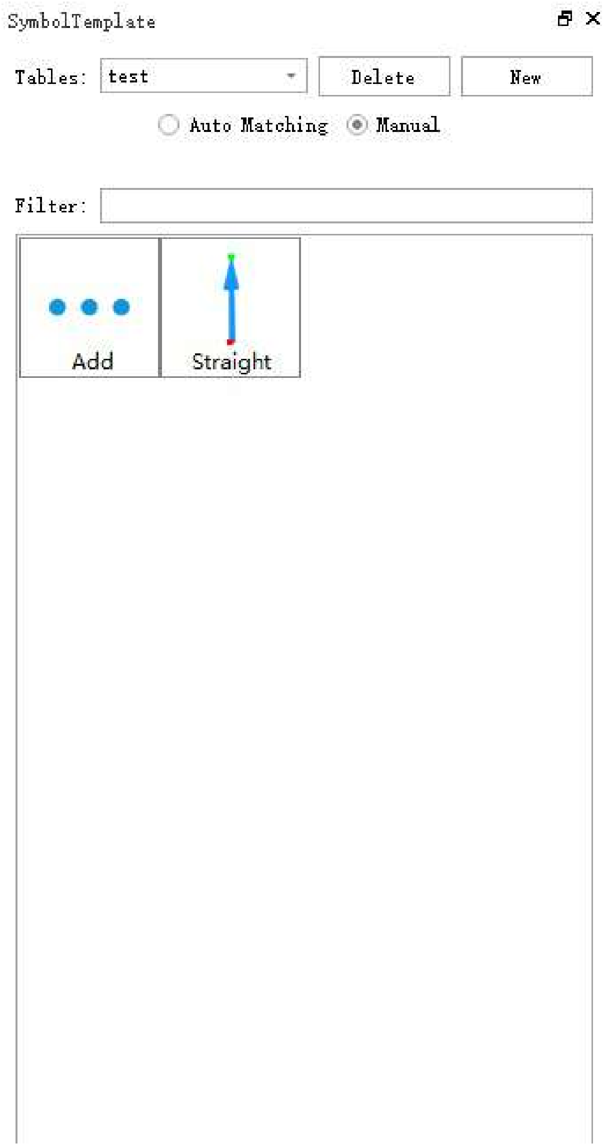
- **TemplateName**: Template name
- **Template Color**: Set the color of the template
- **Template line width**: Set the template line width
- **fill**: whether to fill the template
- **Template Geometry Type**: Set the geometry type of the template
- **description**: text description
- **Real-time preview**: 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 in turn

> 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 live preview

6.Click the **OK** button to save the template



New template result

7.Click the **Reselect** button to redraw

Batch Ground identification

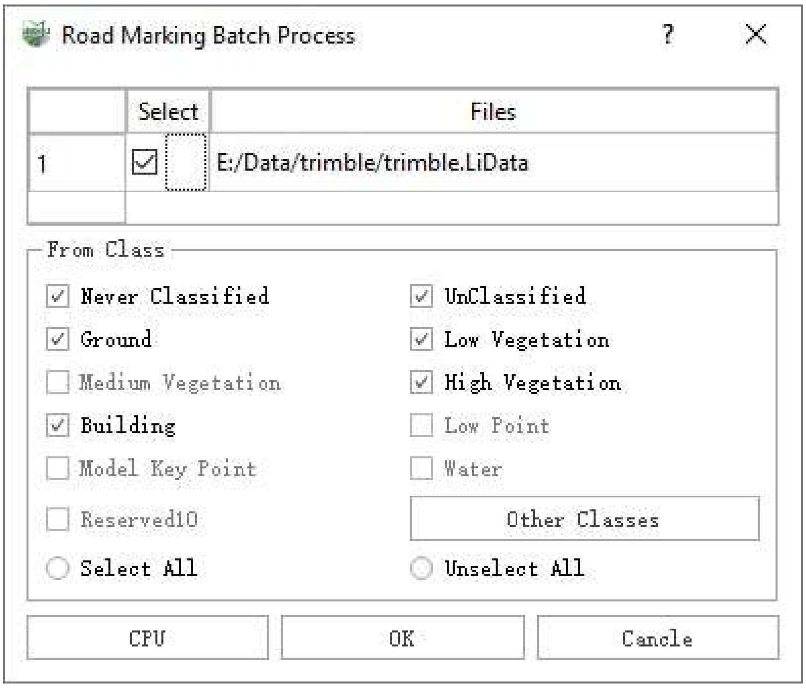
Function description: Automatic, deep learning-assisted ground identification extraction for a large amount of point cloud data.

> **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 category, which can effectively improve processing efficiency and detection result accuracy

STEP

1.Click **Ground Mark Batch** in the Map Elements panel  button.

2.Pop up **Parameter setting dialog**



Ground marking batch

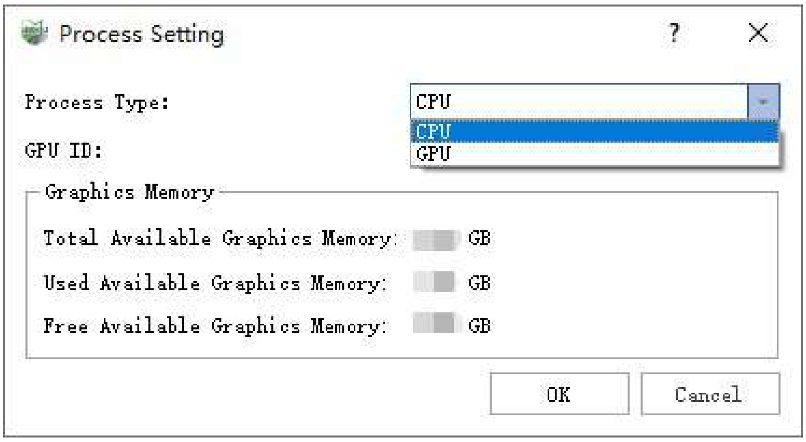
Parameter Description

- **Select**: Check/uncheck the point cloud data to be processed
- **Source Category**: Select the category point that needs to be calculated, you can select multiple
  - **Other categories**: Click to show other categories
  - **Select All**: Click to select all categories
  - **Cancel All Selections**: 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 category where the ground point is located as the input category, which can effectively improve processing efficiency and detection result accuracy

- **Mode: CPU, Mode: GPU**: Set the operation mode, you 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.

Road facilities

Road facilities

The road facility module is used for **vectorization and manual** production of **road appurtenances**.

Includes the following functions：

- Rod detection
- Power line detection
- Add traffic signs
- Add parking space
- Add drain grate
- Add manhole cover

Pole Detection

Distinguished by **operating mode**, divided into four modes:

- Single point mode
- Two-point mode
- Polygon mode
- Grip mode

Pole Detection-single point mode

Function description : Single-point mode for pole detection, manually click on the top of the pole as the key point, and automatically generate the rod by setting the search radius.

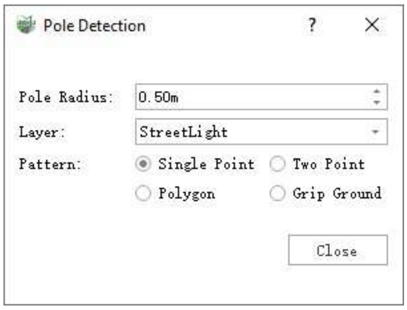
> The generated rod is **precisely perpendicular** to the ground, while the real rod is **not necessarily perpendicular** to the ground due to the **instrument scanning unilateral, its own tilt, the bottom thick top thin**, etc.

Step

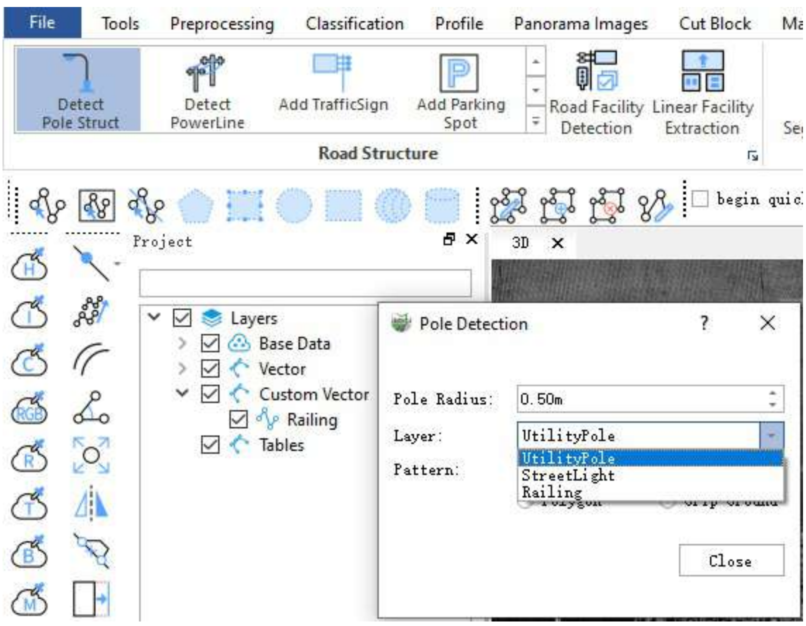
1.In the Road Facilities panel click on the drop down logo on the right and select **Pole Detection** button.

2.Pop up the**Pole detection setting dialog box**

- Single point mode



Single Point Mode Dialog



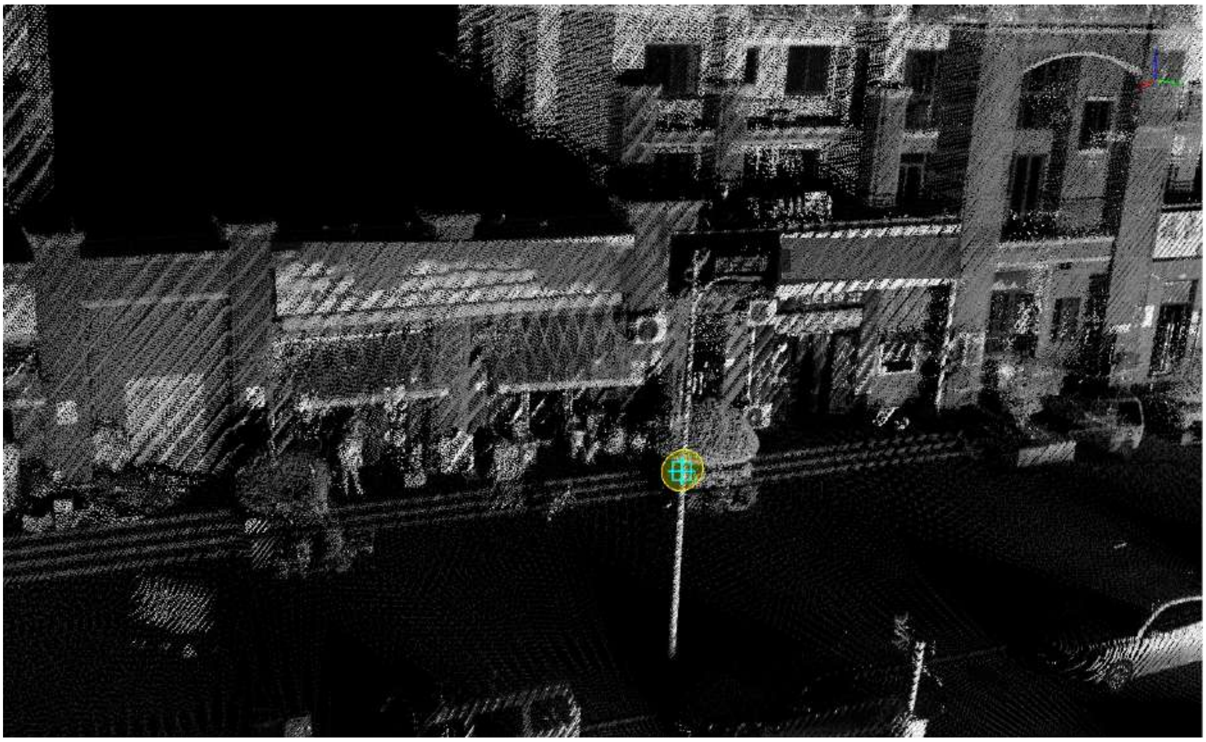
Single Point Mode Dialog Custom Line Layer

Parameter description:

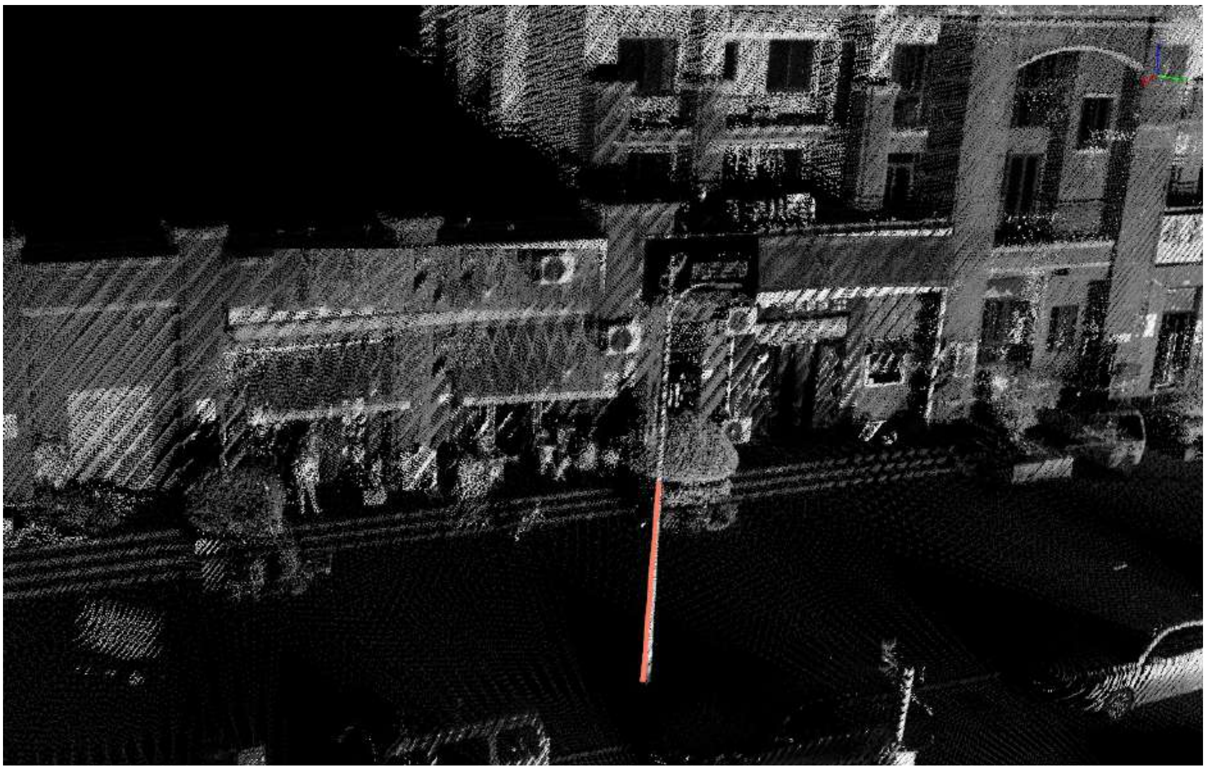
- **Rod radius**: The default is 0.5 meters, which means that the self-clicked **key point** is the reference point, and the box is selected **Rod radius is the radius, and the vertical downward** cylinder.

- **Layer**: The default is street light, the options are: **streetlights, 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 with the left mouse button** and click **on the top** of the pole **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

Pole detection-two-point mode

Function Description: The two-point mode of pole detection, manually click on the top of the rod and the bottom of the pole as the key points to generate the pole vectorization result.

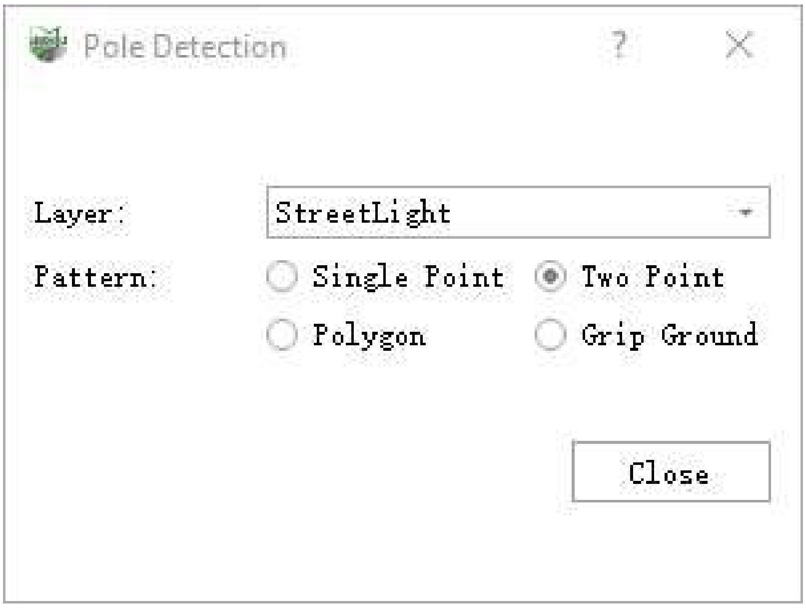
> The generated bar is **exactly in line with the two points clicked**.

Steps

1.In the Road Facilities panel click on the drop down logo on the right and select **Pole Detection** button.

2.Pop up the **Pole detection setting dialog box**.

- Two-point mode



Two-point modal dialog box

Parameter description:

- **Layer**: Default is street light, optional: **street light, utility pole**, you 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 mode**, which means the operation mode of **Clicking on key points**.

3.**left mouse button click** click **top of the rod**, details as shown in the figure, **yellow point** that is the mouse click **key point**.

Click on the key point at the top of the pole

4.**Left mouse button click** Click **At the bottom of the rod**, the details are shown in the figure, **Yellow dot** 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.

Pole detection-Polygon mode

Function Description: The polygon mode of pole detection is used to select multiple rods at once, and manually click on the polygonal area where the poles are located as the key point to automatically extract the vectorization results of the rods.

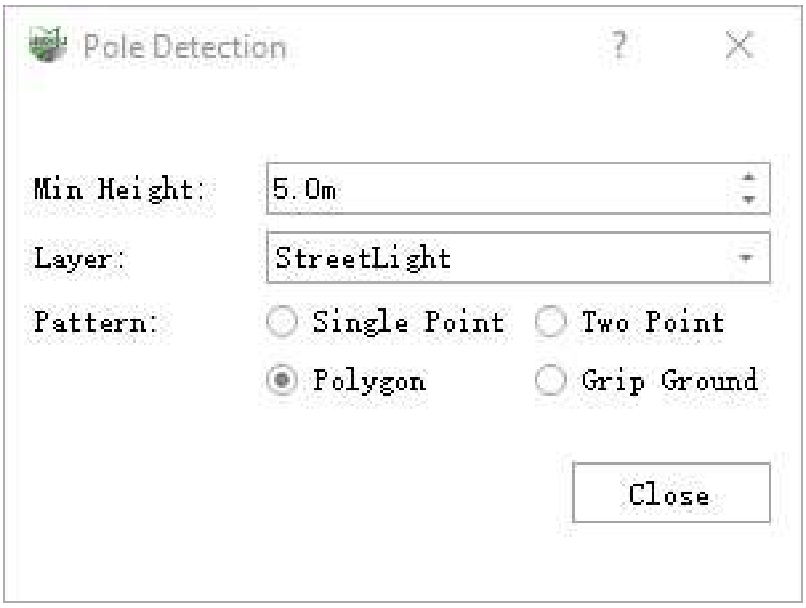
> The generated rod is **precisely perpendicular** to the ground, while the real rod is **not necessarily perpendicular** to the ground due to the **instrument scanning unilateral, its own tilt, the bottom thick top thin**, etc.

Steps

1.In the Road Facilities panel click on the drop down logo on the right and select **Pole Detection** button.

2.Pop up the **Pole detection setting dialog box**:

- Polygon mode



Polygon Mode Dialog box

Parameter description:

- **Minimum height**: The default is 5.0 meters,**below** set height of the rod **will not** be extracted, at the same time, if **set too low a height, 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**.

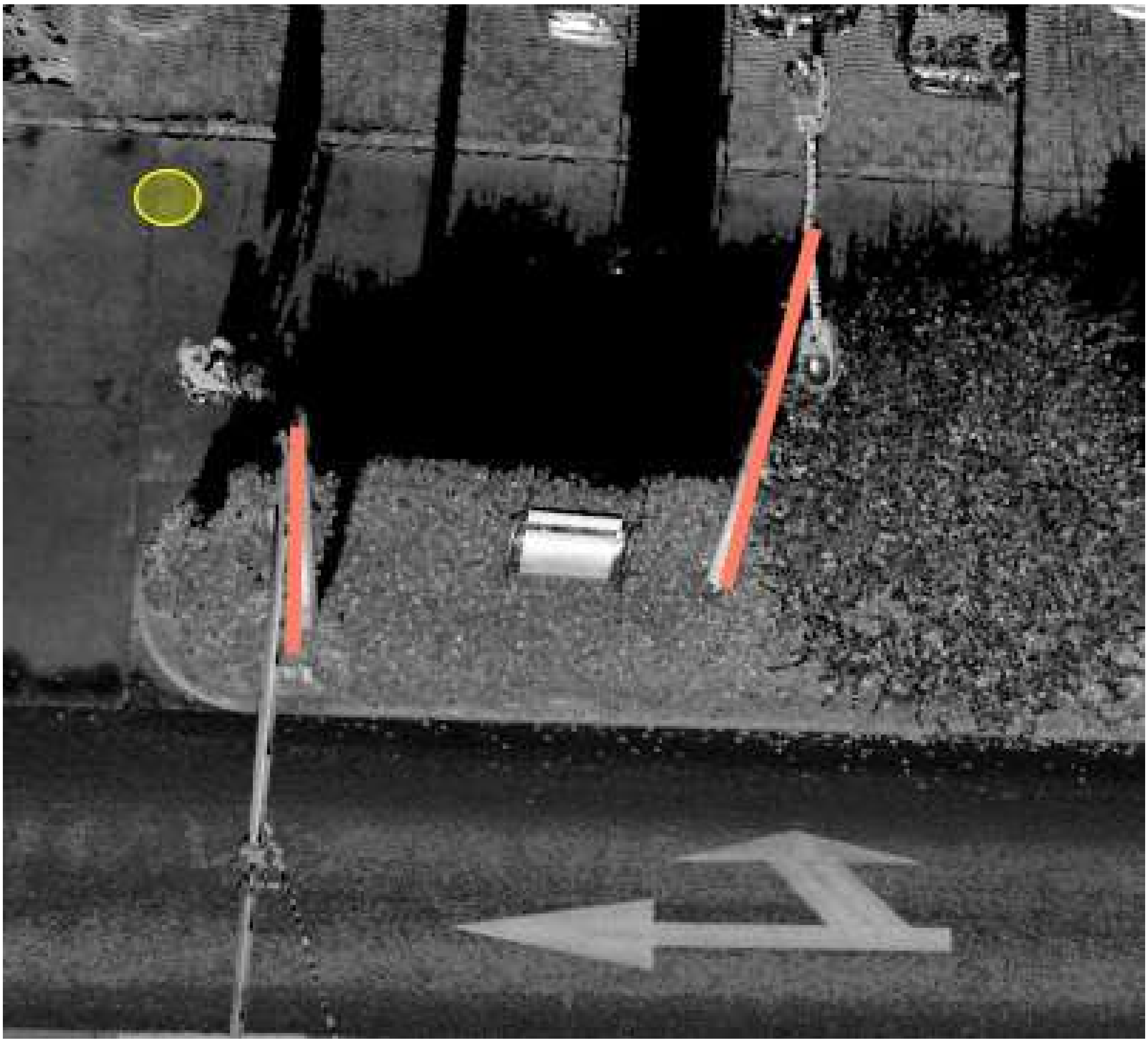
3.**left mouse button in order to click** click **the polygon area where the rod is located** to **left mouse button double click to end** polygon area box selection, automatically start **multiple rod detection**, details as shown in the figure, **yellow point** that is the mouse click **key point**.

> **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 **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, the rest are pop-up **Detection Complete**.


Detection completed

Polygon mode effect

Pole detection-grip mode

Function description : The grip point mode of pole detection, 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.

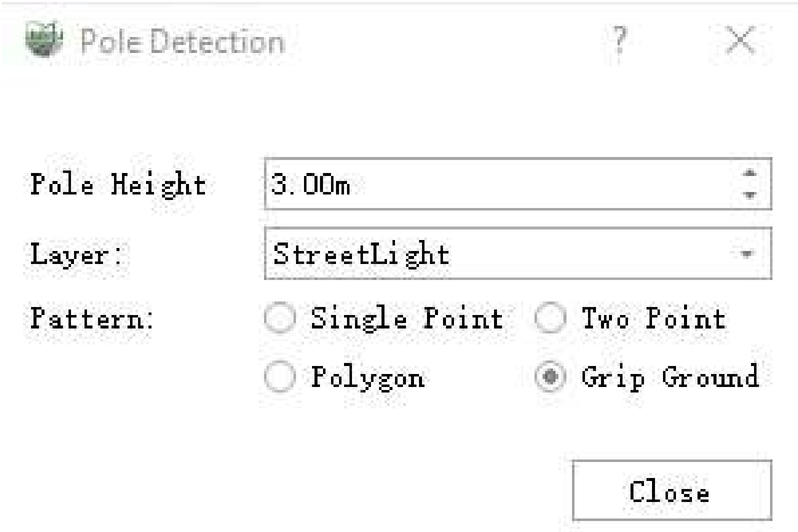
> The generated pole **rises from the clicked ground point, with a precisely set height**.

Step

1.In the Road Facilities panel click on the drop down logo on the right and select **Pole Detection** button.

2.Pop up the **Pole detection setting dialog box**

- 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, 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.**Left mouse button click** click **any point on the pole**, details as shown in the figure, **yellow dot** that is the mouse click **key point**.

Click on the key point of the lever

4.**Left mouse button click** Click **Surrounding ground points**, the details are shown in the figure, **Yellow dot** 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

Power line detection

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

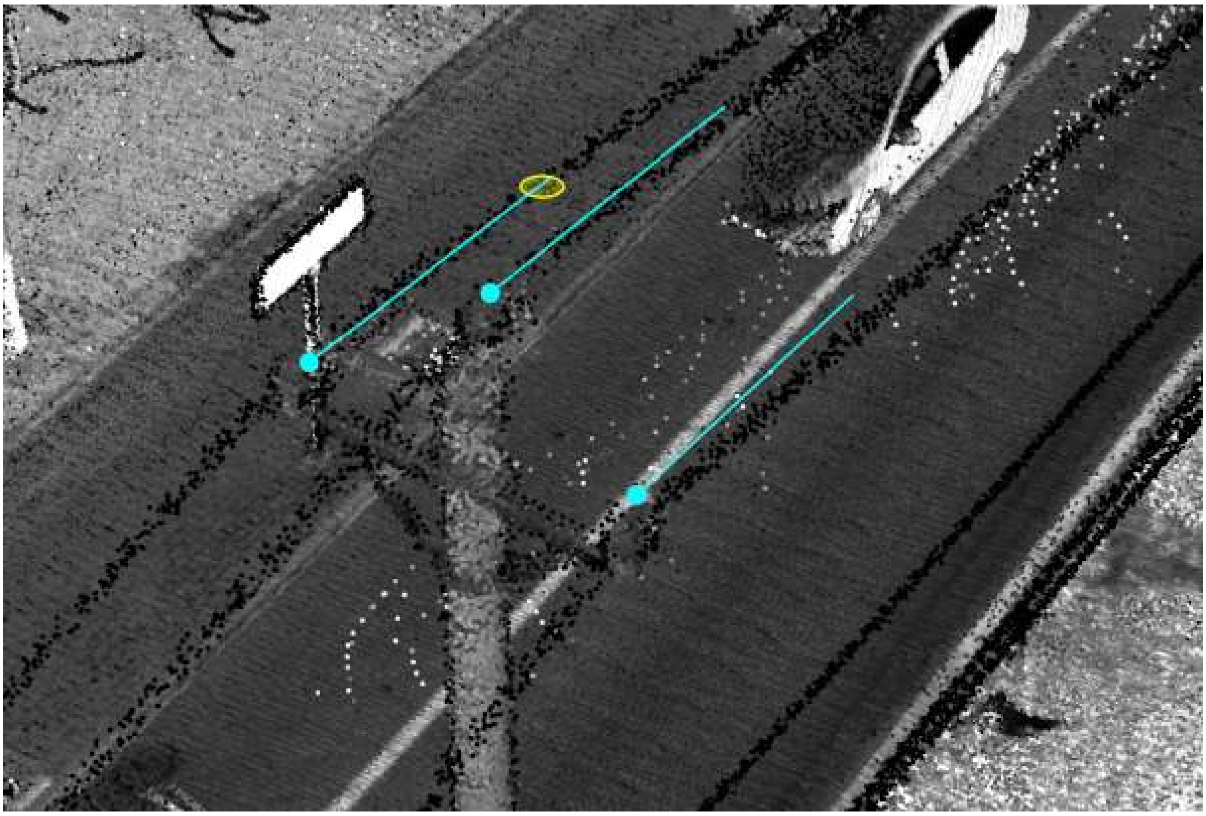
1.In the Road Facilities panel click on the drop down logo on the right and select **Power Line Detection** button.

2.**Left mouse button click** Click **The starting point of the growth of each power line**, 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

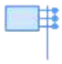

Click on the growth direction

Add traffic sign

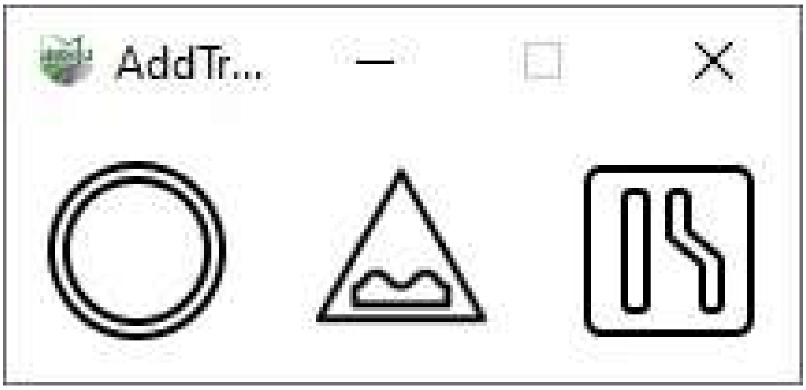
Function description : Manually add traffic signs.

> In order to obtain the plane position correctly, you 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 Traffic Signs**  button.

2.The **Settings dialog box** pops up, select the shape you want to draw:



Dialog box

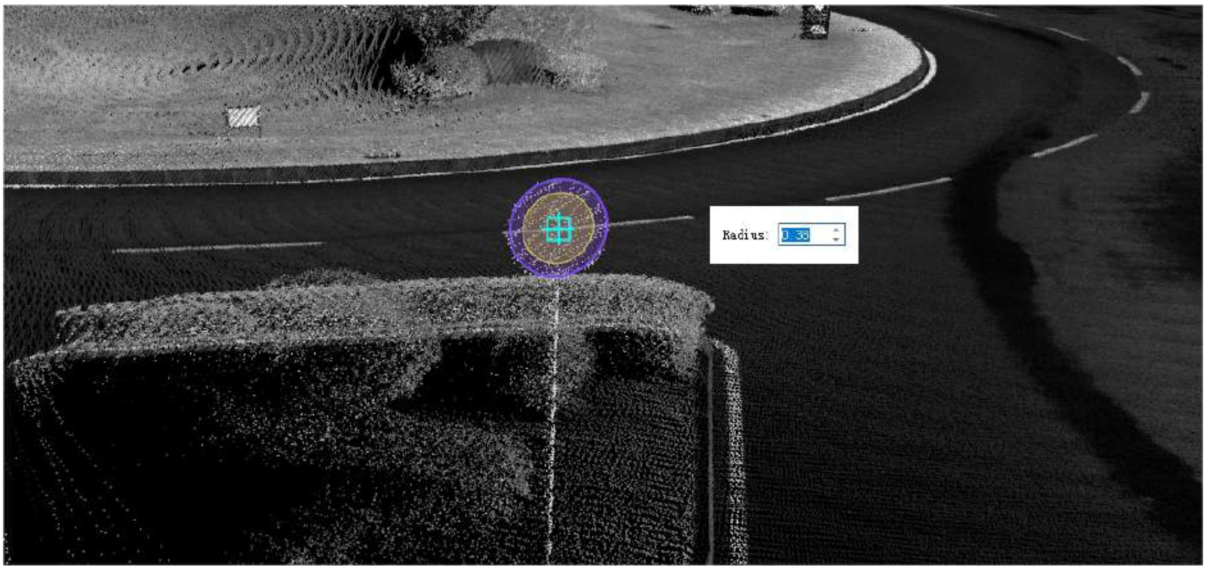
Parameter description:

- 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 in real time and then the mouse, place the mouse in the center of the sign 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 sign

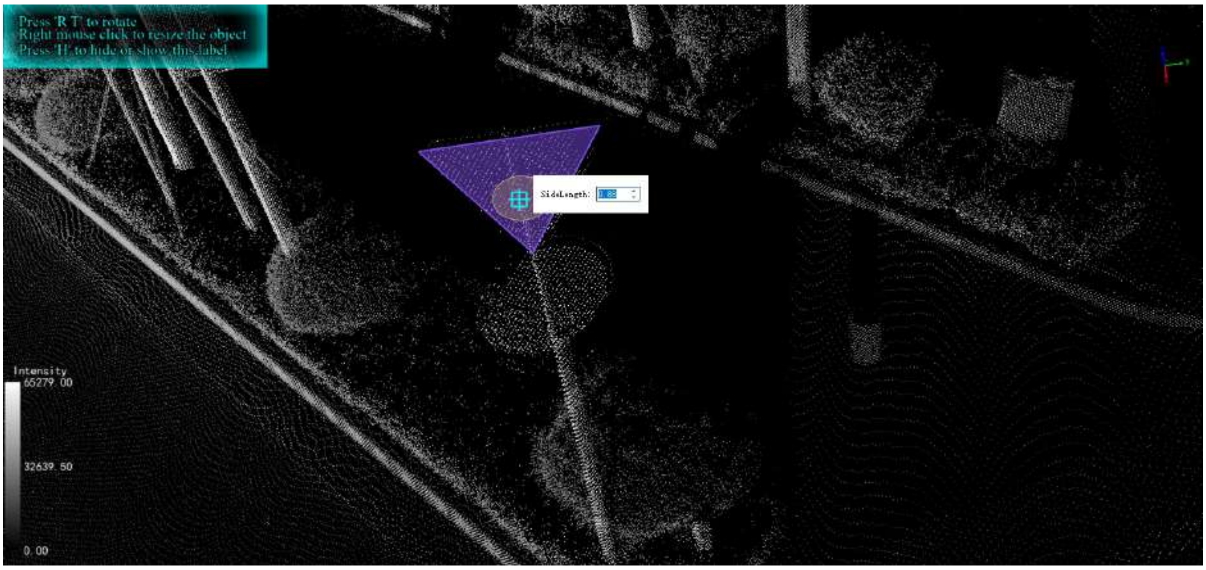
Shortcut key name	Function
r	Rotate counterclockwise
q	Rotate clockwise
Right mouse button	Change the size of the vector template

Round sign



Round

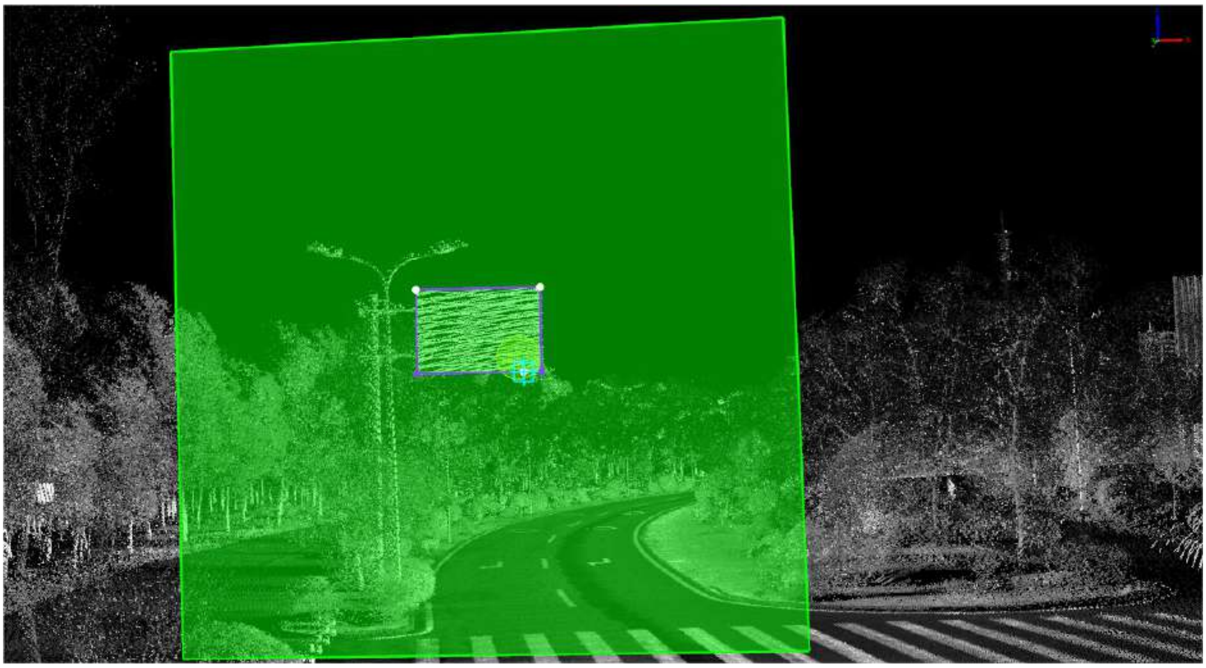
Triangle sign



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. Rectangle drawing.

Rectangular sign



Rectangle

Add parking space

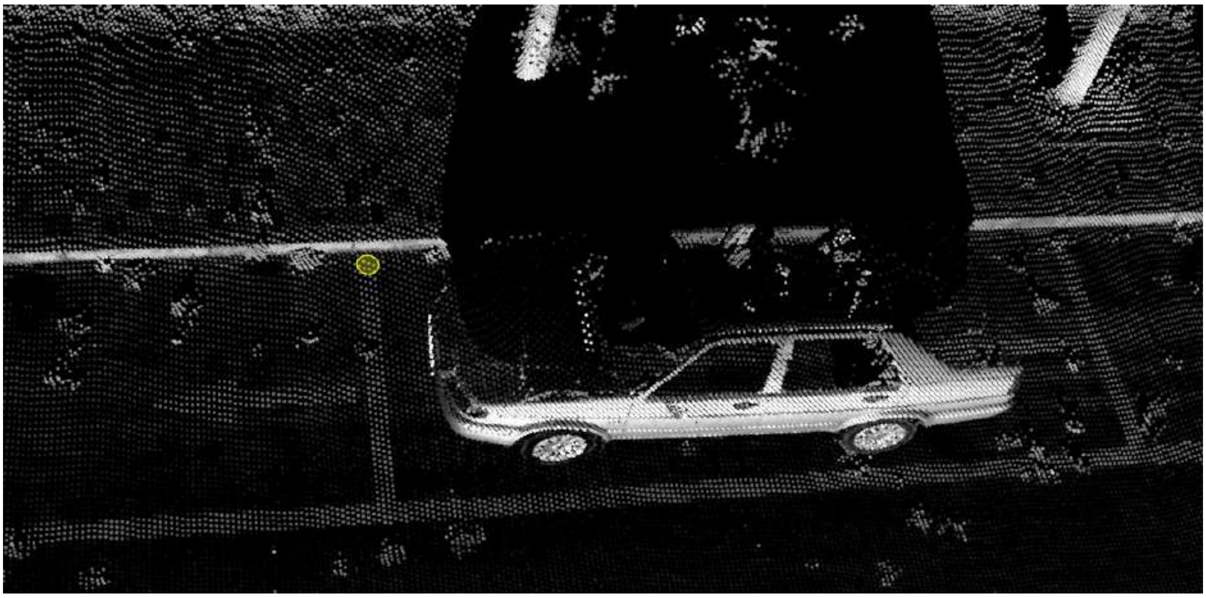
Function description : Manually generate parking space vector.

> Four points to determine multiple parking spaces at the same time.

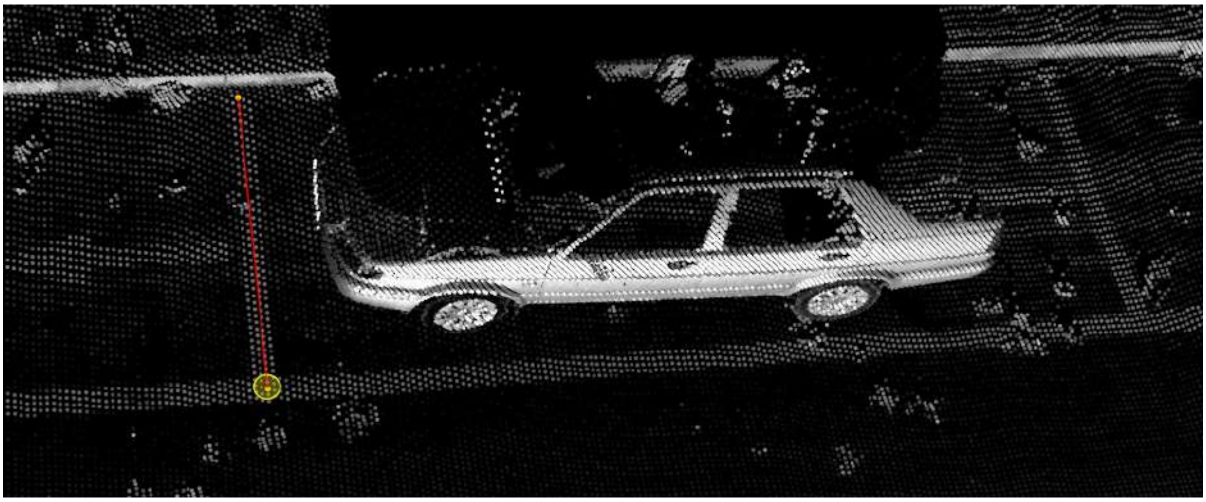
Steps

1.In the Road Facilities panel click on the drop down sign on the right and select **Add parking space**  Button

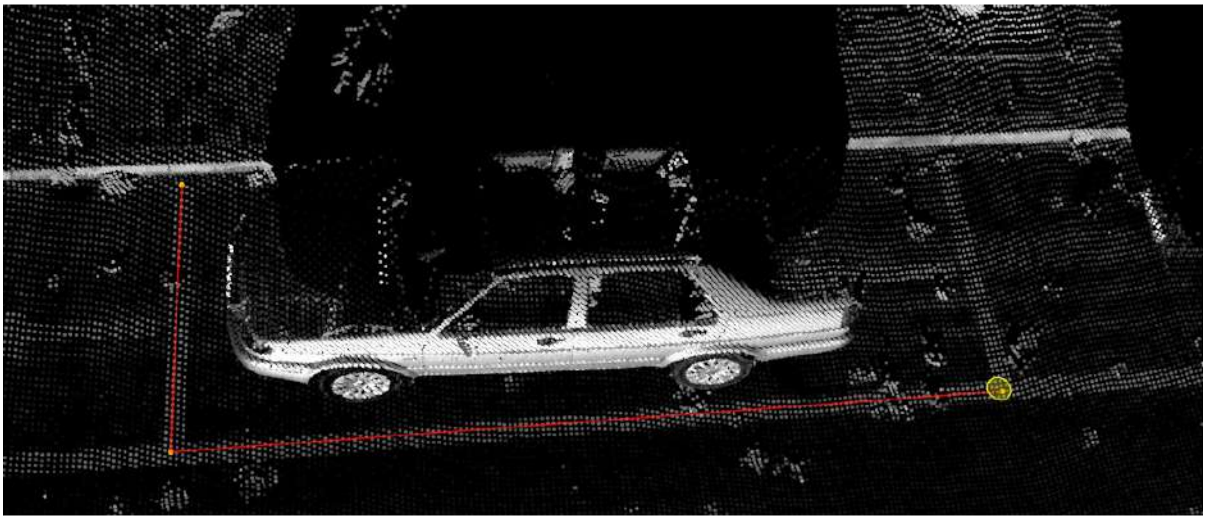
2.**Left mouse button** Click on a single parking space to get three corners



Click on the key point

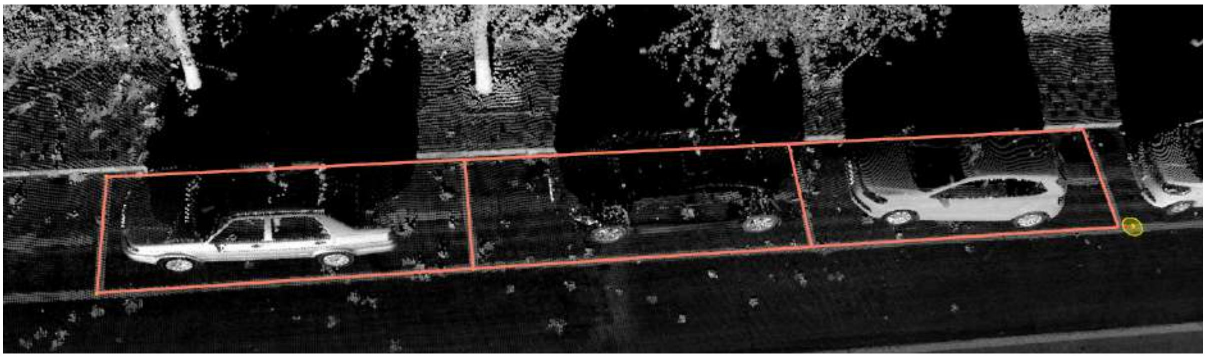


Click on the key point



Click on the key point

3.Three points to determine a single parking space, move the mouse, will automatically follow the length of the second key point in the fourth key point, the number of parking spaces updated in real time display, **double-click** the fourth key point to complete the production



Renderings

Add drain grate

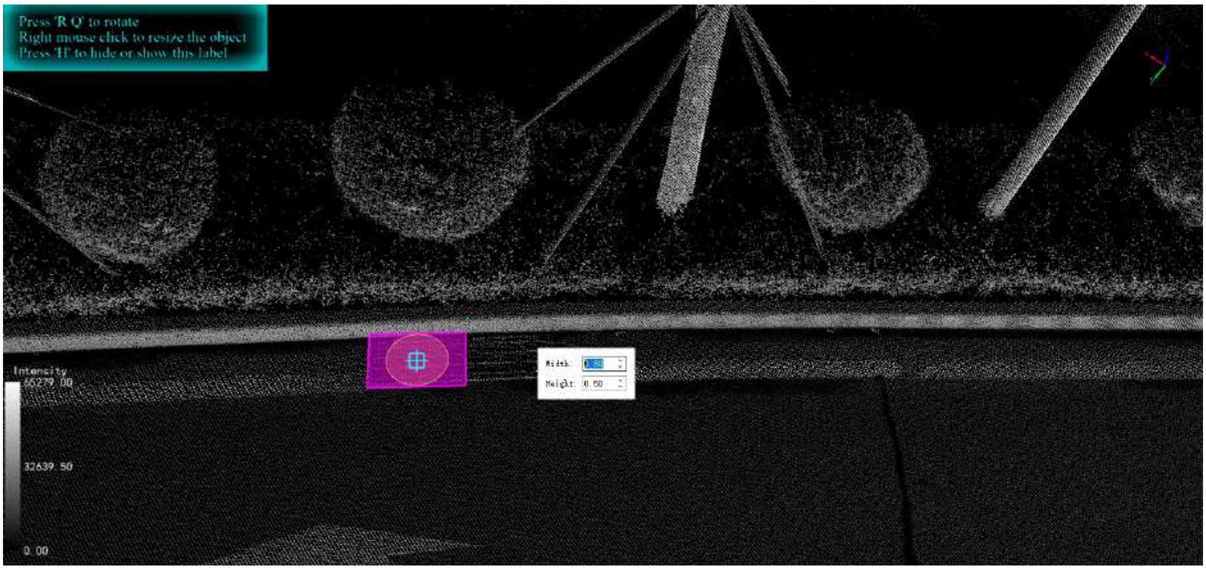
Function description : Manually add a drain grate.

> In order to obtain the plane position correctly, you 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 drain grate** button.

2.The graphics to be drawn will be displayed in real time and then the mouse, place the mouse in 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

Add manhole cover

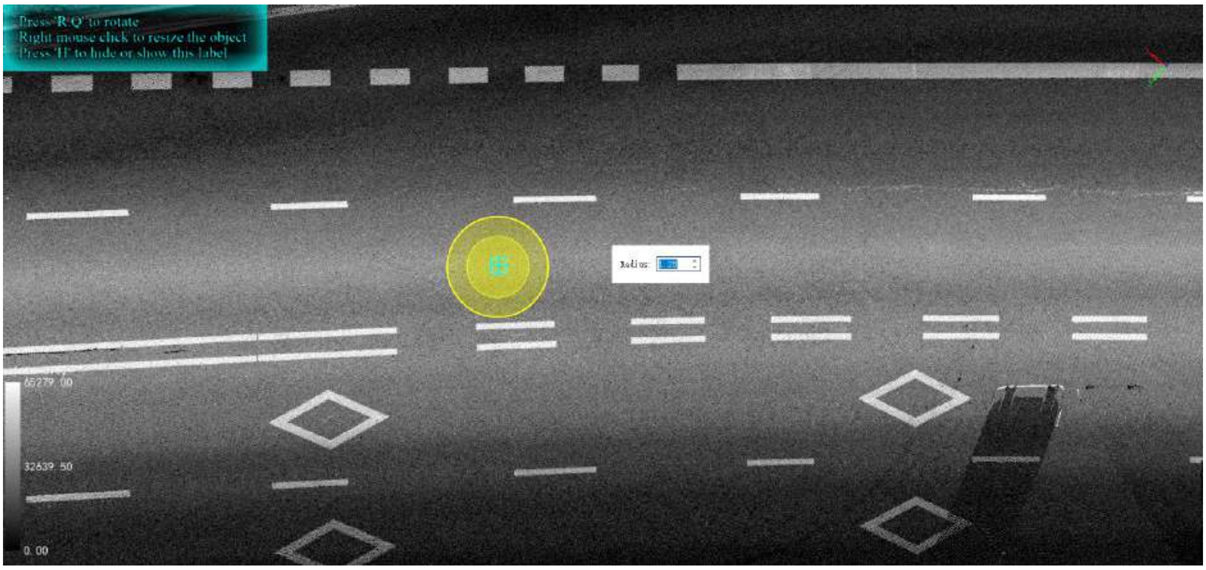
Function Description : Manually add manhole covers.

> In order to obtain the plane position correctly, you need to turn on the mouse hover

Step

1.In the Road Facilities panel click on the drop down sign on the right and select **Add manhole cover**  button.

2.The graphics to be drawn will be displayed in real time and then the mouse, place the mouse in 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:

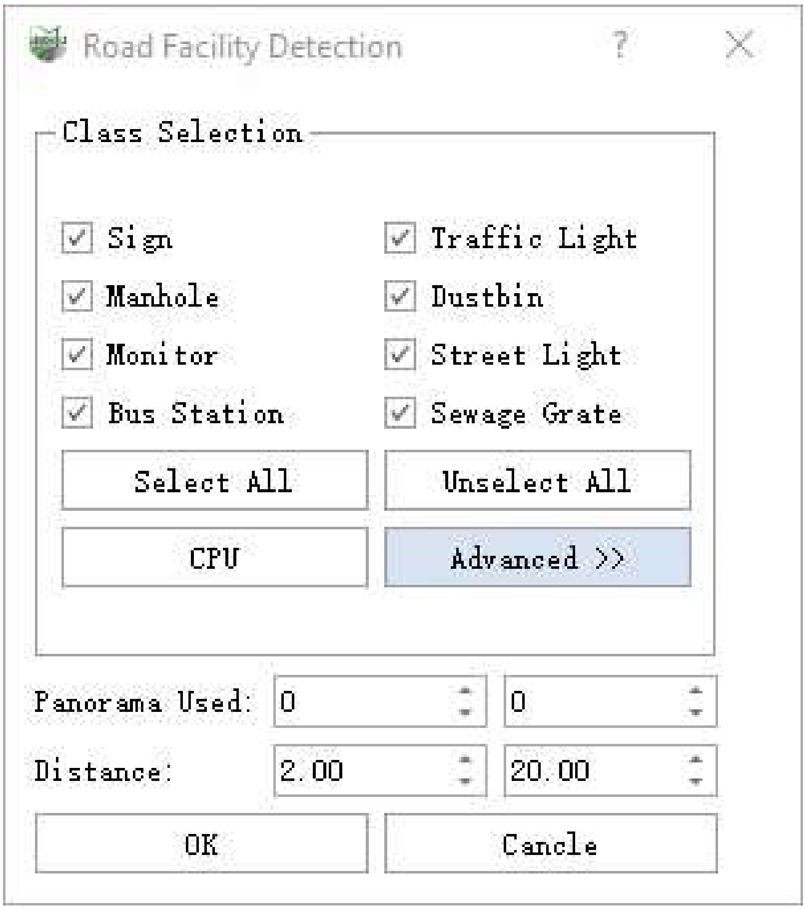
Shortcut key name	Function
r	Rotate counterclockwise
q	Rotate clockwise
Right mouse button	Change the size of the vector template

Batch processing of road facility recognition based on panoramic images

Function Description: 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, pop-up box：



Parameter dialog box

Parameter description:

- **Category selection**: Displays the extraction categories supported by the current function, and you can check/uncheck the extraction categories 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.

- **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.

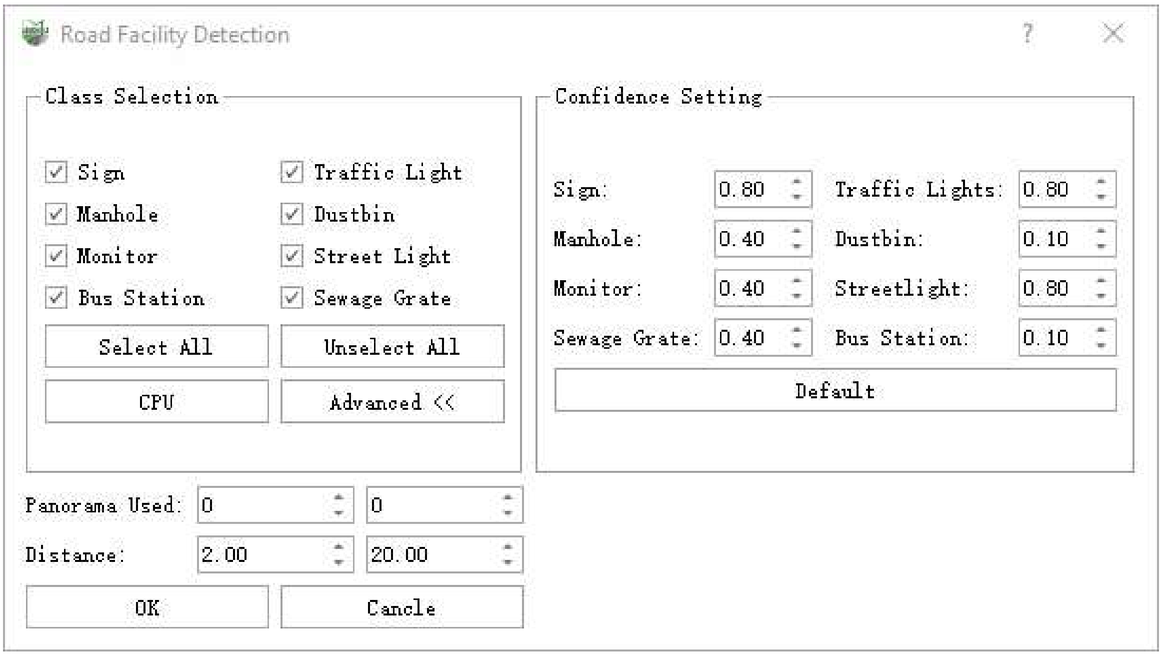
- **Use image**: Used to determine the picture number used in this processing, the default starts from 0, and the last picture in the project ends

  > 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**.

  > The appropriate distance can help reduce incorrect extraction. It is recommended to use the default value

- **More**: Show/close the confidence setting window

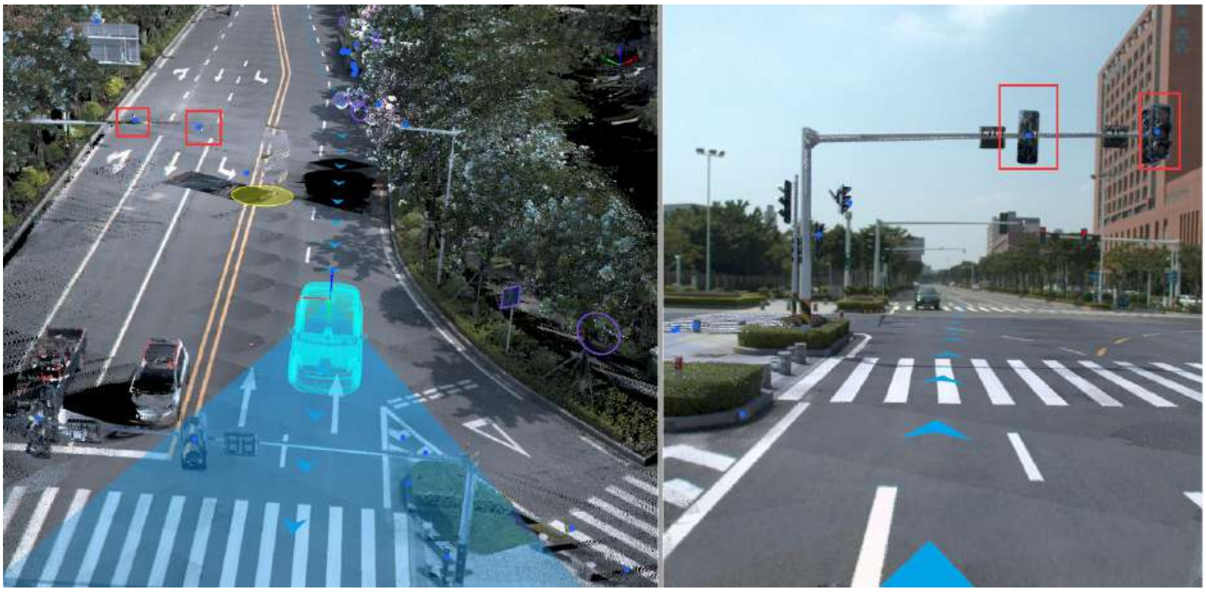


More dialogs

  - **Confidence setting**: For the categories supported by the current function, set the corresponding confidence level s eparately. 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

Strip extract

Function Description: Banded 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 use manual or semi-automatic detection and extraction.

Template editing and selection operations

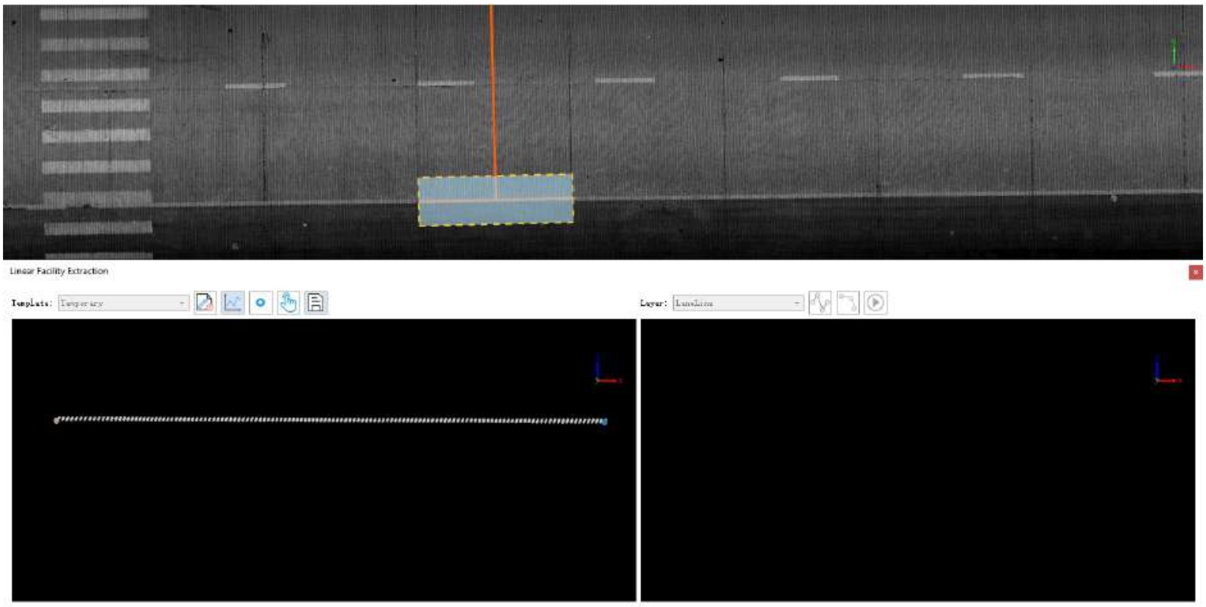
- **Template view**: The template view is located on the left and is used to display the currently applied template point cloud and key points.
- **Template selection**: When the function starts, the program automatically loads the template file located in the installation directory and places it in the template selection box. The user selects the template that matches the detected object.
- **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：

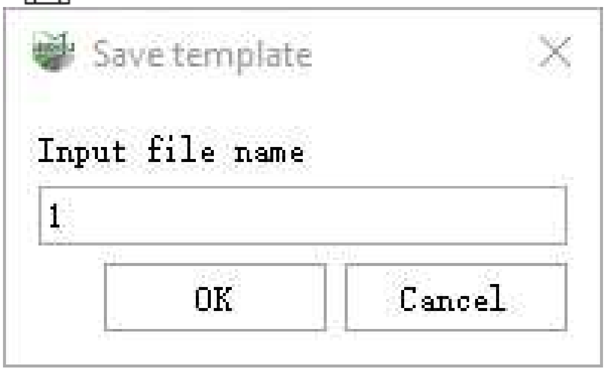
1.In road facilities In the panel, click **Strip extract** button, open the editing mode.

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, the save template dialog box pops up, set the file name, and click OK.



5.Click **to complete the edit**  button to close editing.

Strip extraction operation

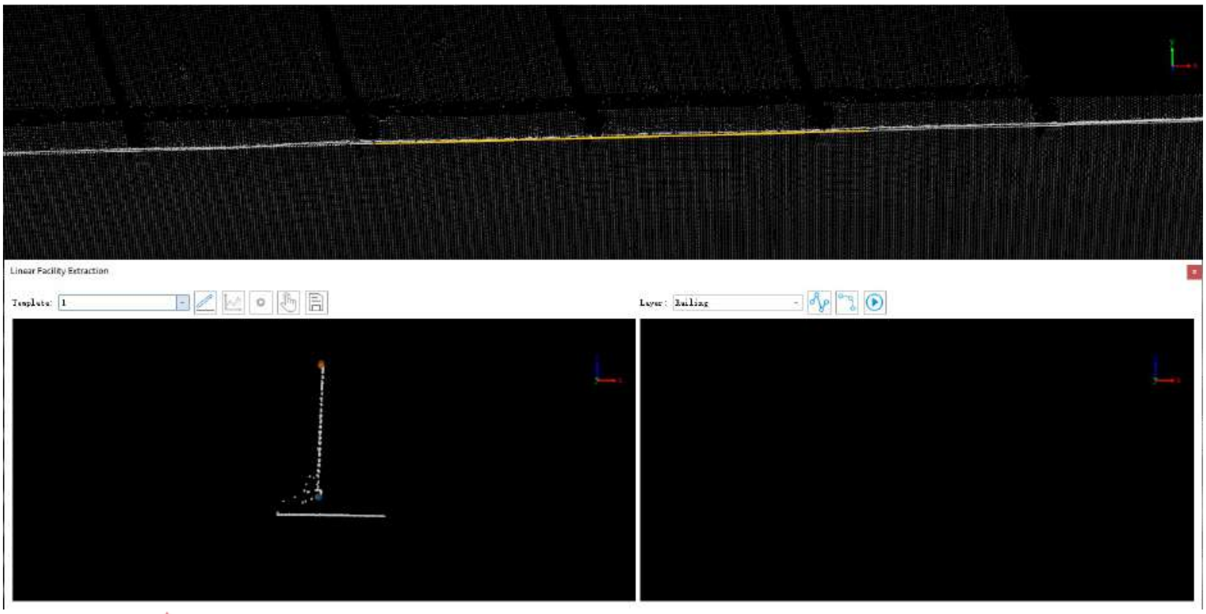
- **Result view**: The result view is located on the right, which is used to display the results of each extraction, and supports adjustments to the key points of the extraction results.
- **Layer selection**: Select the result to save the layer (line layer).
- **Draw a straight line**: After activation, draw a straight line along the strip-like object in the 3D view, 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 the'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 1.5s.If the detection result does not meet the requirements, you 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, Enter can be used to end.

Operation steps：

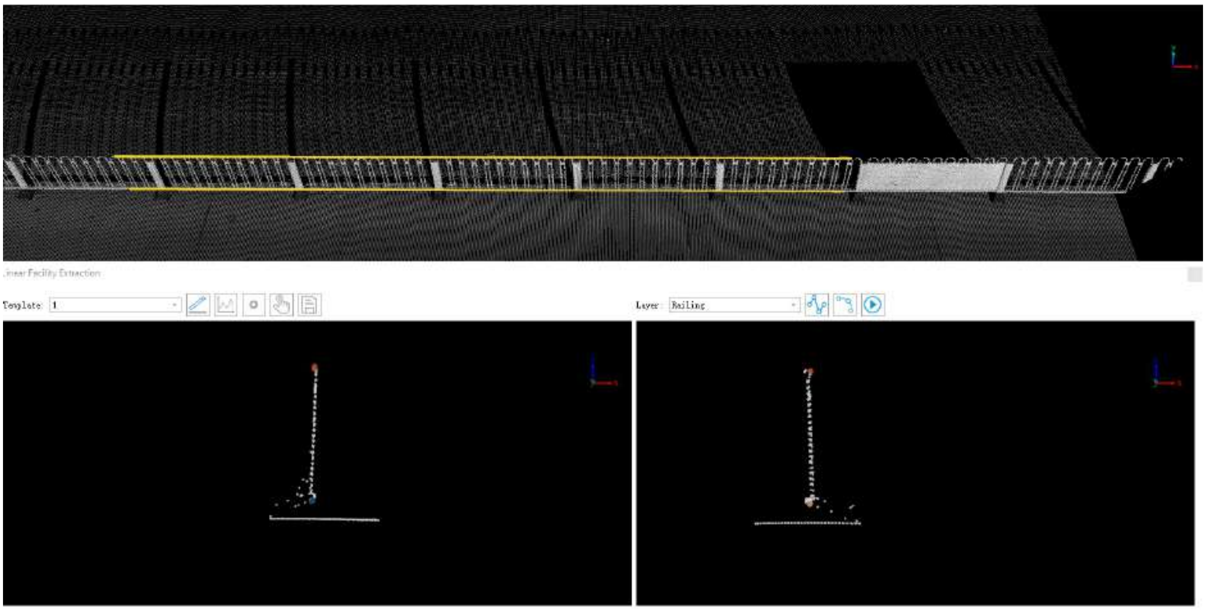
1.Select the template in the template drop-down box on the left side of the strip object extraction, and select the layer you need in the layer on the right.

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, can be automatically detected.

> You 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

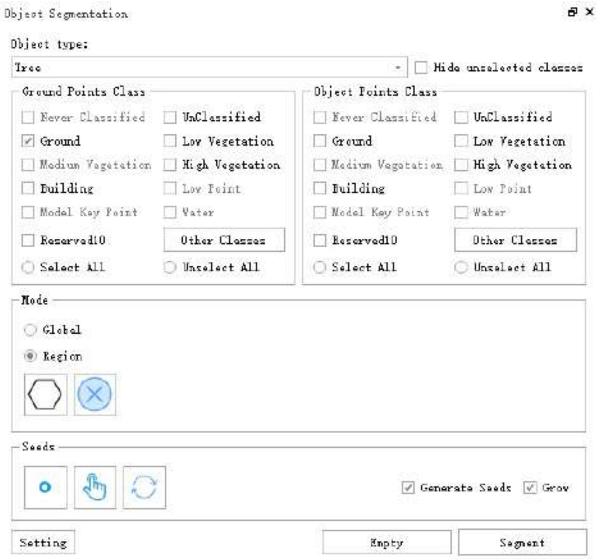
- Monomer segmentation
- Individual Editor
- Vertical Ground Parameter Extraction

Monomer segmentation

Function Description: Supports the segmentation and removal of two types of monomers, tree and rod, which will be described in detail below.

Step

1.Click **Monomer split** button, pop up the pop-up box：



Monomer segmentation

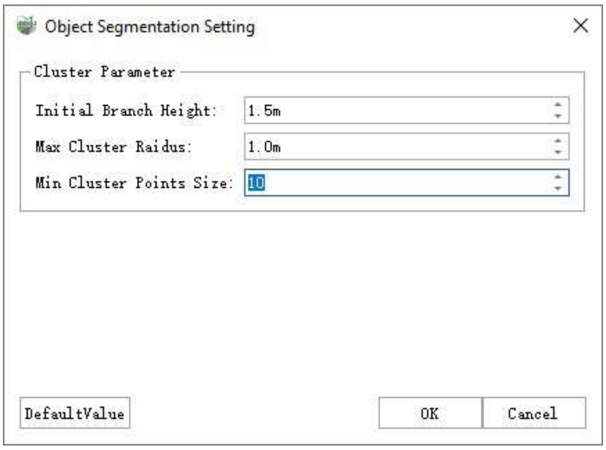
- **Type selection**: Both tree and rod types can be supported currently.
- **Hide unselected categories**: After checking, the unselected category points will be hidden and can be used with any rendering method to improve the convenience of seed point operation and point cloud observation.
- **Ground point category**: 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).
- **Monomer category**: The category of the monomer to be divided (for example, when the tree is divided, check the category of the tree point).
- **Mode selection**: 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.
- **Seed point**：
  - 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

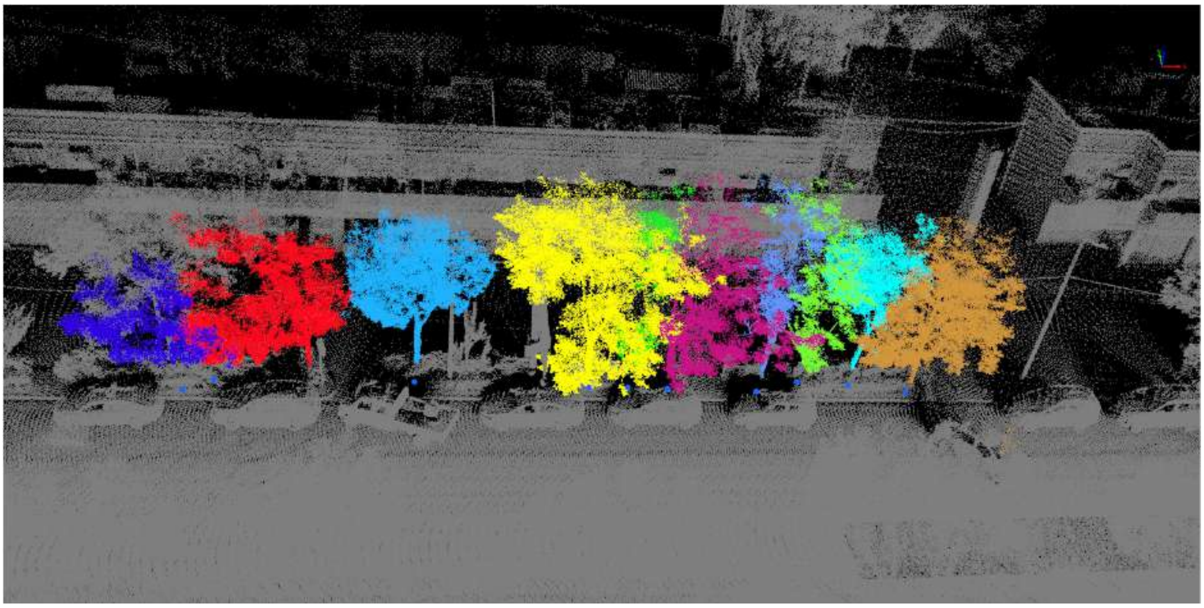
> Tip: When adding and moving, you can use it in combination with [Hide unselected categories] and [Quick horizontal section] for easier operation.

- **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.



  - The initial trunk height (only valid for tree segmentation), set to the minimum trunk height in the area to be divided.
  - Minimum number of clusters, minimum number of points for a single cluster.
  - Maximum clustering distance: the minimum distance between clusters.

2.After clicking **Split**, the split result is shown in the figure：



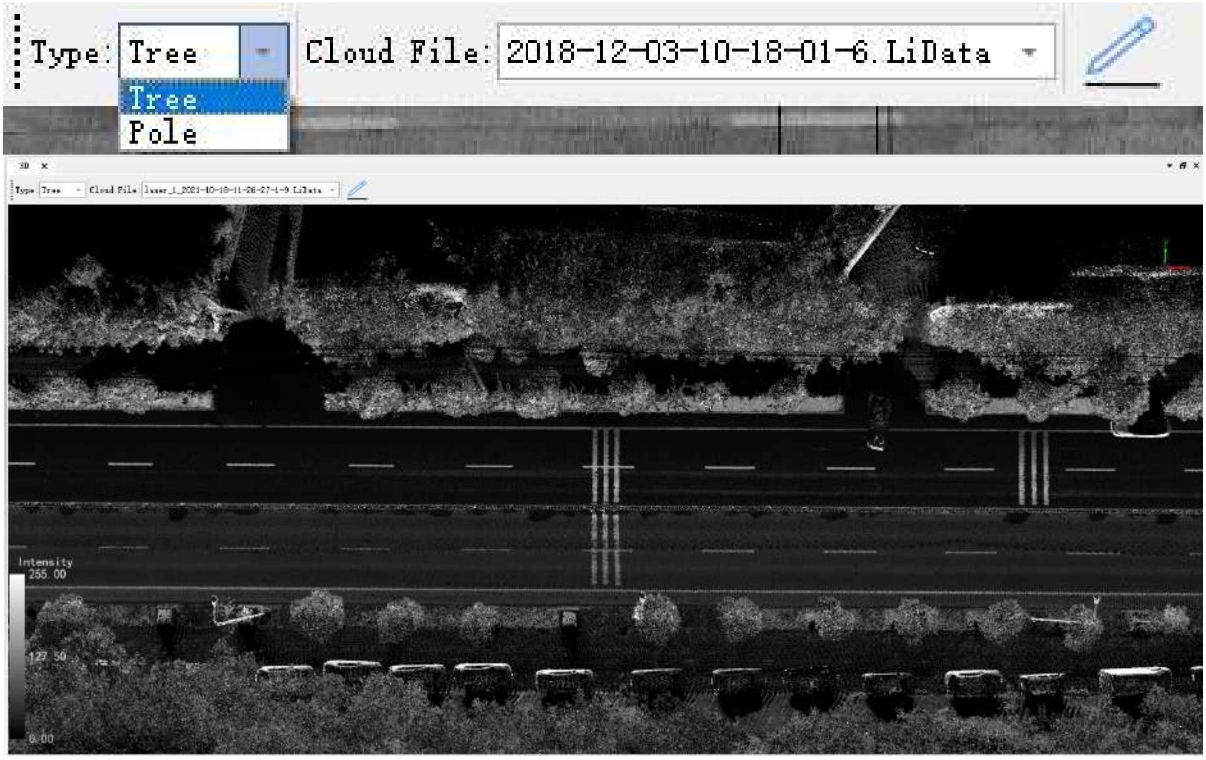
3.Click **Clear**, all objects of the specified type in the target area will be cleared, and the point object ID will be set to zero.

Monomer edit

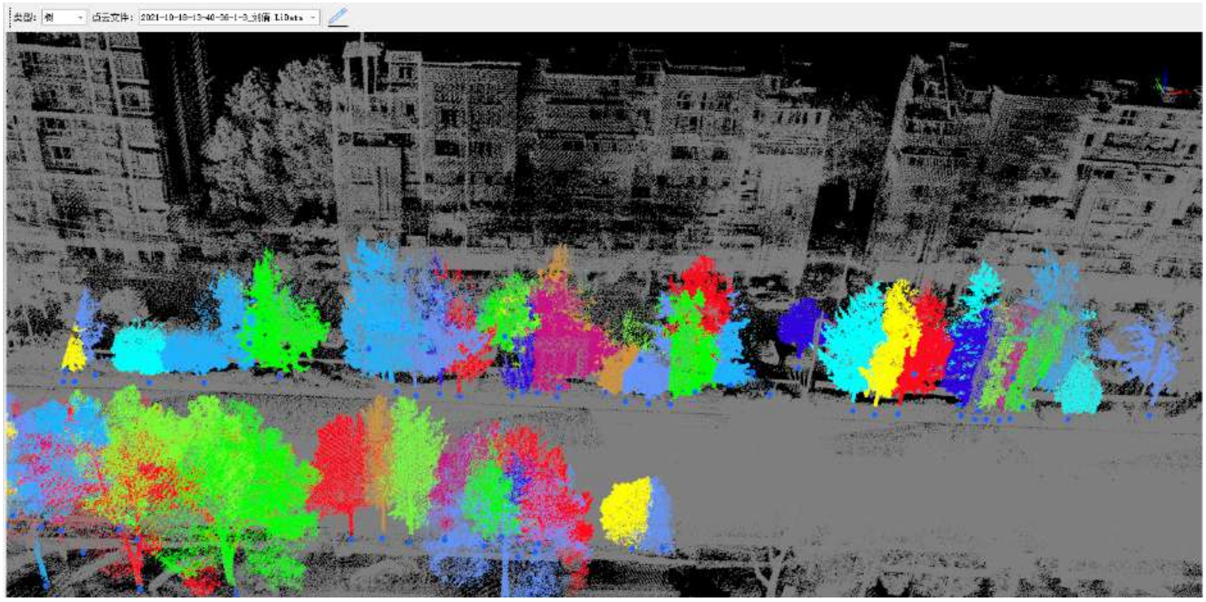
Function description : Supports manual editing of tree and Pole monomers, including the functions of creating, deleting, and setting monomers. The following will be described in detail.

Step

1.Click **Single 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.



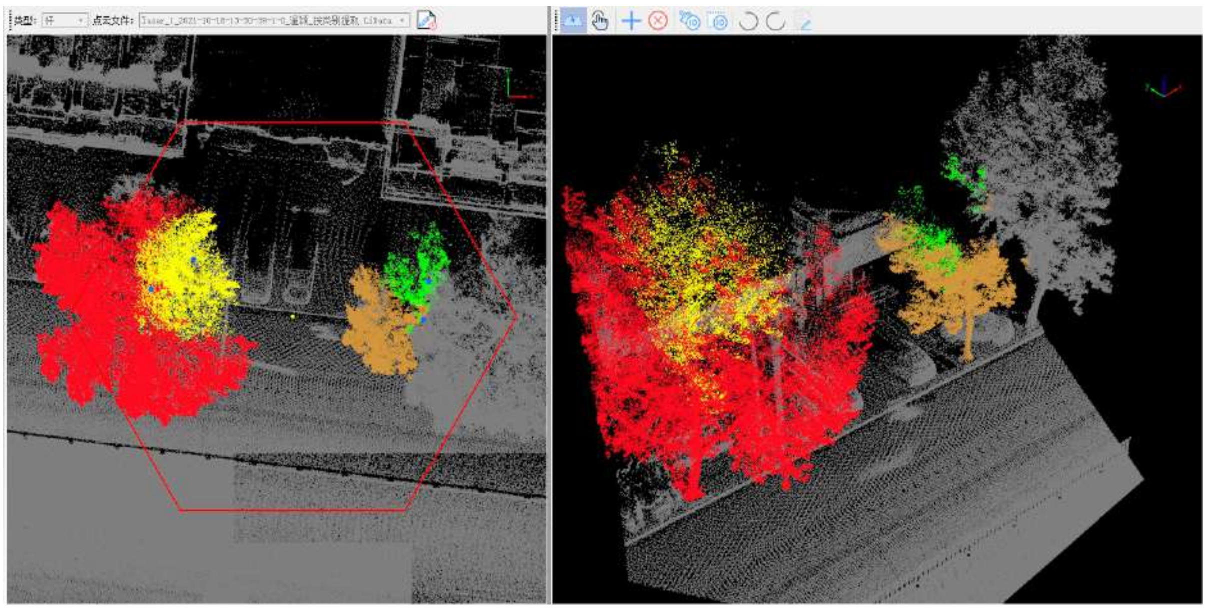
Before you start editing



After you start editing the tree

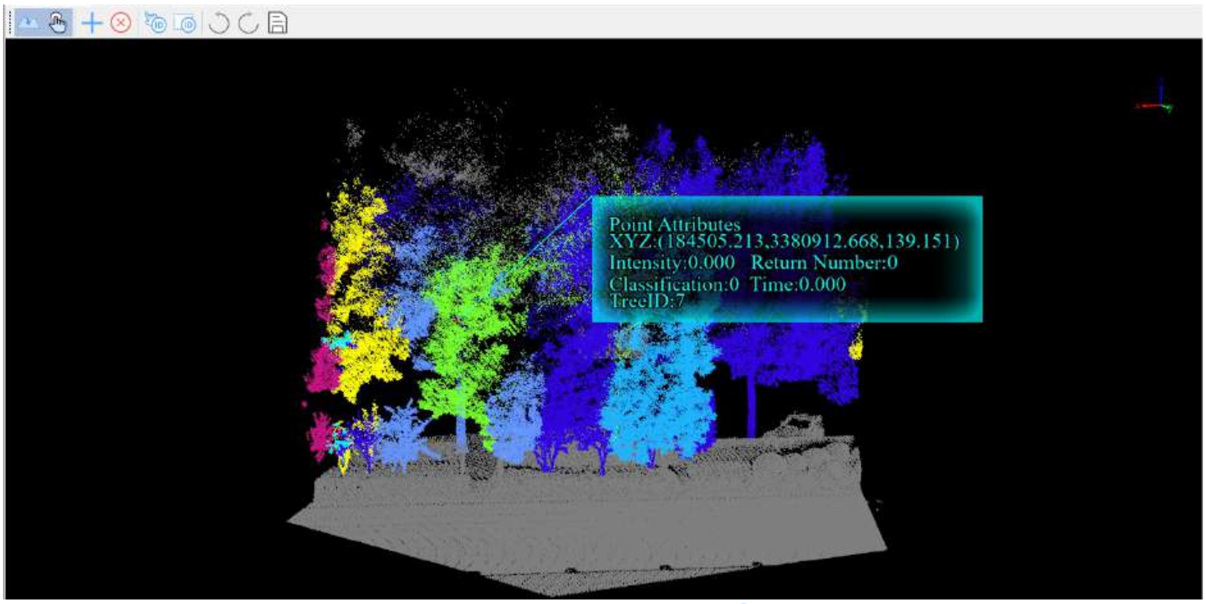
2.After starting editing, a single body editing window will pop up automatically, in which you can add, delete and reduce single body operations.

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.

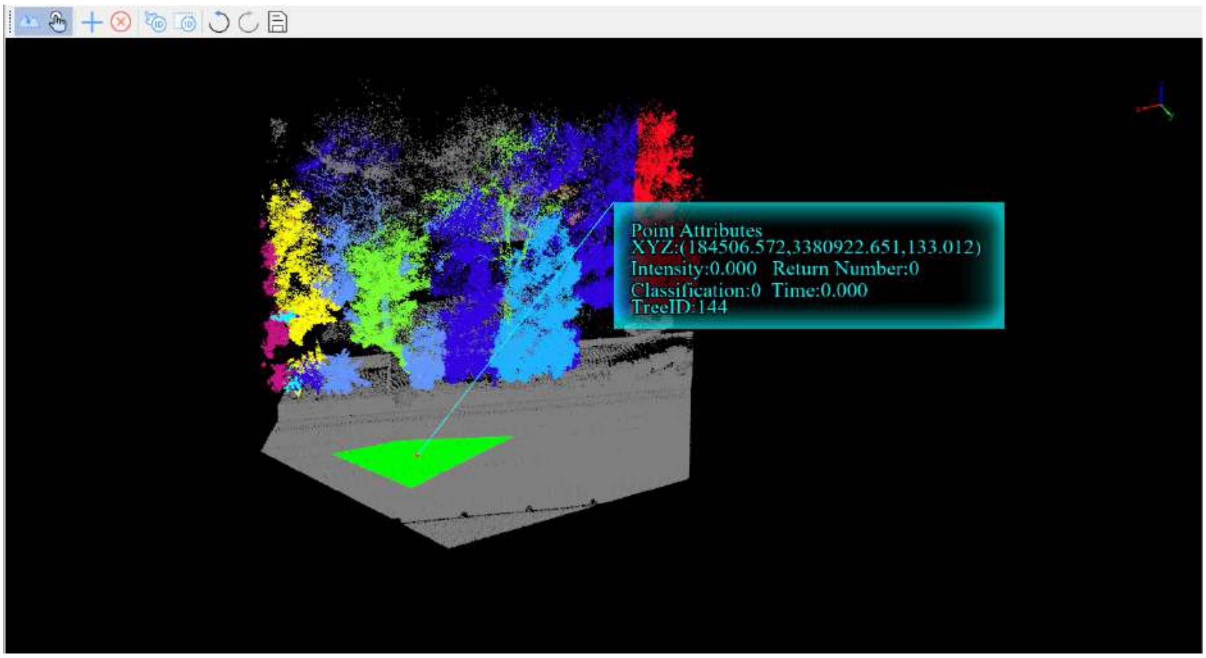


4.**Load Edit Area**:In the Single Edit window，Click **Load Edit Area**  button. Add the point cloud of the area to be edited to the memory for editing. Editing is not supported if it is not loaded.

5.**Pick Object**:In the Single Edit window，Click **Pick Object**  button. You can view the property information of the selected point, including 3D coordinates, GPS time, intensity, RGB, object ID and other information.



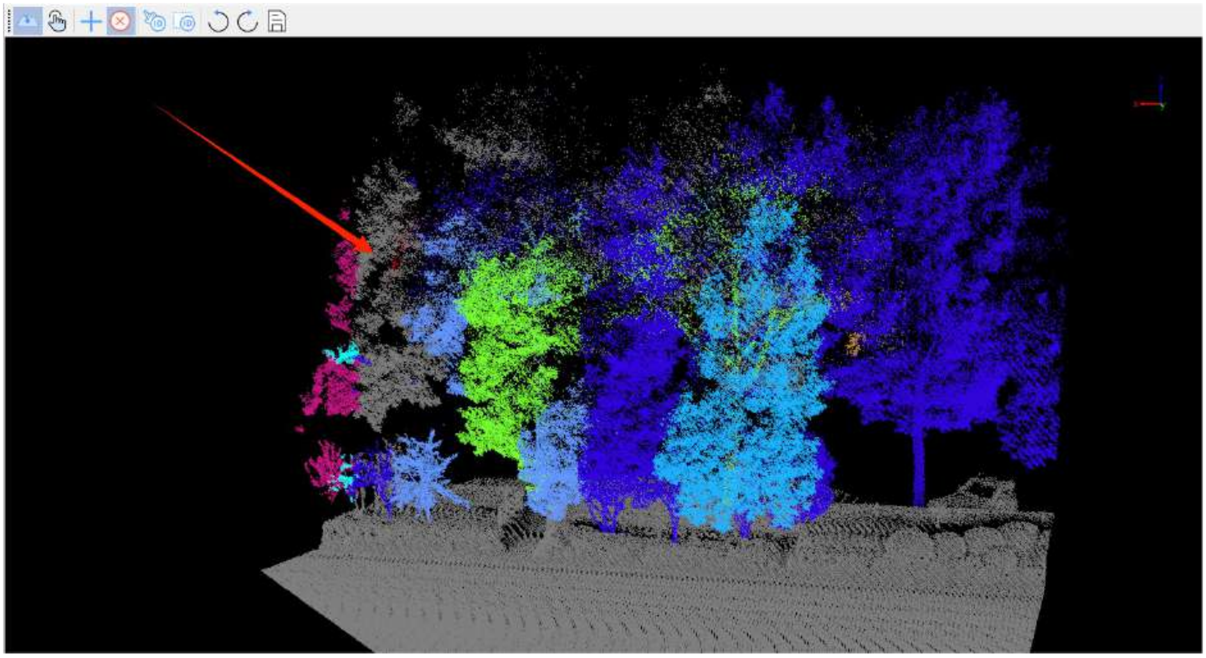
6.**Create Object**：In the Single Edit window，Click **Create Object** button.By boxing the point cloud where the single object is located to complete the creation of a single object, the point object ID in the boxed area will be set to the new object ID, and an object record is automatically added to the property table. To fine-tune the point cloud, you can use the [Point Set ID] and [Box Set ID] operations.



Newly built monoliths

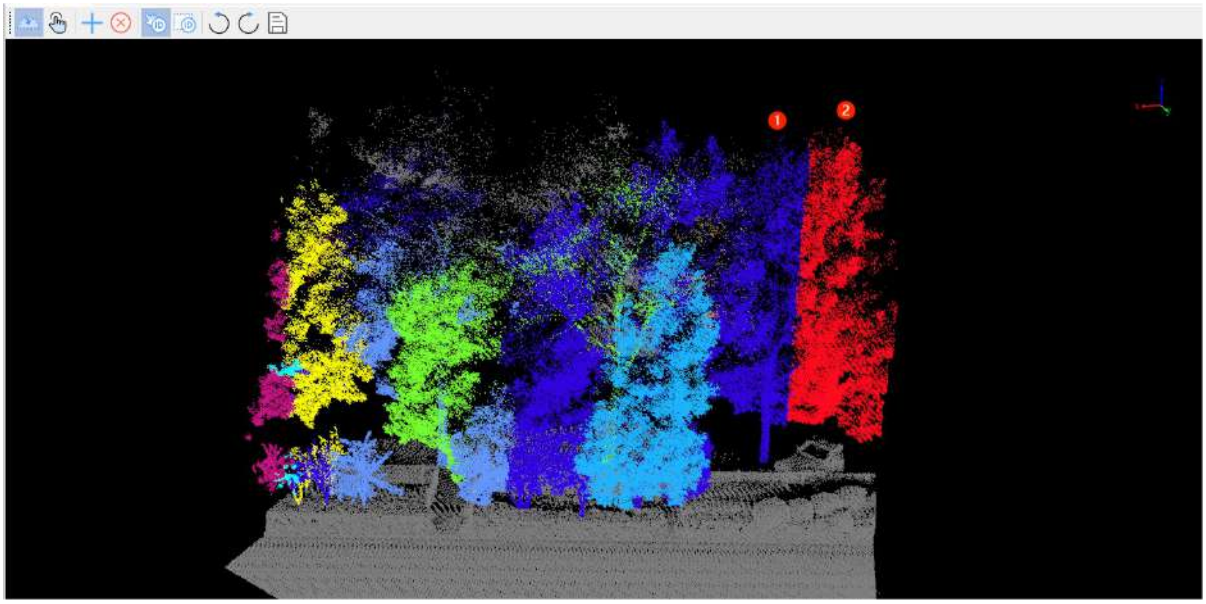
7.**Settings**：In the Single Edit window，Click **Settings** button.In the setting window, you can check the box to display only the points to be edited from two dimensions, category and single ID, to reduce the interference of the background point cloud to the editing process and improve the convenience 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.

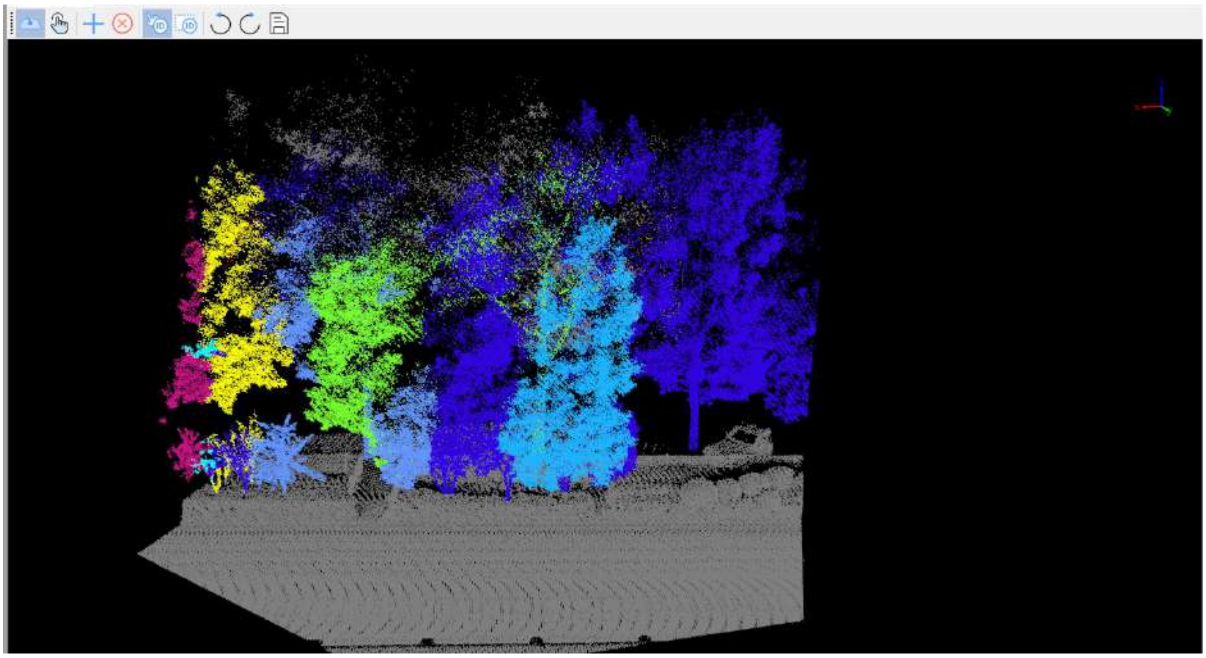


Detele Object

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. can be used to achieve the editing area of two single point cloud merge, a single point cloud delete.

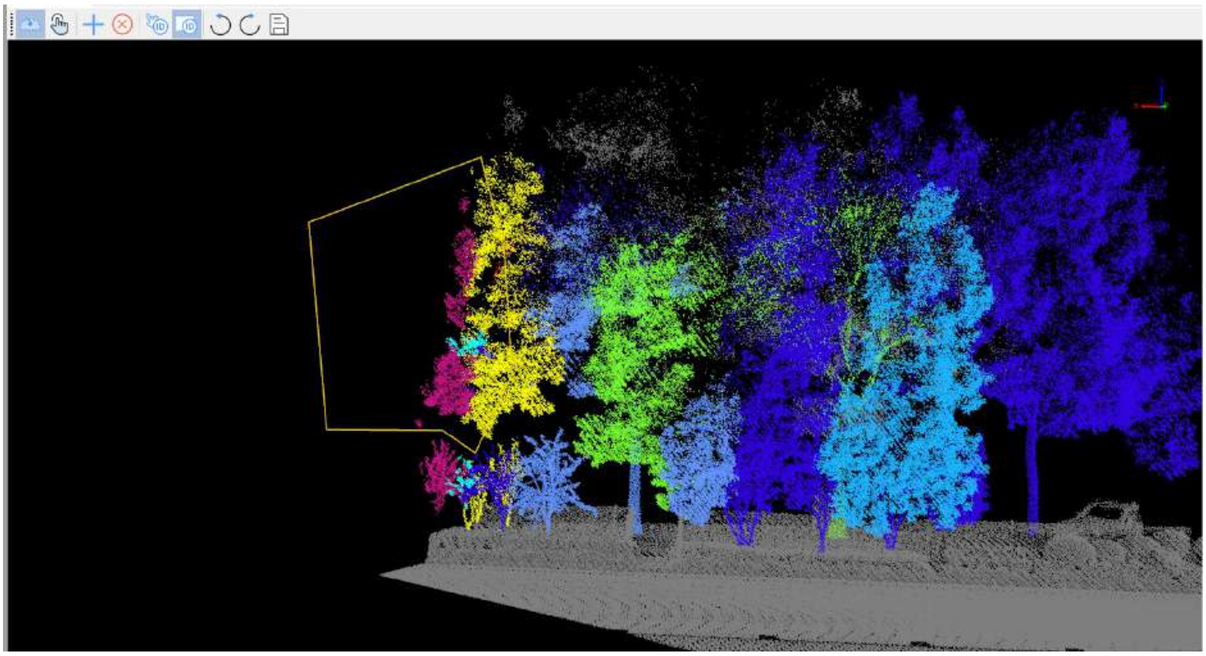


Before tapping Set ID

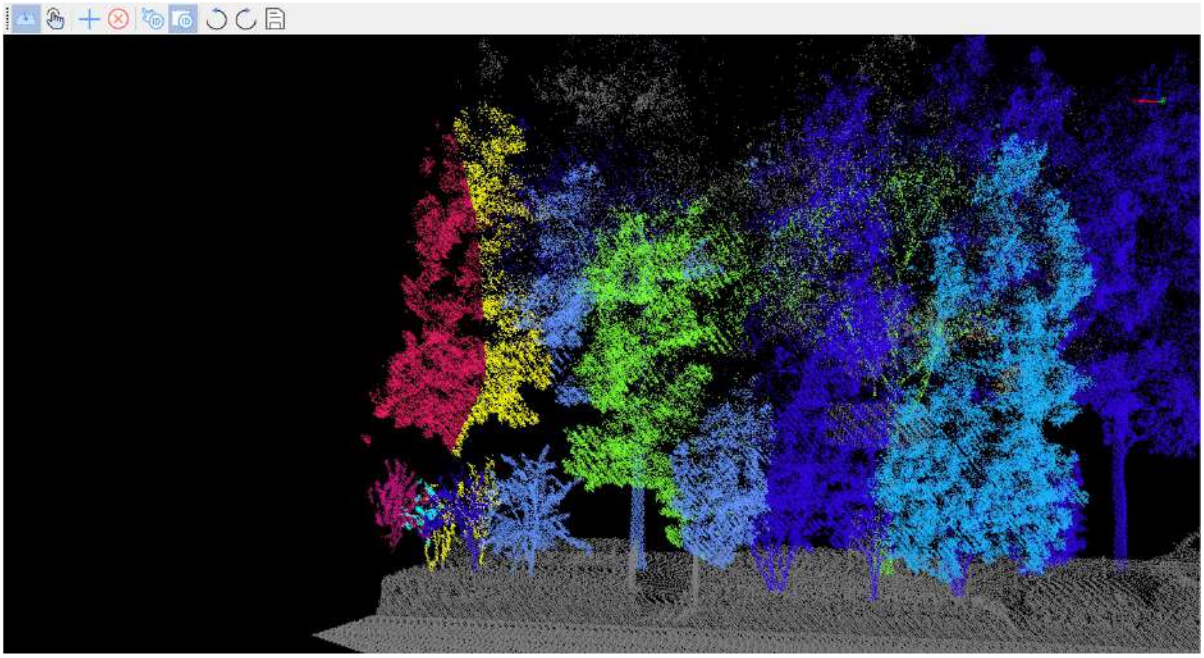


After tapping Set ID

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, support "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.

13.**Save**：In the Single Edit window，Click **Save** button. Save the edit to the file, the left 3D view will refresh the result, support "Ctrl+Shift+S" shortcut keys.

Vertical object parameter extraction

Function description : You can modify the vertical object parameters chest diameter, height, angle, and crown width.

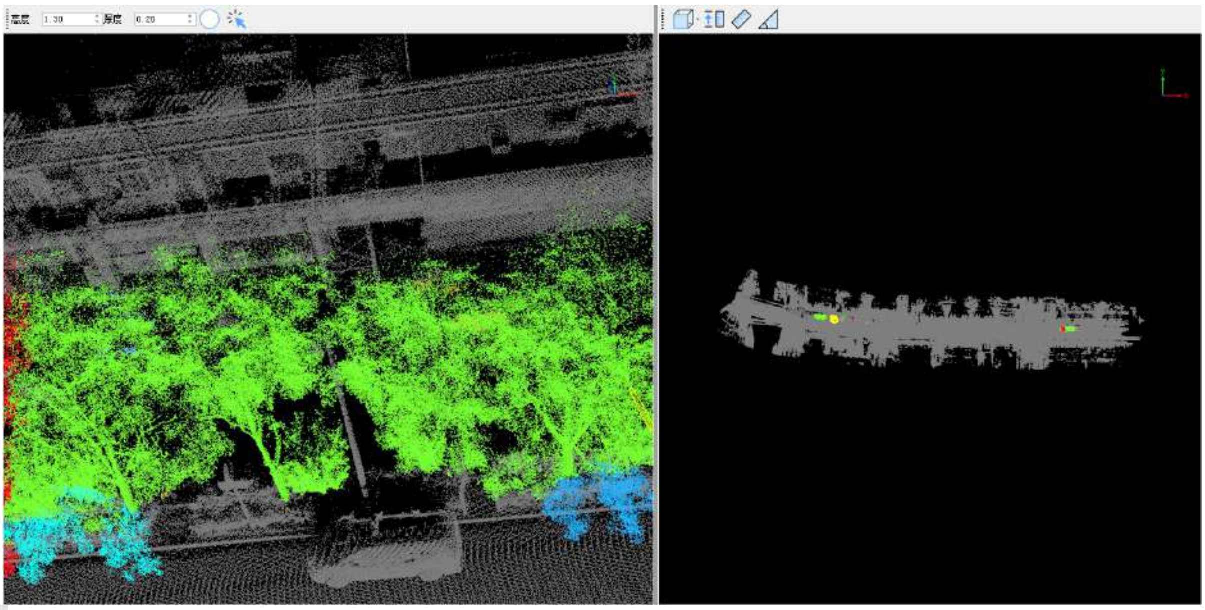
Steps

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.：

	Object ID	DBH(m)	Height(m)	Angle(°)	CrownWidth(m)
1	1	1.140	5.252	59.932	3.421
2	2	1.091	4.797	25.252	3.208
3	3	1.118	4.814	61.052	3.273
4	4	0.966	7.348	13.663	3.721
5	5	0.962	4.317	75.557	2.692

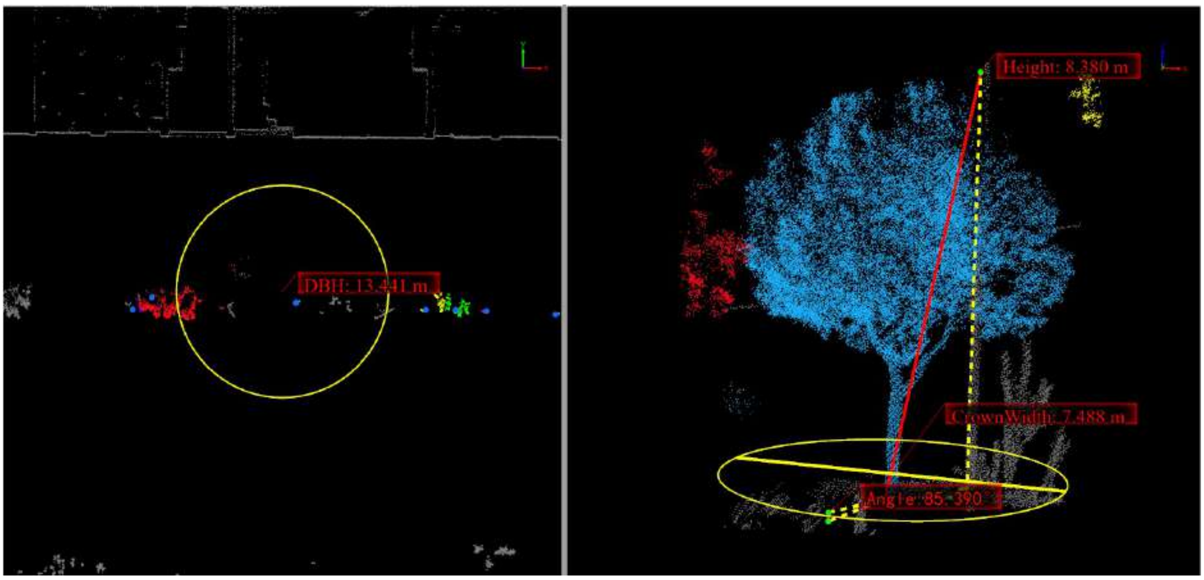
> Note: Clicking on the white box in the upper left corner is the full menu body ID.

2.Click **To extract the vertical object parameters** button, enter the function interface：



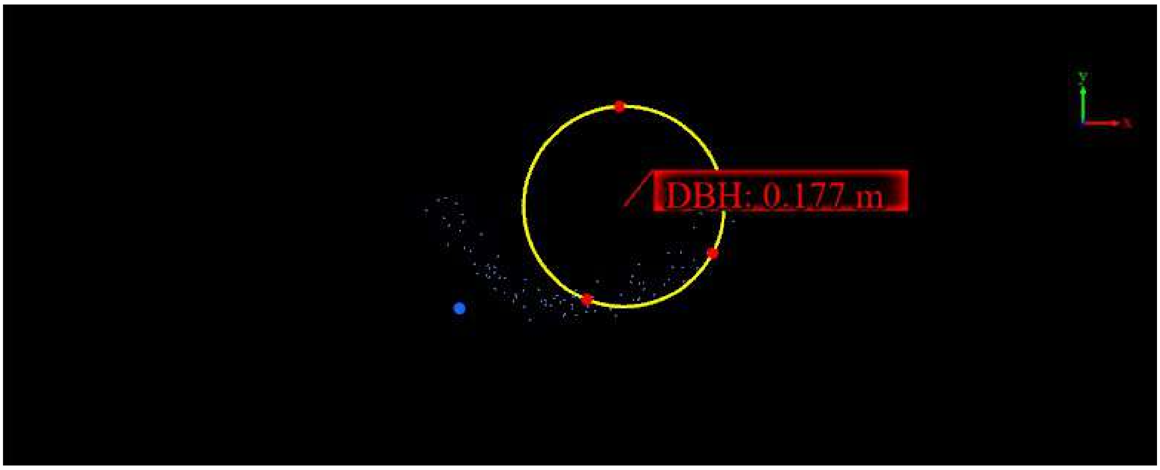
> Note: You must first complete the monomer segmentation and editing operations.

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.

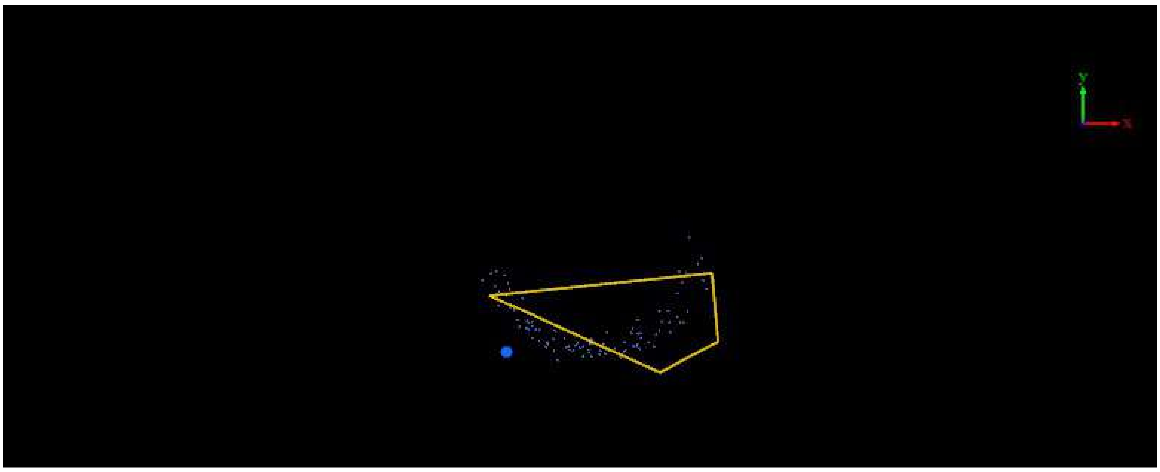


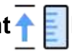
4.**DBH Measurement**：There are two ways to refit the DBH:

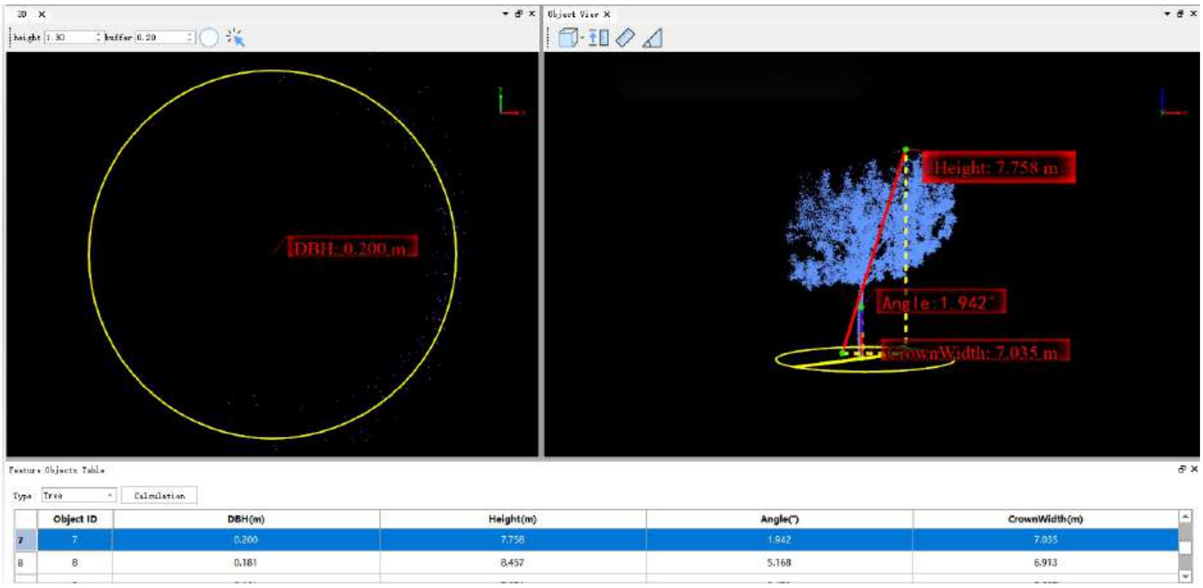
- **3DView on the left**: Display DBH parameters: First click on the monomer ID you want to view, and then click **DBH three-point fit** button, click three points with the left mouse button to determine the fitting surface you want to view, as shown in the figure:



- **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.

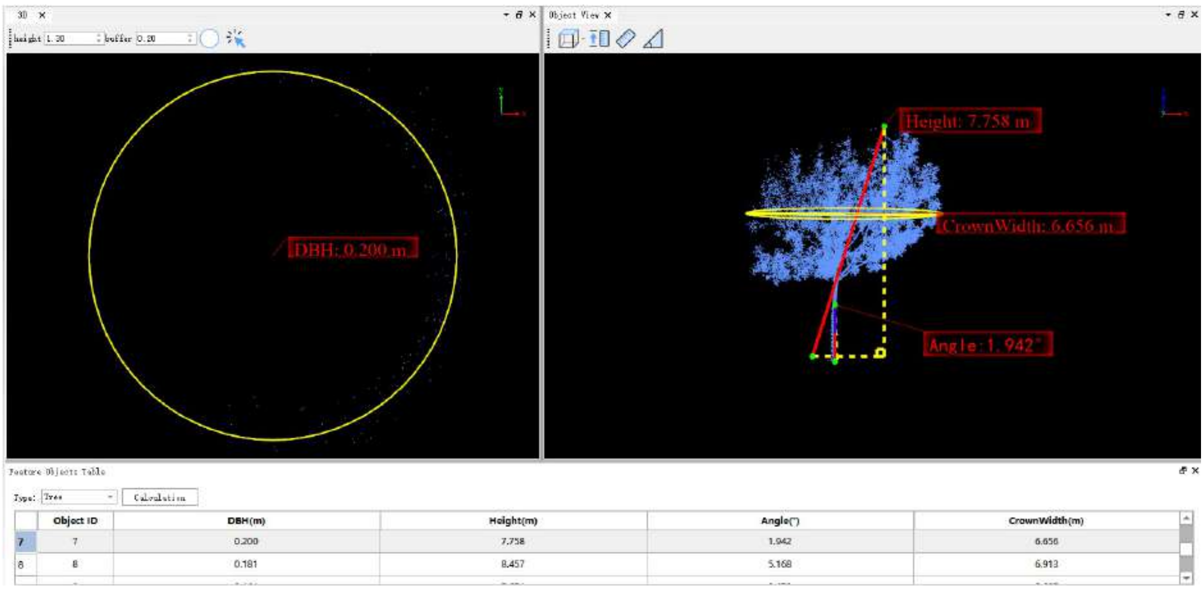


5.**Height measurement**:In the Object View window, click **Height Measurement**  button，Using a two-point approach, the height of the monolith is re-measured on the point cloud and the new measurements are updated in real time in the corresponding list of monolith tables.

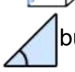


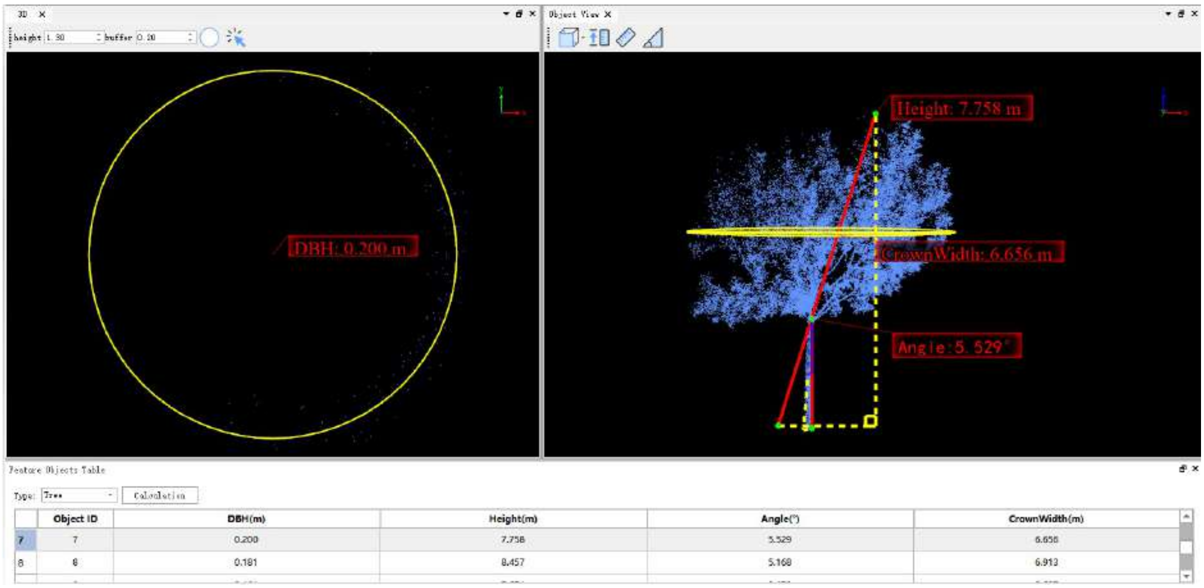
Value after height re-measurement

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 Editing

Vector editing supports modules such as vector drawing, element editing, and node editing.

Draw Vector

Edit Element

Edit Node

Draw

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

- Draw Point
- Draw Multiple Points
- Draw Line
- Draw Segments
- Draw Arc
- Draw Polygon
- Draw Rectangle
- Draw parallelogram
- Draw Circle
- Draw Hexagon
- Draw Octagon

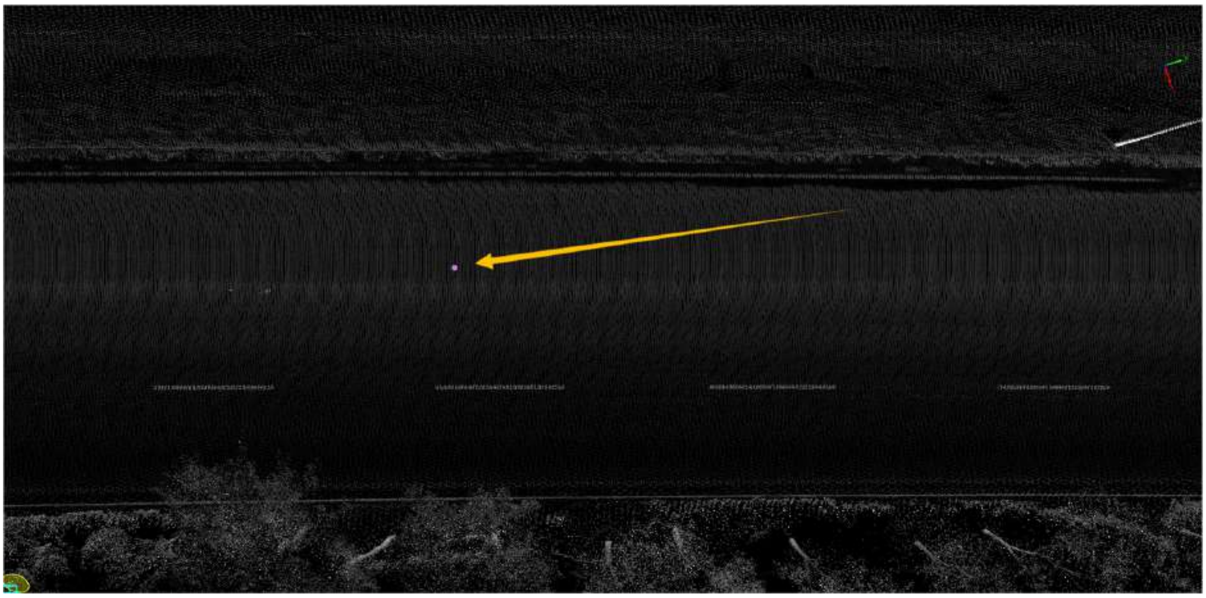
Add point

Function Description：Add vector point objects.

Steps

1.Select a point layer, such as a surveillance camera point layer

2.Click **Add Point** in Vector Edit button, 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 meaning of layers, see Layer Management

Add Multiple Points

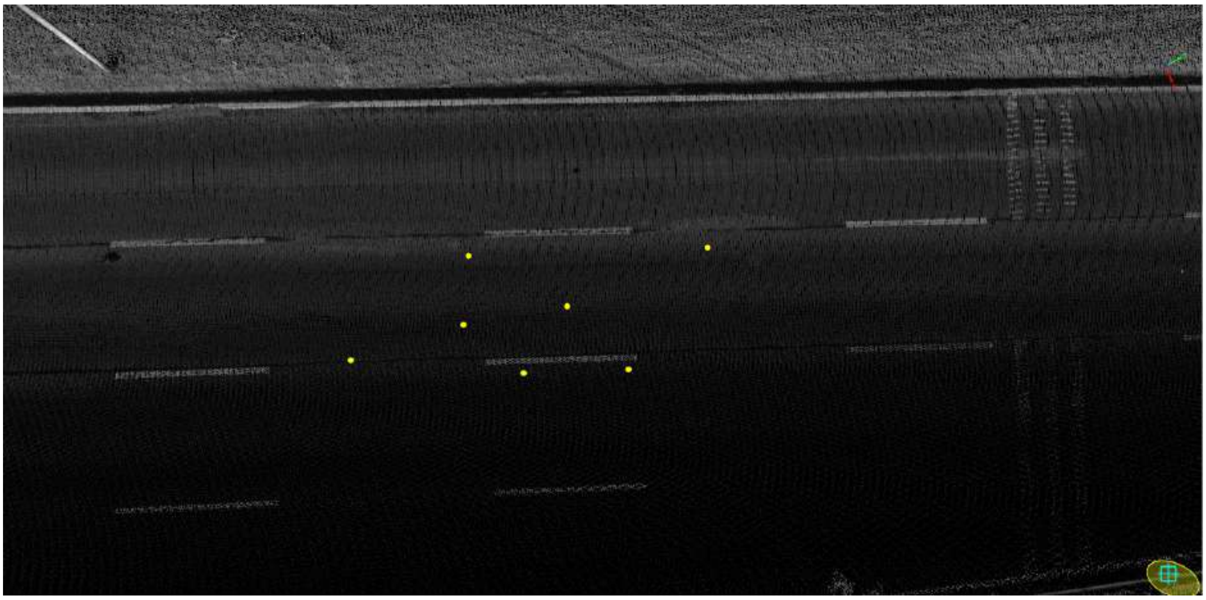
Function Description：Add vector multipoint objects.

Steps

1.Select a point layer, such as a surveillance camera point layer.

2.Click **Add MultiPoint** in Vector Editing button, 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 Multipoint 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 meaning of layers, see Layer Management

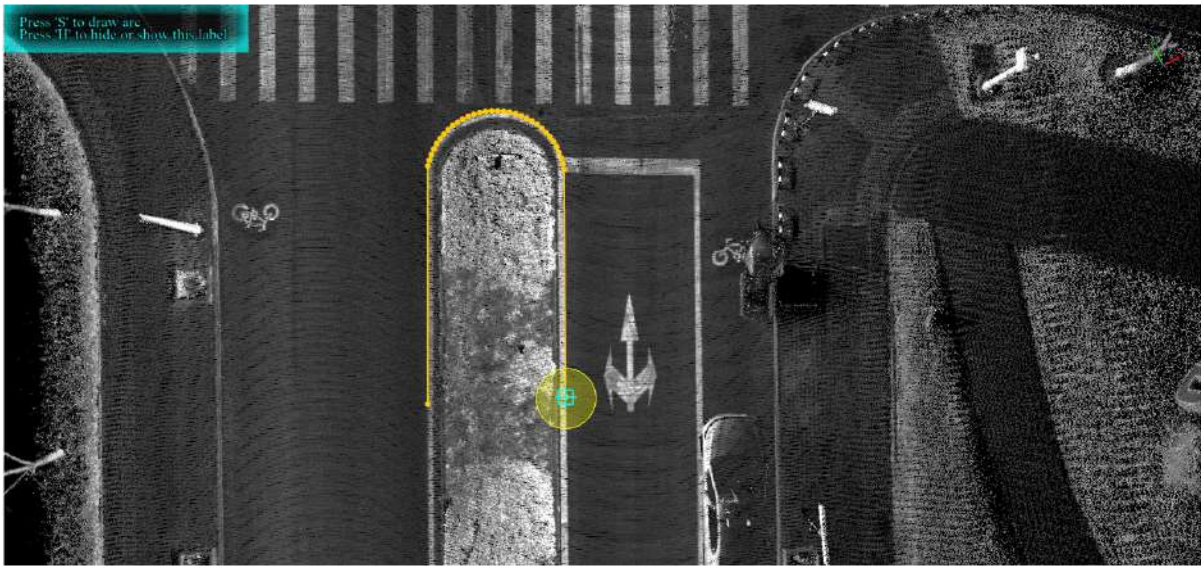
Add Line

Function Description：Add vector line objects.

Steps

1.Select a linear layer, such as a lane line layer.

2.Click **Add Line** in Vector Editing  button. 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 meaning of layers, see Layer Management
> - The right mouse button and shortcut key are the same as those of map elements during drawing，seeRoad Lane line

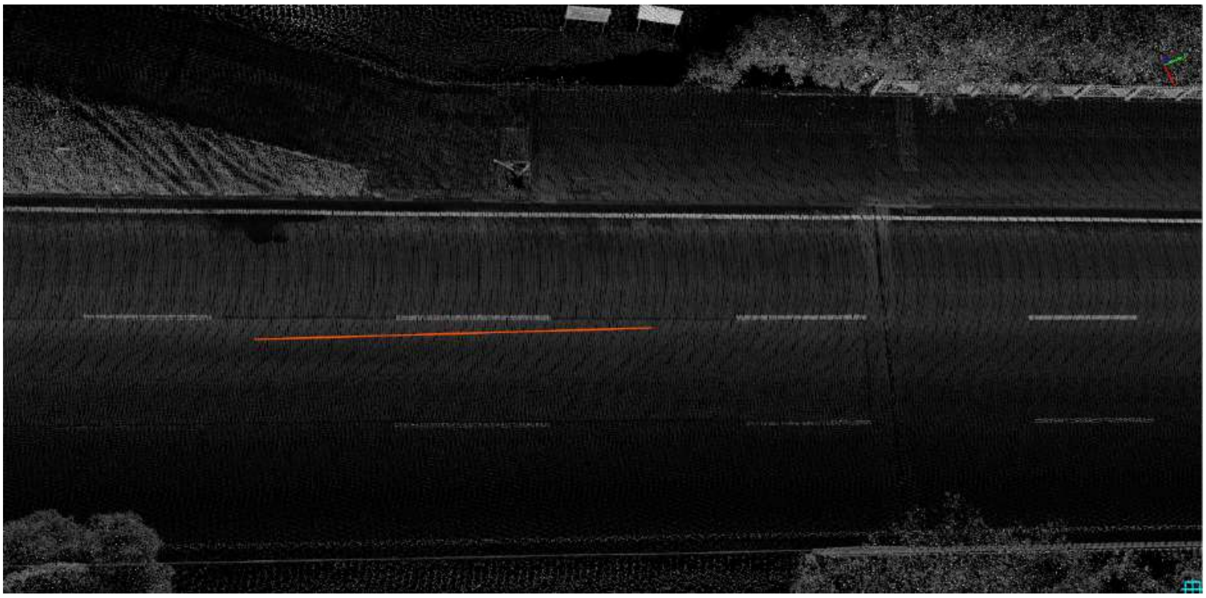
Add Segment

Function Description：Add a vector line segment object.

Steps

1.Select a linear layer, such as a lane line layer.

2.Click **Add Line Segment** in Vector Editing  button. 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 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 meaning of layers, see [Layer Management]

Add Arc

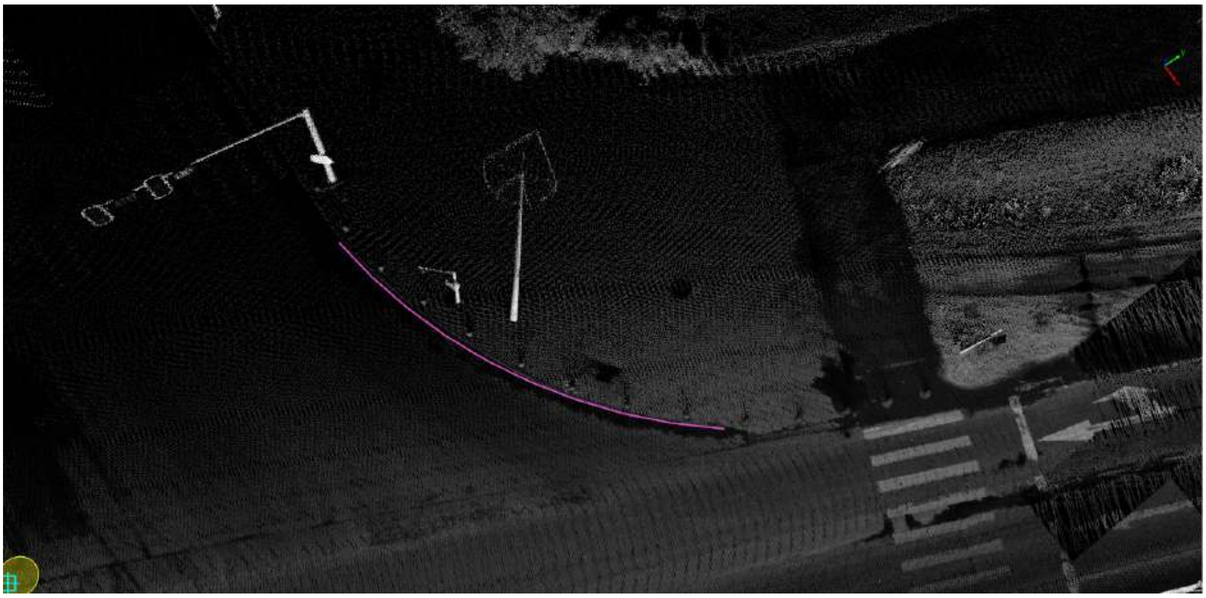
Function Description：Use the three key point method to add arc-shaped vector objects.

Steps

1.Select a surface layer, such as a surface facility layer.

2.Click **Add Arc** in Vector Editing  Button

- 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 meaning of layers, see Layer Management

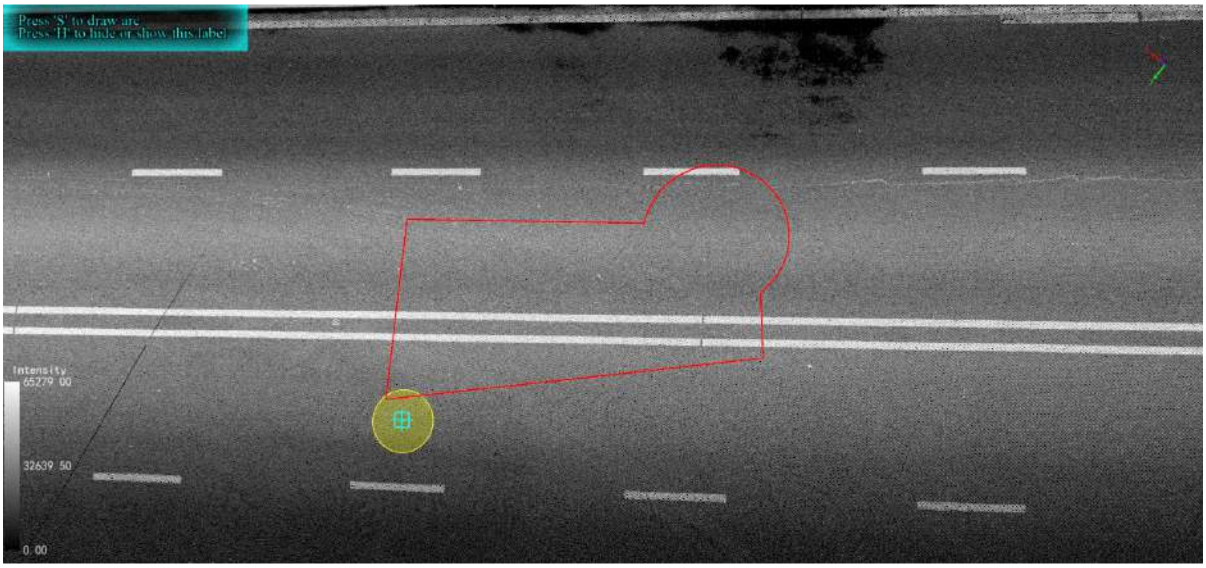
Add Polygon

Function Description：Add vector polygon objects.

Steps

1.Select a surface layer, such as a surface facility layer

2.Click **Add Polygon** in Vector Editing  After the button, use the mouse to click multiple points in succession to confirm the nodes of the polygon, and double-click the left mouse button to end the drawing.



Draw polygon interface

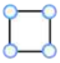
> Note:
>
> - The geometric characteristics of the vector result must be consistent with the geometric characteristics of the layer to be added. For example, the point-like vector result must be added to the point-like layer, otherwise it will be prompted that it cannot be drawn. For the meaning of layers, see Layer Management.
> - The right mouse button and shortcut key are the same as those of map elements during drawing，seeRoad Lane line

Add Rectangle

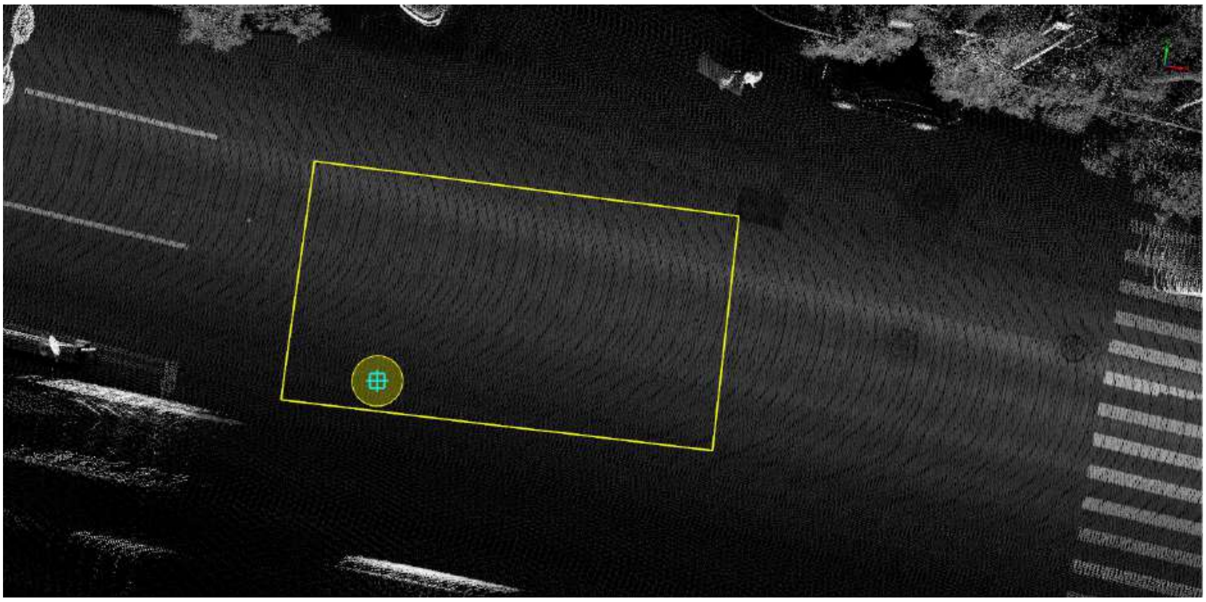
Function Description：Use the three key point method to add vector rectangular objects.

Steps

1.Select a surface layer, such as a surface facility layer

2.Click **Add Rectangle** in Vector Editing  Button:

- 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

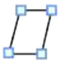
> Note: The geometric characteristics of the vector result must be consistent with the geometric characteristics of the layer to be added. For example, the point-like vector result must be added to the point-like layer, otherwise it will be prompted that it cannot be drawn. For the meaning of layers, see Layer Management

Add parallelogram

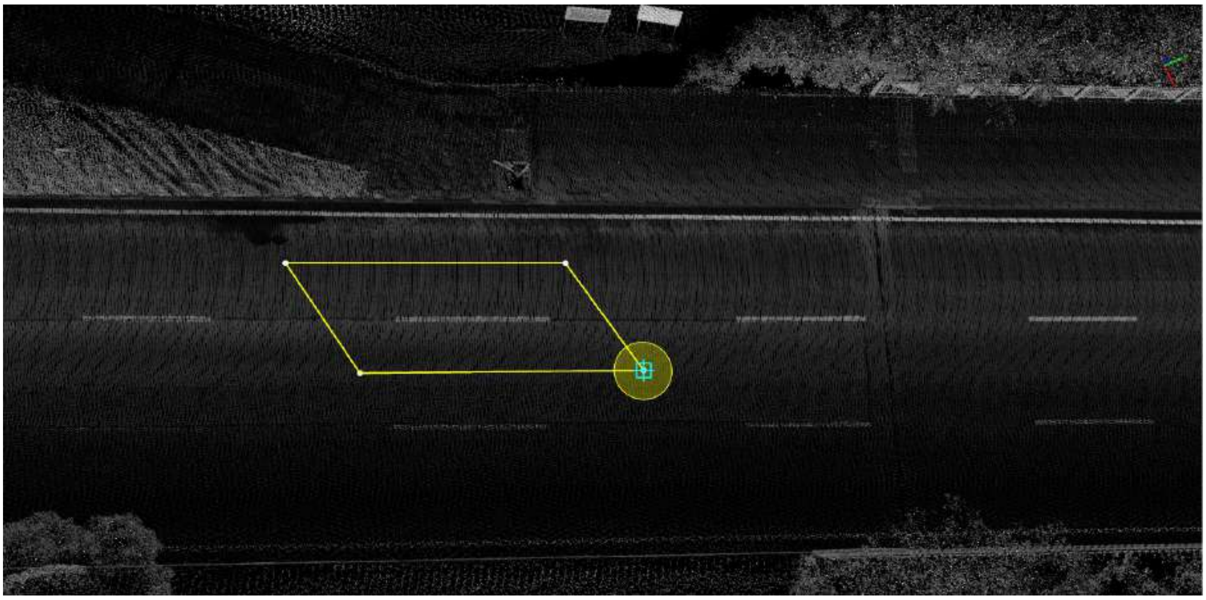
Function Description: Use the three key point method to add a parallelogram vector object.

Steps

1.Select a surface layer, such as a surface facility layer

2.Click **Add Parallelogram** in Vector Editing  Button:

- 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 meaning of layers, see Layer Management

Add Circle

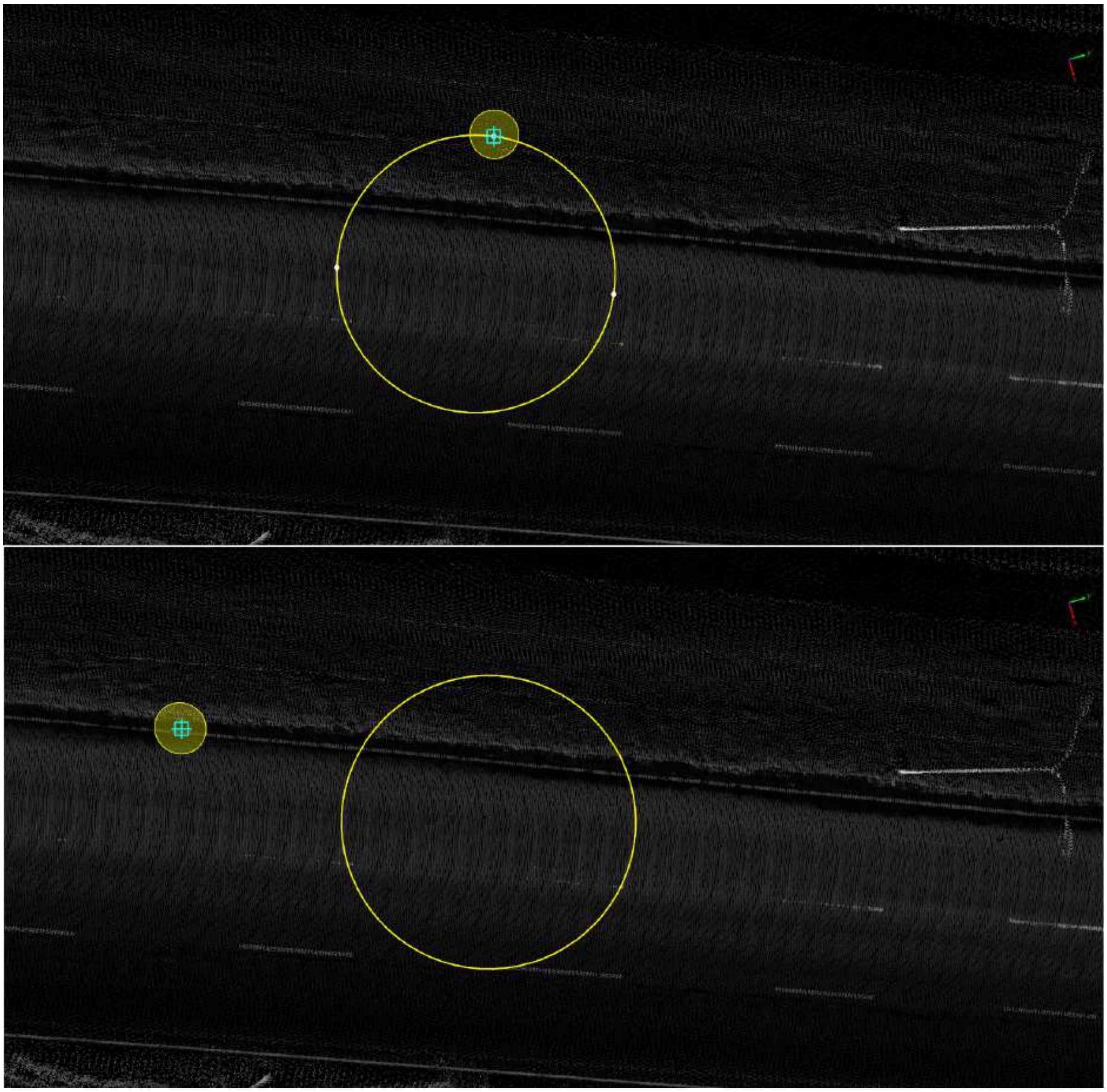
Function Description: Use the three key point method to add circle-shaped vector objects.

Steps

1.Select a surface layer, such as a surface facility layer.

2.Click **Add Circle** in Vector Editing Button

- 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 circle-shaped vector object.



Draw circle interface

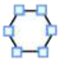
> Note: The geometric characteristics of the vector result must be consistent with the geometric characteristics of the layer to be added. For example, the point-like vector result must be added to the point-like layer, otherwise it will be prompted that it cannot be drawn. For the meaning of layers, see Layer Management

Add hexagon

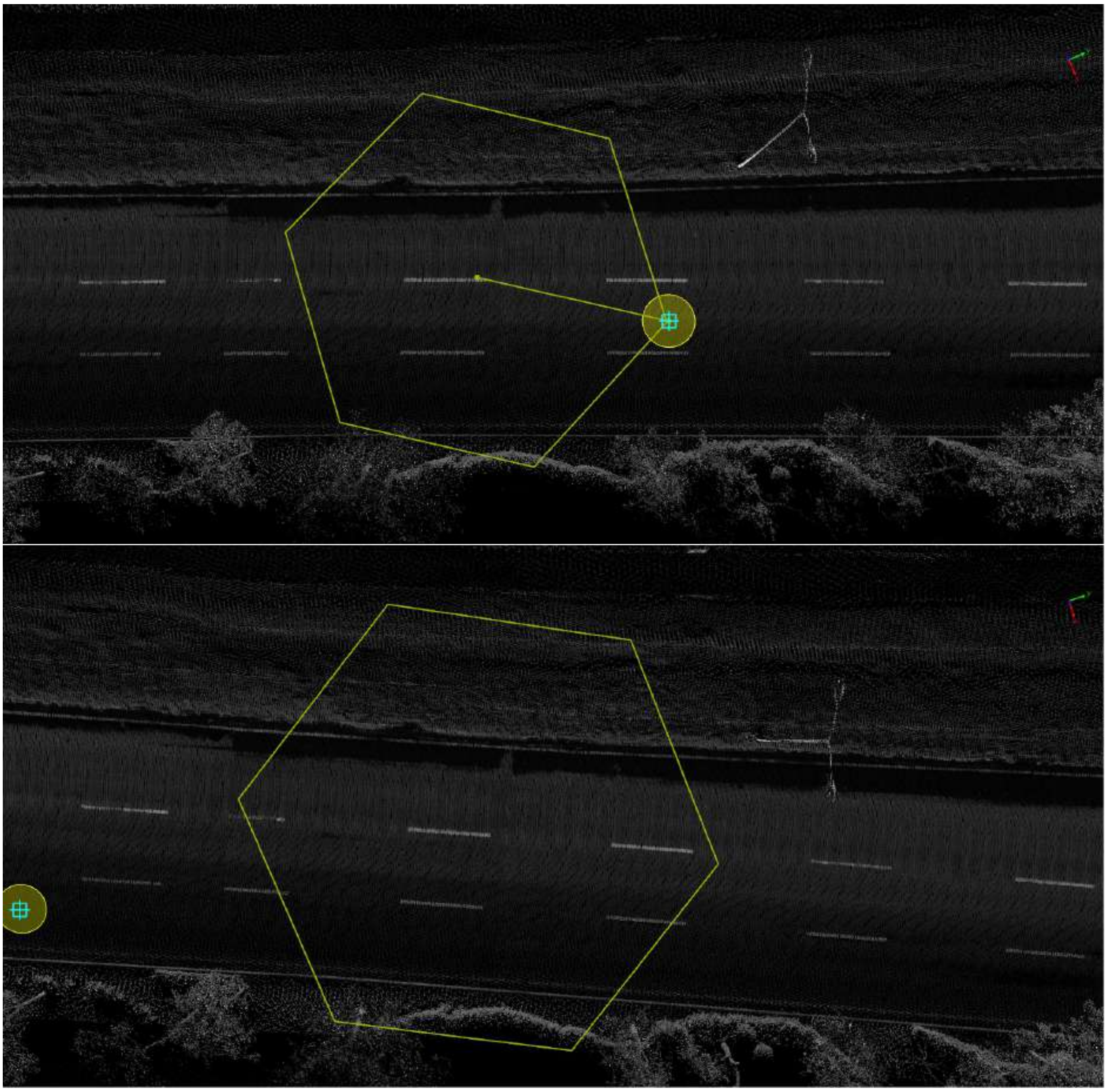
Function Description: Add a hexagonal vector object by using the center point and dragging the mouse.

steps

1.Select a surface layer, such as a surface facility layer

2.Click **Add Hexagon** in the vector editor  button:

- 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 Edge vector object.



Draw a 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 meaning of layers, see Layer Settings

Add octagon

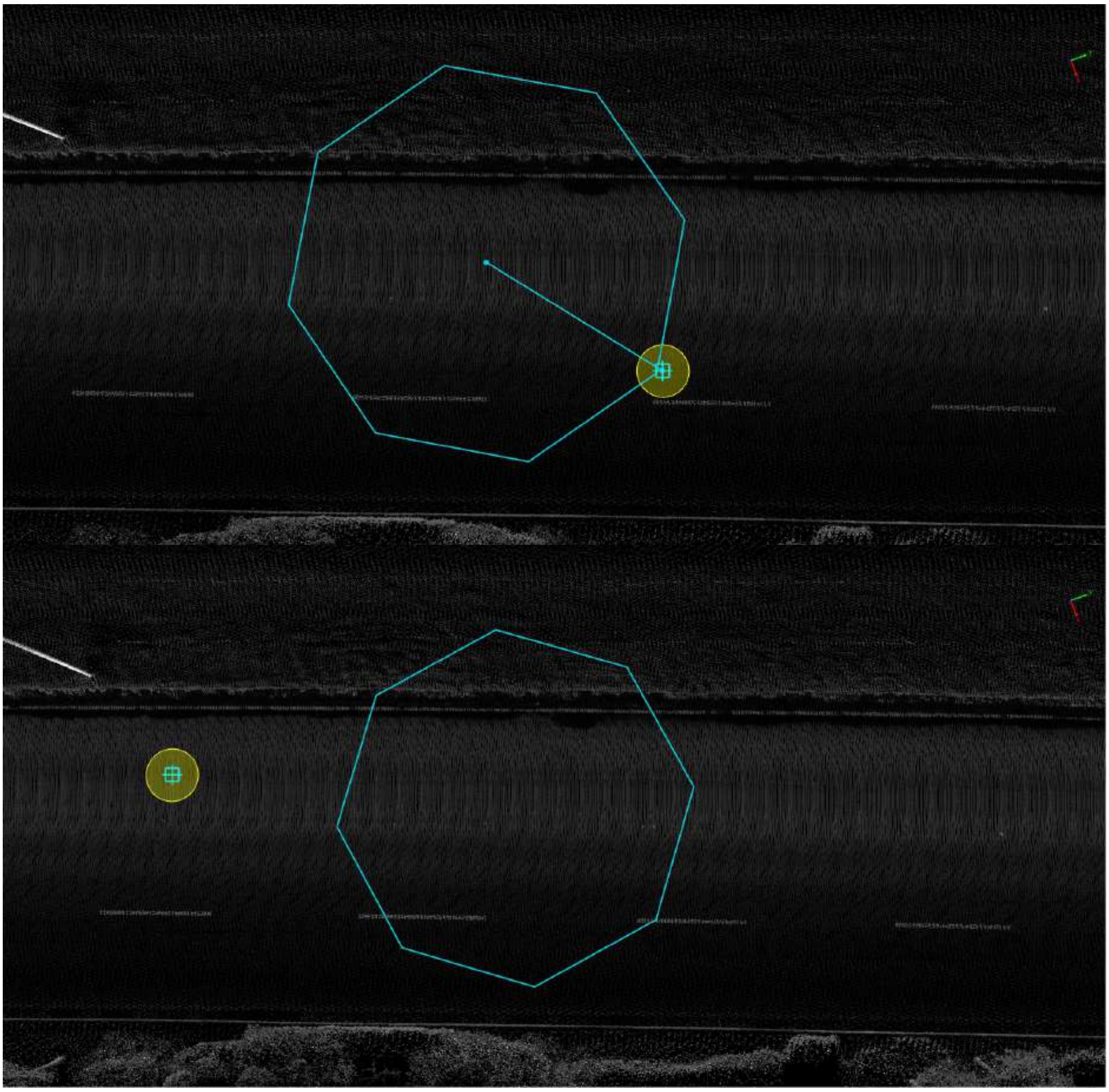
Function Description: Add an octagon vector object by using the center point and dragging the mouse.

steps

1.Select a surface layer, such as a surface facility layer

2.Click **Add Octagon** in vector editor  button:

- 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 an 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 meaning of layers, see Layer Settings

Edit Elements

The element editing 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
    - Vector Select
    - Secttion Select
    - Line Select
    - Polygon Select
    - Lasso Select
    - Circle Select
    - Box Select
    - Sphere Select
    - Cylinder Select
- Break Line
    - Point Break Line
    - Line Break Line
    - Polygon Break Line
    - Line Break Objects
- Align
    - Move vector
    - Parrallel Copy Vector
    - Rotate Vector
    - Scale Vector
    - Mirror Vector
- Reshape
    - Reshape Line
    - ReshapeRect
    - Merge Lines
    - Break Lines
    - Split Lines
    - Stretch/Cut Line

- Merge/Split Polygons
- Hook Lines
- Extend Lines
- Fillet
- Split Multipolygon
- Combination Polygons Points

- Auxiliary Functions
  - Grounding
  - Trace Line
  - Trace Polygons
  - Draw Intersection
  - Draw Perpendicular

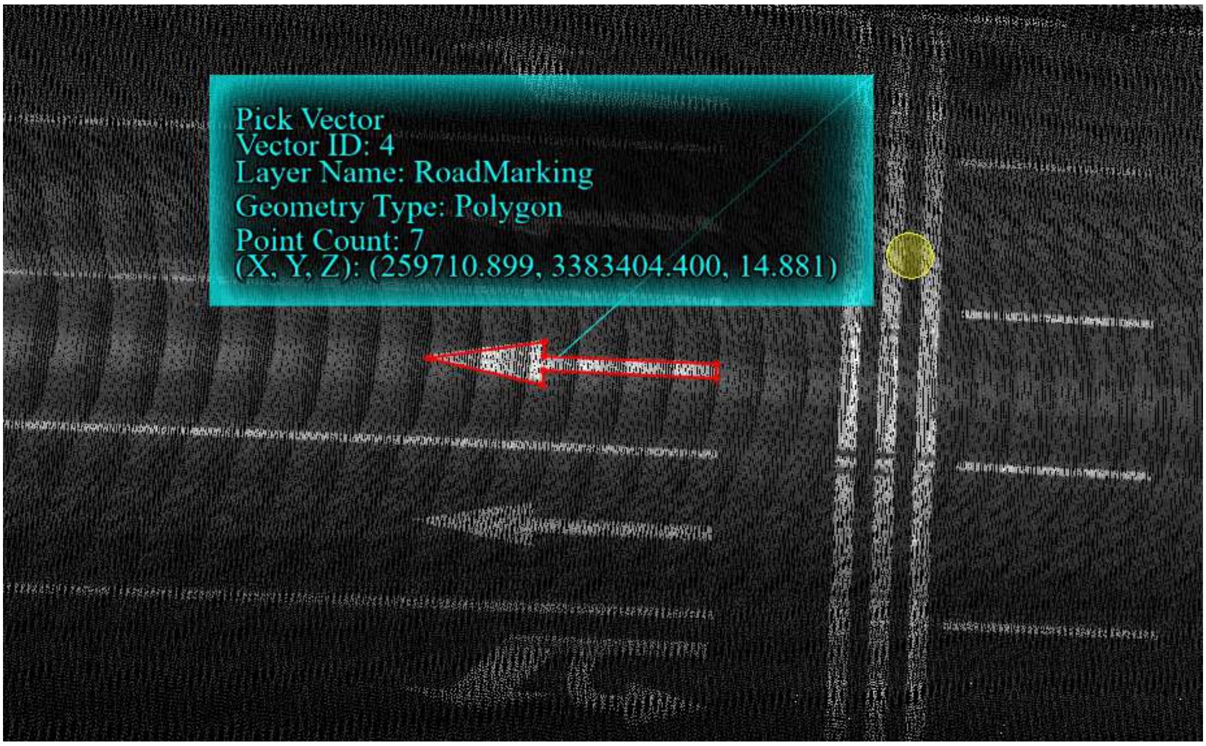
Select Vector

Function Description：The select vector function allows the user to select a vector object and obtain its detailed information.

Steps

> Note: Select the shortcut key to copy and delete during editing Vector Select.

1.After clicking the **Select Vector** button  in the vector editing, 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 and geometry information, the attribute table window will navigate to the row where the attribute field of the currently selected vector is located.



Select vector

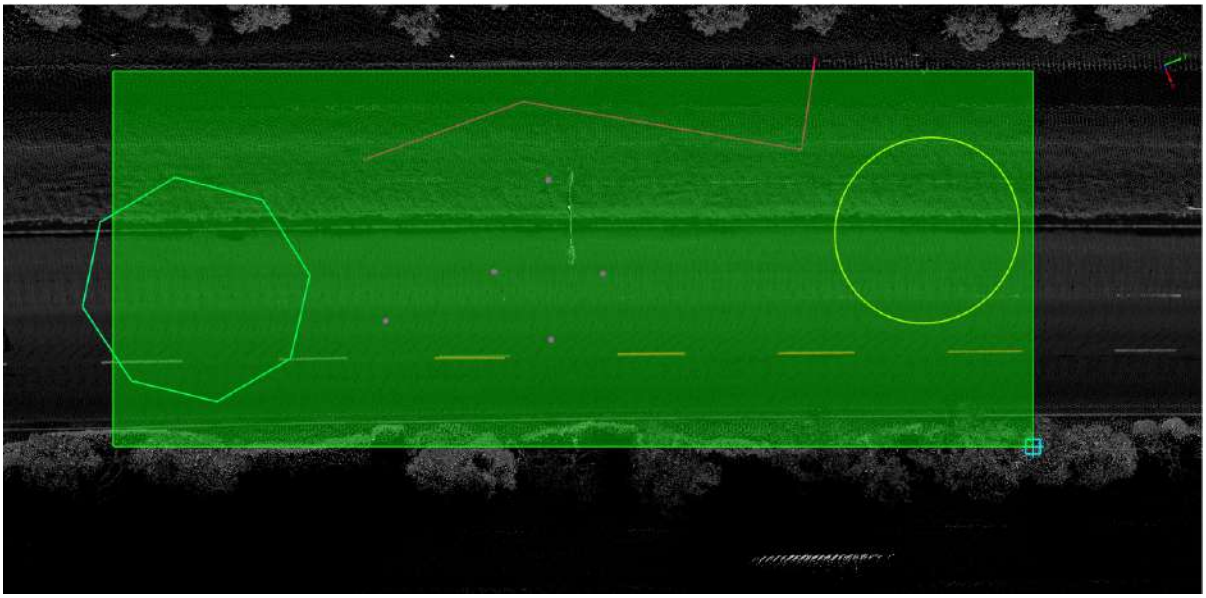
> Note: Select the shortcut key to copy and delete during editingVector Select。

Section Selection

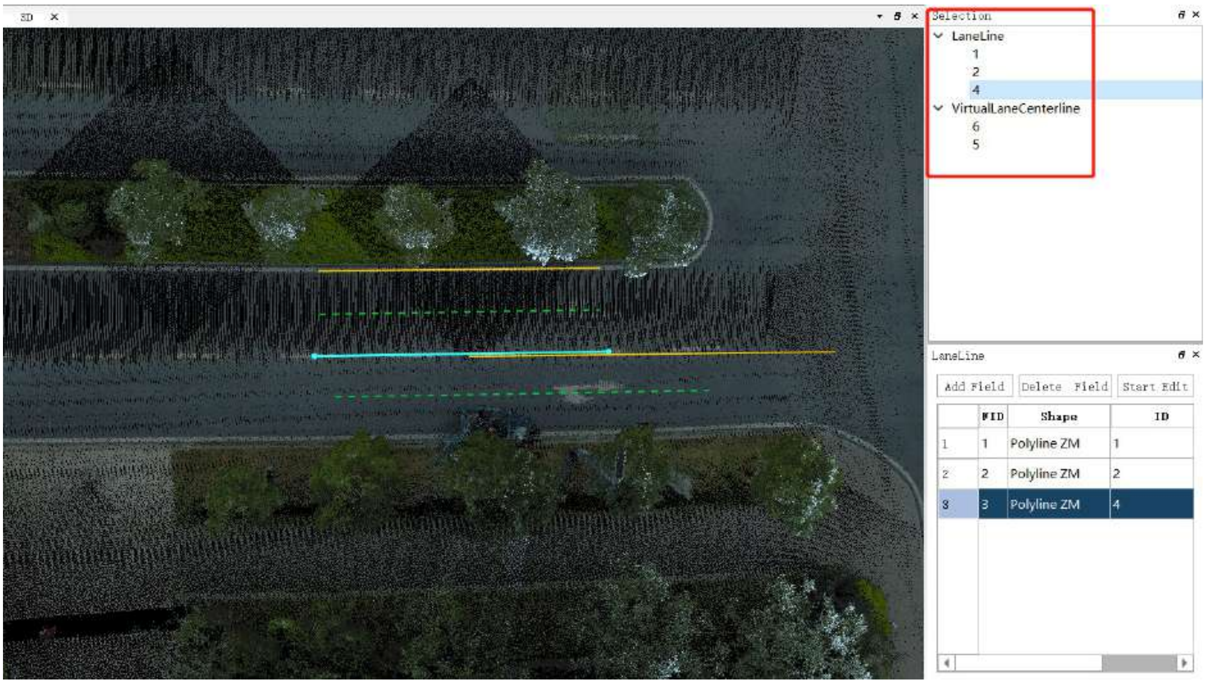
Function Description：The section selection function supports users to select a series of vector objects by using a rectangle frame.

Steps

1.Click **Section selection** in vector editing , 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 tree

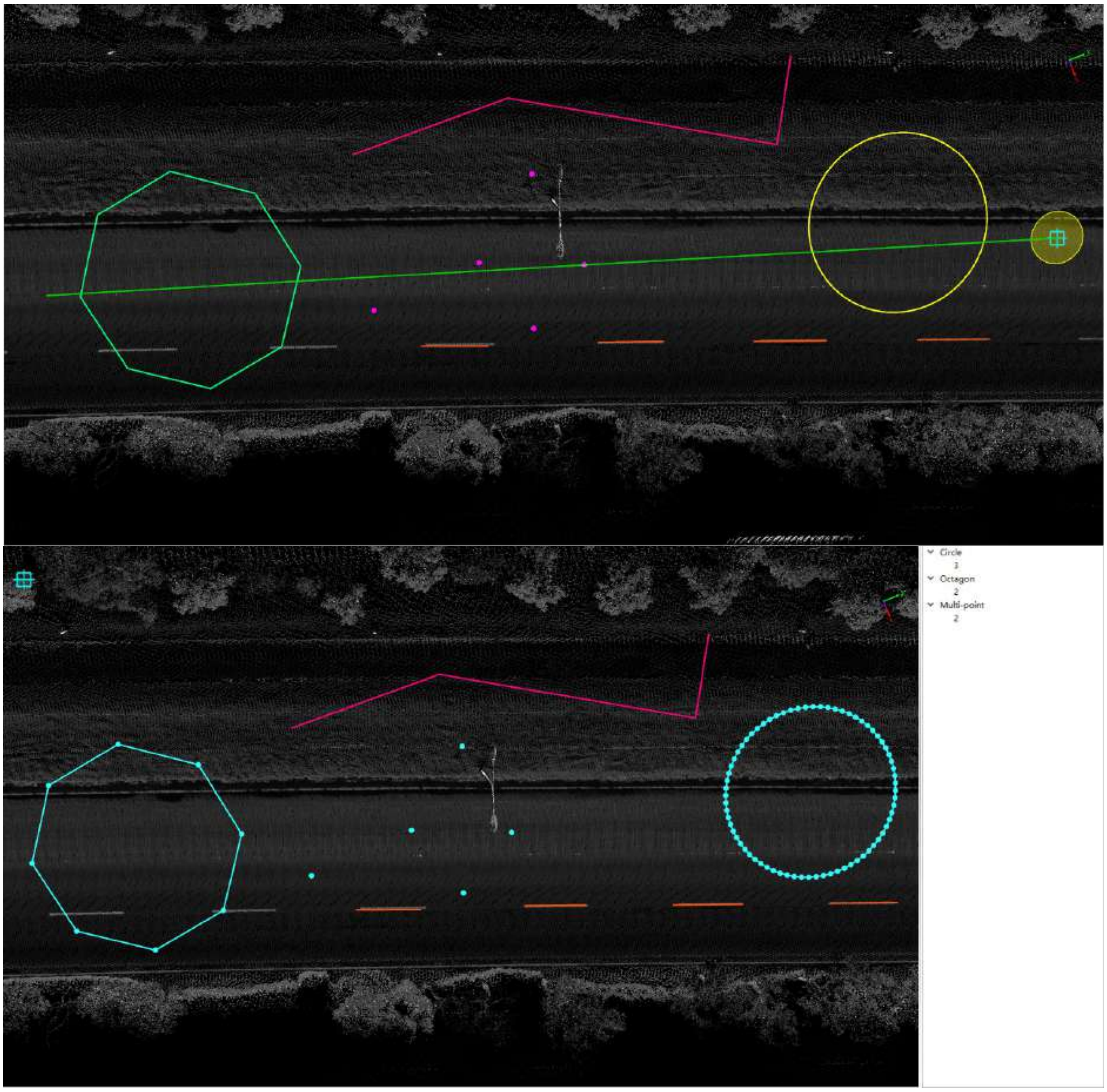
> Note: Select the shortcut key to copy and delete during editing Vector Select。

Select by Line

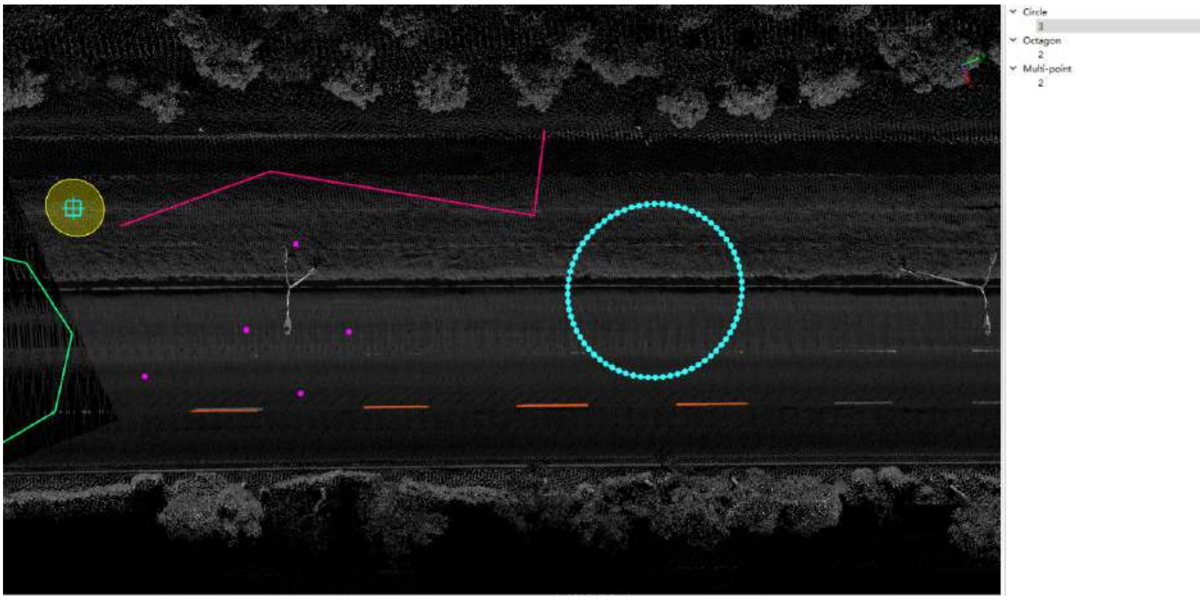
Function Description：The line selection function allows users to select multiple objects by drawing lines and operate them.

Steps

1.After clicking the line selection button  in the vector editing, 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

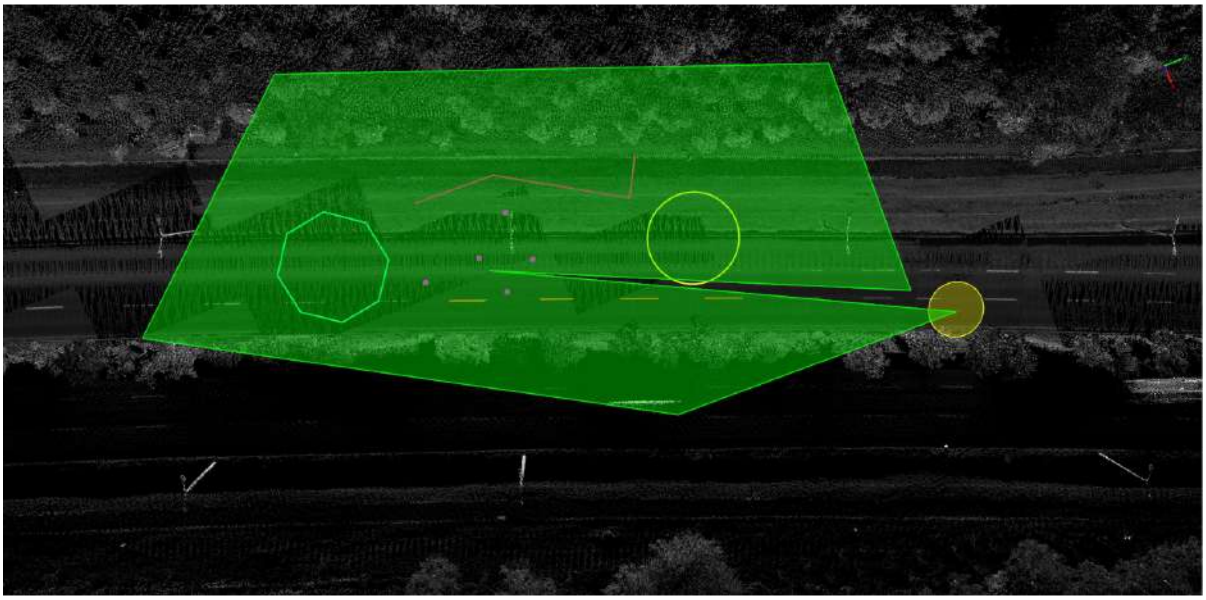
> Note: Select the shortcut key to copy and delete during editing Vector Select。

Polygonal selection

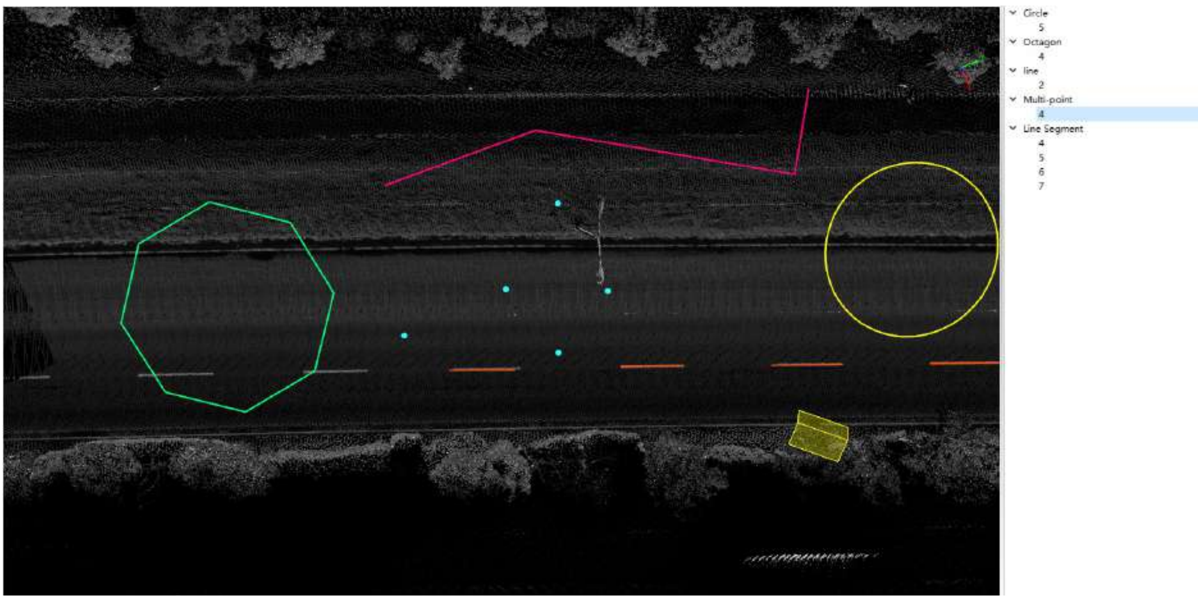
Function Description: The polygon selection function allows users to select a series of vector objects by using a polygon frame.

steps

1.Click **Polygon Select** in the vector editor  button, 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

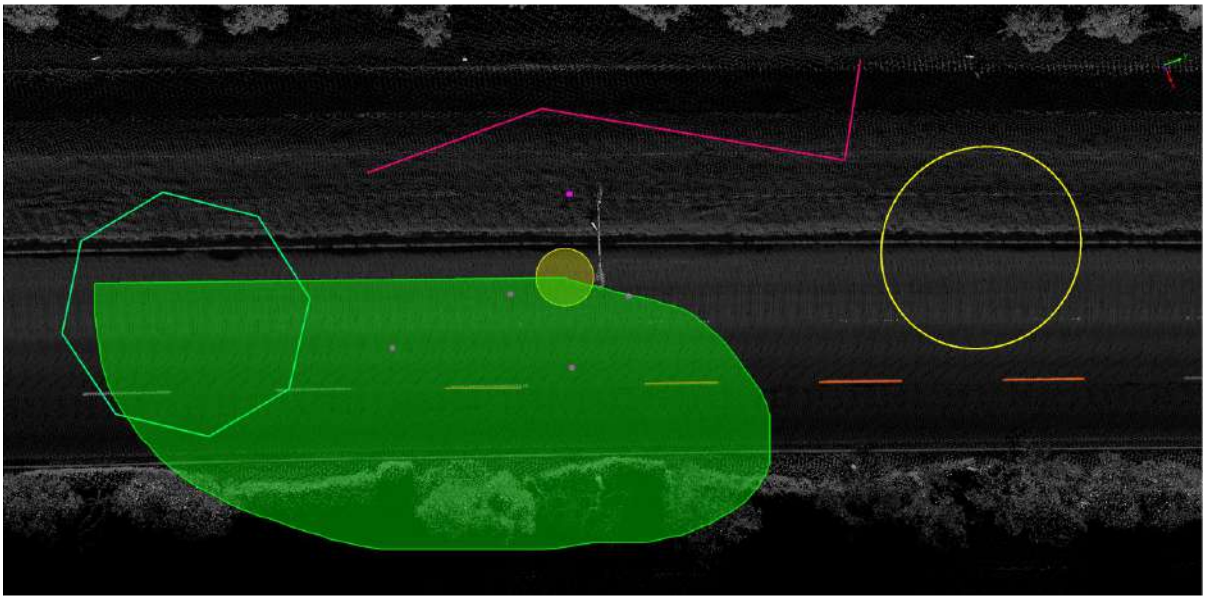
> Note: Select the shortcut key to copy and delete during editing Vector Select。

Circle selection

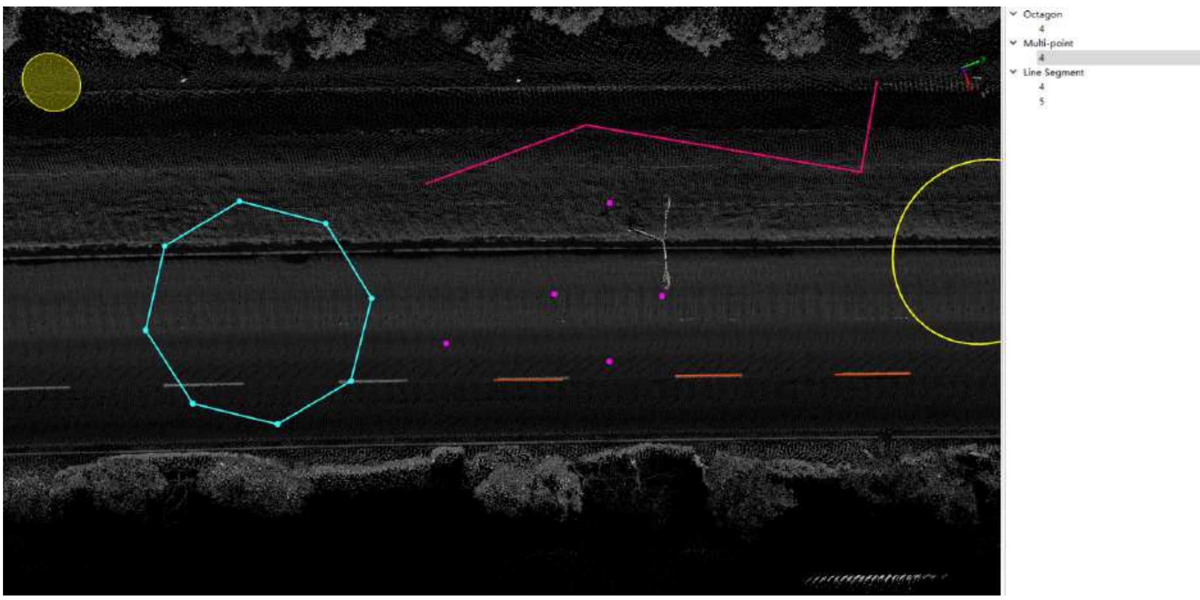
Function Description: The circle selection function allows users to move the mouse to select a series of vector objects.

steps

1. Click **circle selection** in the vector editor  button, 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. .



Circle selection vector



Circle selection vector - vector tree

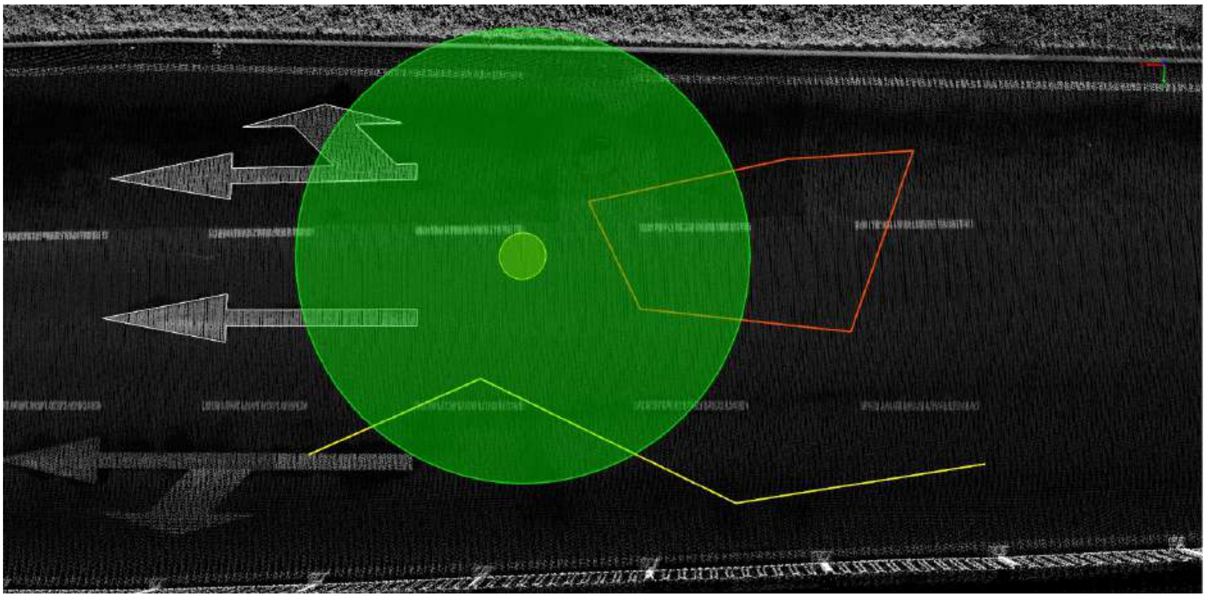
> Note: Select the shortcut key to copy and delete during editing Vector Select。

Cicle selection

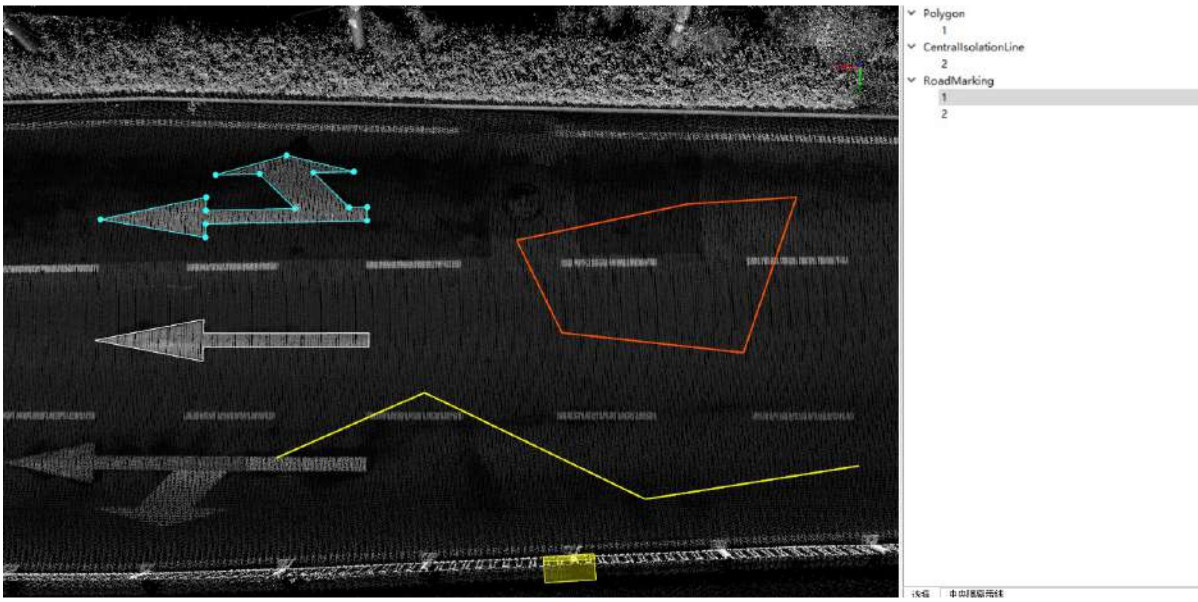
Function Description: The circle selection function allows users to use circles to select a series of vector objects.

steps

1.Click **circle selection** in the vector editor  button, 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. is 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

2.After the circle selection, you can directly use the shortcut keys to copy, delete and other operations

Shortcut	function
Ctrl+C	Copy
Ctrl+V	Copy to the same layer
Ctrl+Alt+V	Copy to any layer with the same geometry (the popup layer is optional)
Delete	Delete

Bounding box selection

Function description: The bounding box selection function allows users to select a series of vector objects by using a rectangle.

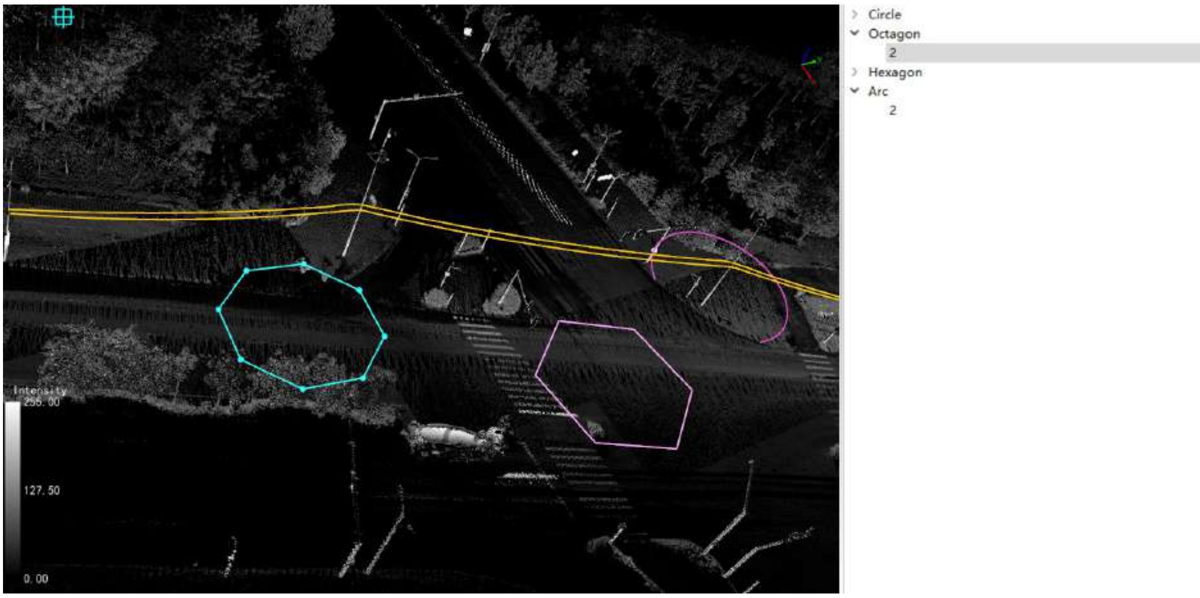
steps

1.Click **Bounding Box Selection** in Vector Editing  button:

- 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



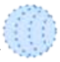
Bounding box selection vector-vector tree

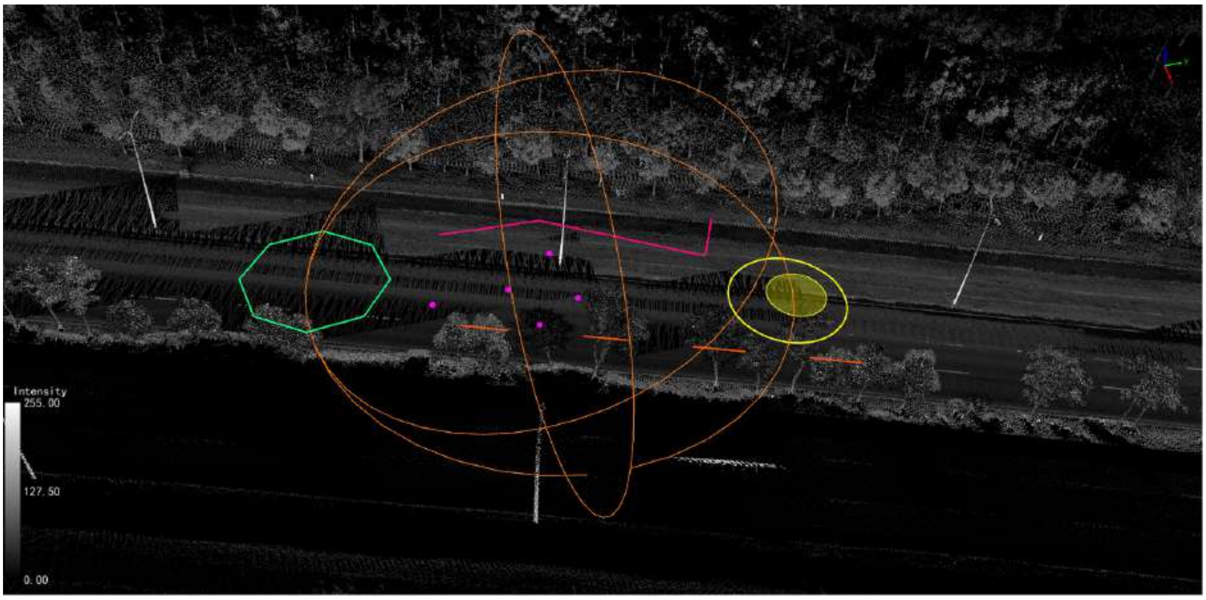
> Note: Select the shortcut key to copy and delete during editing Vector Select。

Spherical selection

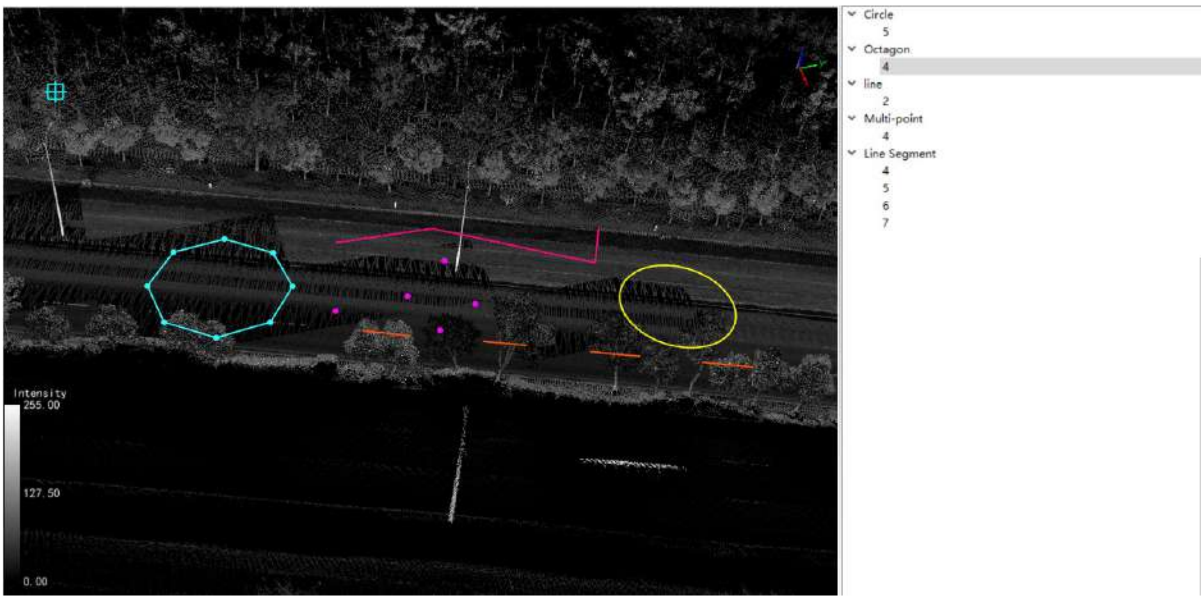
Function description: The sphere selection function allows users to select a series of vector objects using spheres.

steps

1.Click **ball selection** in vector editor  button, 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 ball 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.



Ball selection vector



Ball selection vector-vector tree

> Note: Select the shortcut key to copy and delete during editingVector Select。

Cylindrical selection

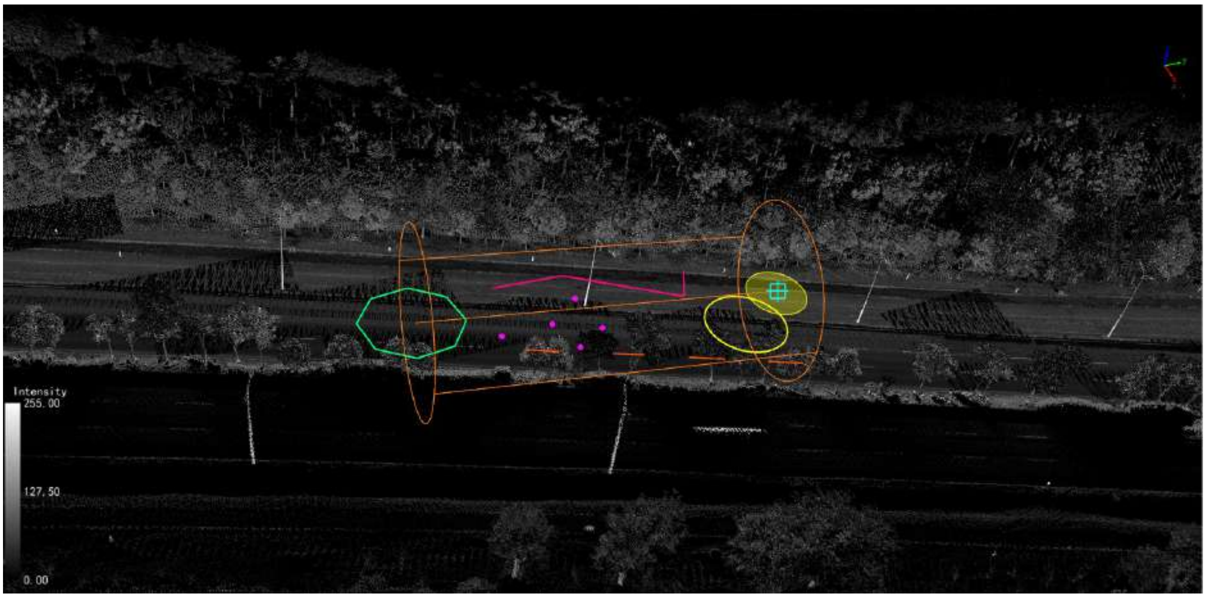
Function Description: The cylinder selection function allows users to select a series of vector objects using cylinders.

steps

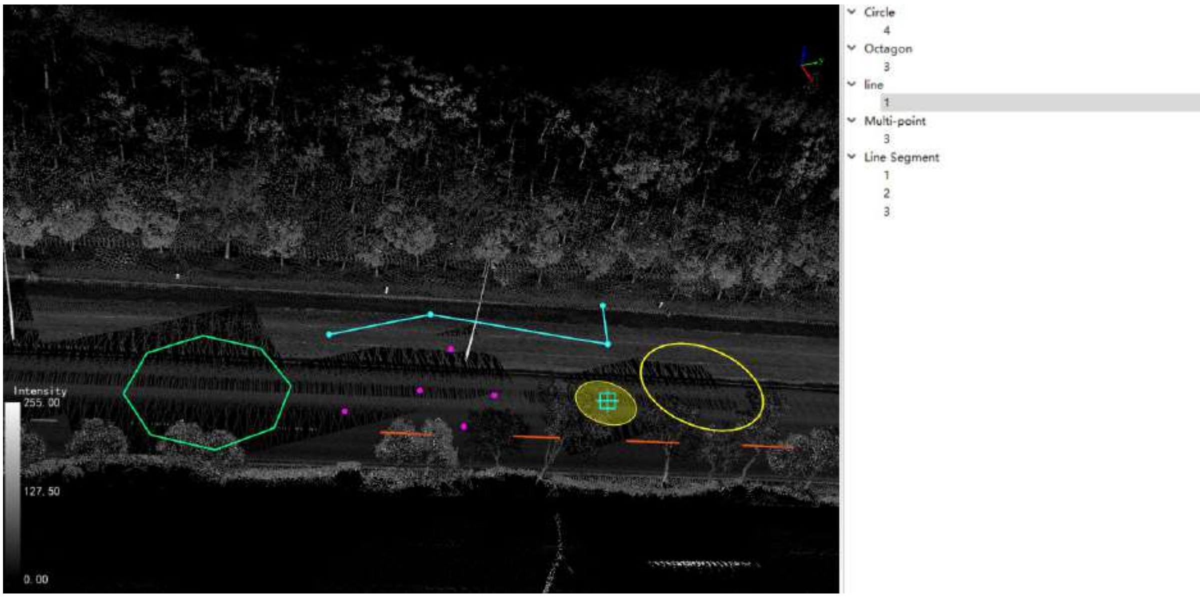
1.Click **Cylinder selection** in the vector editor  button:

- 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 and delete during editing Vector Select。

Point Break Line

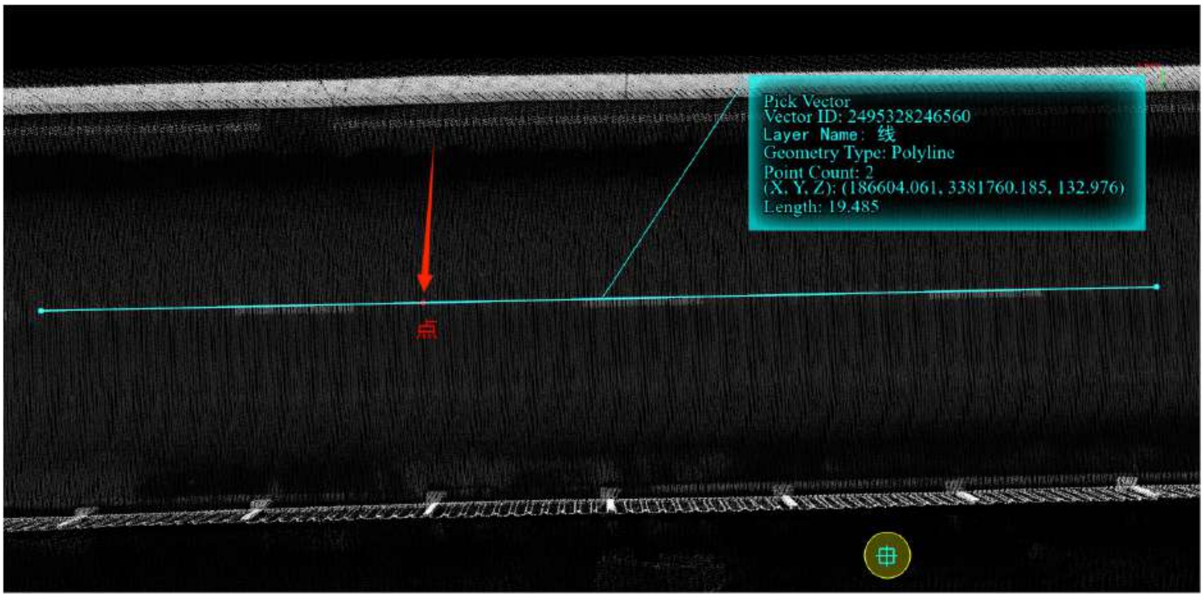
Function Description：The point break line function supports breaking a line object into two end-to-end line objects through a point on the line.

Steps

1.After clicking the **Point Break Line**  button in vector editing, 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.

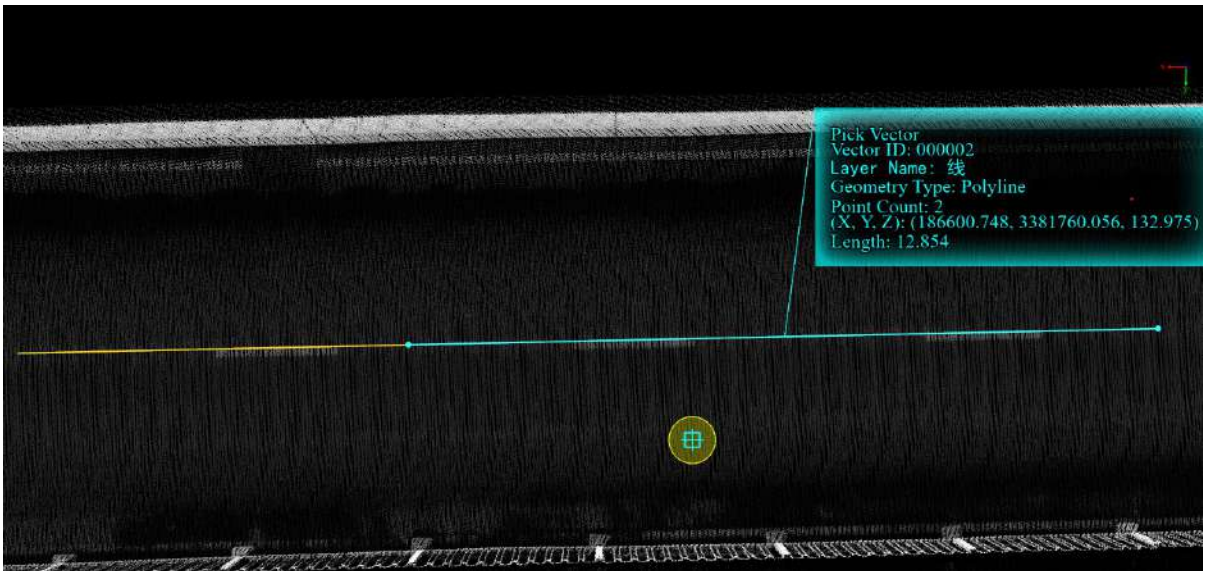
> If the point object is not on the line object, interruption is not allowed, and there will be a prompt in the output window.

- **Original line segment**: a single line object.



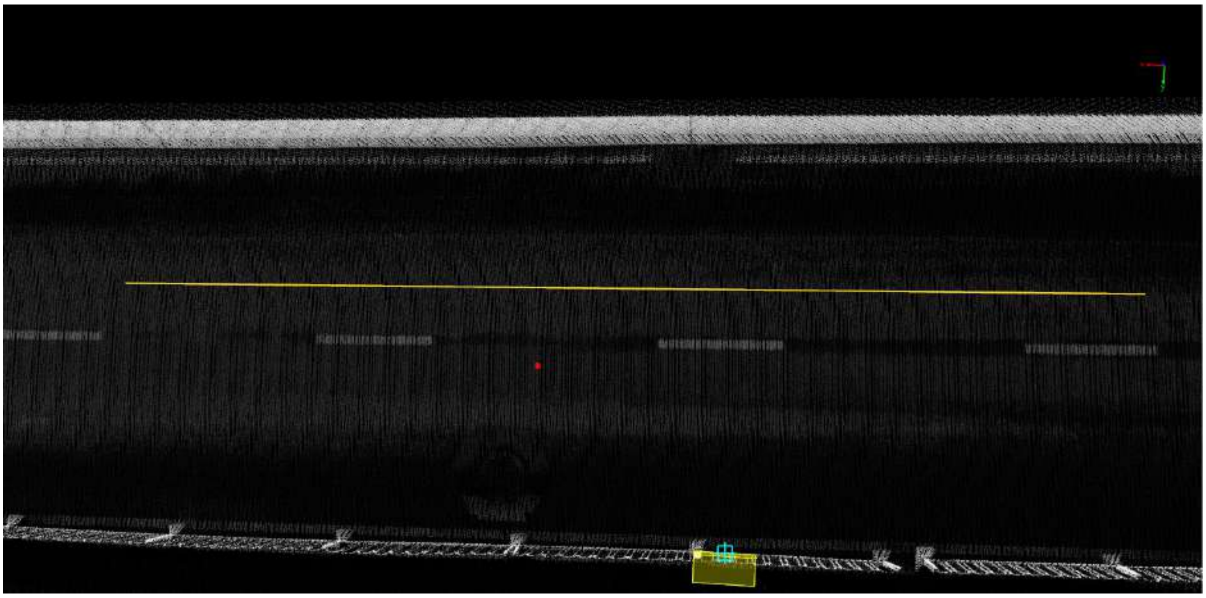
Original line

- **Break at the point object on the line segment**: The two lines after the break are connected end to end at the point object.

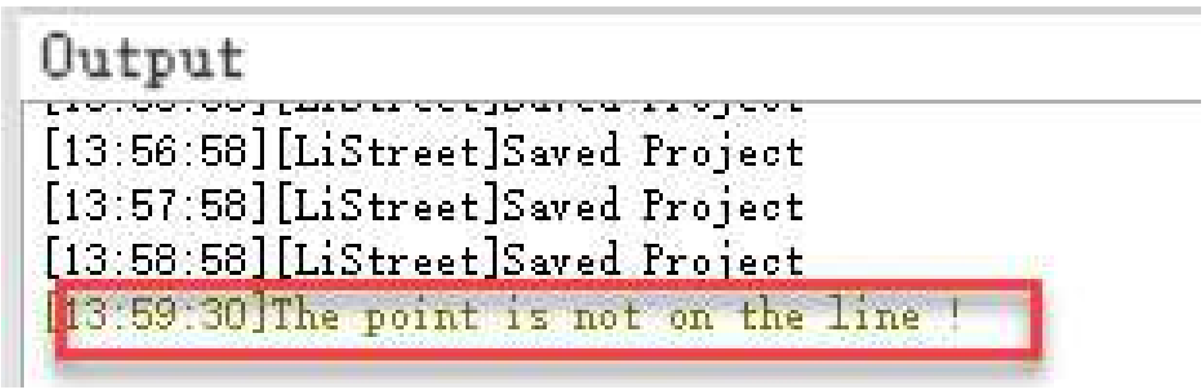


After break

- **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 Break Line

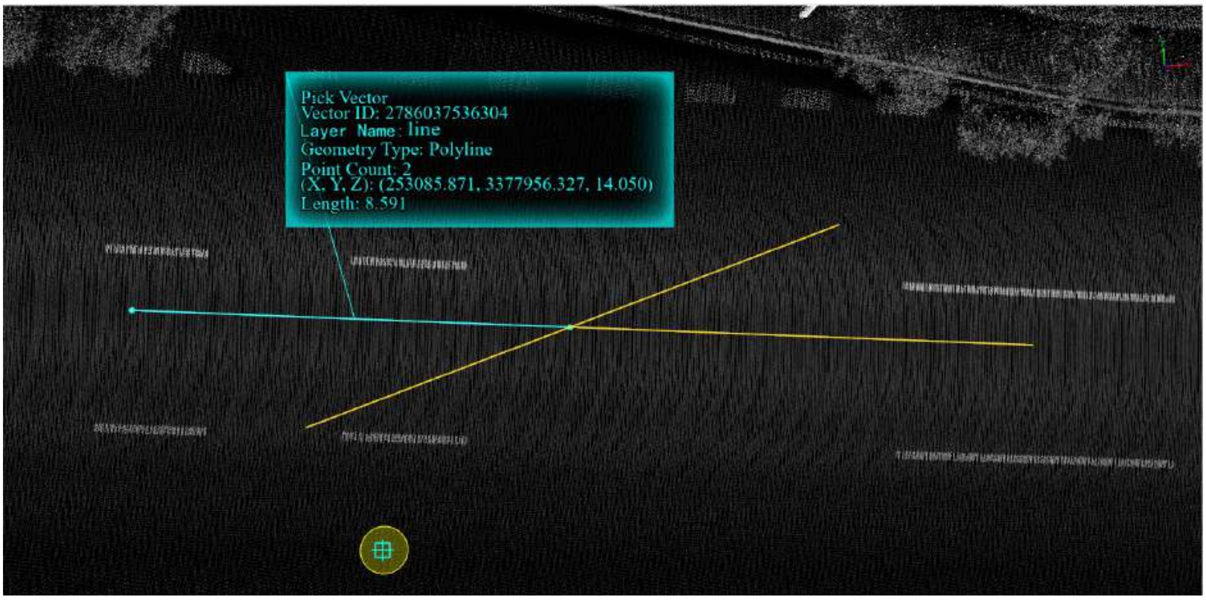
Function Description： The Line Break Line function supports by intersecting one of the two lines, the other line is broken into two lines connected end to end.

Steps

1.Click **Line break line** in vector editor  button, 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.

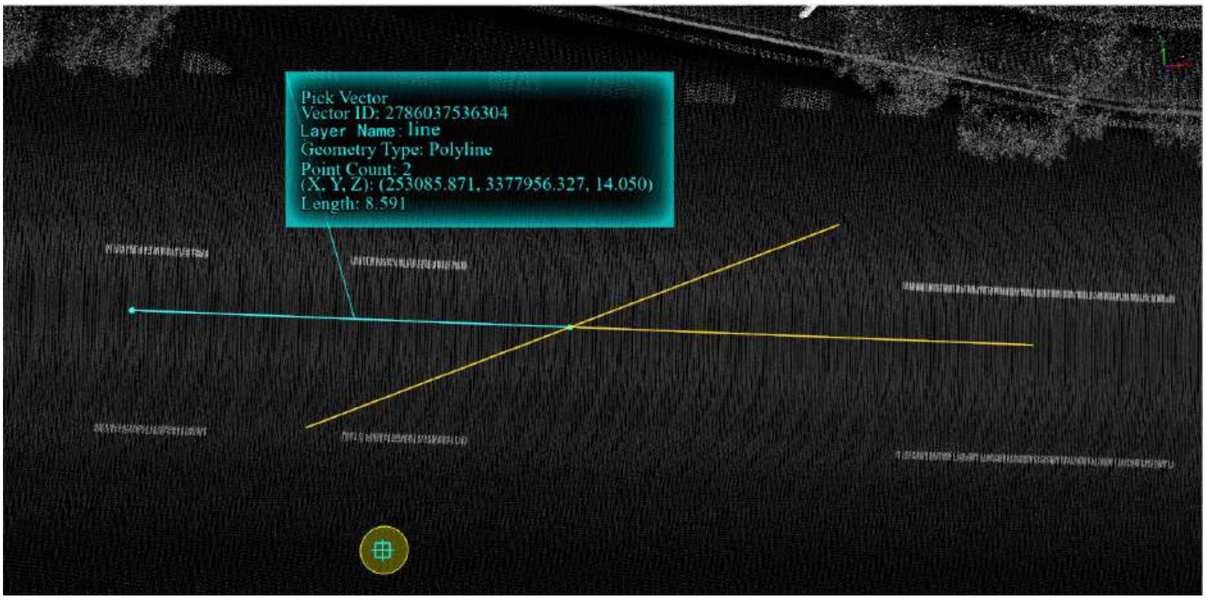
> If the two line objects do not intersect, line break will be invalid, and there will be a prompt in the output window.

- **Original intersecting lines**: Two intersecting line objects.



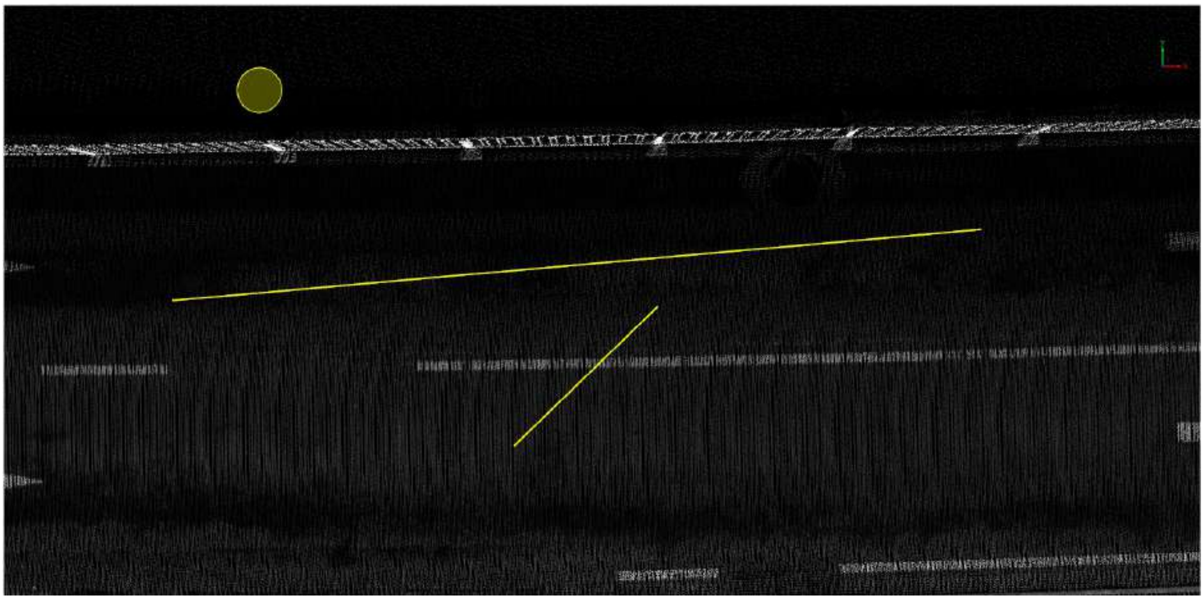
Original line

- **Break at intersection point**: The two lines after the break are connected end to end at the intersection point.

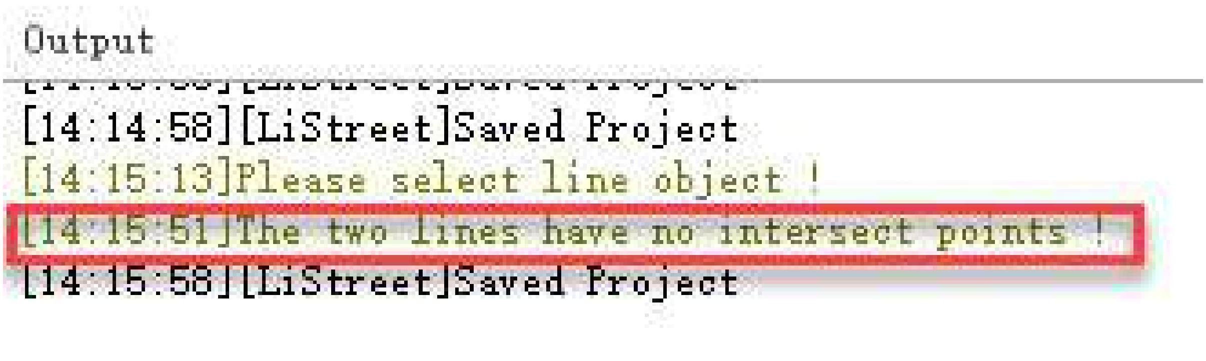


Line after break

- **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

Polygon Line Break

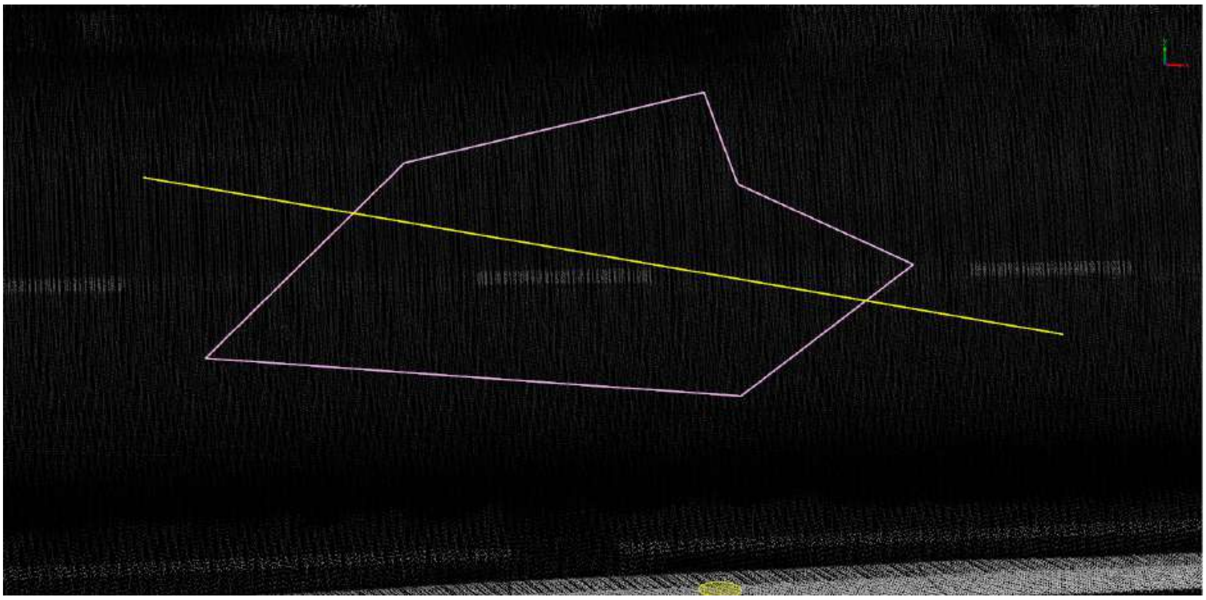
Function Description：The Line Break Polygon feature supports the breaking of an area object into two two-area objects by intersecting a line in the area object.

Steps

1.After clicking the **Line Interrupting Surface**  button in 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.

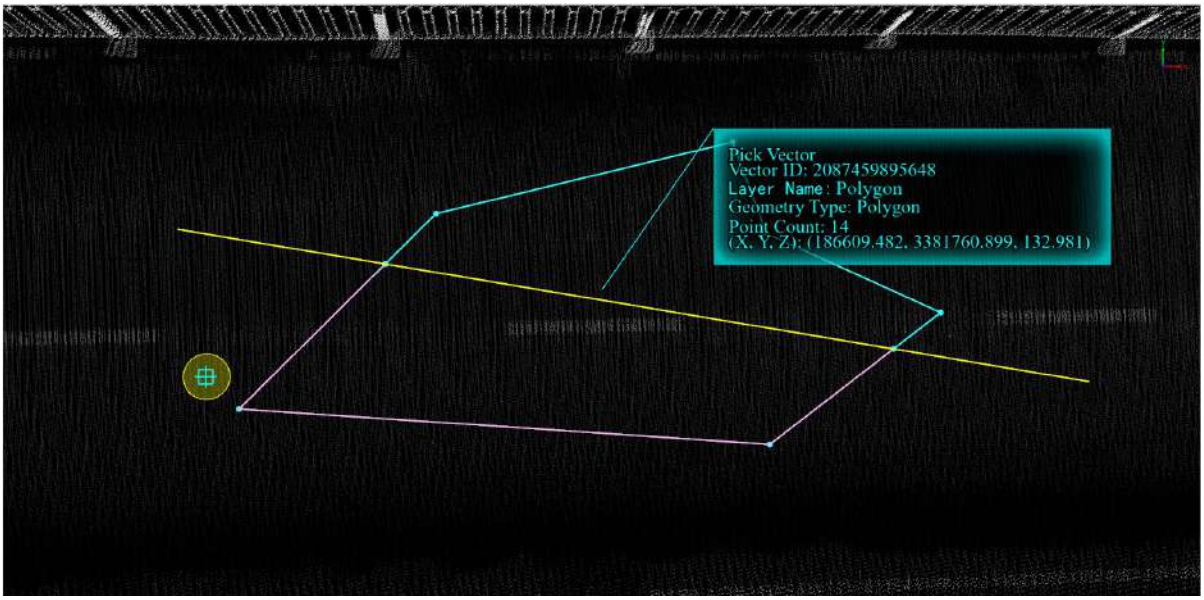
> If the intersection of the line and the polygon is less than two, line break will not be allowed, and there will be a prompt in the output window.

- **Original Intersecting Lines and Polygon**: Two intersecting line and area objects.



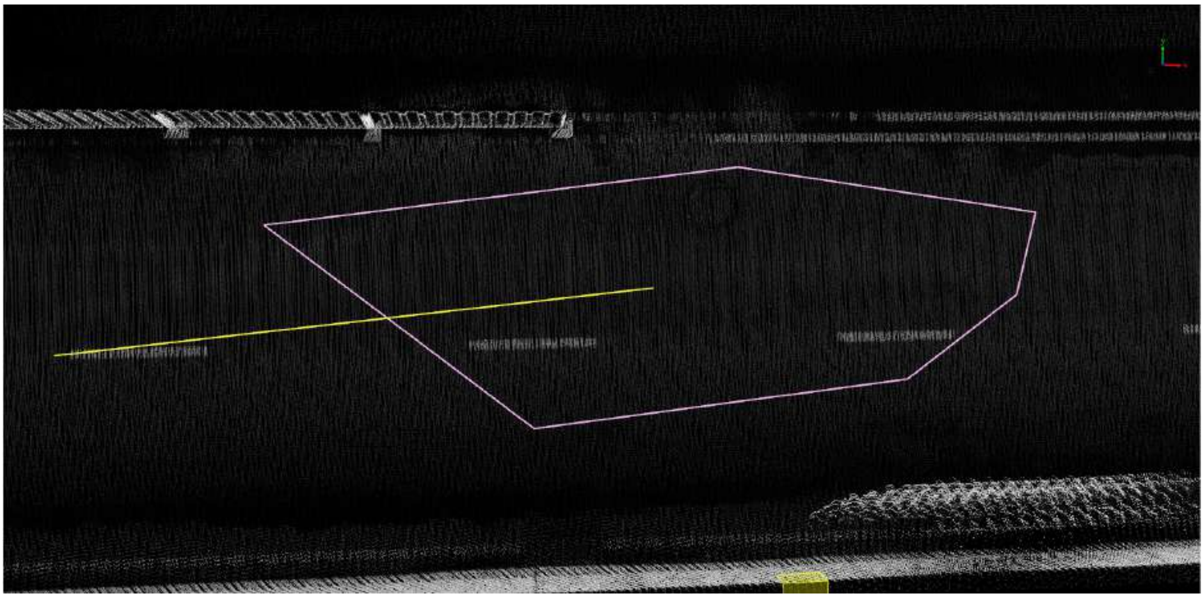
Original lines and polygon

- **Broken Polygon**: Two polygons that are connected after being broken.

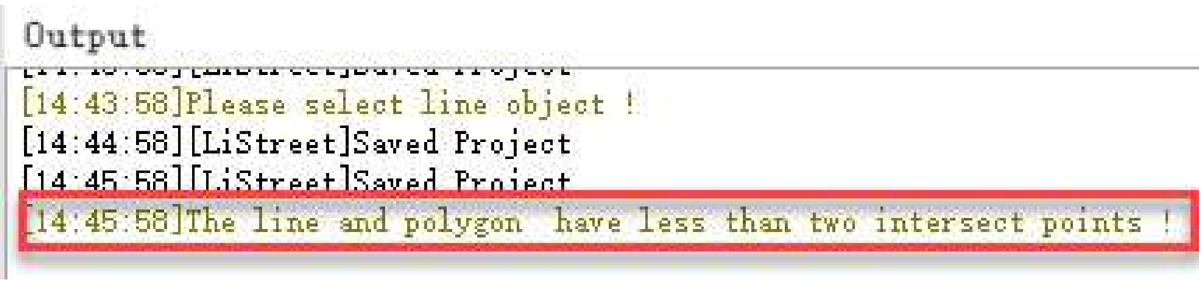


Two polygons connected after break

- **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

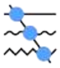


Line-polygon intersection is less than two points

Line break multiple objects

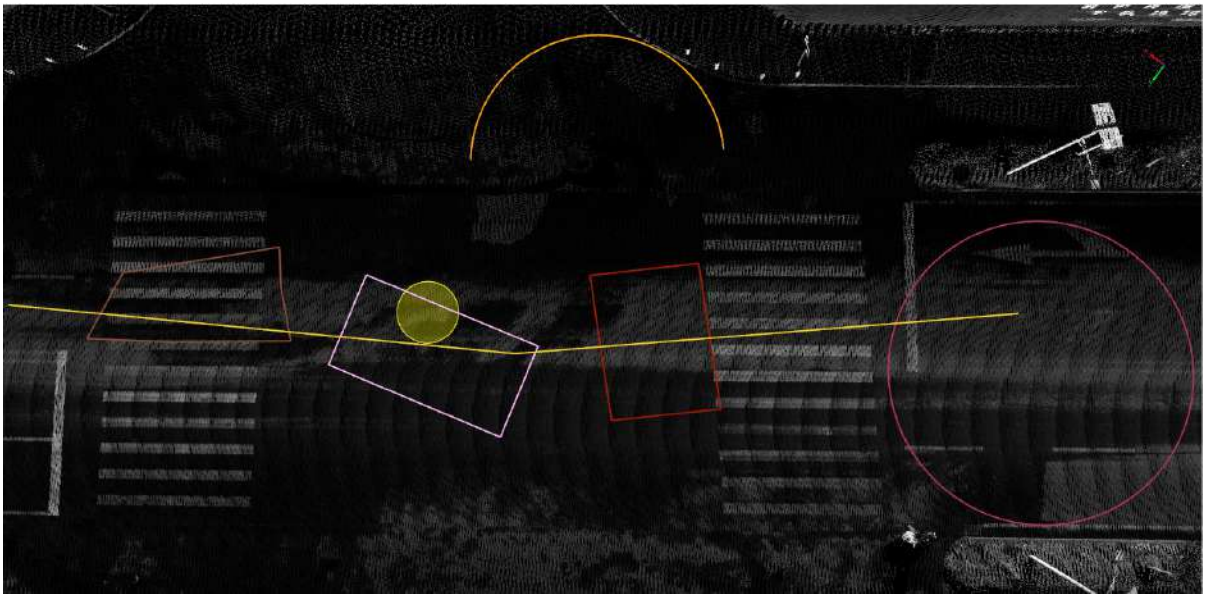
Function description: The function of breaking multiple objects with a line supports breaking the intersecting objects through a line, and breaking a line or area object into two objects.

steps

1.Click **Line break Multi Objects** in the vector editor  button, 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 Check Highlight. 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.

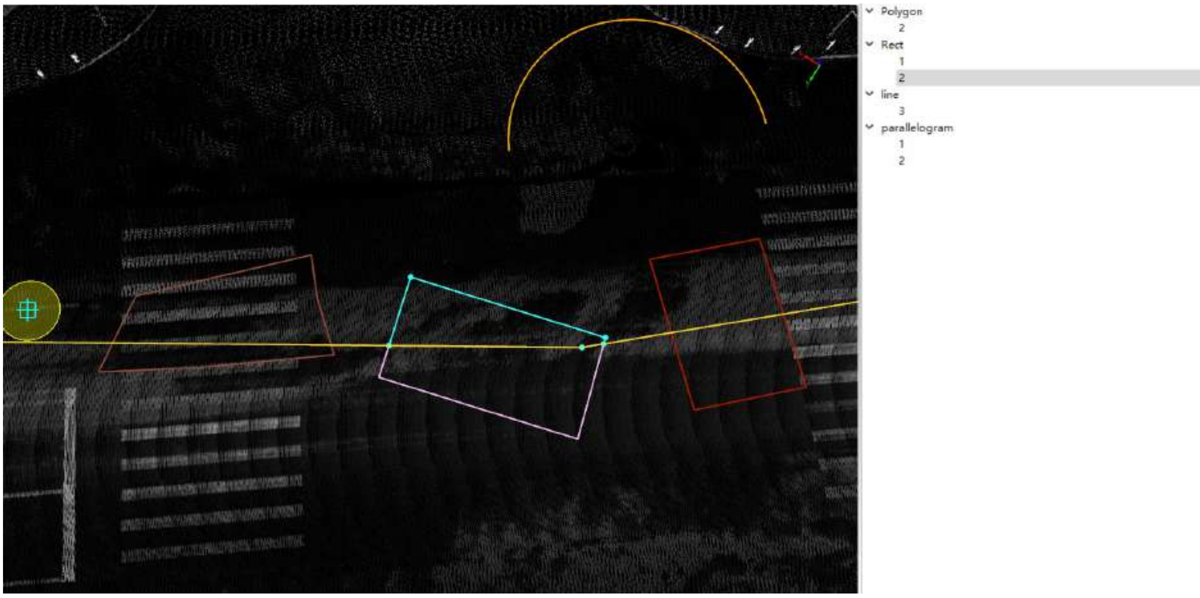
> If the intersection of the line and multiple objects is less than two, interruption is not allowed, and there will be a prompt in the output window.

- **Original intersected line with multiple objects**: Line object intersects with multiple objects.



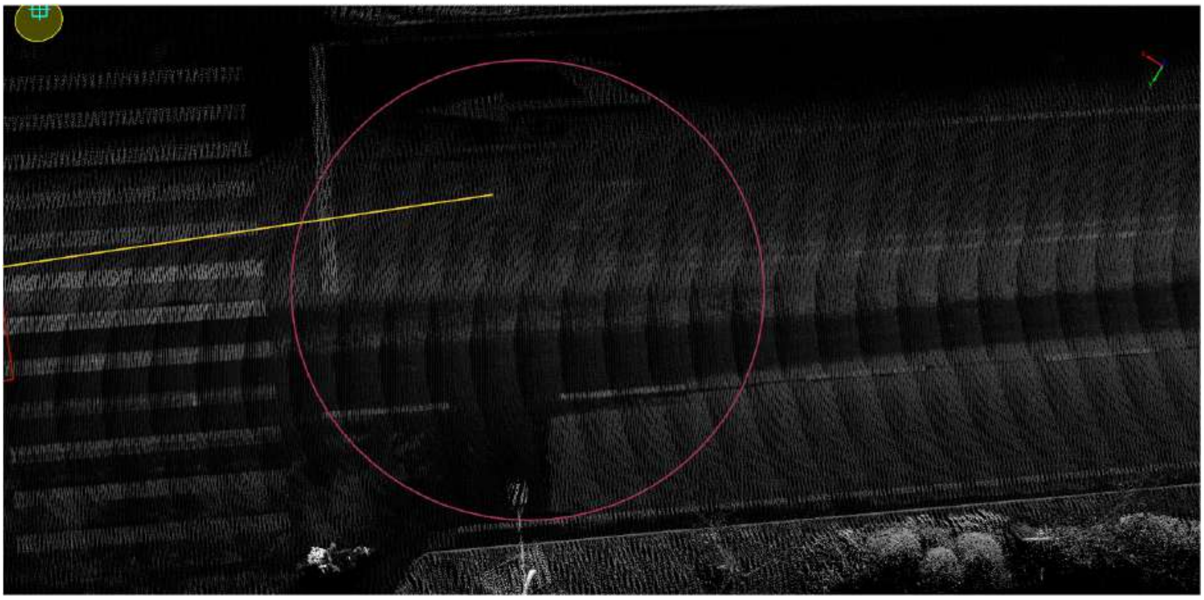
Original intersecting lines with multiple objects

- **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

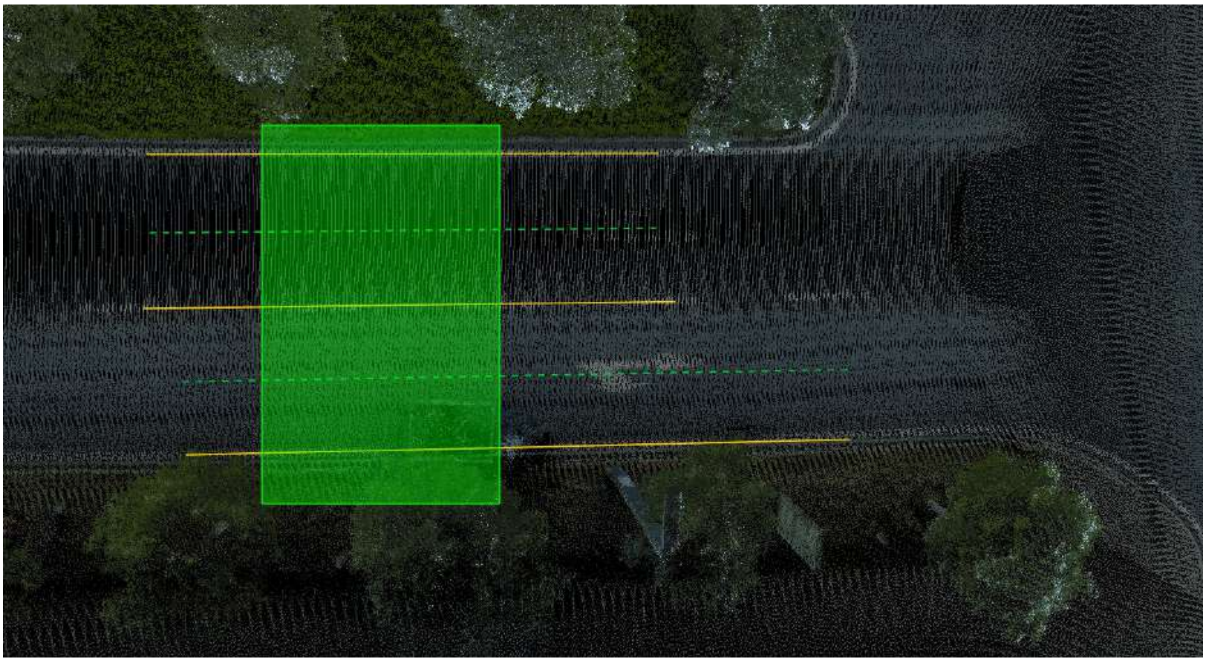
The intersection of the line and the object is less than two prompts

Move Vector

Function Description：The move function supports the user to move a vector object to a new position.

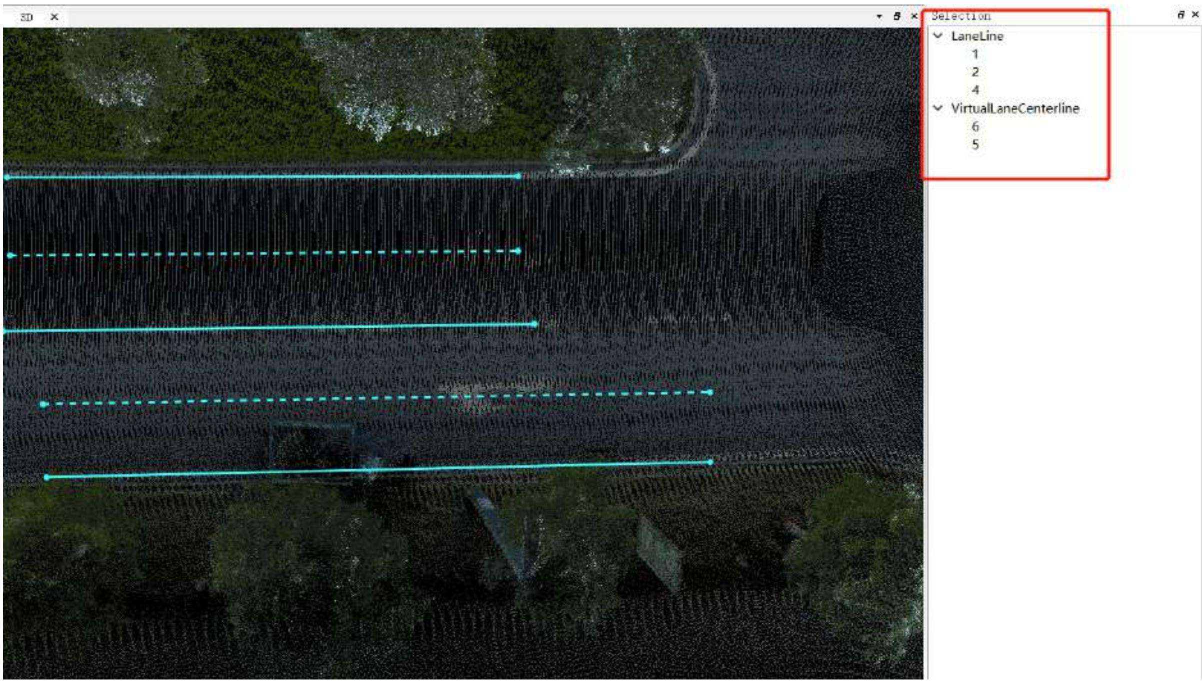
Steps

1.After clicking the **Move**  button in the vector editing, frame the area where the vector to be moved is located.



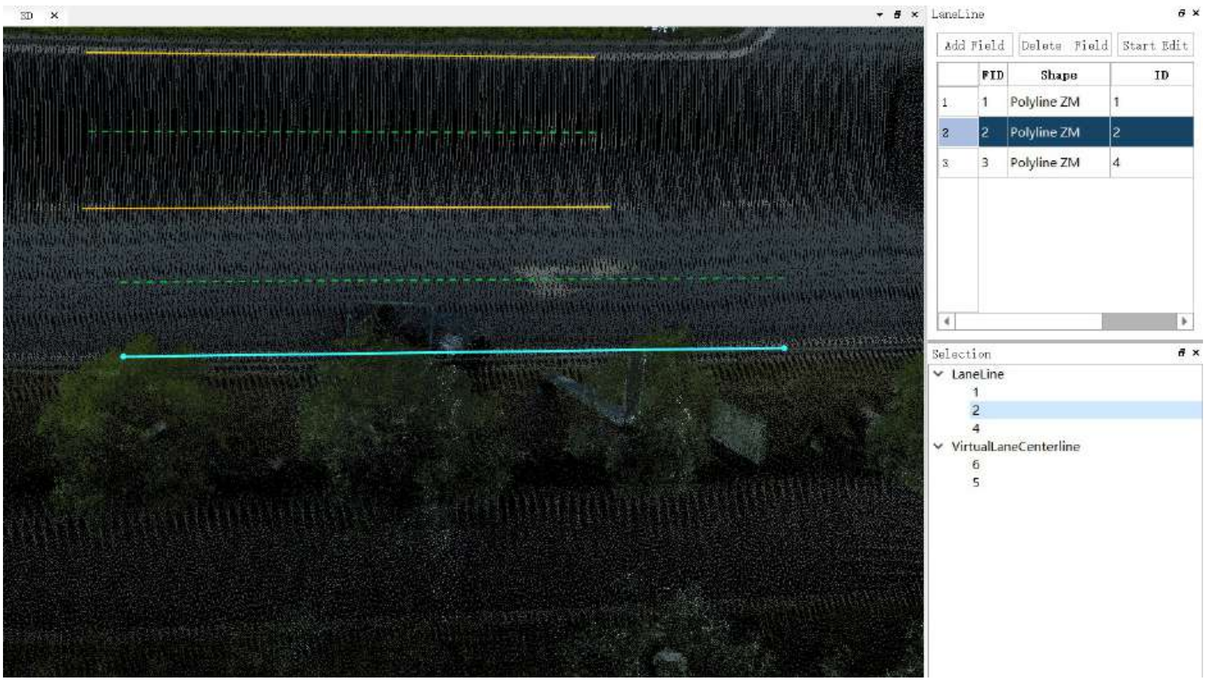
Frame selecttion of vectors

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.



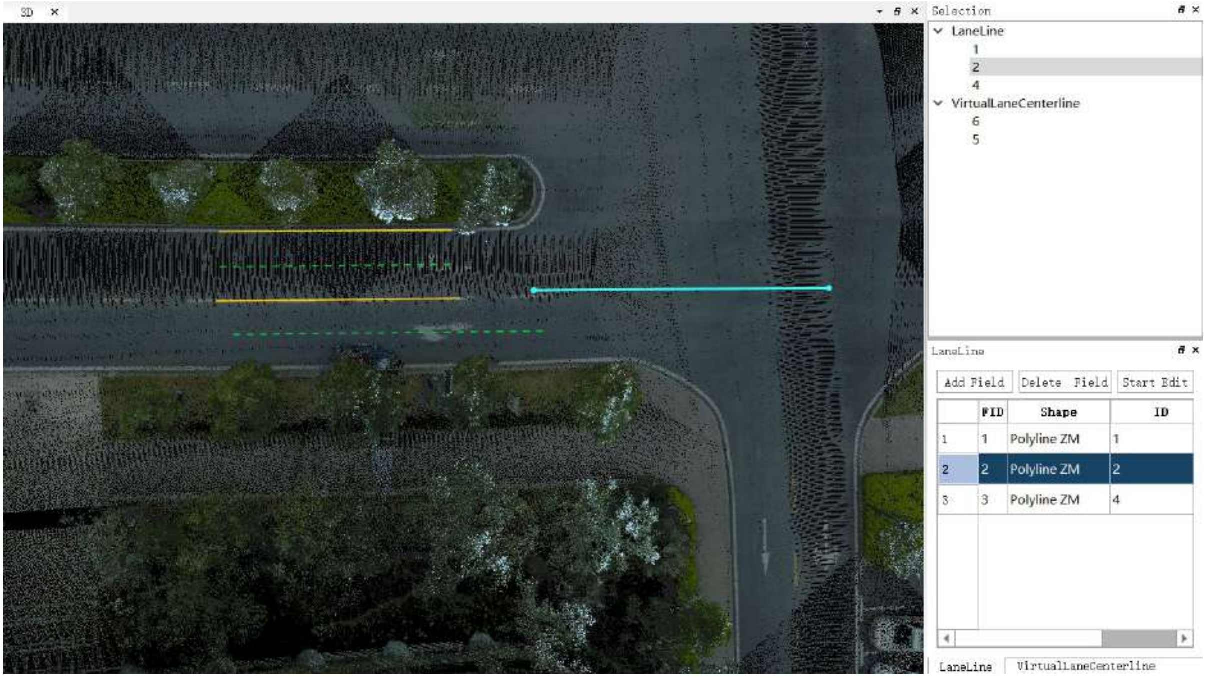
After frame selection of vectors

3.Click to select the details of a vector on the right, other vector highlights will be cleared, and the selected vector will be highlighted.



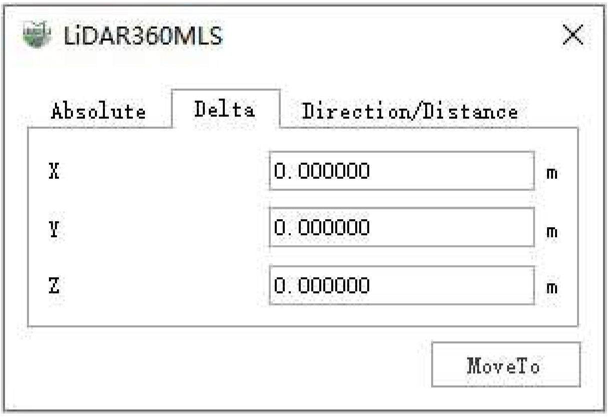
Select a vector

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 pop up the pop-up box



- **Absolutely**：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.

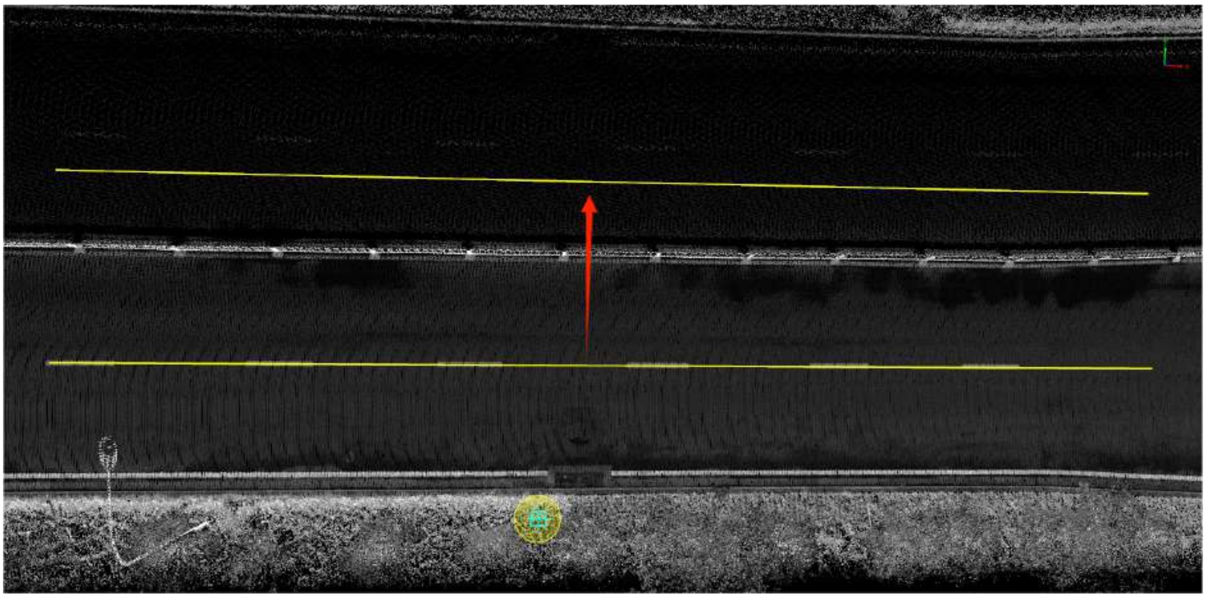
> Note: After you box select a vector object, the selected vector will be highlighted, and the details of all the selected vectors, including the vector layer and ID, will not be displayed on the right side of the main window at the same time.

Parallel Vector Copy

Function Description：Parallel copy function supports users to copy a vector object in parallel.

Steps

1.After clicking the **parallel copy**  button in 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.



Rotate Vector

Function Description：The rotation function supports the rotation of vector objects with a certain point as the center.

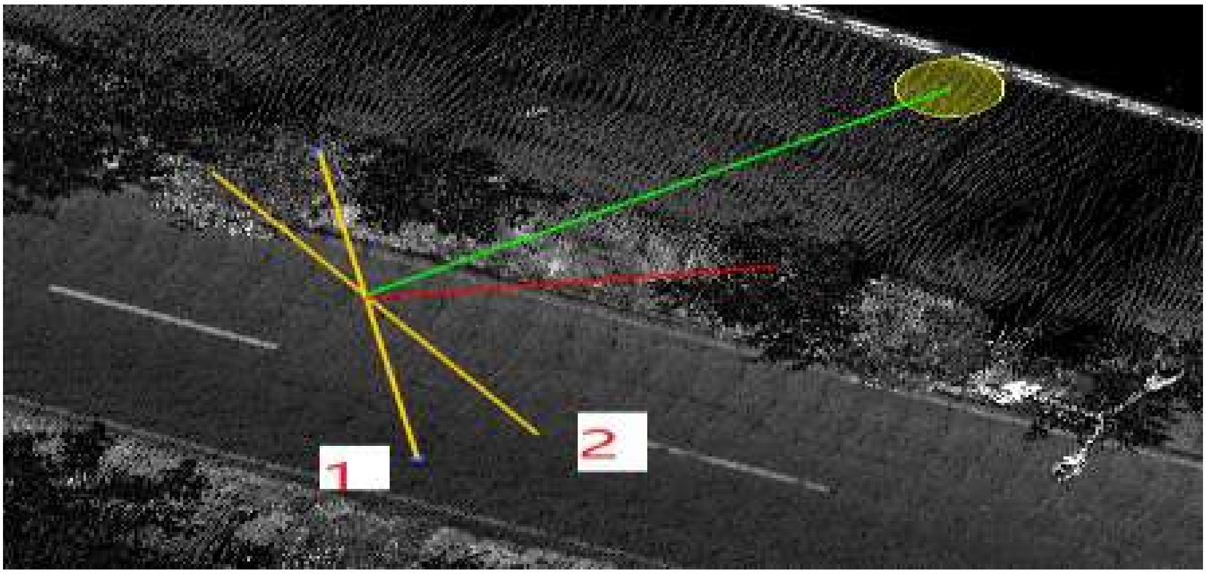
Steps

1.Click **Rotate** in vector editing  After pressing the button, 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.



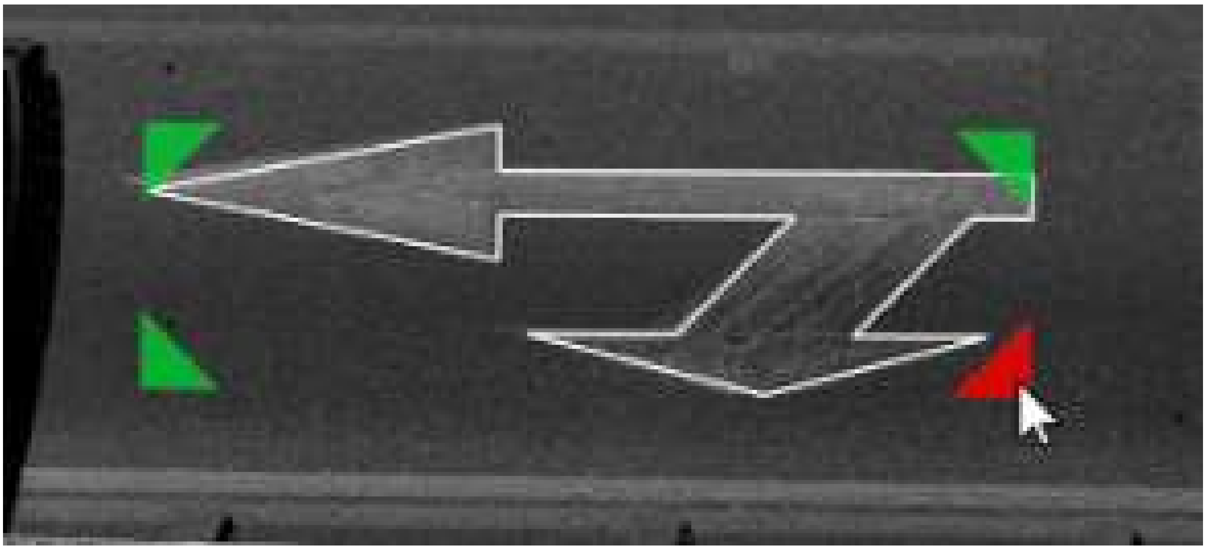
Rotate vector

Polygon Scale

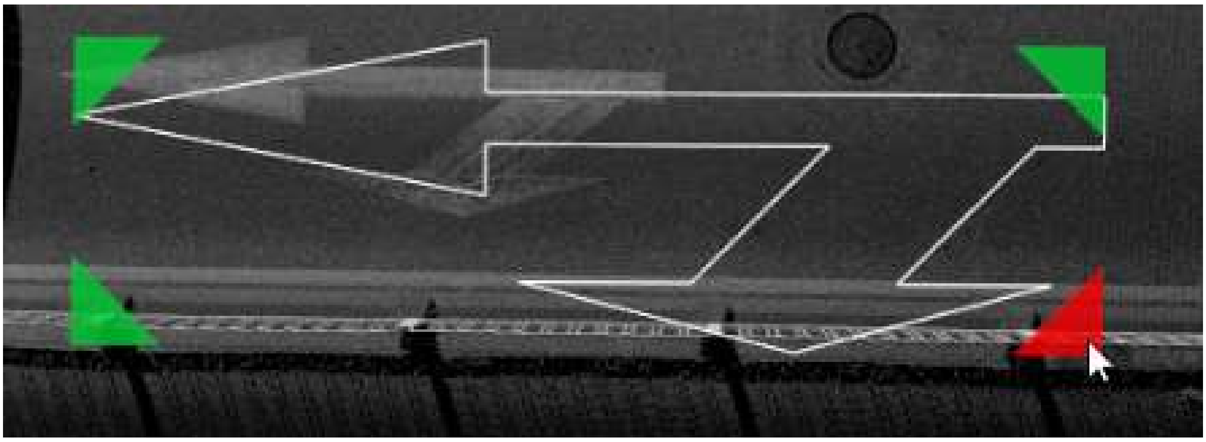
Function Description：Polygon scaling function supports scaling polygons in equal proportions along the horizontal direction.

Steps

1.After clicking the **Polygon Scale** 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.



Polygon Scale



Polygon Scale

Mirror Vector

Function Description：The mirror function supports moving a vector object to a new position in a mirror-symmetrical manner.

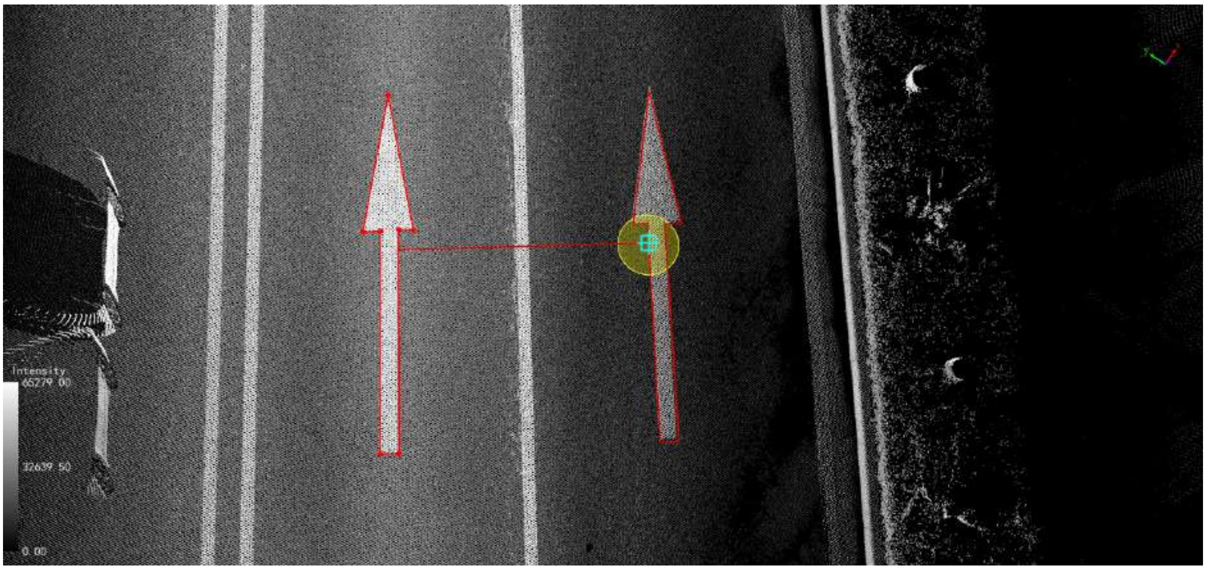
Steps

1.Click **Mirror** in vector editing  Button

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.

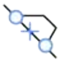


Mirror Vector

Reshape Line

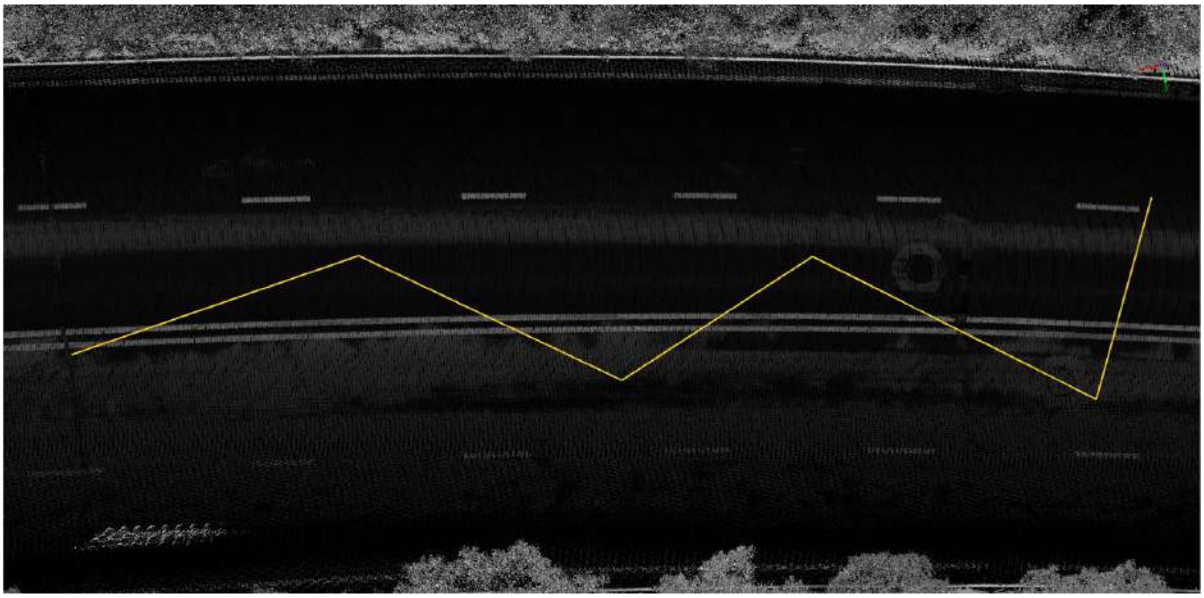
Function Description：Line reshape supports the adjustment of existing line objects.

Steps

1.After clicking the **Line Reshape**  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.

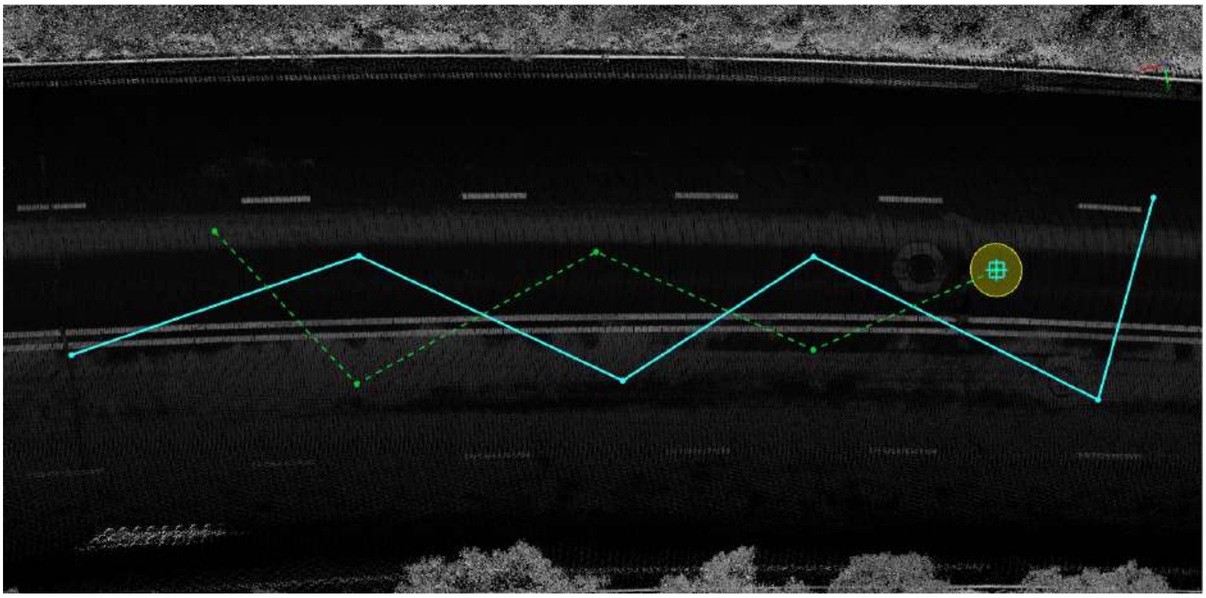
> Please ensure that the newly drawn polyline has more than two intersection points with the original line object, otherwise it will not be redrawn.

- **Original line object**: The object to be redrawn.



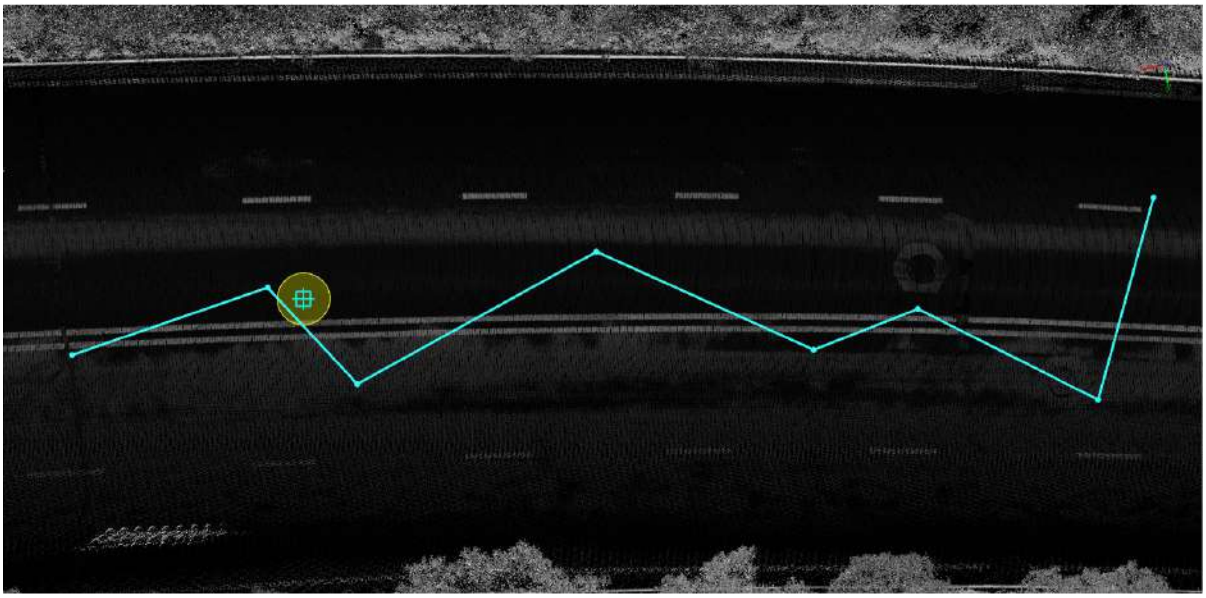
Original line

- **Drawing process**: The same steps as drawing a polyline, click on each point to be redrawn in turn, 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.

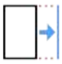


After reshape

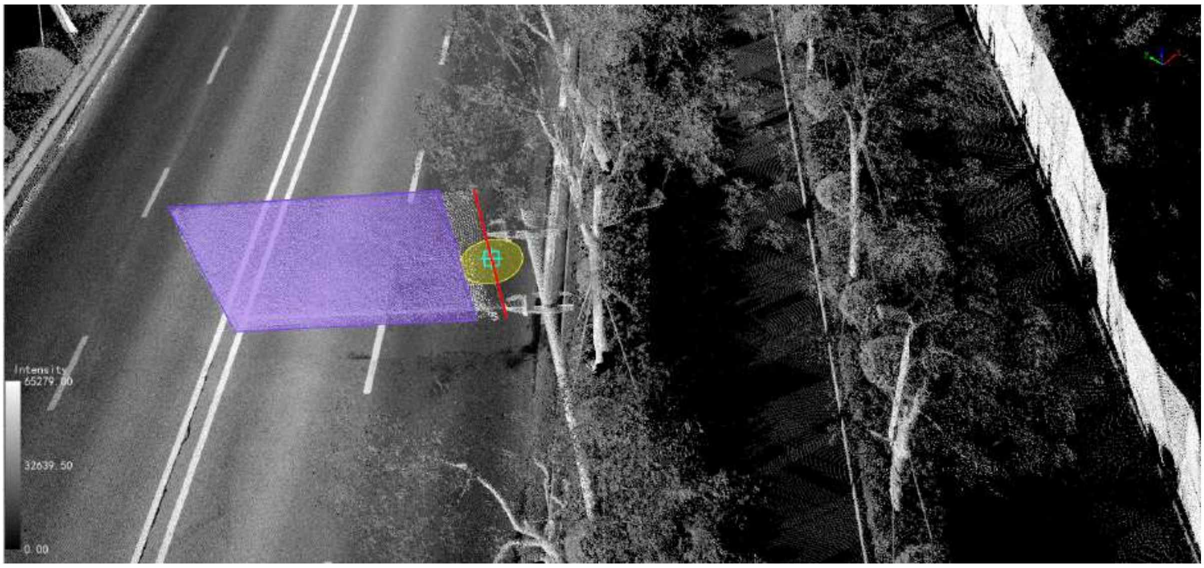
Parallel push edge

Function Description: The parallel edge push function supports moving one edge of the quadrilateral object in parallel to the position of the mouse.

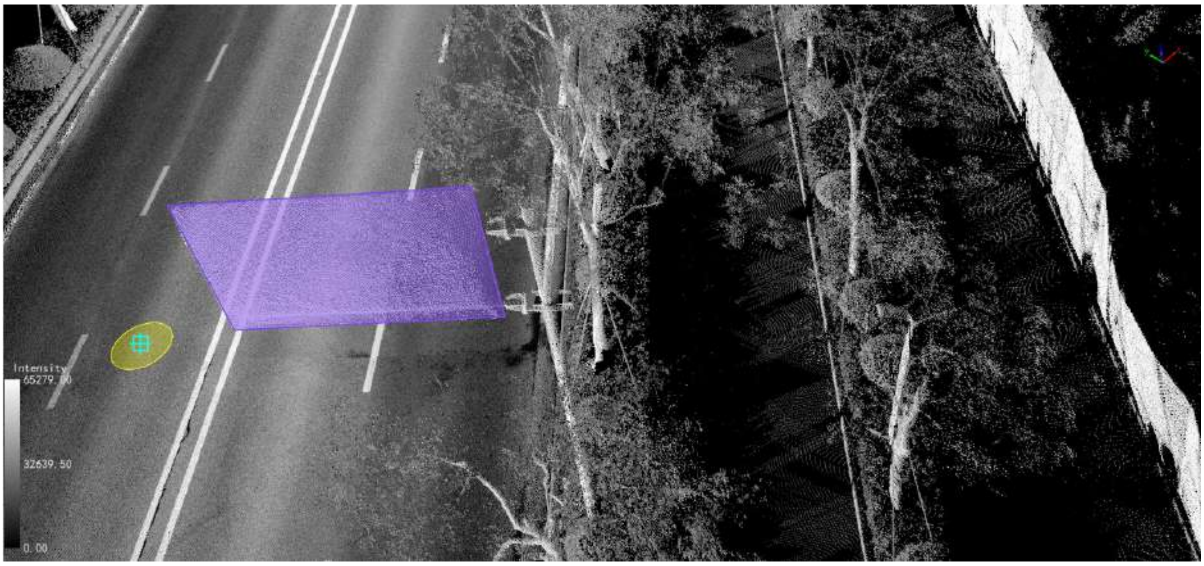
steps

1.Click **parallel push edge** in the vector editor  button;

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;



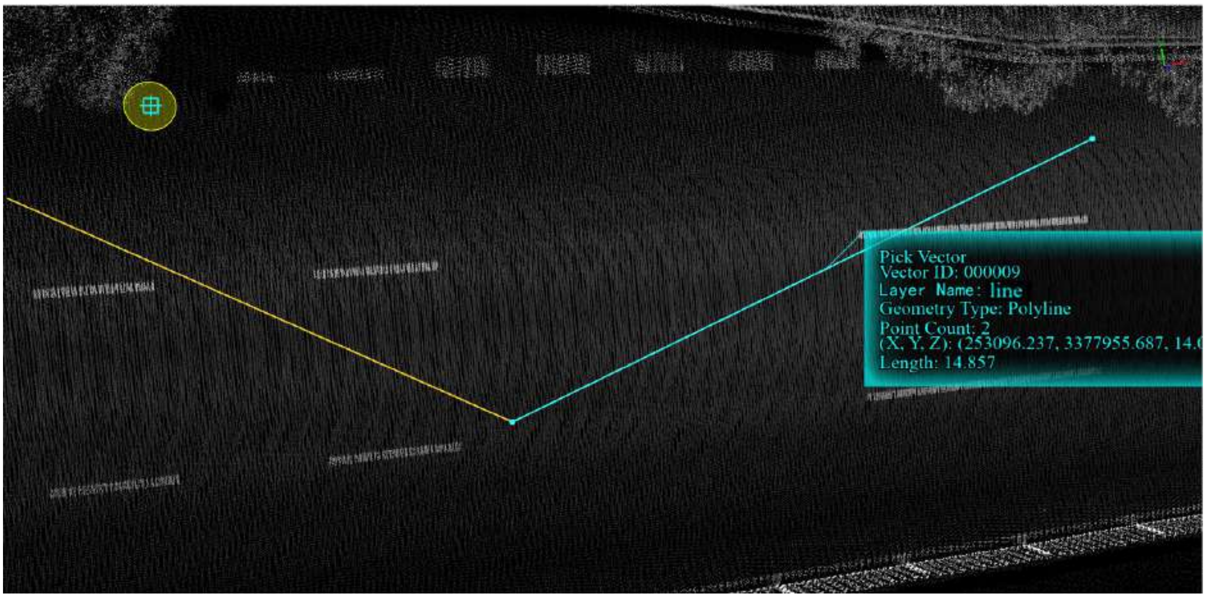
Merge Lines

Function Description：The merge line function supports merging two lines connected end to end into one line object.

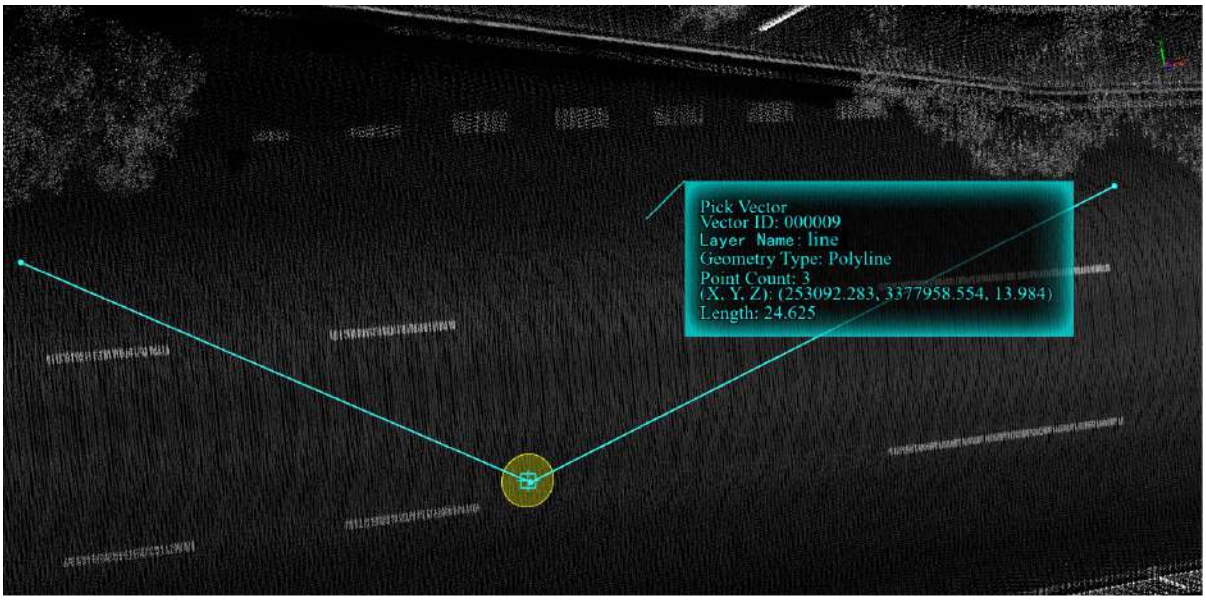
Steps

Click **Merge Line** in vector editing , 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 lines

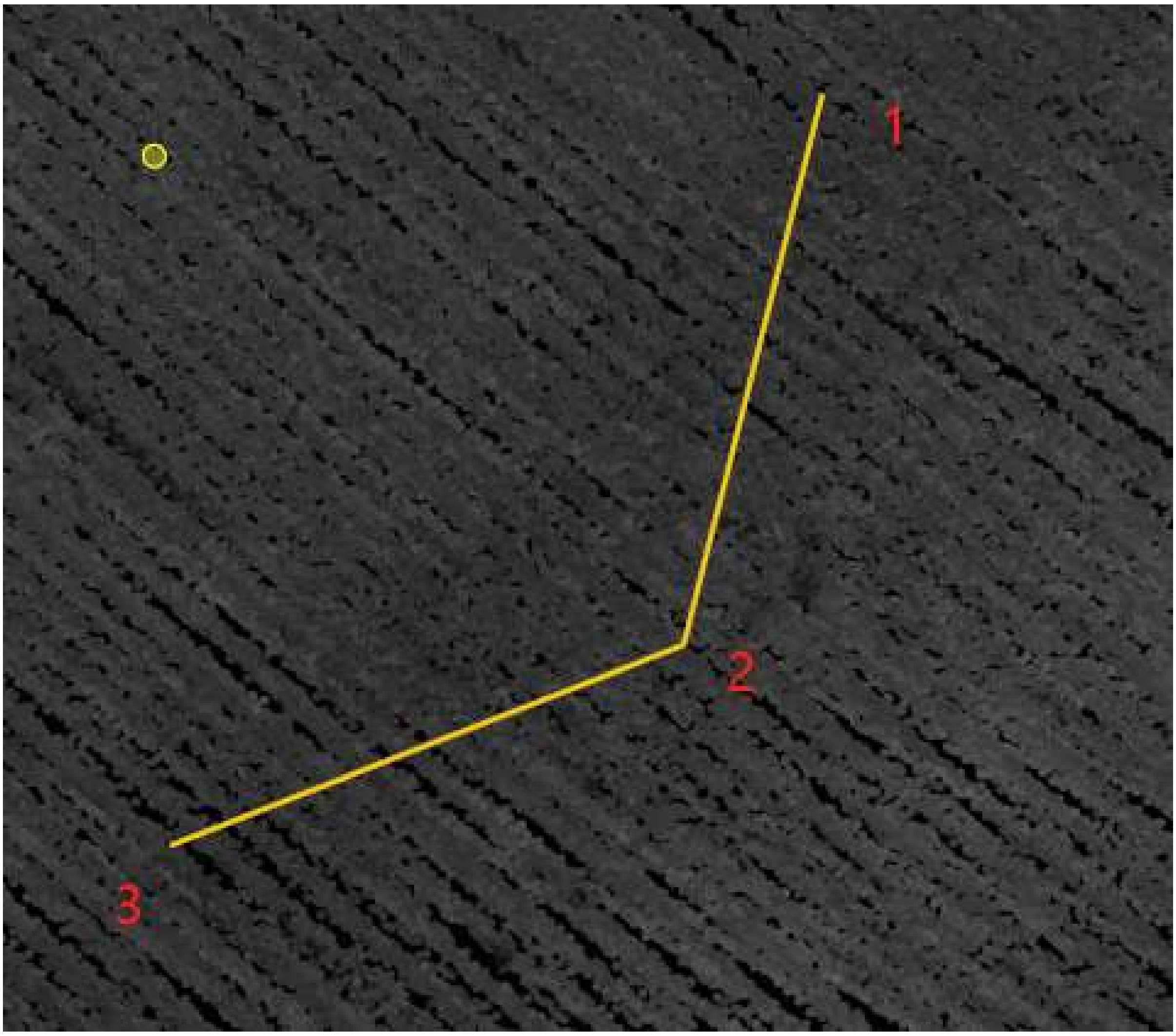
Function Description：The break line function supports breaking a line object into two line objects connected end to end.

Steps

Click **Break Line** in vector editing  button, move the mouse to the vector, select the place you want to interrupt, and left click to complete the interruption of the line.

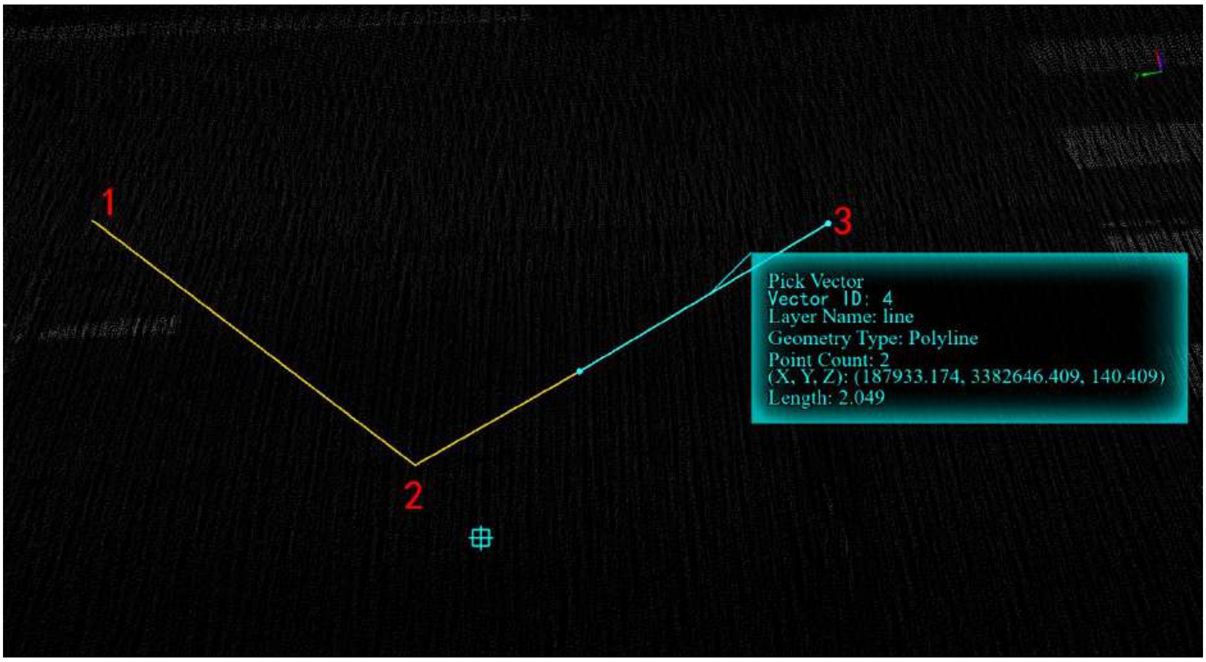
> If the selected node is a node, it will break from the node and change from a line string to two lines; if the mouse selected is not a node, insert a node at that position and break it into two lines.

- **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 Lines

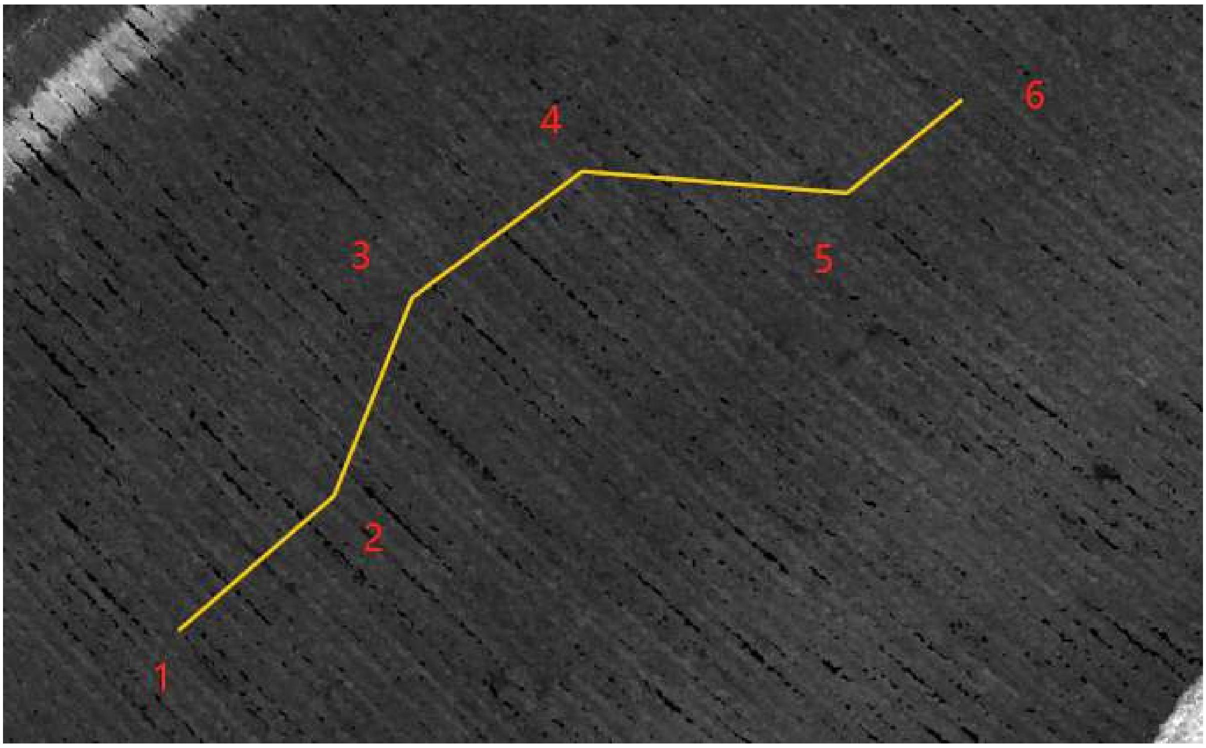
Function Description：The split line function supports splitting a solid line object into a dashed line object. See the picture below for details.

Steps

1.Click **Split lines** in vector editing  button.

2.Left click the line you want to split to complete the line segment splitting.

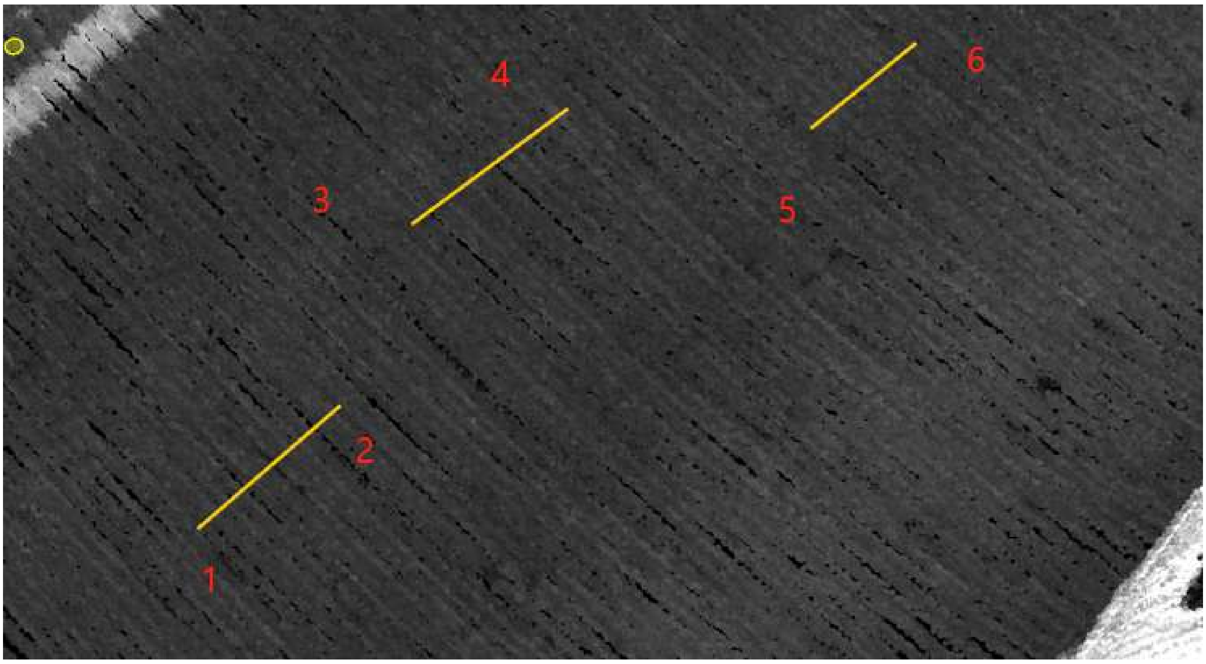
- **Before vector line splitting**: The original solid line consists of 6 nodes.



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

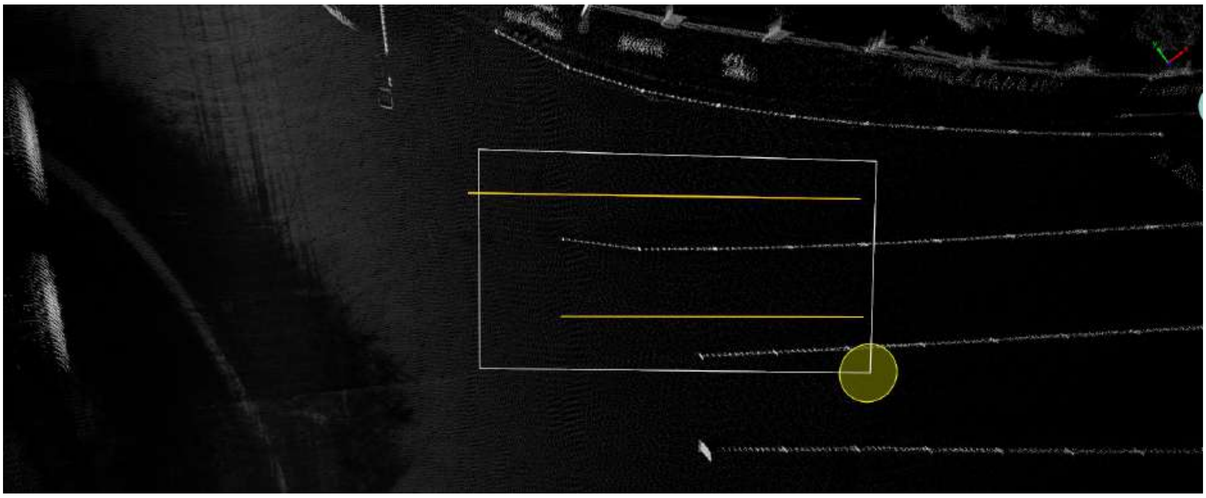
Modify Line Length

Function Description：Stretch or cut the start or end of a series of lines based on the target line drawn by the user.

Steps

1.Click **Modify Line Length** in vector editing  button, 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).



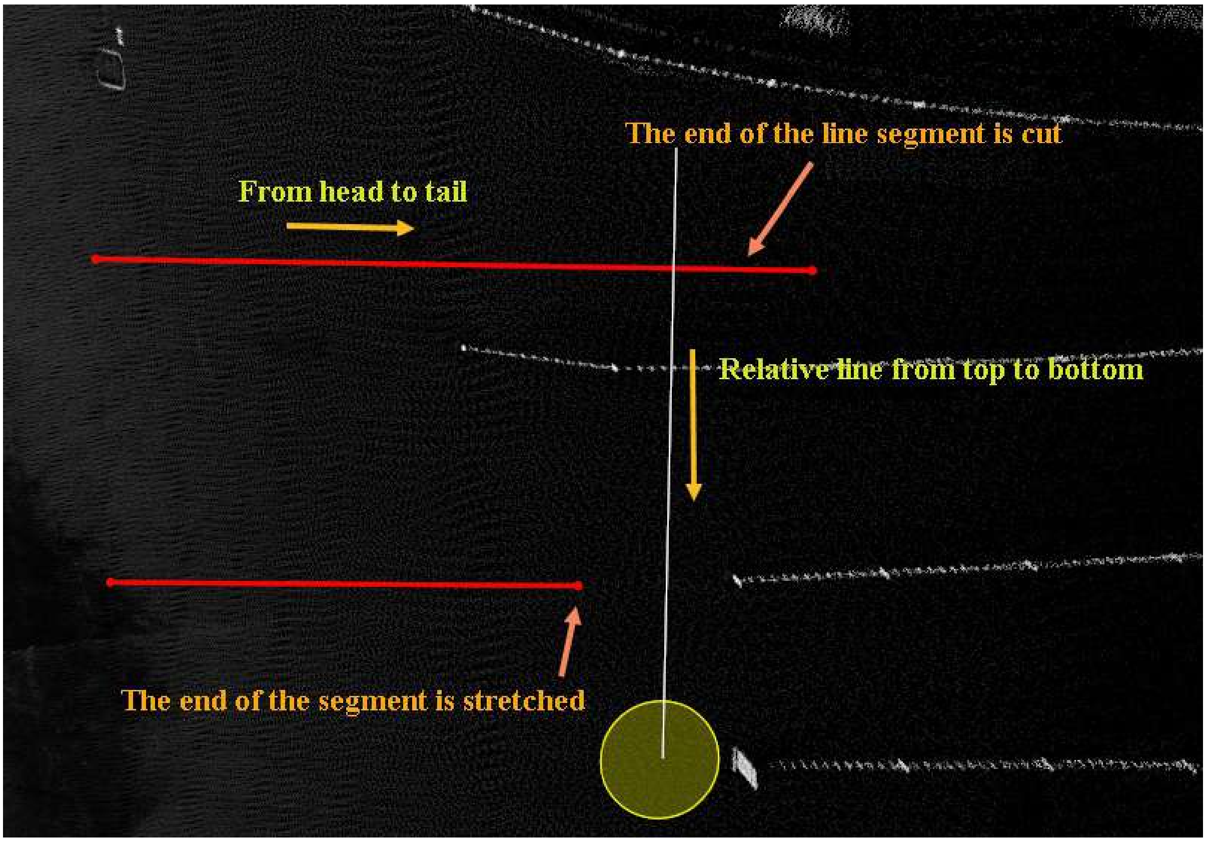
Frame selection of line objects

2.Right-click to select two points in turn to form a **cutting/stretching** reference line to complete the cutting or cutting. 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

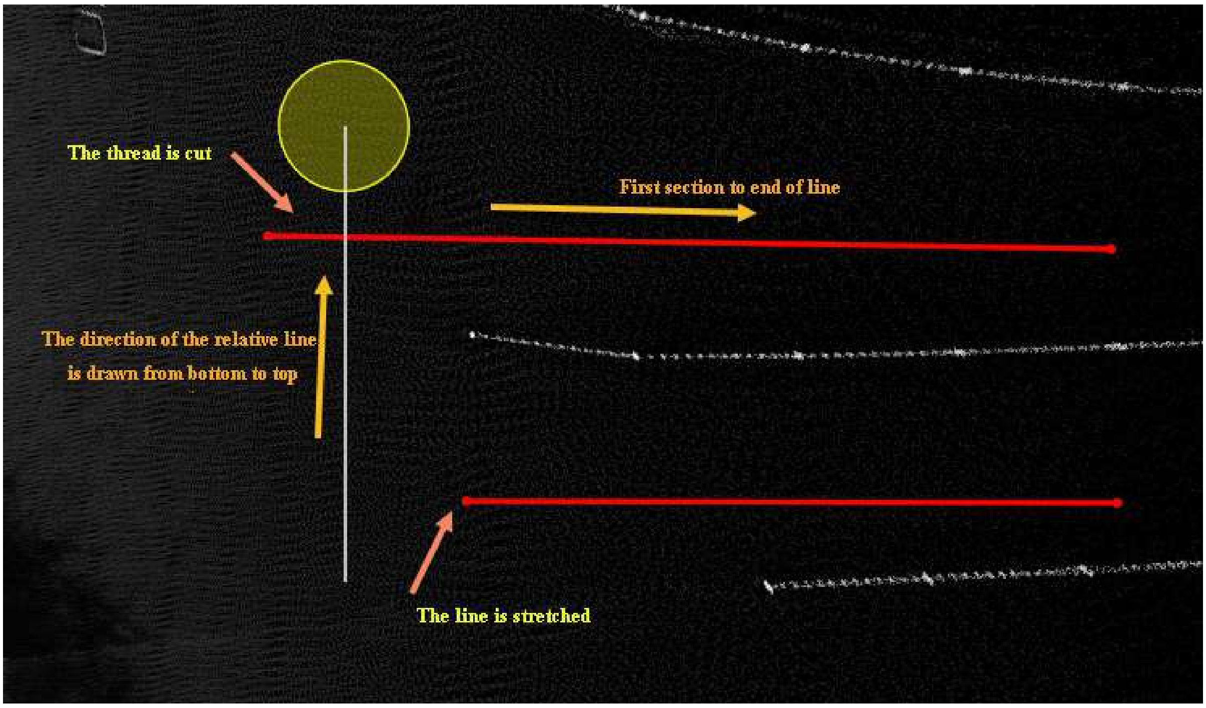
- Draw from left to right relative to the direction of the selected line



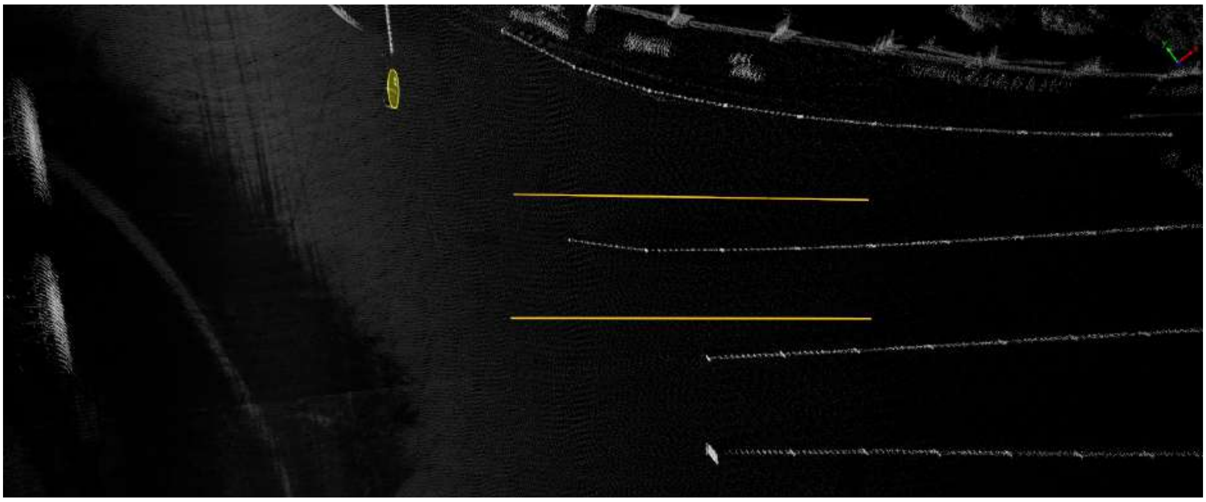
Draw the target line from left to right


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

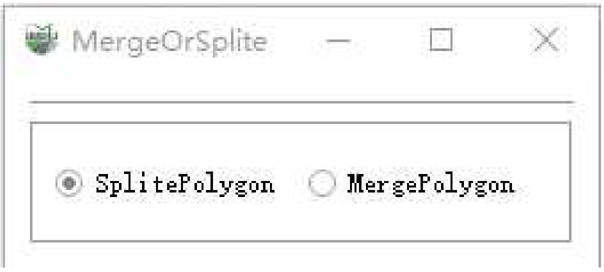
Cut and Merge Polygons

Function Description：Cut 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 **Polygon cut and merge** in vector editing , the operation mode selection dialog box will pop up.



The pop-up window selects the current operation as cutting or merging

Parameter Description:

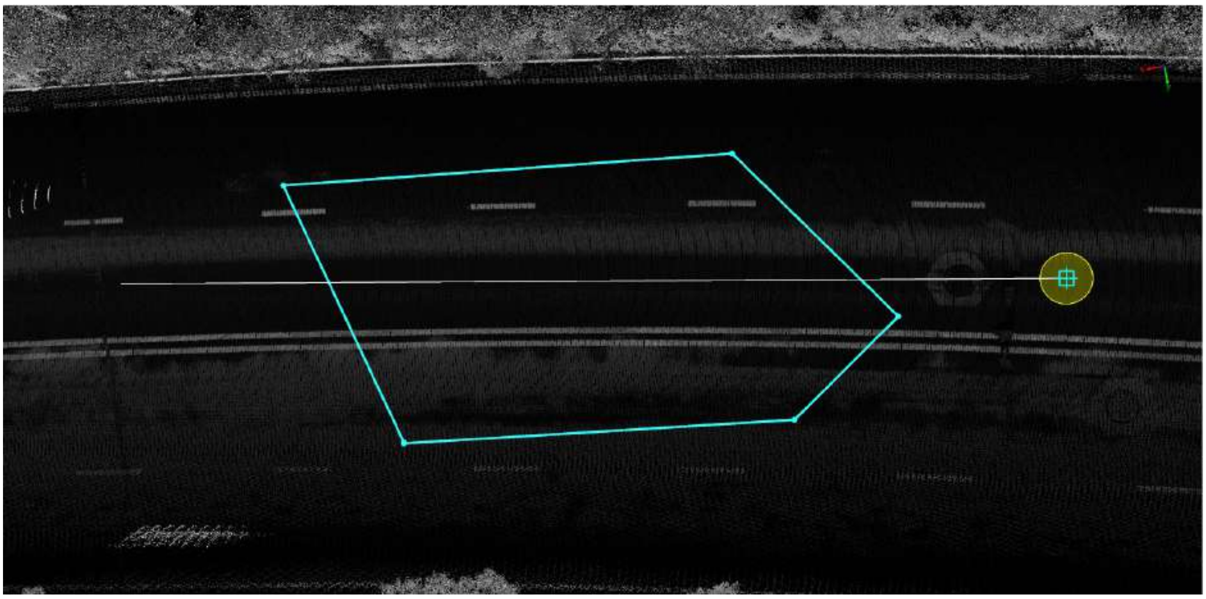
- **Cut Polygons**: Cutting mode.
- **Merge polygons**: merge mode.

Crop Polygons

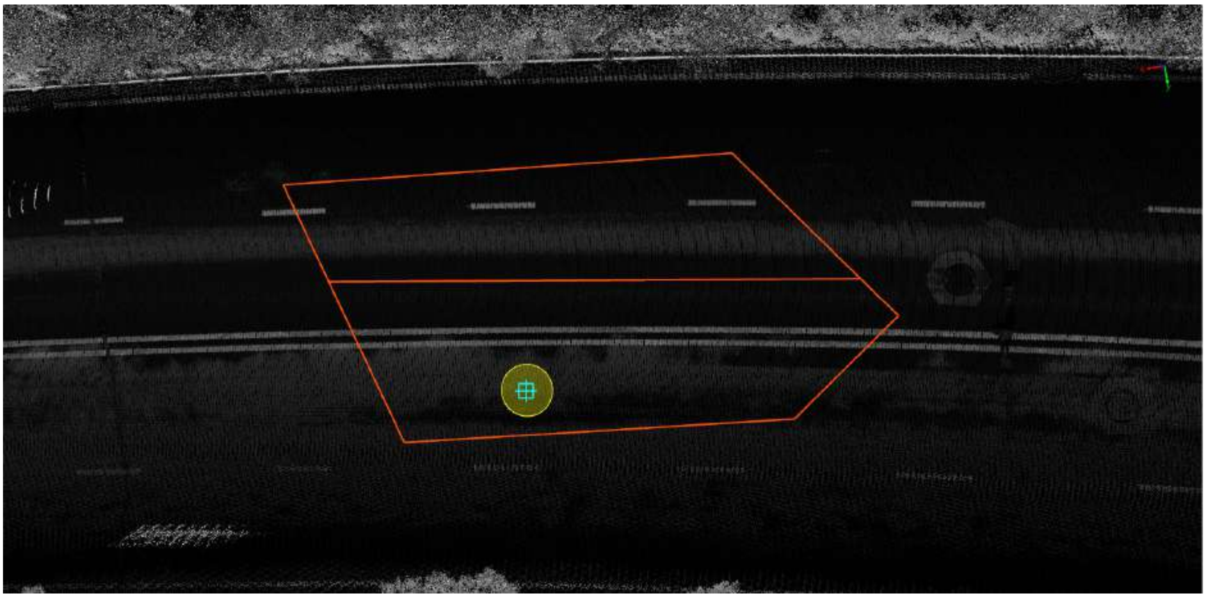
1.Left click to select the polygon to be cut, and the polygon will be highlighted in red.

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

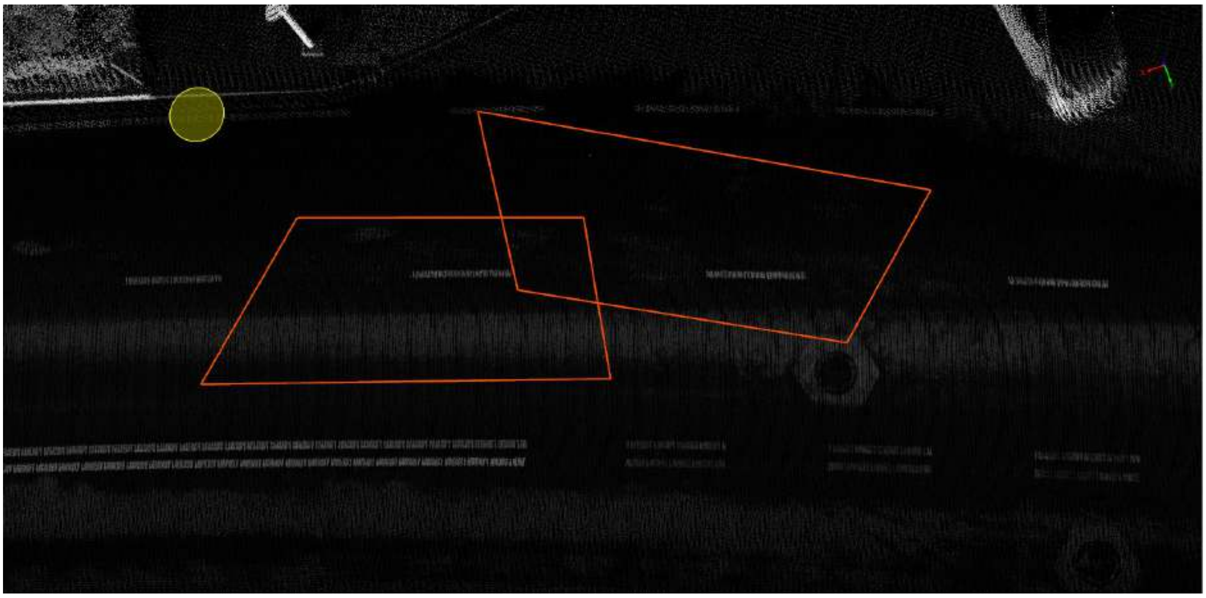


Cutting result

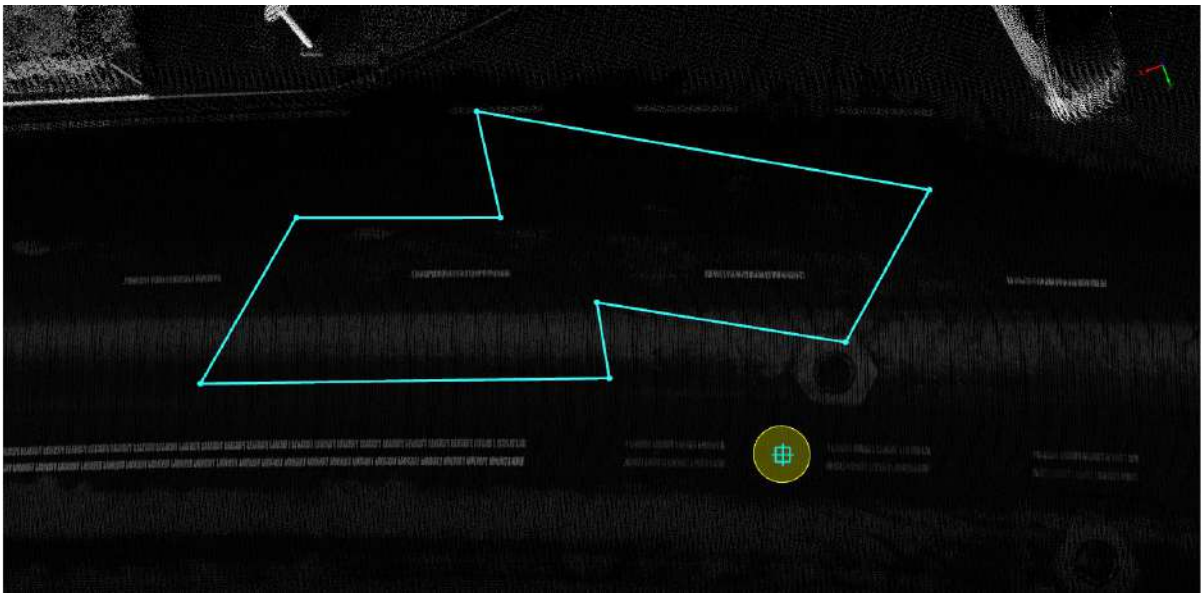
Merge Polygons

1.Left click the two polygons that need to be merged to complete the merging of the polygons.

> Note: The polygons that need to be merged do not need to have intersections



Merge the first two polygons



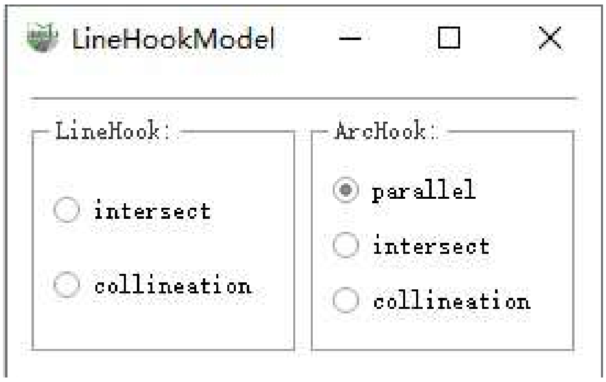
Merge result

Linking Lines

Function Description：Linking 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 Lines** in the element editing function group button, the link mode pop-up window will pop up, select the corresponding mode to perform the link operation, see the detailed description below for details.



The pop-up window selects the connection mode and the relative relationship of the lines

Parameter Description

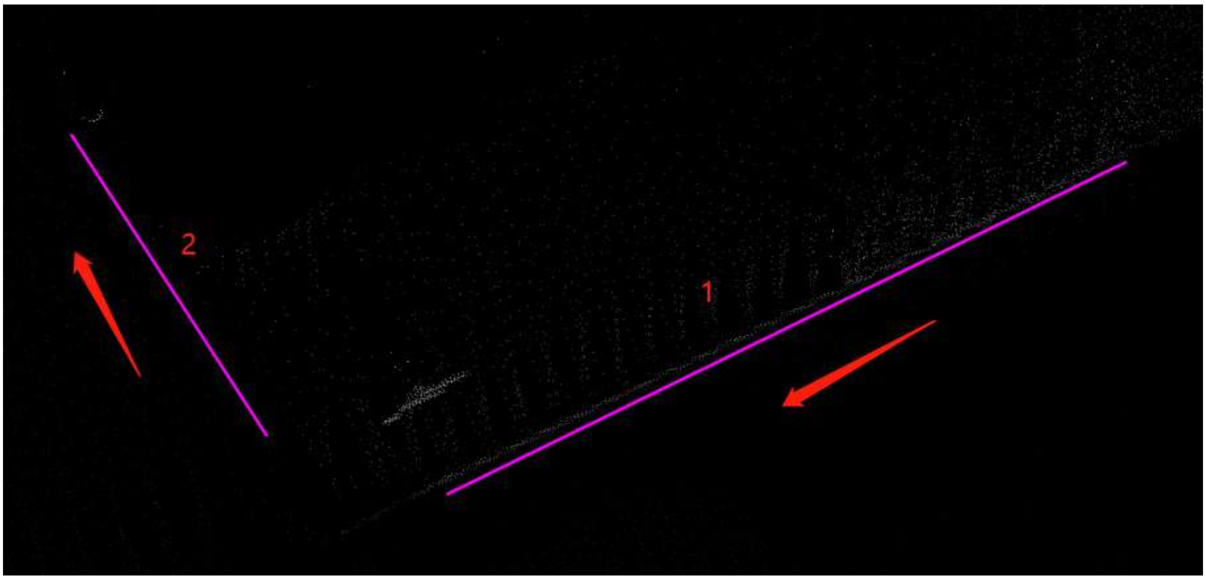
- **Line connection**: According to the spatial geometric relationship of the two lines, they are connected end to end or extended to intersect.
- **Arc linking**: According to the spatial geometric relationship of the two lines, the end points of the two lines plus a user-defined point are selected, 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.**Line linking**

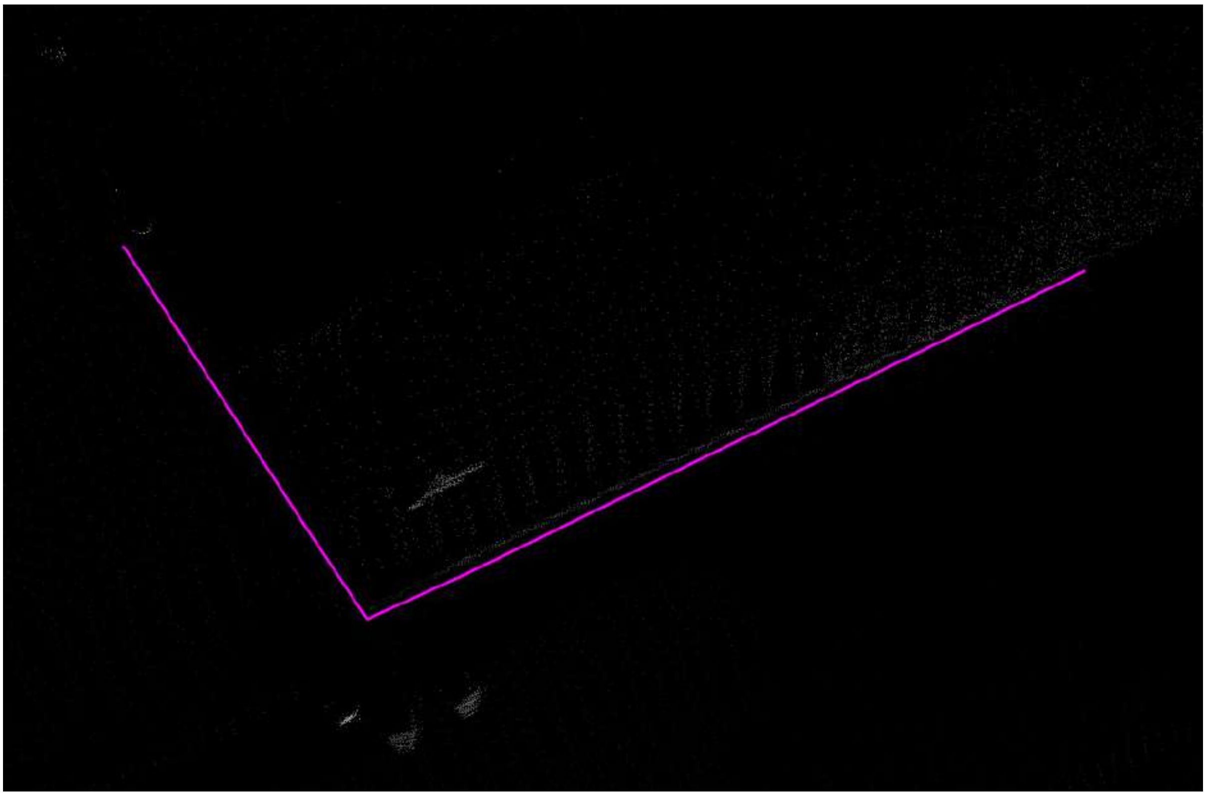
（1）**intersect**

Select **Line linking**, 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 linkied 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: Intersection means that the straight lines passing through two line segments can intersect at one point.



Line linking

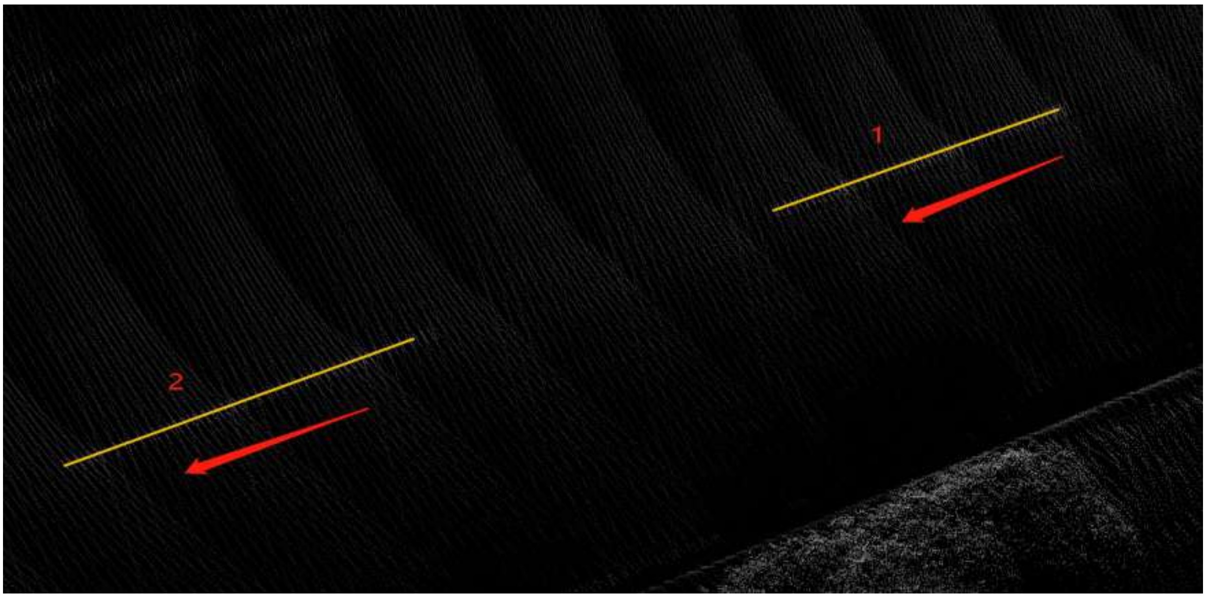


Linking result

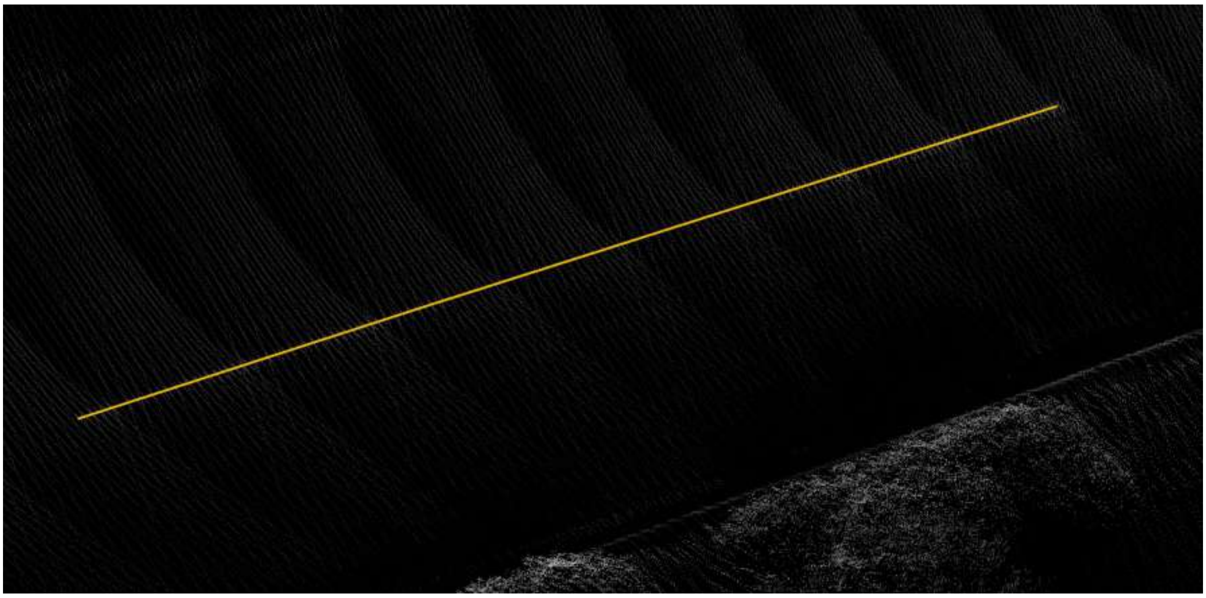
（2） **Collinear**

Select **Line Connection**, and the relative relationship between the lines is **Co-Line**, 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: Collinear means that the straight line passing through the two line segments is approximately on a straight line.



Line linking

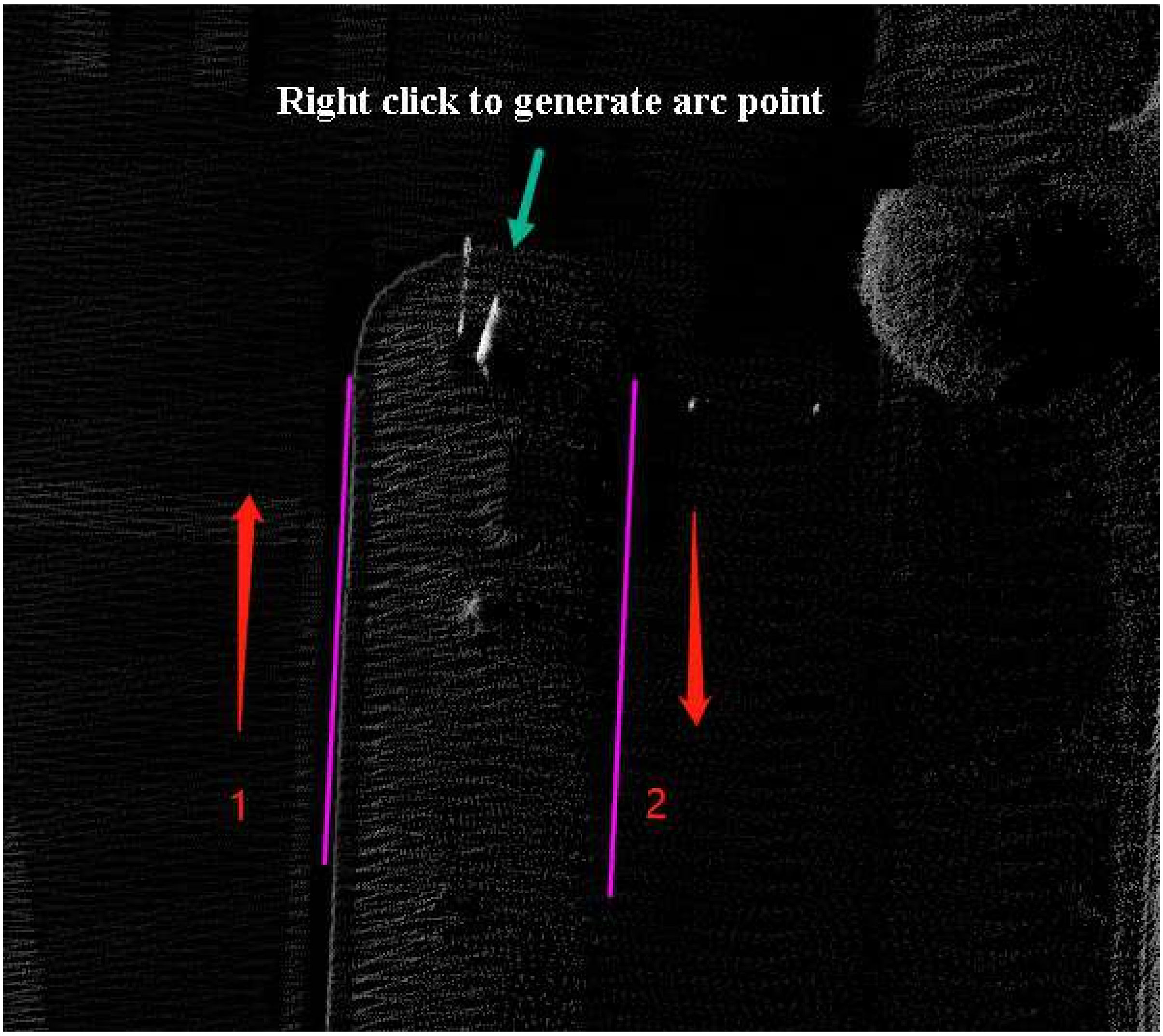

Linking result

3.**Arc hooking**

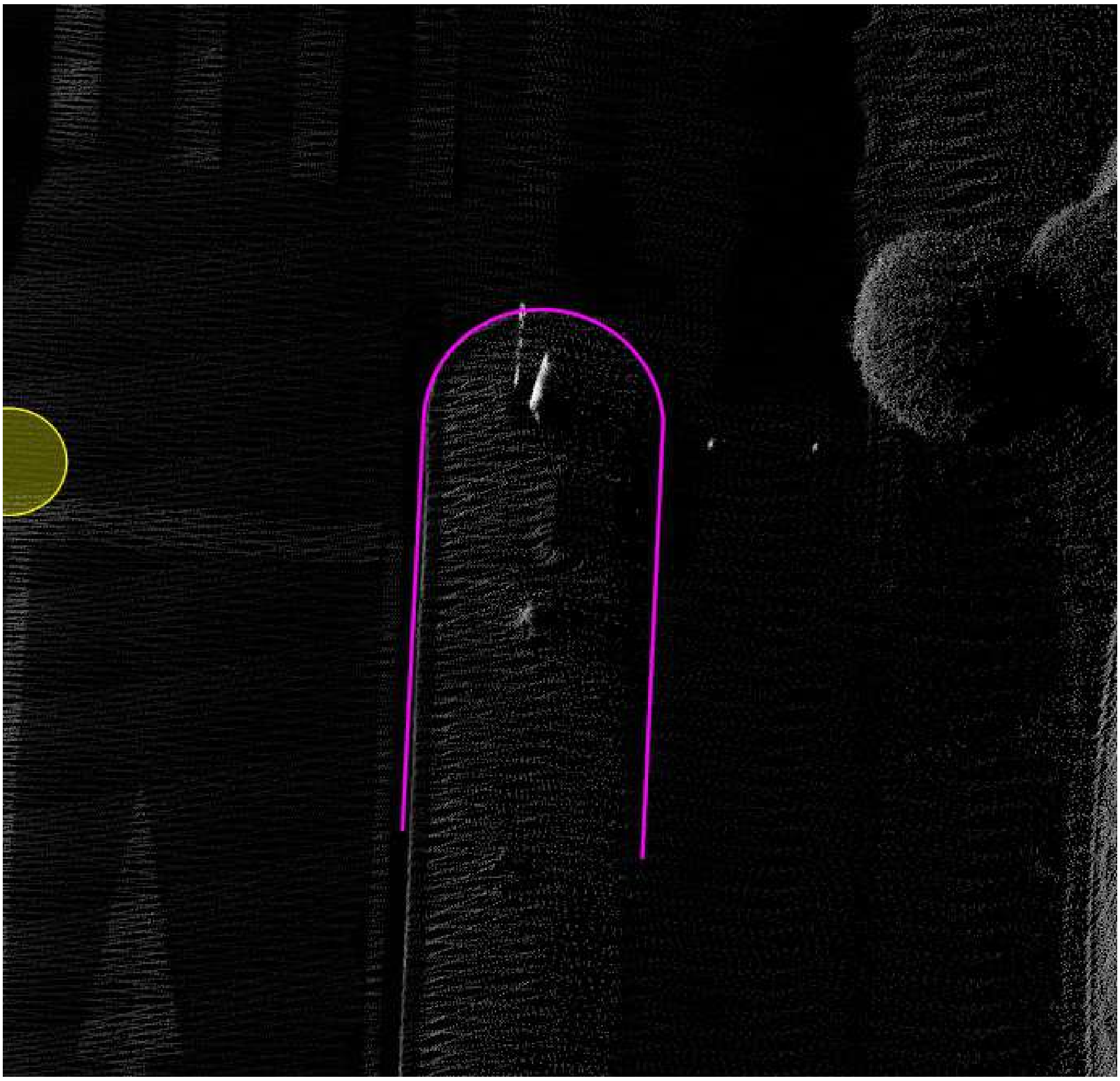
（1）**parallel**

Select **Arc Hook**, 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 As shown, 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.



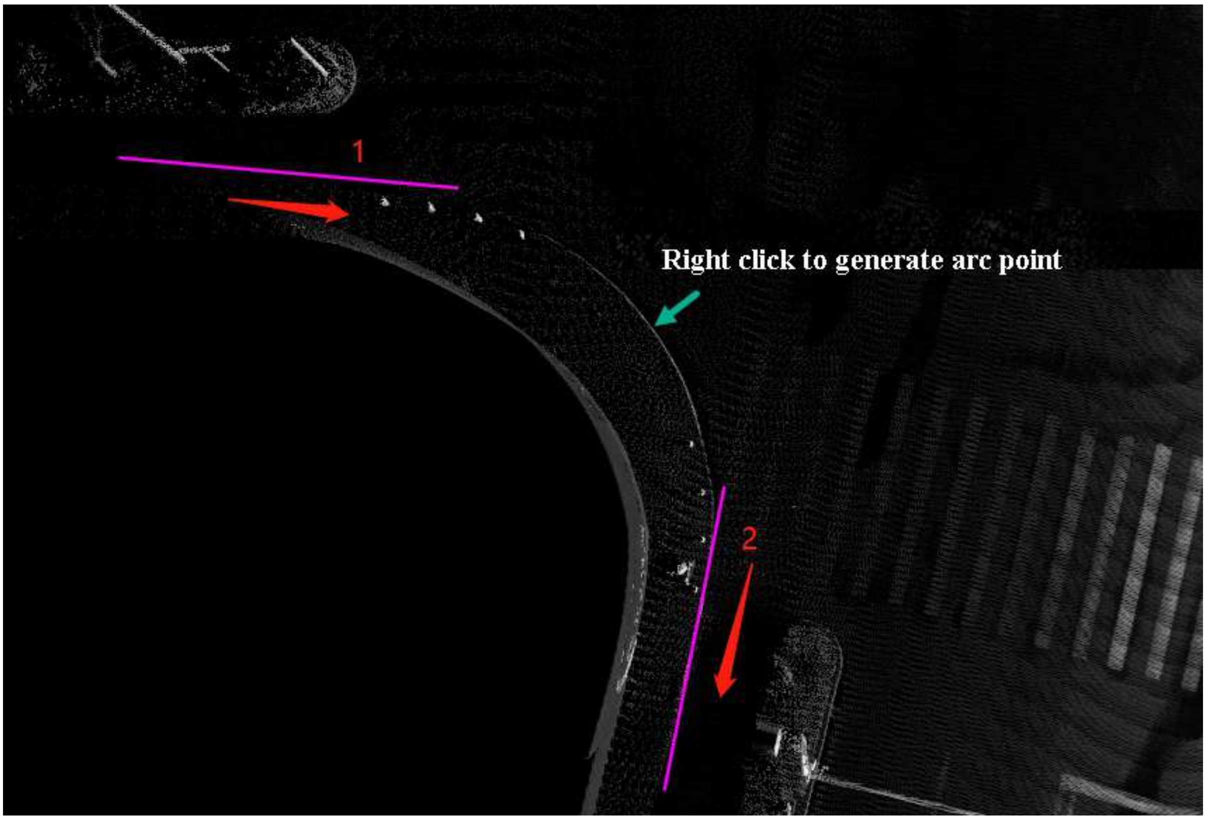
Line Linking


Linking result

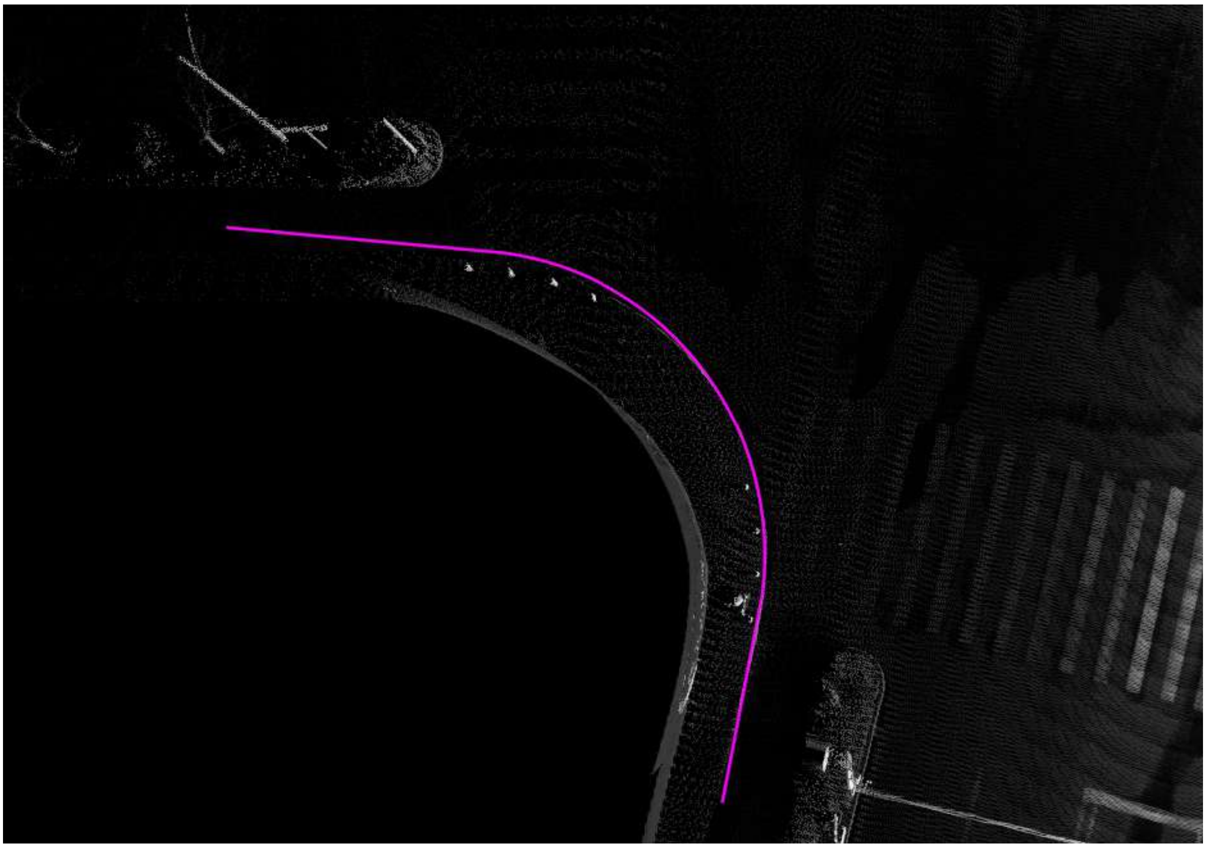
（2）**intersect**

Select **Arc Hook**, 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 As shown, 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: Intersection means that the straight lines passing through two line segments can intersect at one point.



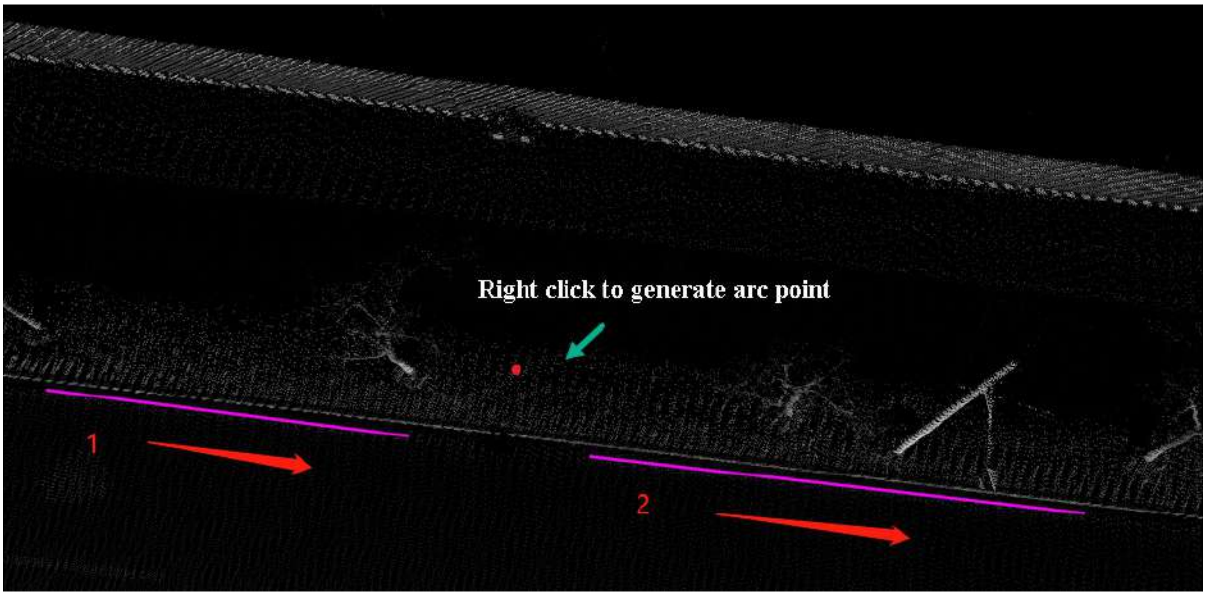
Line linking


Linking result

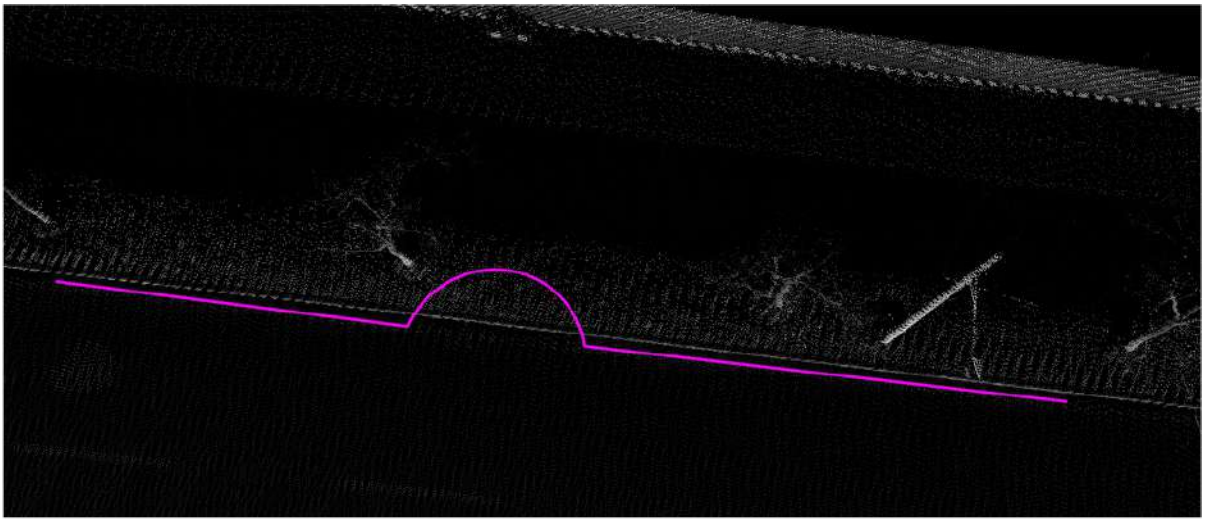
（3）**Collinear**

Select **Arc Hook**, and the relative relationship of the lines is **collinear**, 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 follows 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 **collinear mode** can be completed.

> Note: Collinear means that the straight line passing through the two line segments is approximately on a straight line.



Linking lines


Linking result

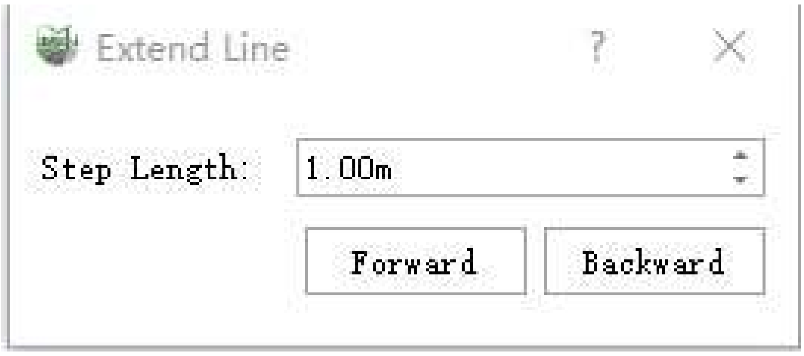
Line Extension

Function Description：The ine extension function can 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.**Line Extension** dialog box pops up



Line xxtension dialog box

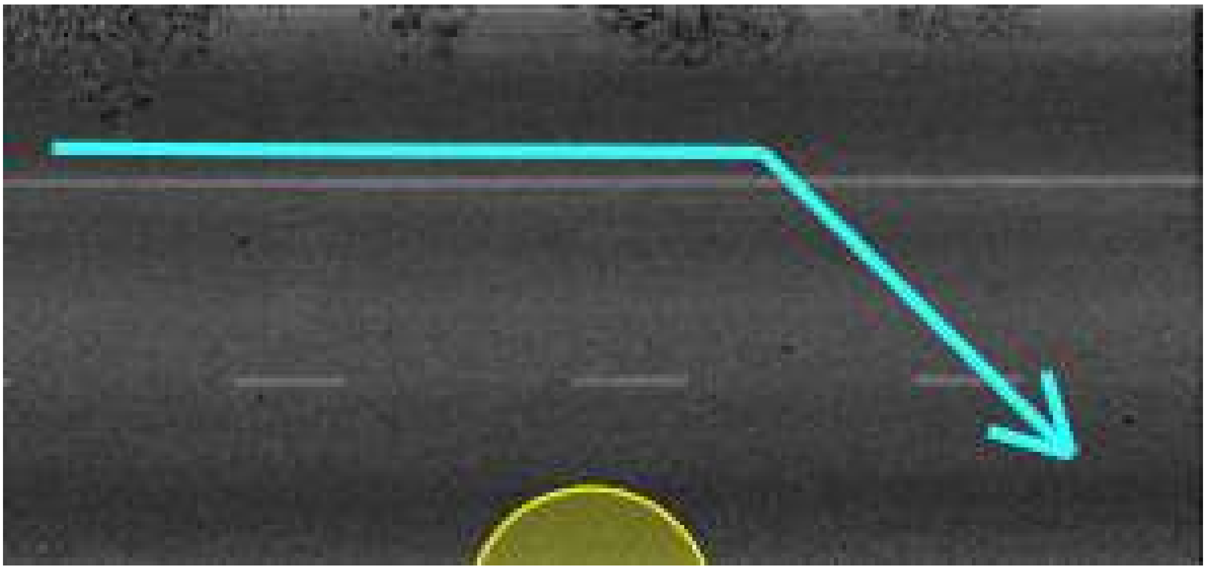
Parameter description:

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.

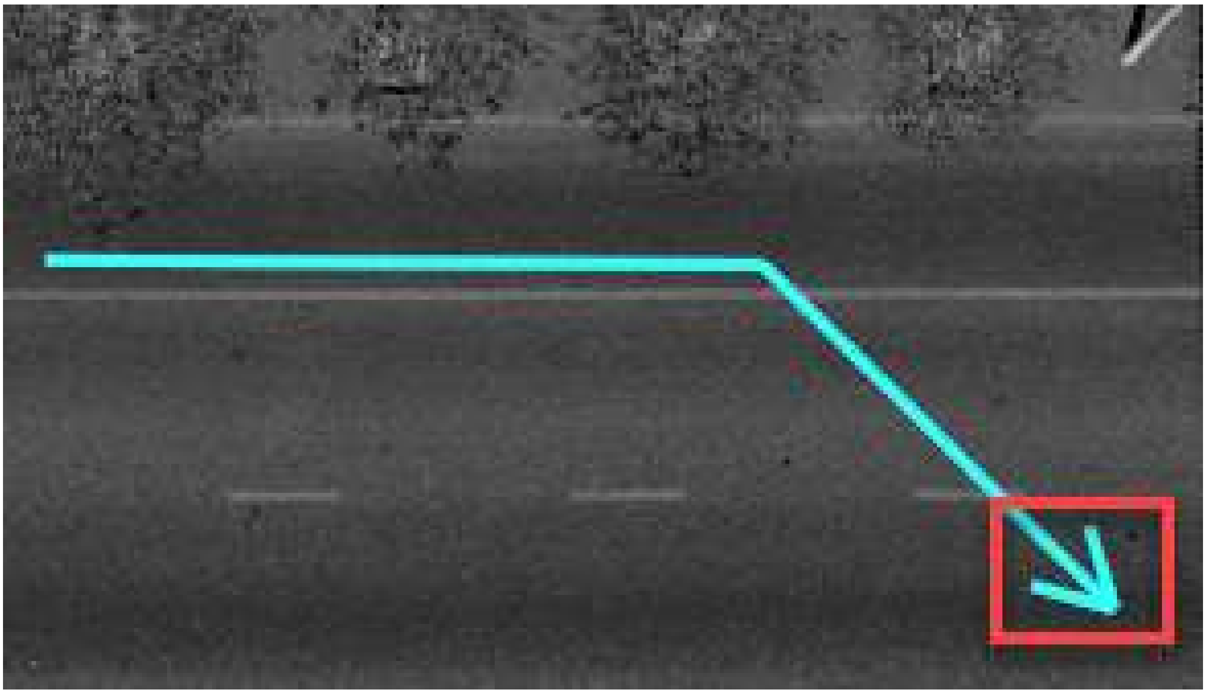
Shortcut key	Function
F	Forward
B	Backward
Shift+z	Fallback extension
H	Hide shortcut key pop-up

3.Click the mouse to select the line object to be extended.

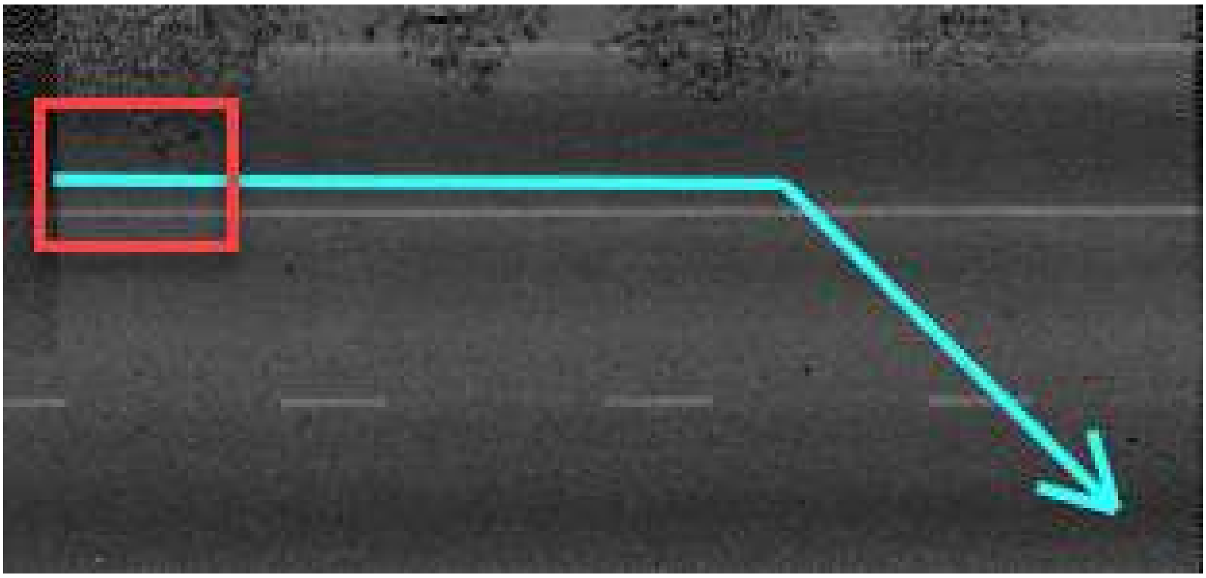


Select line to be extended

4.Click the forward or backward button in the dialog box, and the line object will be extended in the corresponding direction.



Extend forward



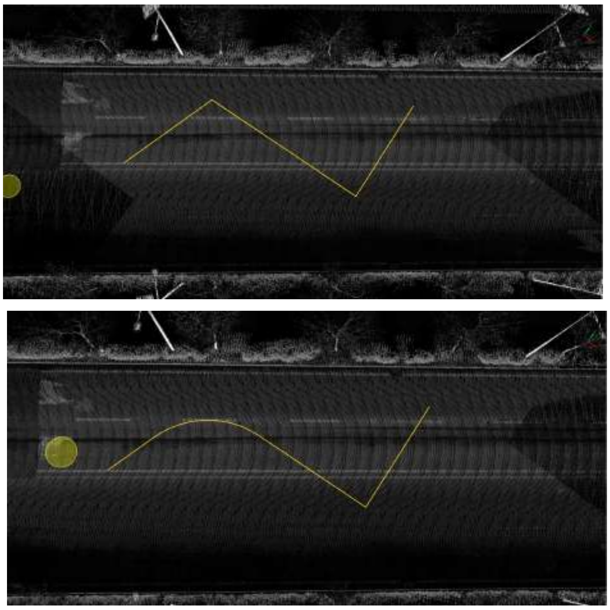
Extend backwards

Inner Circle Angle

Function Description: The inner corner circle function adjusts the size of the angle through the edge of the same node.

steps

1.Click **Inner Fillet** in the vector editor  button, click the left mouse button to click the two intersecting line segments in turn, move the mouse, determine the required angle according to the requirements, and double-click the mouse to complete.



Round corners

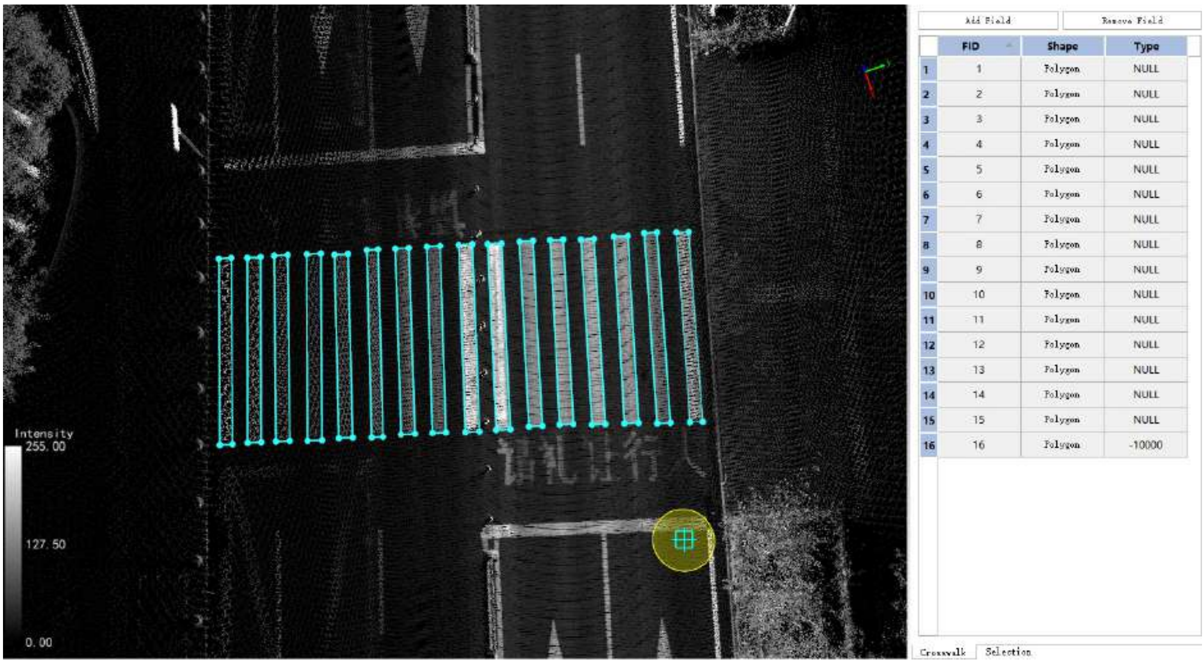
Split object

Function Description: Split the vectors of the same layer into a single vector object.

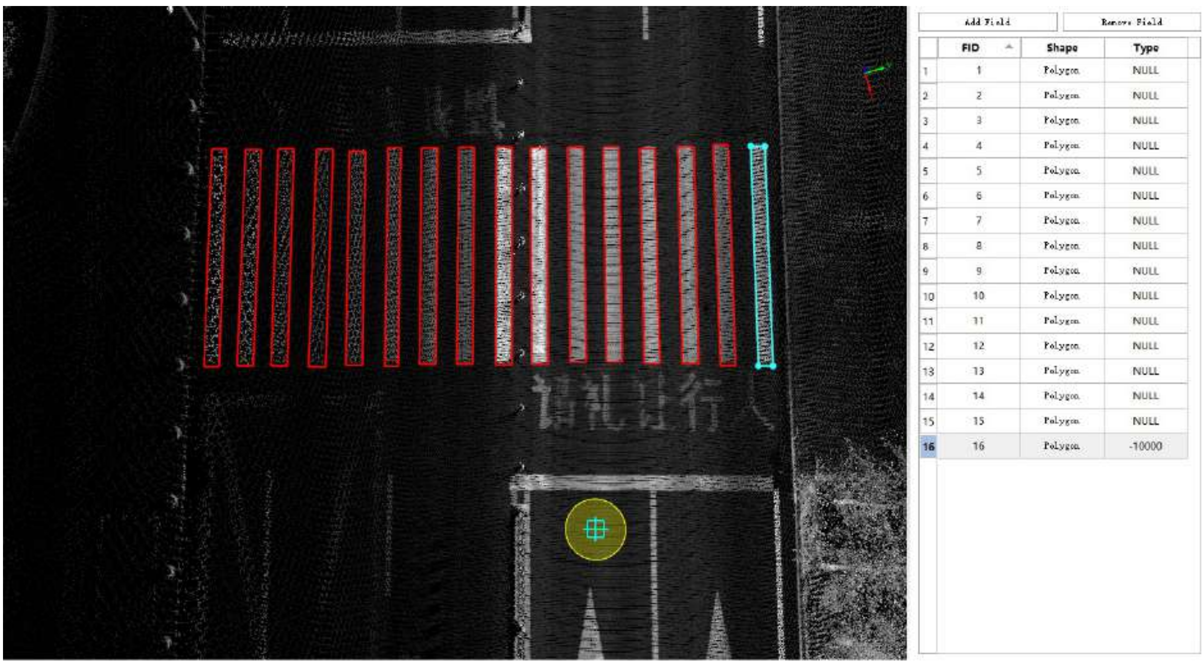
steps

1.Click **Split Object** in the vector editor  button, click the left mouse button and drag the mouse to select the vector object to be split.

> This function can only process the vectors of the same layer, the details are as follows:



Vector to be split



Split vector

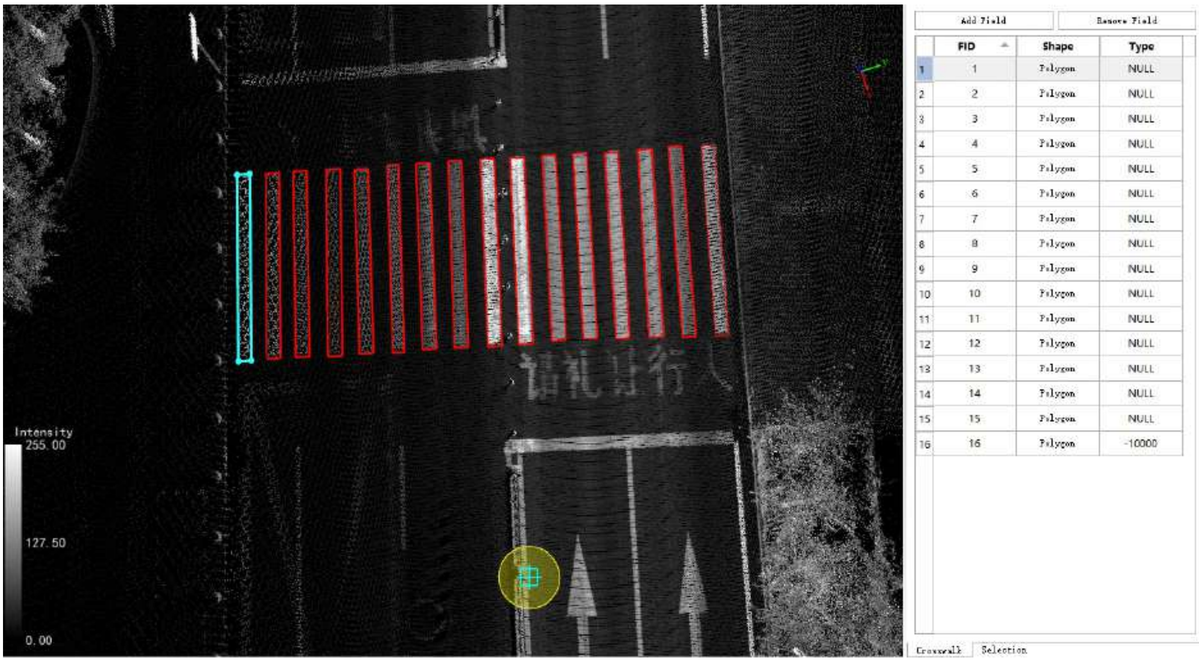
Combining objects

Function Description: Combine different vectors of the same layer to form a vector object.

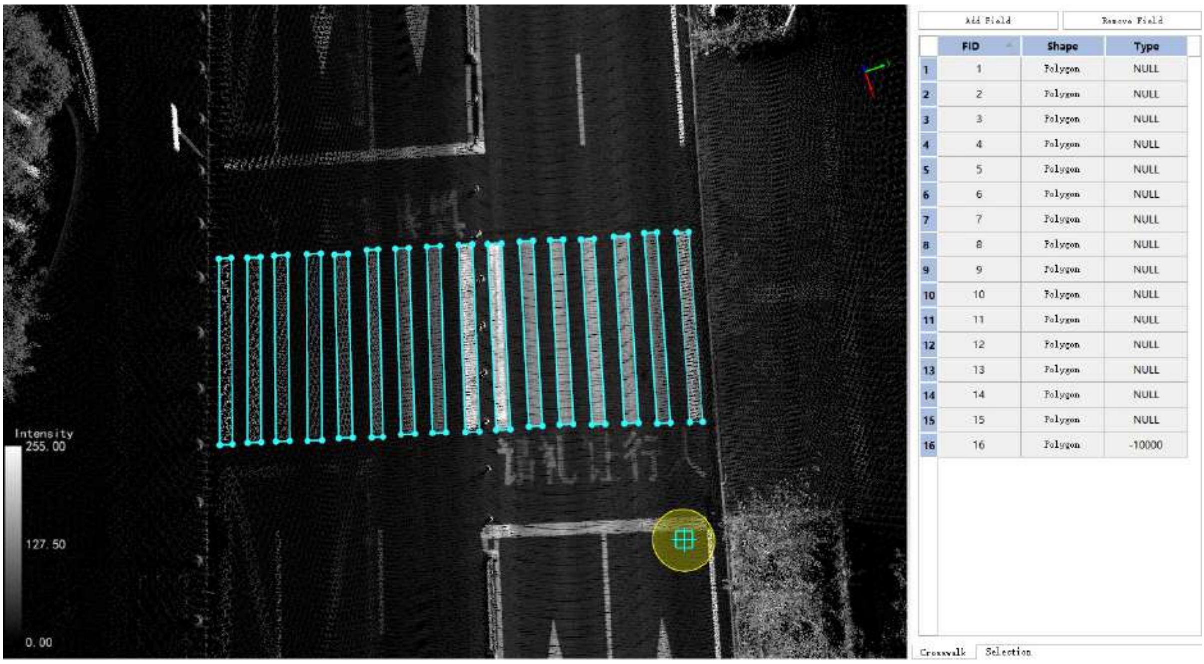
steps

1.Click **Combination Object** in the vector editor button, click the left mouse button and drag the mouse to select the vector objects that need to be grouped together.

> This function can only process the vectors of the same layer, the details are as follows:



Vectors to be combined



Combined vector

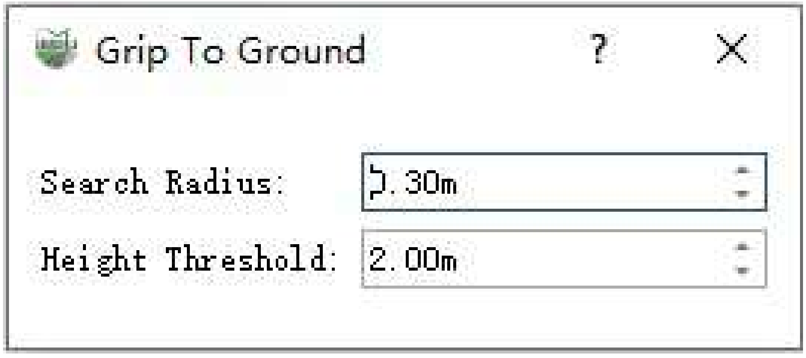
Grounding

Function Description: The grounding function can make the vector objects that are not completely grounded or floating in the air completely grounded to ensure the accuracy of the vector data elevation.

Steps

1.Click the **Grounding**  button in the vector editor.

2.The **Grounding** dialog box will pop up.

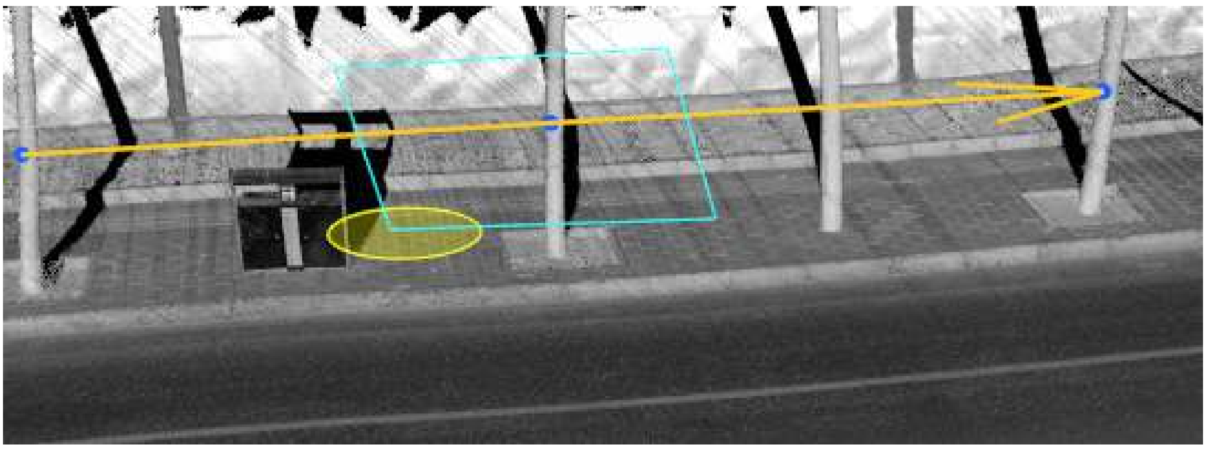


Grounding Dialog

Parameter Description:

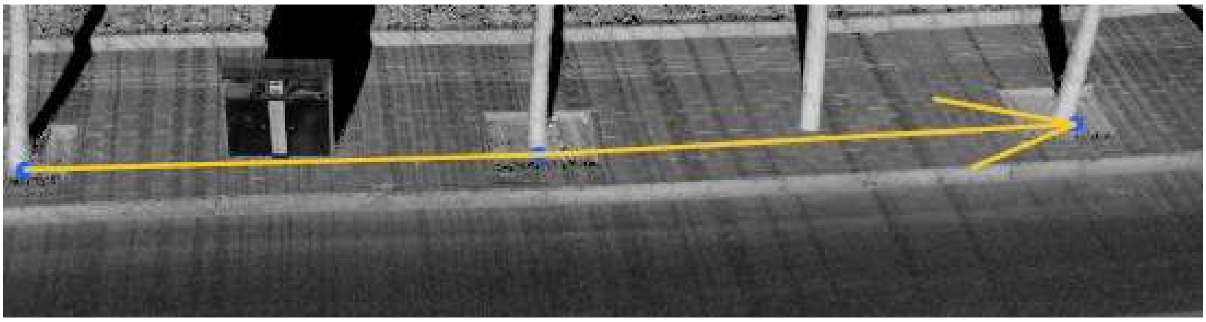
- **Search Radius**: The default is 0.3 meters. Set the radius to find points on the perimeter plane.
- **Altitude Threshold**: The default is 2.0 meters.Indicates the distance between the current vector point to be pasted to the ground and the real ground. If the actual distance between the real ground and the vector point is greater than the set distance, it cannot be pasted

3.Use the mouse to select objects that need to be grounded.



Select the objects that need to be grounded

4.After selection, double-click and the program will automatically calculate and stick the selected vector object to the ground.



Renderings of objects pasted to the ground

Line Trace

Function Description：Trace the vector in view to collect line features.

Steps

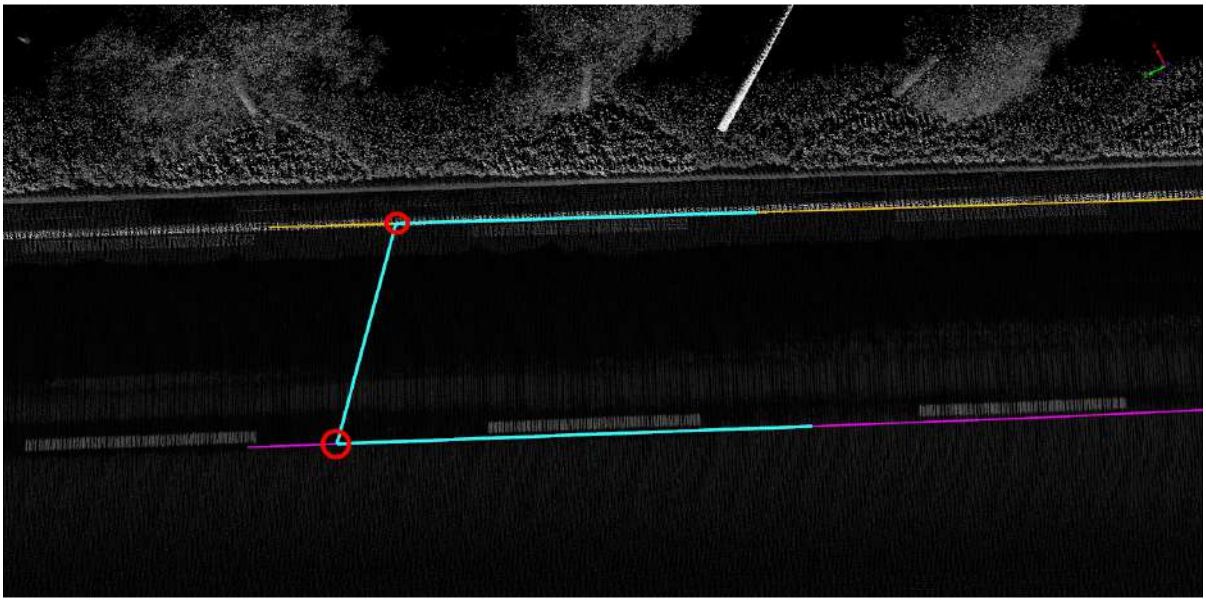
1.First switch the layer to the line layer.

2.After clicking the **Line Tracking**  button in the vector editor, point at any vector in the view, the collection will start at the point closest to the clicked position on the vector.

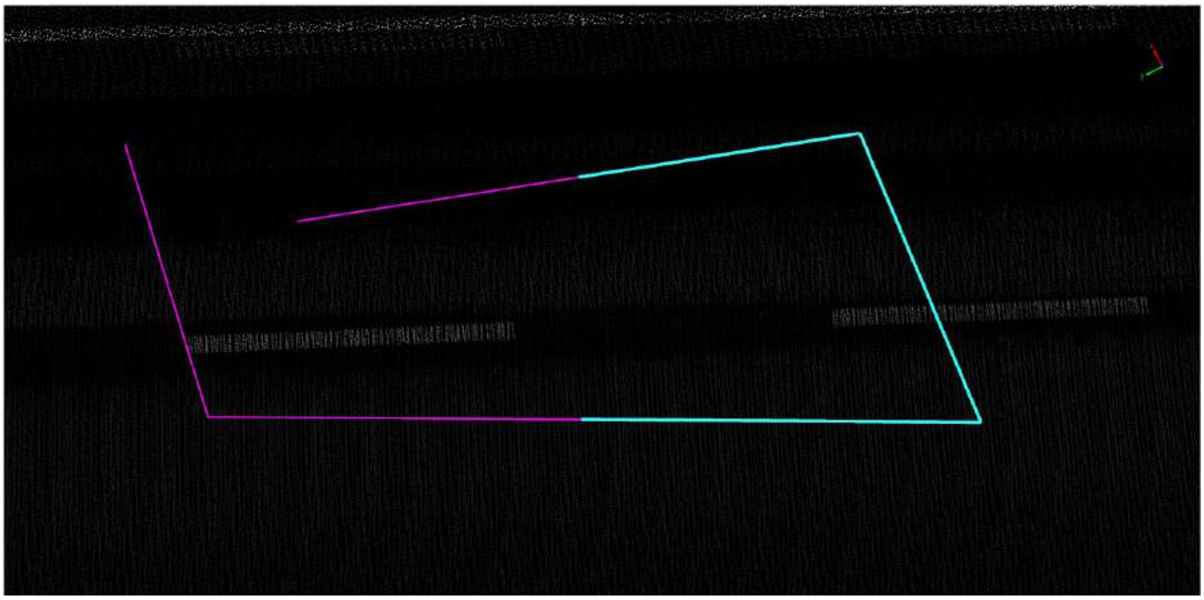
3.If you move the mouse, the currently collected line will track the mouse to add the corresponding key points, and moving in the reverse tracking order will remove the key points.

4.If you need to switch other lines, there are two cases:

- If the two lines are not connected 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 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.

Polygon Trace

Function Description：Trace the vectors in the view to capture polygon features.

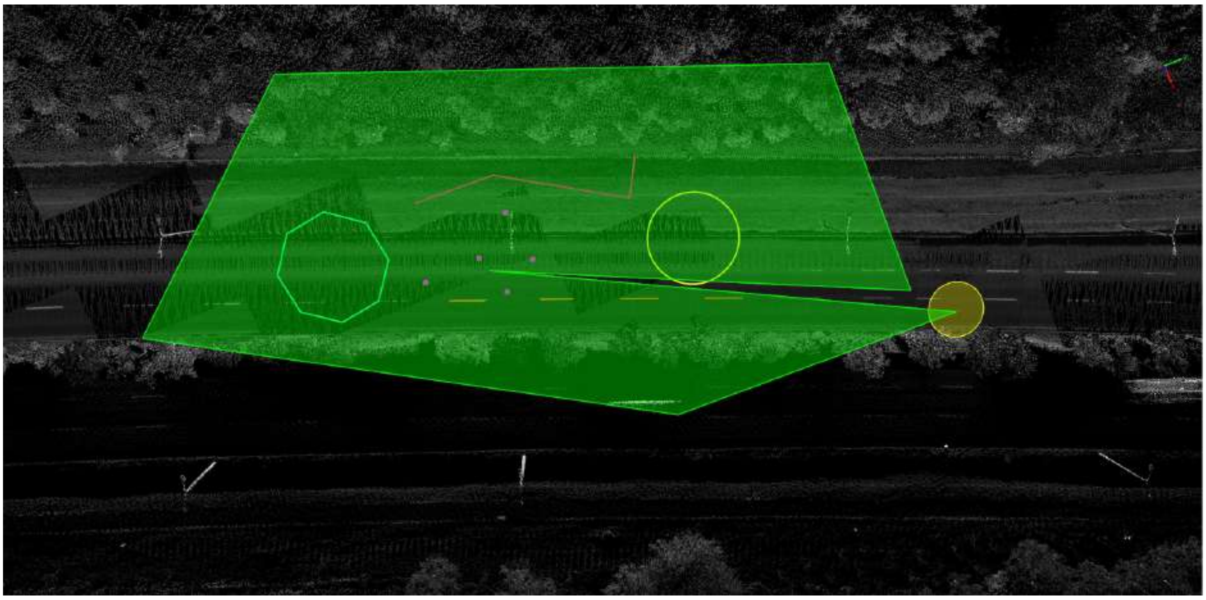
Steps

1.After clicking the **Polygon Trace**  button in the vector editor, under a point near any vector in the view, the collection will start at the point closest to the clicked position on the vector.

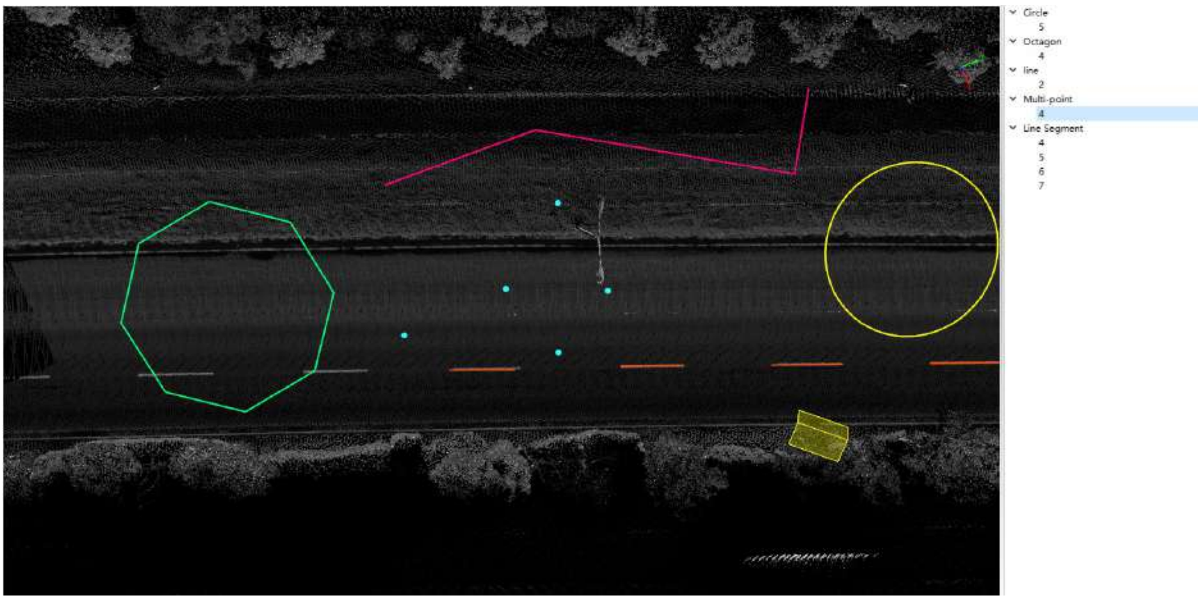
2.If you move the mouse, the currently collected polygon will track the mouse to add the corresponding key points, and moving in the reverse tracking order will remove the key points.

3.Double-clickto end the current trace, and a point is added at the closest point of the double-click.

4.Clicking the right mouse button will clear the current trace acquisition.



Polygon selection vector



Polygon trace

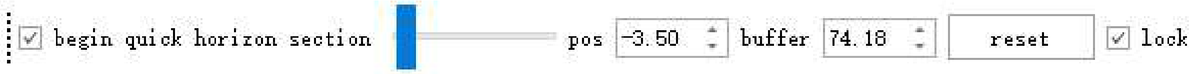
> Note: Select the shortcut key to copy and delete during editing Vector Select。

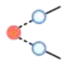
Cross plot

Function description: 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 you need to determine a reference plane first, refer to the locking function of fast horizontal section:



2.Select the line layer or polygon layer, activate **Intersect Drawing**  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, and at the same time the intersection point The line connecting with the 4th point is used 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.

Cross drawing

Draw perpendicularly

Function Description: Draw vertical polylines or polygons with adjacent sides according to the layer type.

steps

1.Select the line layer or polygon layer, activate **vertical drawing**  function.

2.First collect 2 points to determine the current baseline, and then add a point to determine a line perpendicular 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 vertically

Node Edite

Node editing has the functions of vector node movement, vector node insertion, and vector node removal.

Move Node

Insert Node

Remove Node

Edit Node

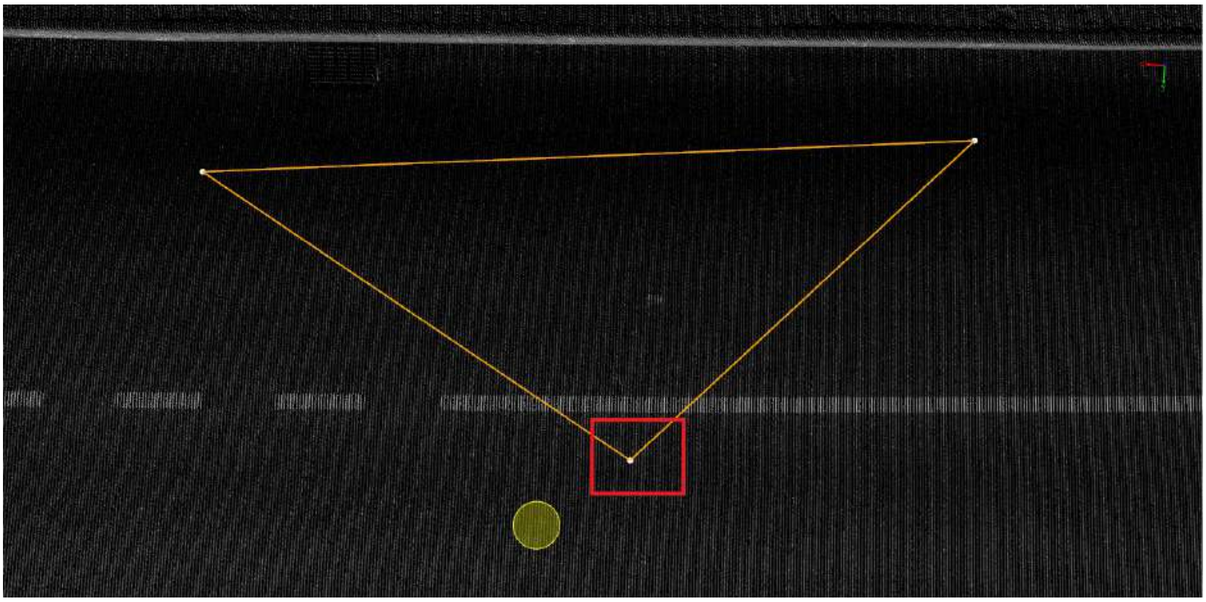
Move Node

Function Description：Move the node in the vector.

Steps

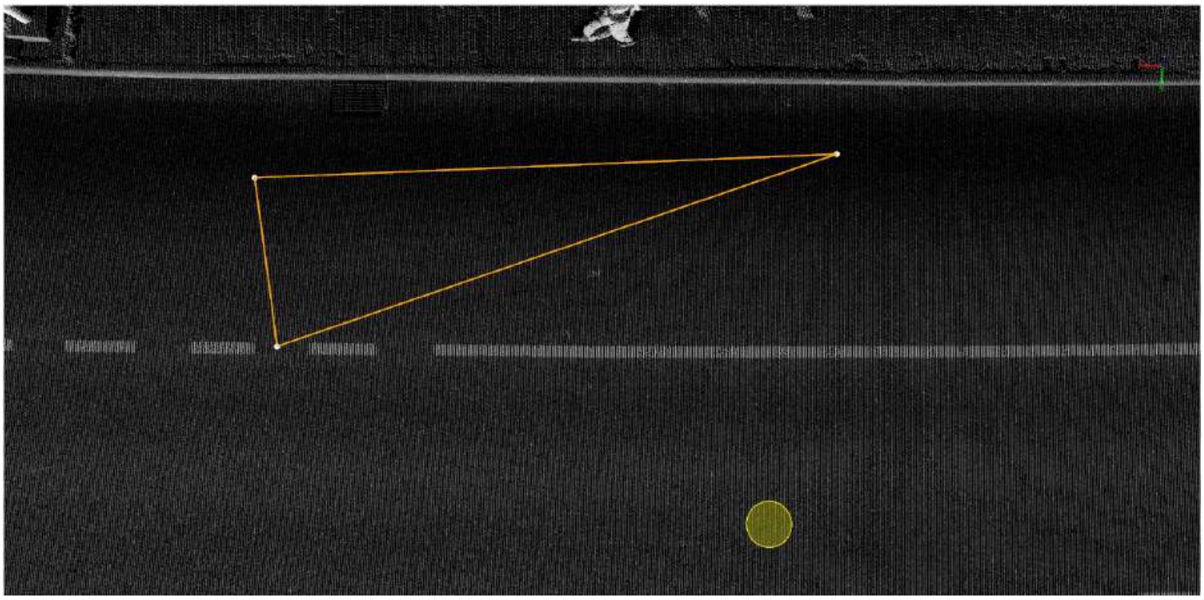
Click **Move Vector Node** in Node Editing  button, when the mouse is close to the vector node, the software will automatically capture the nearest node, and the cross-shaped mouse will be locked by the red rectangle. At this time, you can press the left mouse button and drag the node to the target position. As shown below:

Before node move：



Before node move

After node move：



After node move

Insert Node

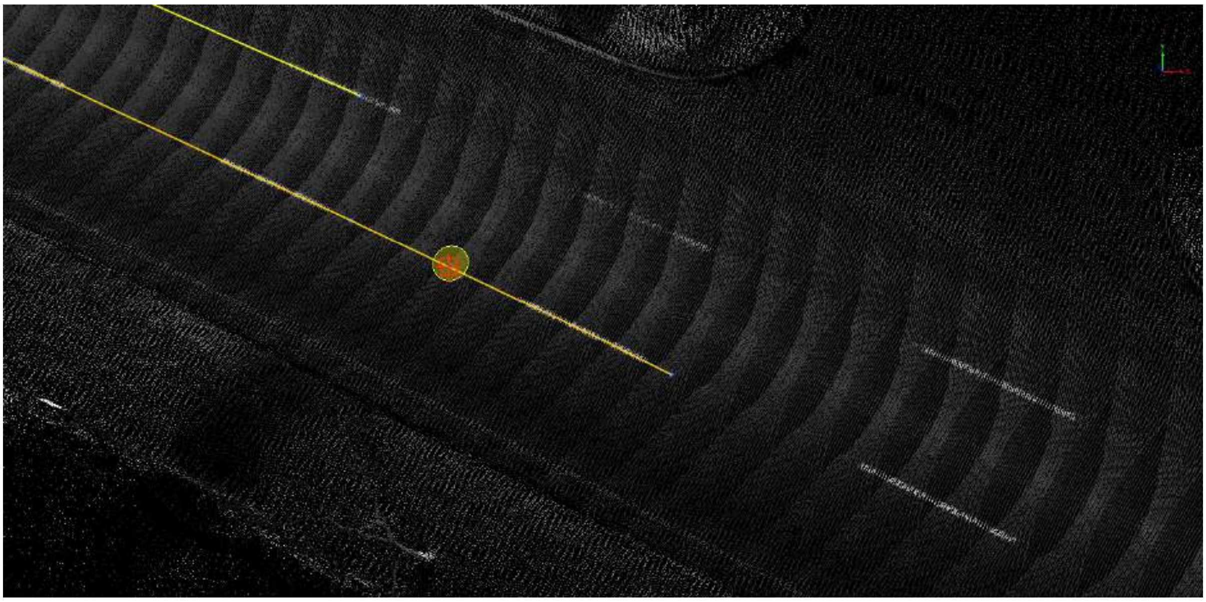
Function Description：Insert a node in the vector object.

Steps

1.Click **Insert Vector Node** in node editing  button.

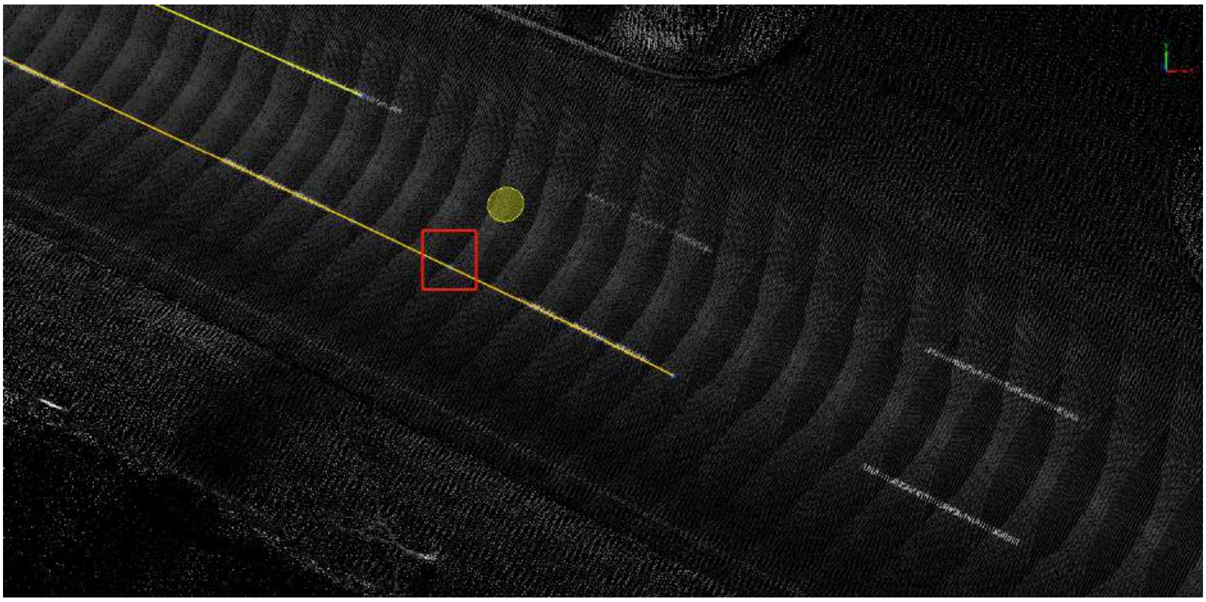
2.Left-click the position where you want to insert a node, and a node will be automatically added at that position.

- Before the node is inserted:



Before the node is inserted

- After the node is inserted:



After the node is inserted

Remove Node

Function Description：Remove nodes in the vector.

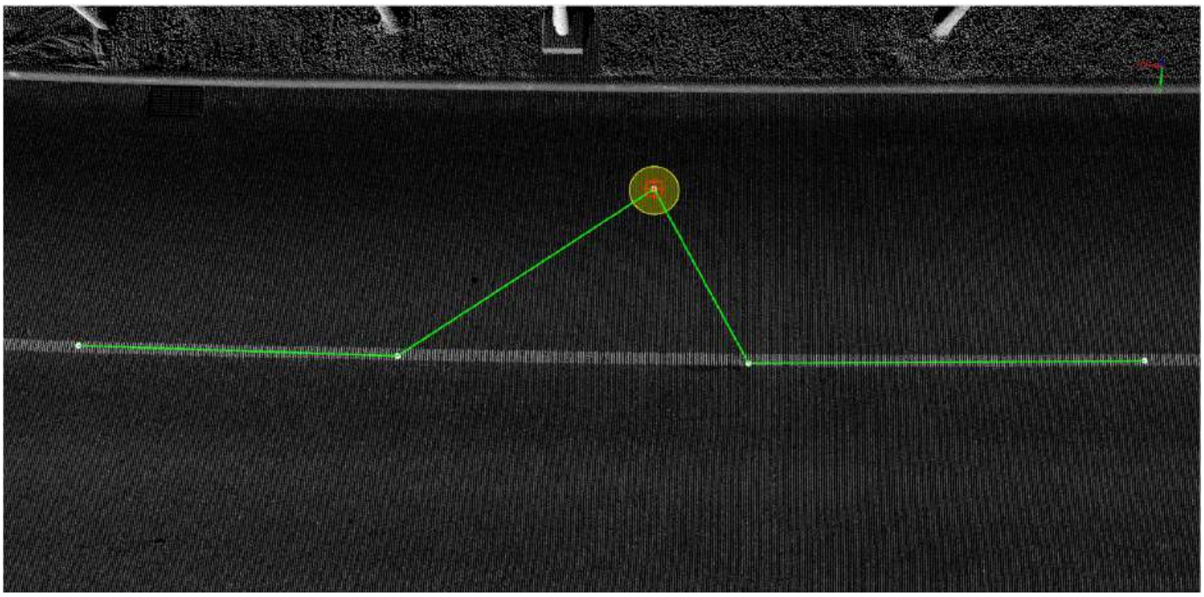
Steps

1.Click **Remove Vector Node** in Node Editing  button

2.**Left click** to select the node you want to delete.

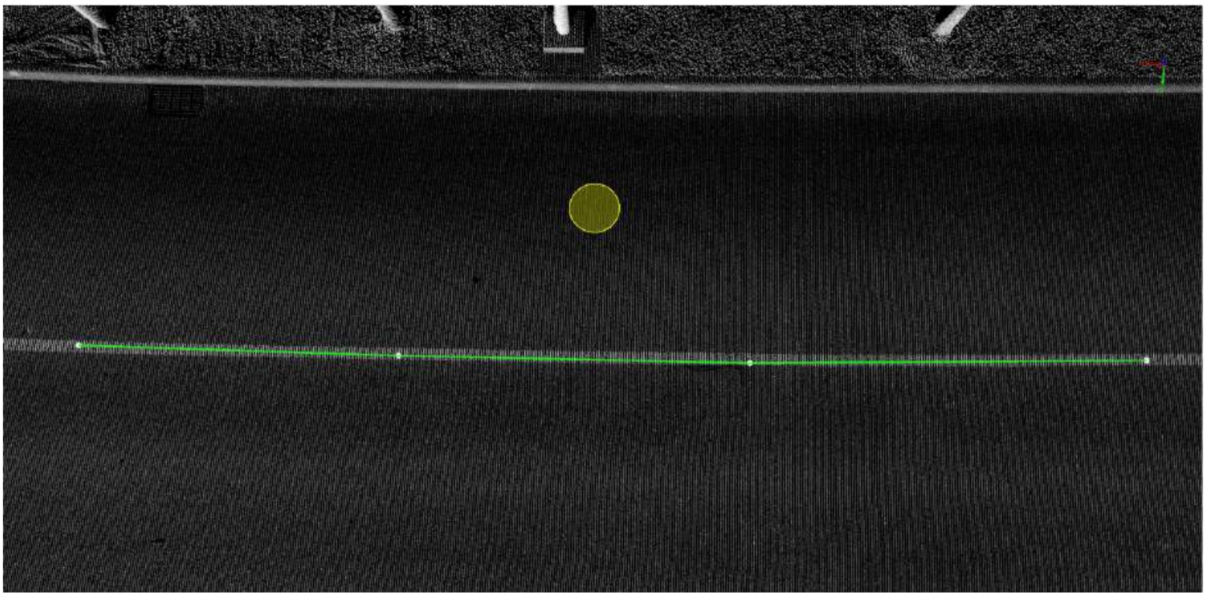
> Note: If the node to be deleted is an internal node, when deleting the current node, the nodes on both sides of the deleted node will be directly connected. The details are shown in the figure below

Before node removal：



Before node removal

After the node is removed：



After node removal

Edit Node

Function Description：Edit the nodes in the vector.

Steps

1.Click the **Node Edit** button in the node editor to activate the dialog function of the vector node editing. When the node editing is activated and the following three situations appear, the node editing dialog box will pop up.

(1) Click to select a vector;

(2) Click on a vector after box selection;

(3) Select a row in the attribute table.

As shown below:

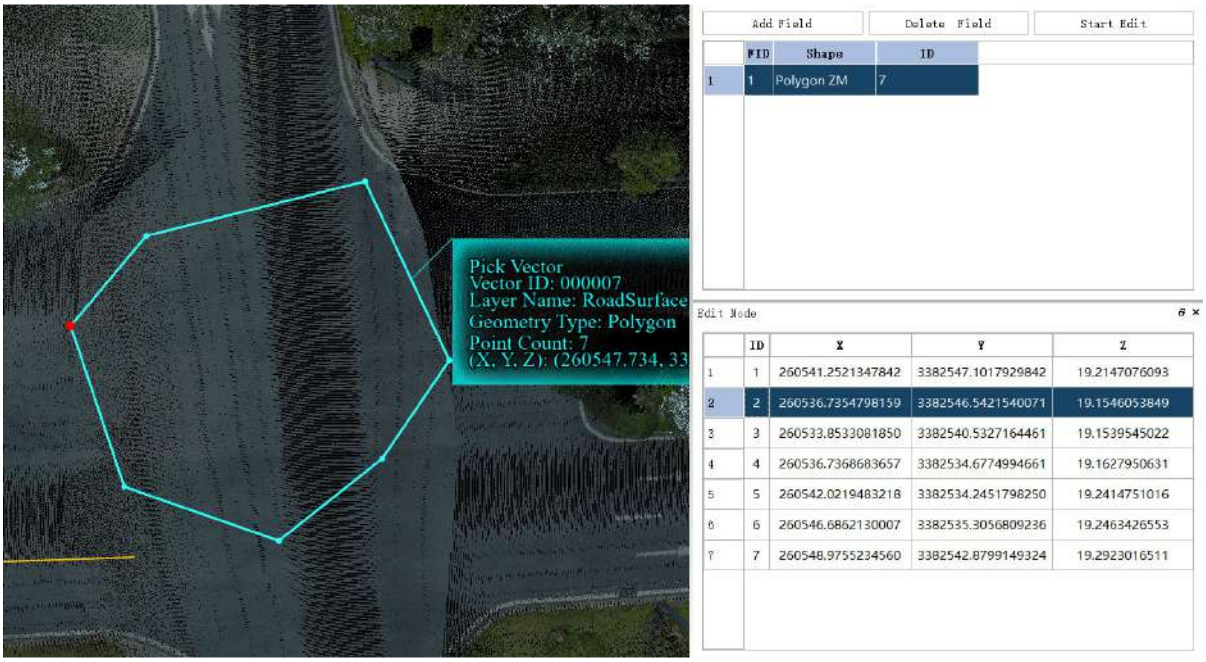
Edit Node

	ID	X	Y	Z
1	1	260541.2521347842	3382547.1017929···	19.2147076093
2	2	260536.7354798159	3382546.5421540···	19.1546053849
3	3	260533.8533081850	3382540.5327164···	19.1539545022
4	4	260536.7368683657	3382534.6774994···	19.1627950631
5	5	260542.0219483218	3382534.2451798···	19.2414751016
6	6	260546.6862130007	3382535.3056809···	19.2463426553
7	7	260548.9755234560	3382542.8799149···	19.2923016511

Node Edit Interface

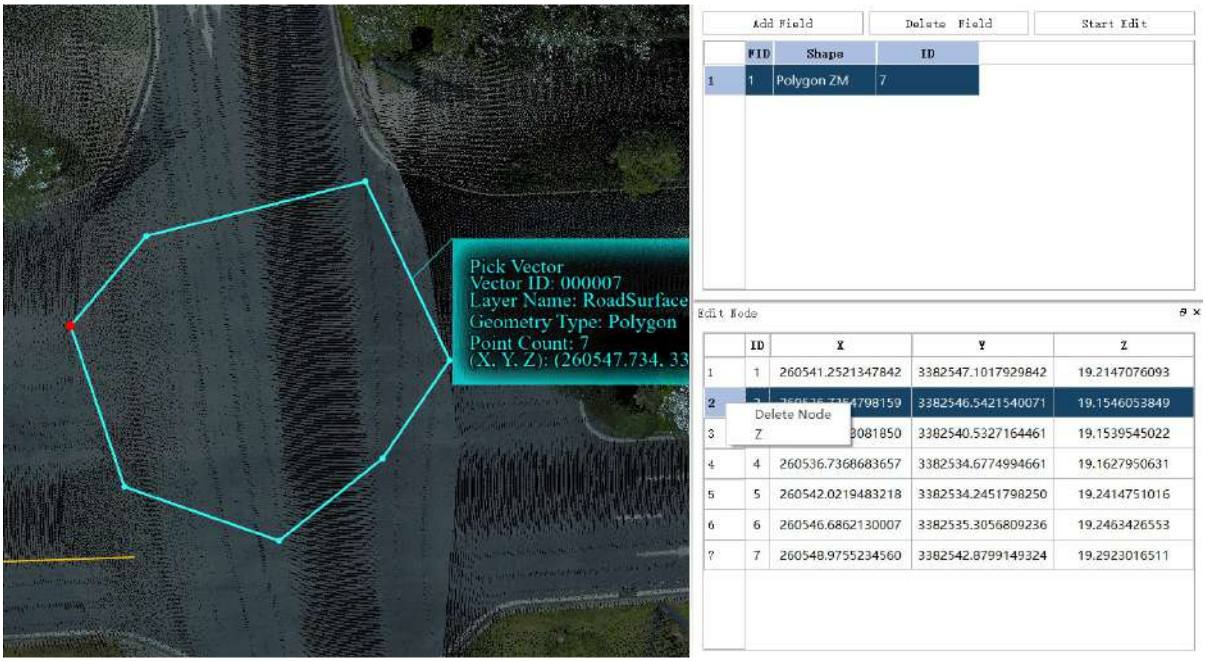
2.The main functions of node editing are as follows：

- **Select highlight**: Click to select one or more rows of the selected node editing table, and the node selected by the vector will be highlighted in the 3D scene.

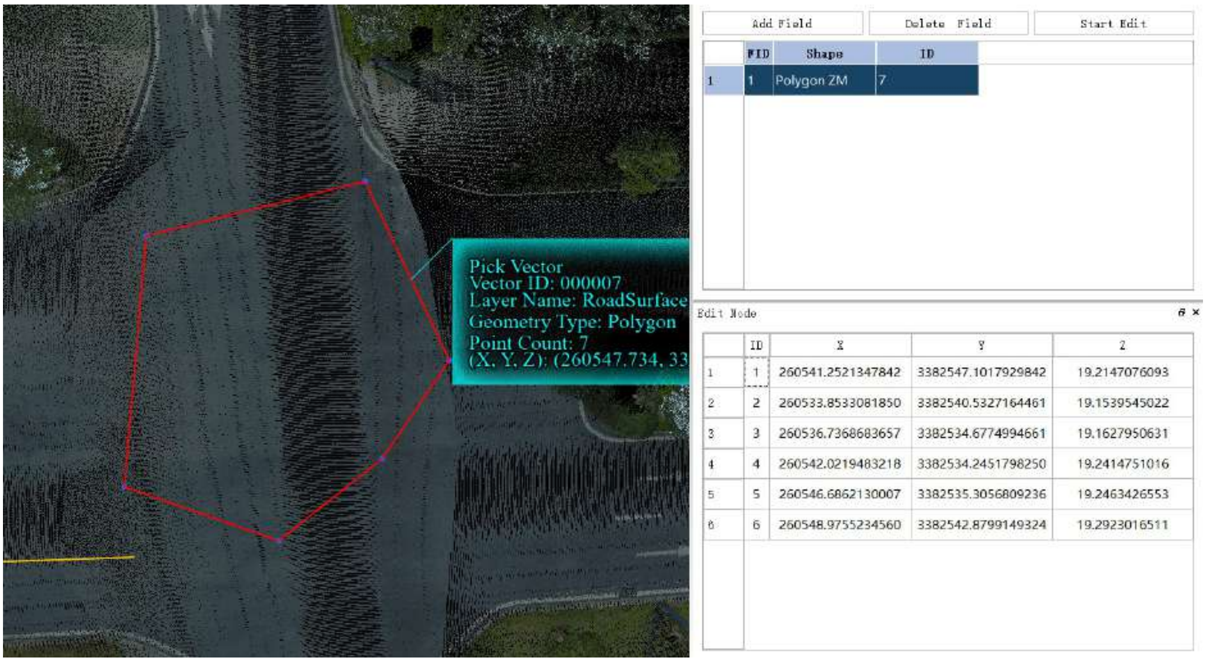


- **Double-click, jump to highlight**: Double-click a row of the node editing table with the mouse to switch the perspective of the 3D scene to this node and highlight the node.
- **Delete node**： Select a row in the node editing table, then right click to pop up the drop-down menu, select the function of "Delete Node", the corresponding row will be deleted in the table, and the selected node of the vector will be deleted in the 3D scene, and redrawn the vector. Note that the point layer retains at least one node, the line layer retains at least two nodes, and the polygon layer retains at least three nodes. As shown below:

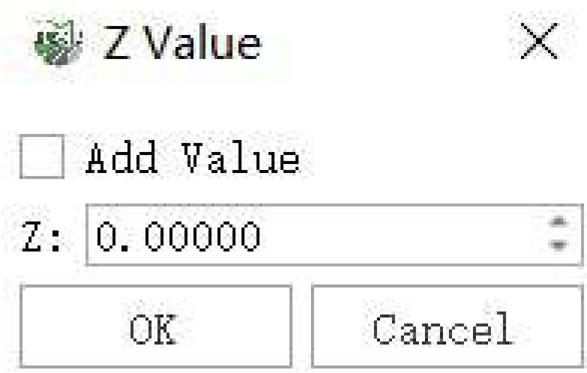
Before node deletion:



After node deletion:



- **Modify Z value**: Select a row in the node editing table, then click the right mouse button to pop up the drop-down menu, select "Z", and a dialog box for modifying the Z value will pop up. The Z value of the selected node is modified to the value in the input box, and the vector is redrawn. If "Accumulate" is checked, a new value will be added to the original Z value of the node, and the vector in the 3D scene will also be redrawn.



Facade Survey

As shown in the following figure, the functions included in the elevation measurement page are:

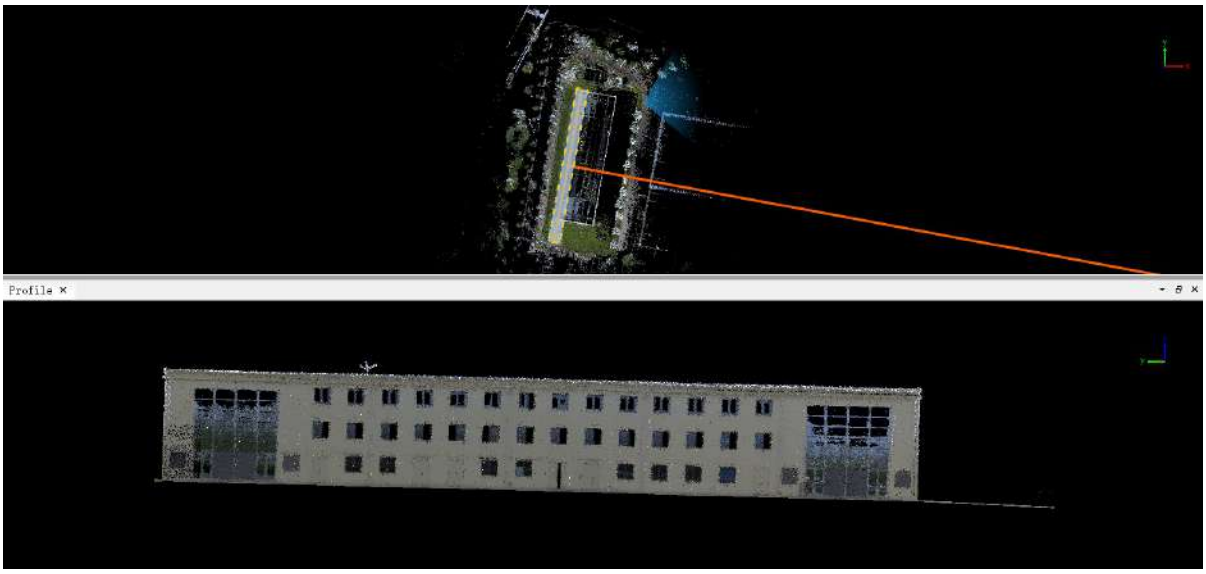
- SetVertical
- Baseline
- BoundaryLine
- Profile
- Drawing
- TectonicCollect
- Array
- Edit
- Export

SetVertical

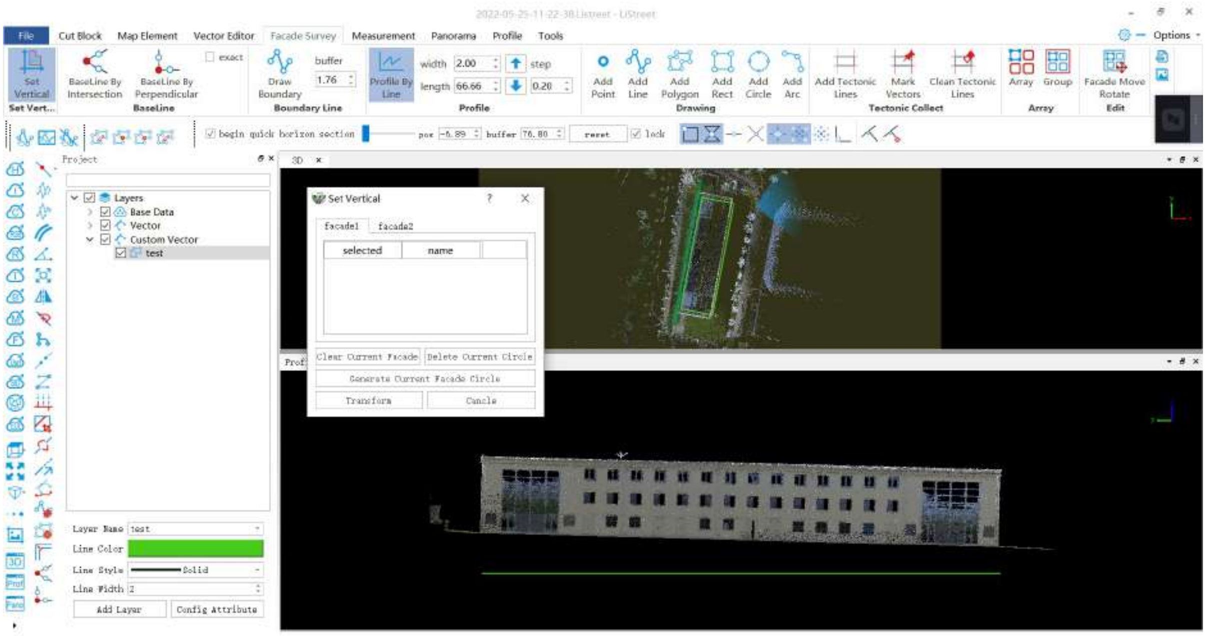
Function Description：The vertical setting function is based on the two side elevations of a building, and the side elevations of the building are vertically corrected.

Steps

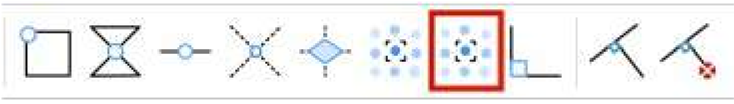
After the data is loaded, you can take a cross-section to check whether the handheld data is horizontal. If the data is as shown in the figure below, and the side elevation is inclined.



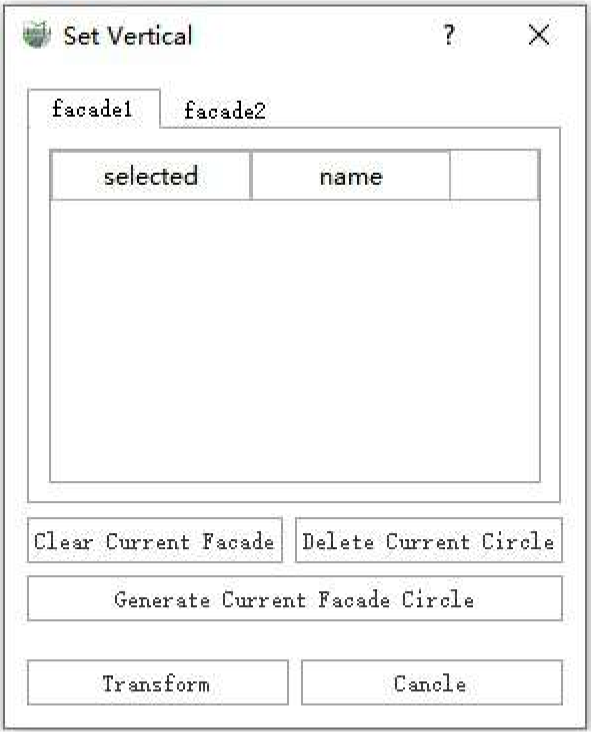
you can activate the "Set Vertical"  function in "Façade Survey" to make the side elevation vertical.



1.Open the point cloud fitting capture tool of the software



2.Click the vertical button to pop up the vertical operation pop-up box



3.Under the "Facade 1" 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 Elevation" to generate the fitted vertical plane of the current side elevation (Figure 2 below).

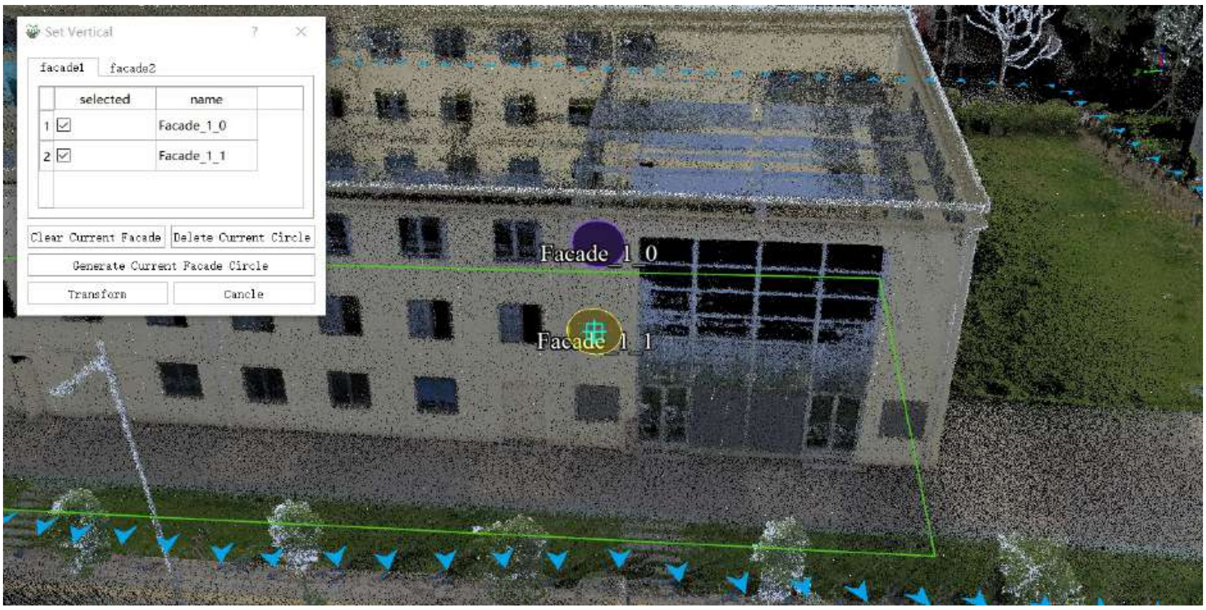


Figure 1 Selecting the Fitting Plane

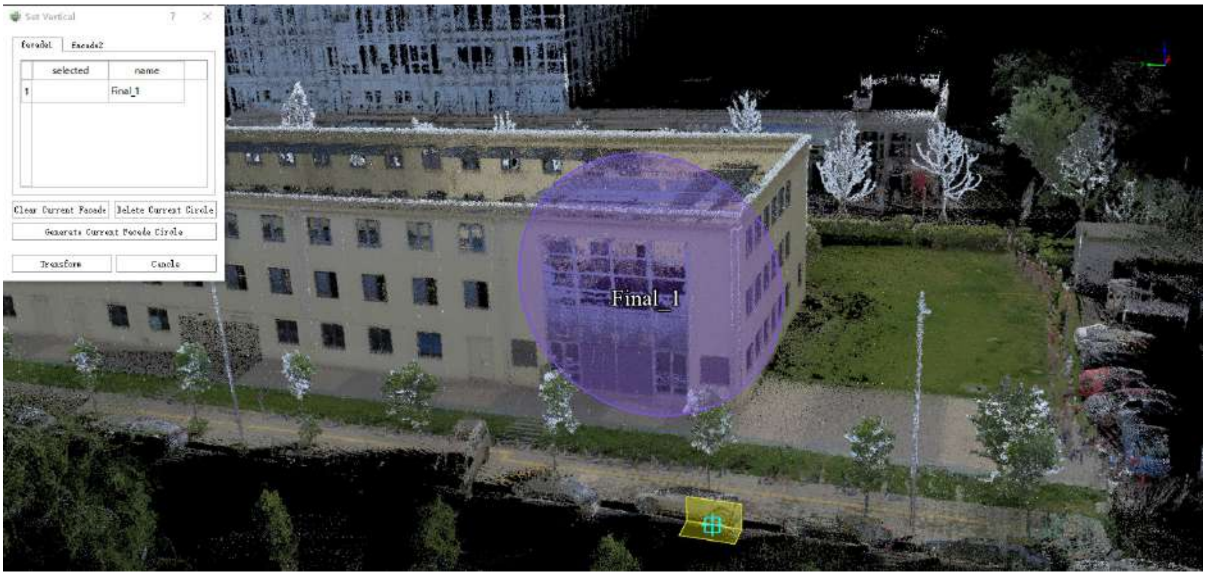
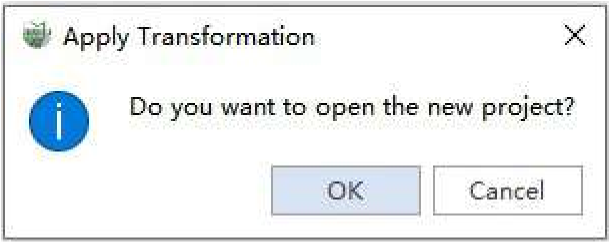


Figure 2 Generate a facade circle

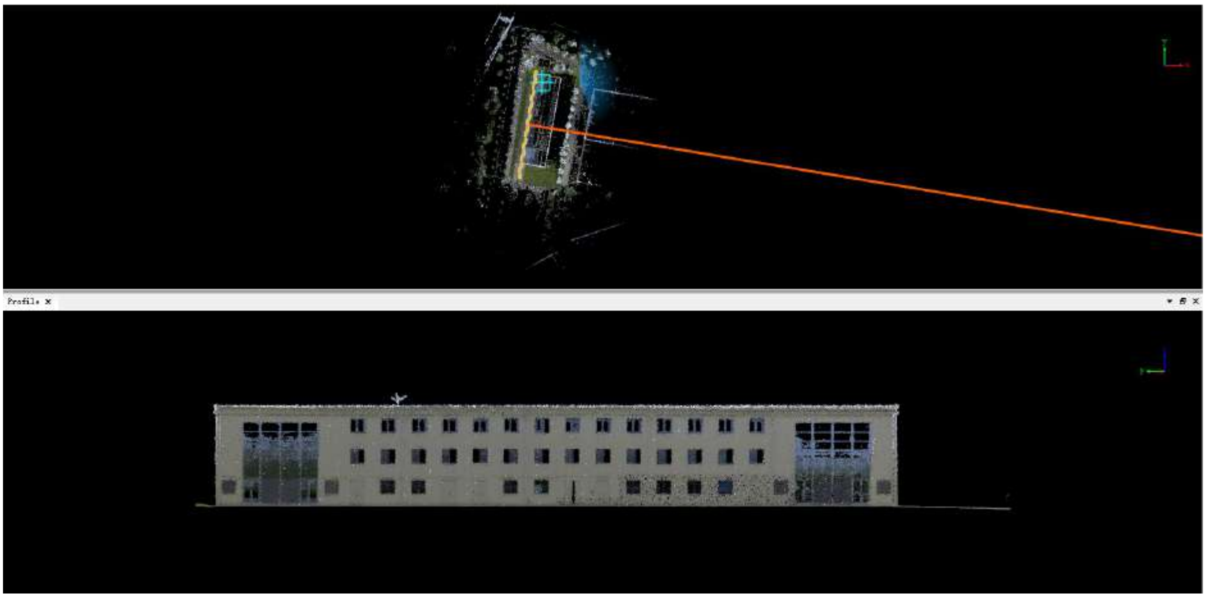
Under the "Elevation 2" tab page, select the side elevation that is connected to the edge of Elevation 1, and then repeat the operation of Elevation 1 to generate a second elevation circle.

4.After the two side elevation circles are generated, confirm that they are correct, and click the "Rotate" button to apply the transformation. Select "Yes", the point cloud, panorama and trajectory (if 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.LiData will be regenerated according to the converted point cloud, panorama and trajectory (if set), and a new project will be prompted after the conversion is completed.



5.The point cloud after resetting the vertical is shown in the figure below



BaseLine

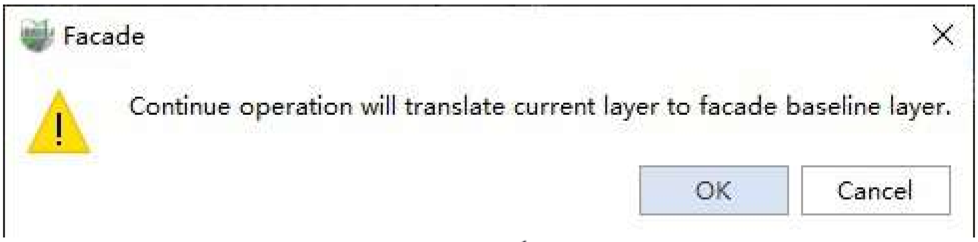
Function Description：Draw the horizontal structure diagram.

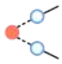
Steps

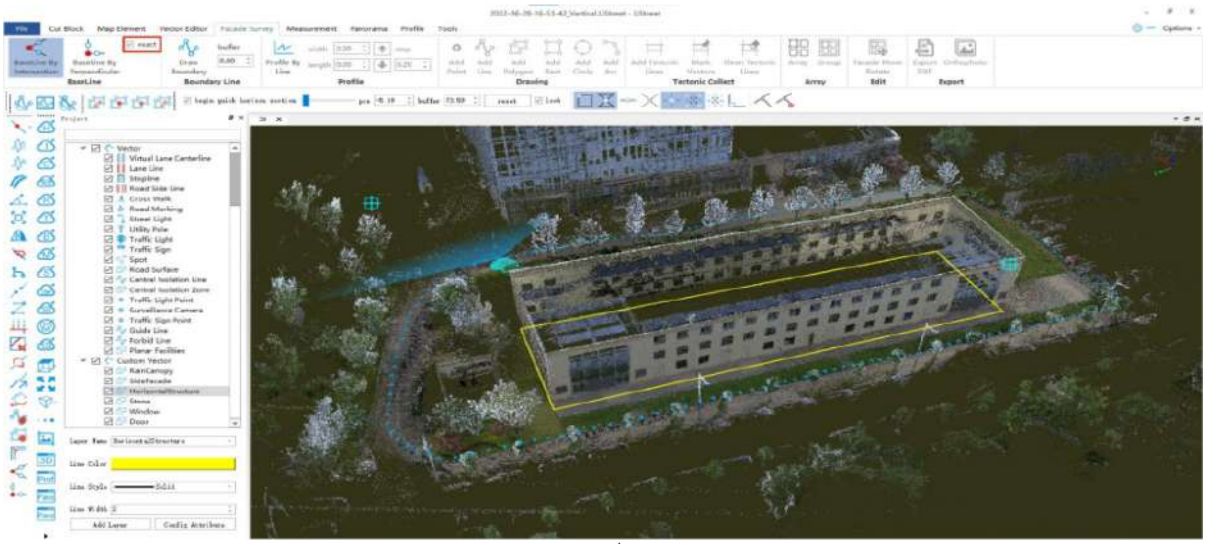
> Attention：You must have a horizontal structure drawing before you can draw the side elevation elements

1.Select the tool in "Baseline Collection" under the "Elevation" menu bar, and "Precise Mode" is checked by default.

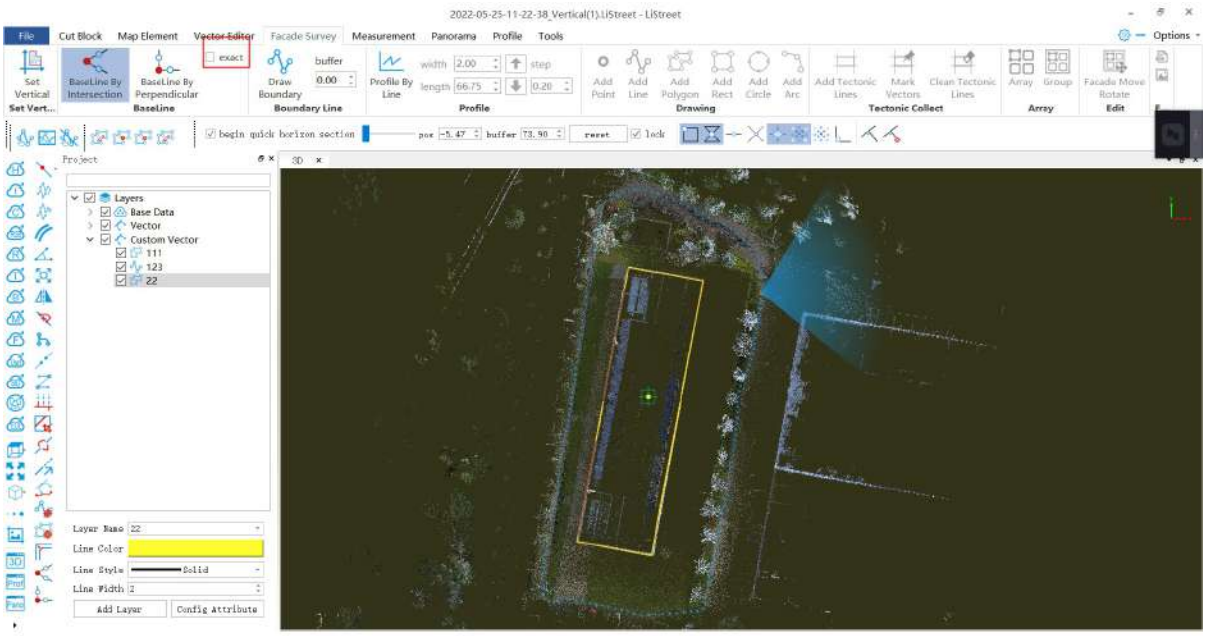
2.If you select any tool again, you will be prompted to convert the selected layer into a reference line layer, because the later selection of the edge elevation controls the layer selection, only the reference line layer and the range line layer can be selected to intercept the section.



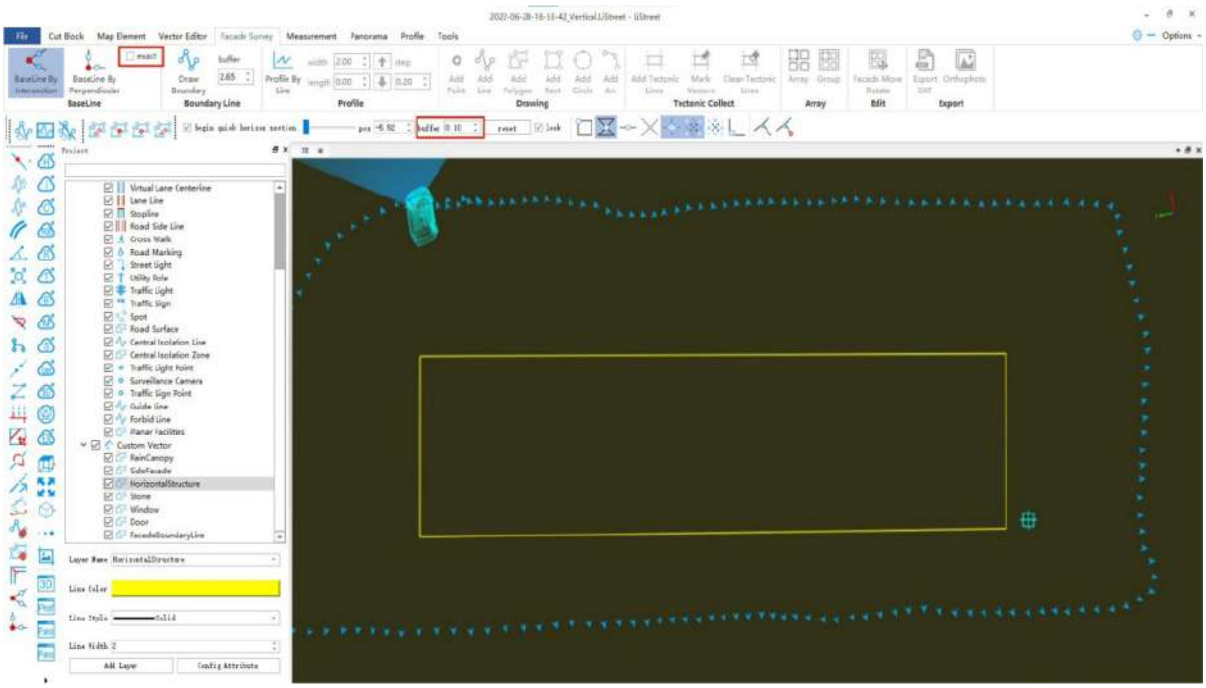
- In precise mode, using the "Intersection Baseline"  tool, you need to click on two points on each side elevation, and double-click on the last point to complete the closure.



- In precise mode, using the "Vertical Reference Line"  tool, you need to click two points on the first side elevation, click the point on the previous point cloud on the other side elevations, and double-click the last point to complete the closure.



- Uncheck "Accurate Mode", the two tools will turn the 3D window into top view + orthogonal projection. At this time, it is recommended to change the buffer in "Quick Horizontal Section" to 0.1 to reduce the thickness of the point cloud, which can increase the Precision of the drawing.

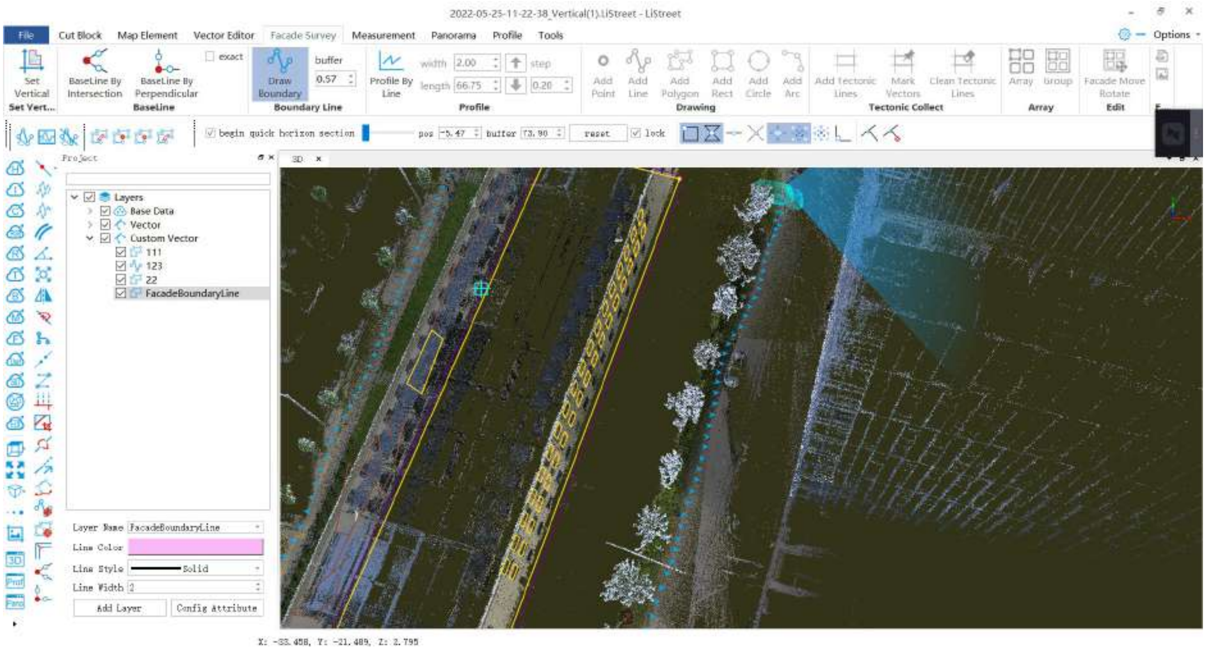


Boundary Line

Function Description: Draw the range parameter line

Steps

1.According to the elevation drawing specifications, if you need to draw a range line, you need to extend a certain range to draw a range line based on the reference line layer. Use the "Draw Range Line"  tool to select a reference line, and you can choose to drag it or draw it out at the length of the range line. Enter a fixed value to define the range line, which will be drawn in a built-in layer.



Facede

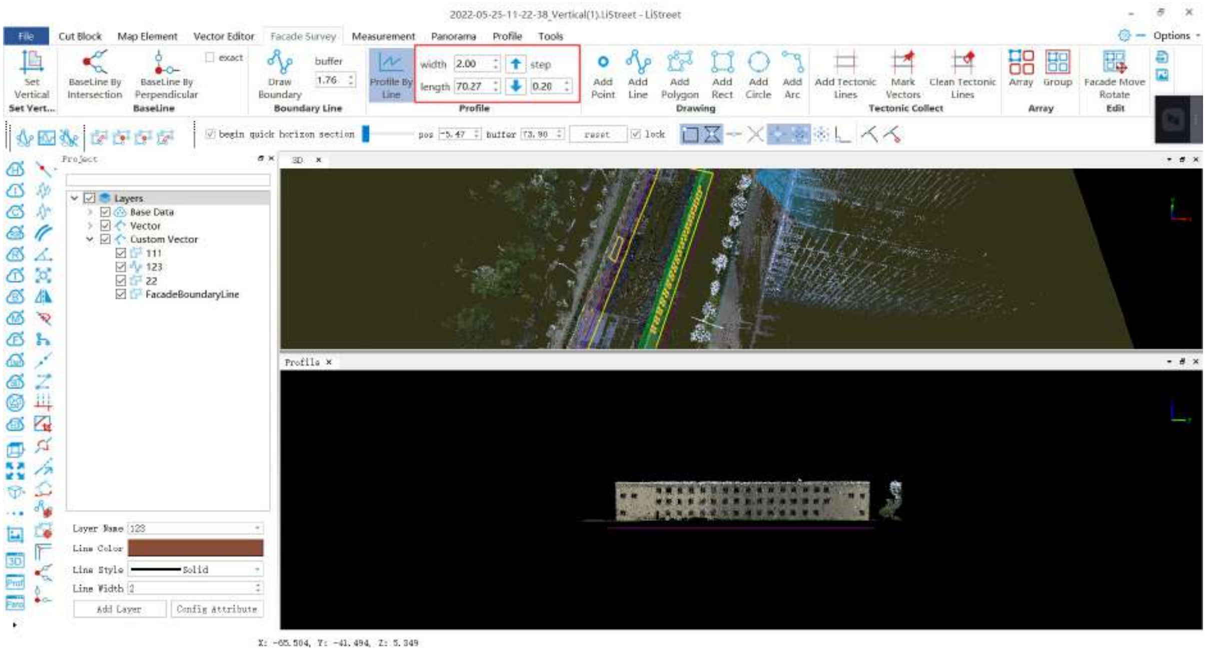
Function Description：Select an edge of the baseline or range line, intercept the point cloud with the default thickness, and generate an elevation view.

Steps

1.Click the **Select Side Elevation**  button to activate the function of generating elevation

2.Click a vector edge of the datum plane drawn in advance, and in the section window, the side elevation corresponding to the current edge can be automatically generated

3.You can adjust the thickness of the point cloud display through Buffer Thickness, and adjust the left and right display range of the point cloud through Buffer Length. Adjust the position of the point cloud display "forward" or "backward":



Drawing

Function Description: The basic element collection function is the same as the drawing function under the vector editing menu page

Steps

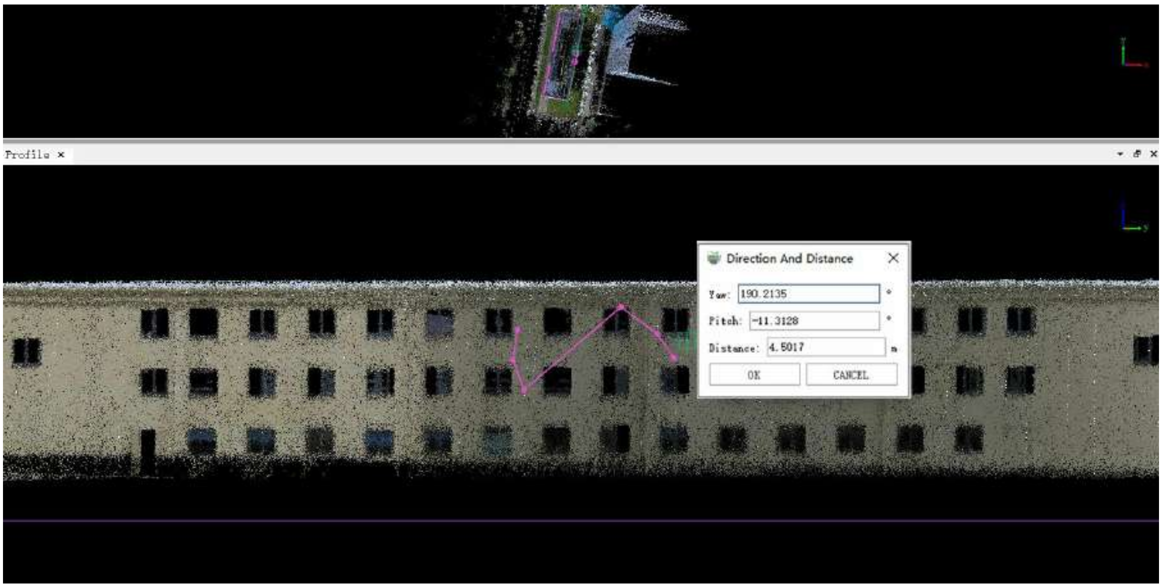
1.Point feature drawing : select a point layer and select a point in the point cloud under the section window with the mouse, and a circle will be drawn at this position according to the current layer.

2.Line element drawing : select a line layer, use the mouse to draw line elements in the point cloud under the section window, and 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:

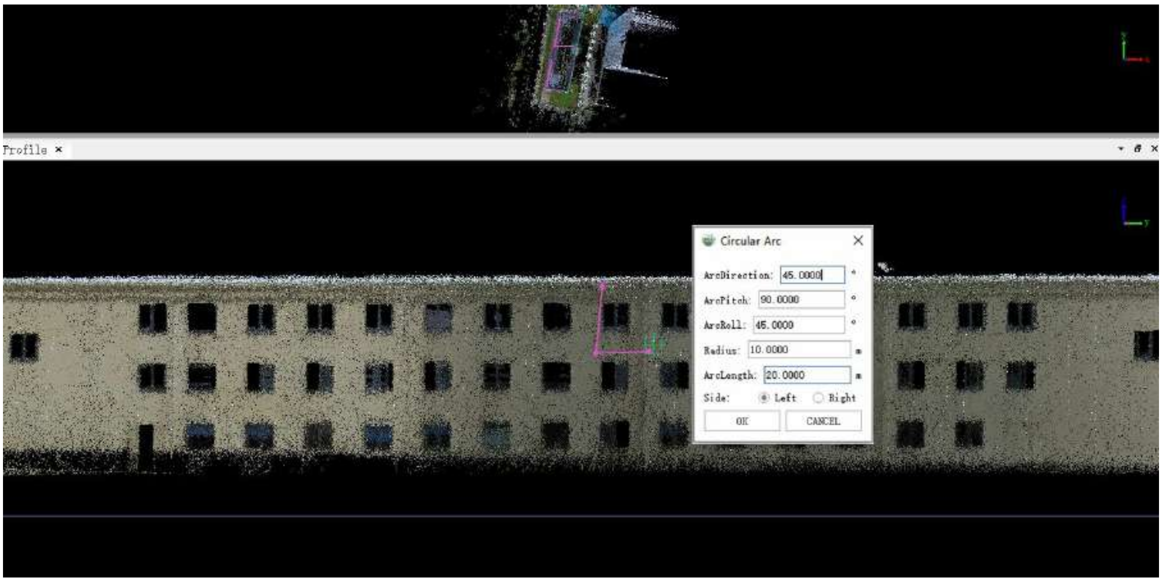
- Capture mode: you can switch various capture modes at any time in the capture mode menu
- Fallback point: click fallback point to fallback nodes in order. (the first node cannot be rolled back). During drawing, the shortcut key of back point is b/b
- Absolute XYZ: with the absolute XYZ function, you can specify the coordinates of absolute XYZ during the drawing process. The coordinate value entered in the pop-up box is the position drawn by the node.

- Relative XYZ: through the relative XYZ 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: through the direction distance function, you can specify the direction 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 angular direction. In the section window, there is no need to modify the deflection angle.



- Arc calculation: through the arc calculation function, an arc with a specified direction, size and position can be generated during the drawing process. As shown in the following figure, the arc direction refers to the angle of the arc relative to the Y axis of the coordinate system; Arc pitch refers to the direction relative to the XY plane of the coordinate axis; Arc rollover 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; Arc length refers to the side length of the generated arc; Edge 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)



- Reverse: during the line drawing process, the direction of the drawn vector line can be reversed through the reverse 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 section 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 2 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 rectangular elements by drawing a rectangle with three points.

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.

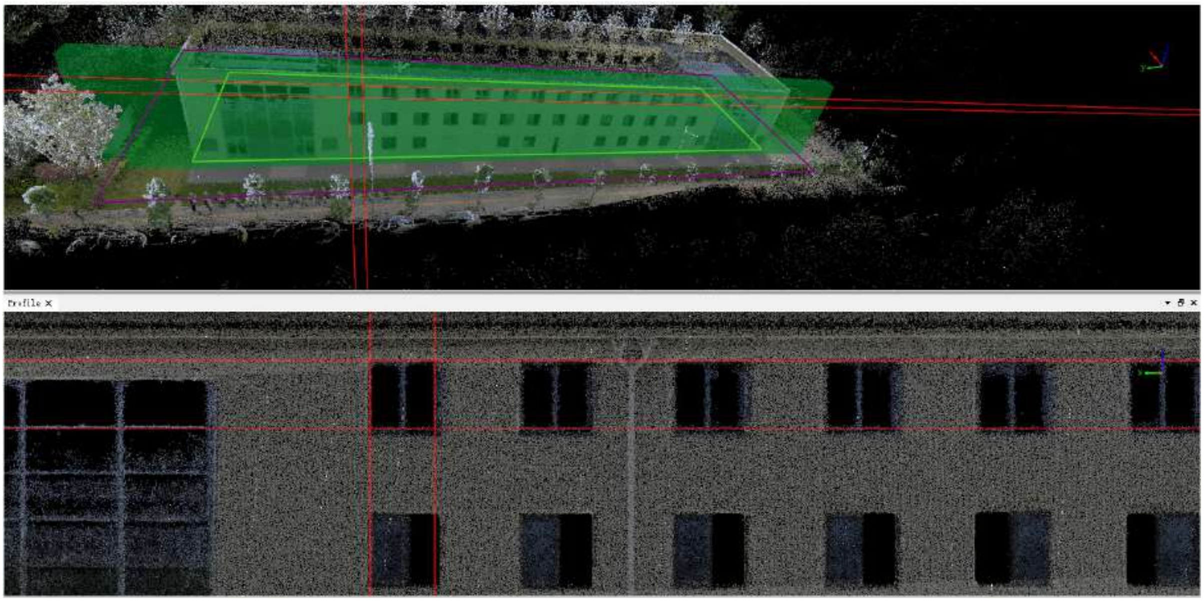
> Note: the use of basic feature drawing tools in elevation is the same as that of basic feature drawing tools in vector editing module

TectonicCollect

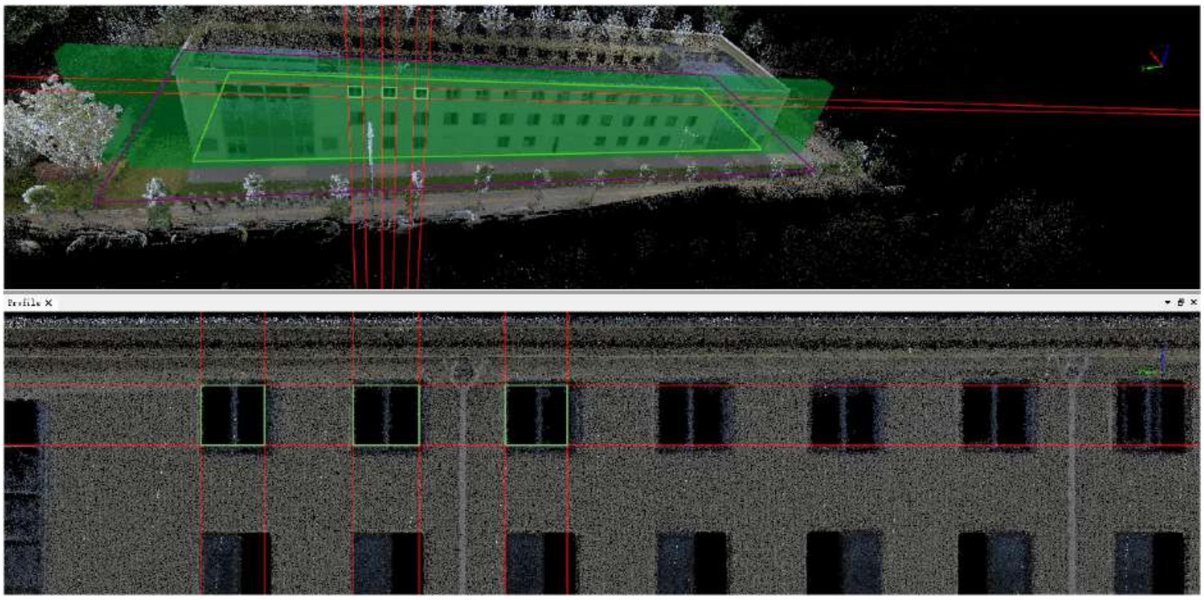
Function Description：Features used to draw side elevation layout rules.

Steps

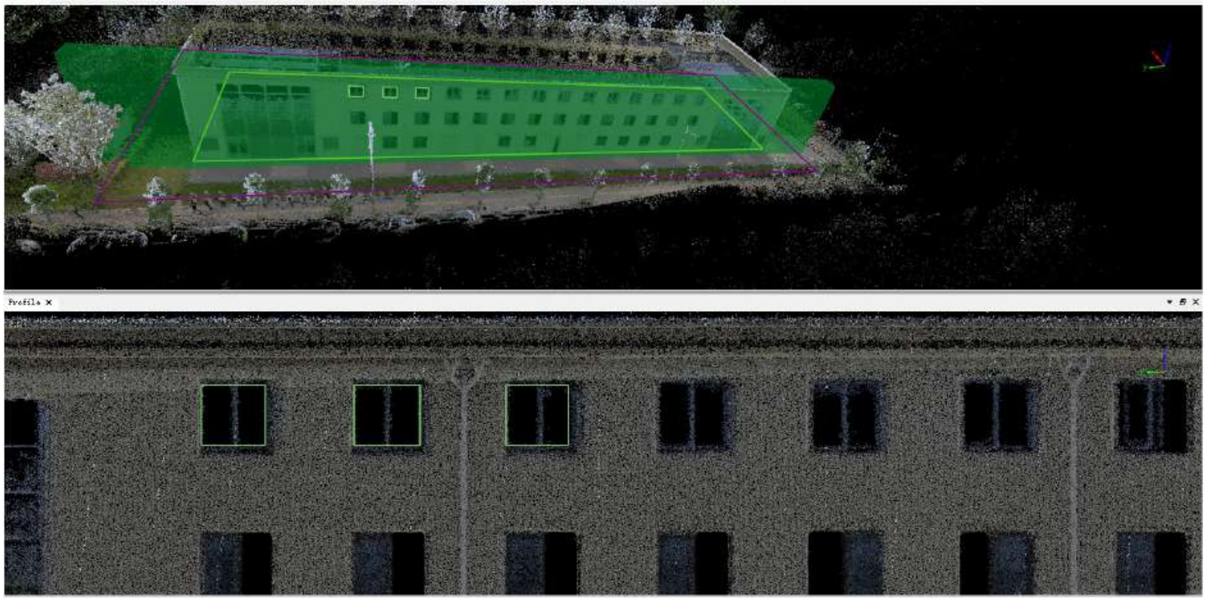
1.Activate "add construction line", left click to add horizontal construction line, ctrl+ left click to add vertical construction line, and select the surface layer to be drawn



2.Click "mark vector" to automatically generate a rectangle in the middle of the four construction lines, which can mark multiple vectors continuously. (When adding a construction line, the right mouse button is used to back out the previous construction line, and you can back out multiple times)



3.Click "clear construction line"  to display only the generated vector.

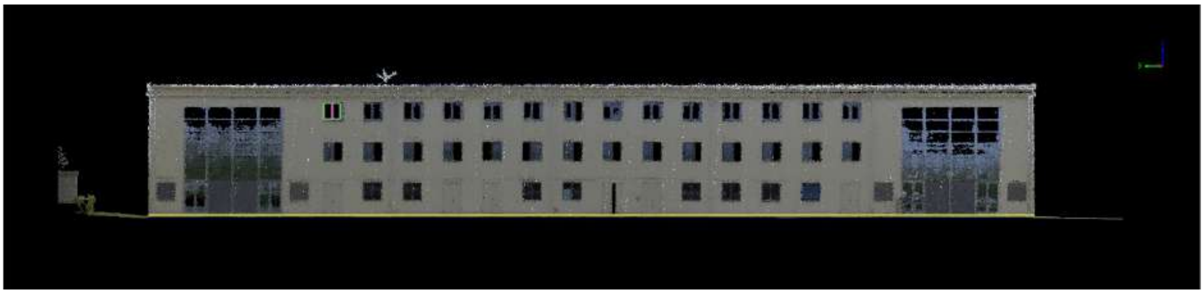

Profile

Array

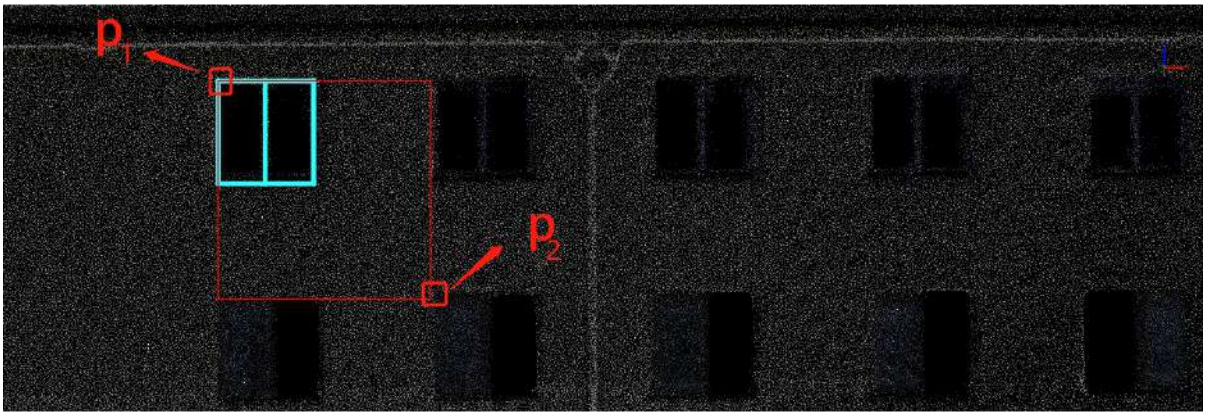
Function Description：The array tool can quickly draw windows with the same structure arranged at equal intervals.

Steps

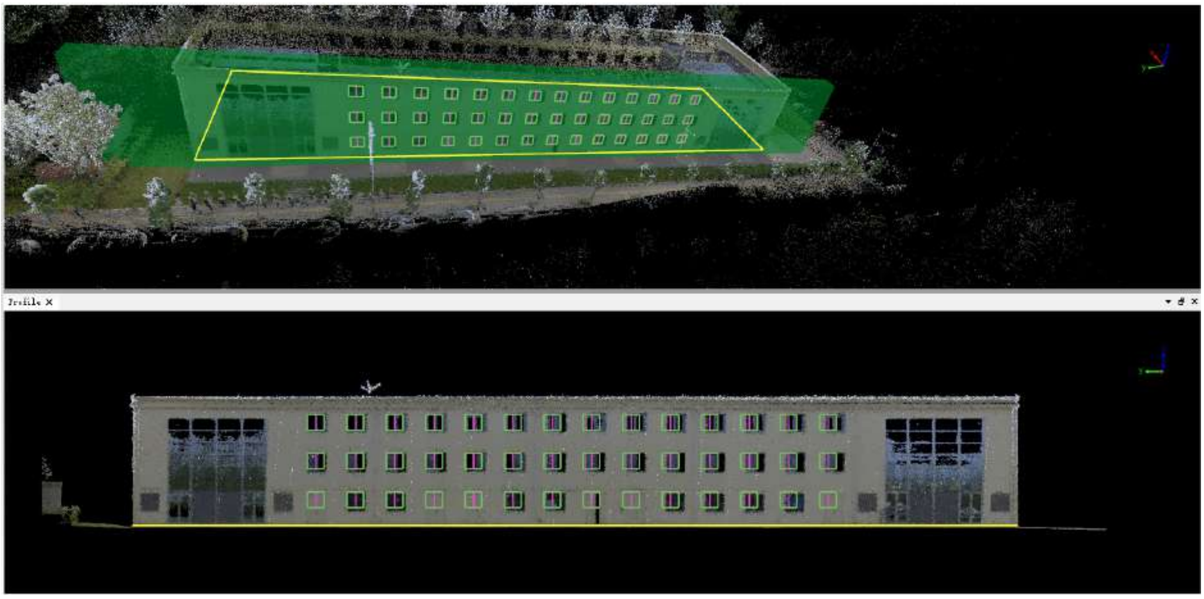
1.As shown in the image below, the side façade windows are arranged nearly equally spaced and have the same window structure.



2.At this point, you can use the "array"  tool, activate the tool, select the objects that need to be arrayed with the left click, and then click the structure group, and pull a moving box. The first point is the starting point, and the second point is the first point that needs to be copied to. Point, you can re-pull the box several times to select the position.



3.After determining the length and width of the box, click Enter to lock it, 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. You can draw a whole wall of windows and their interior elements very quickly.

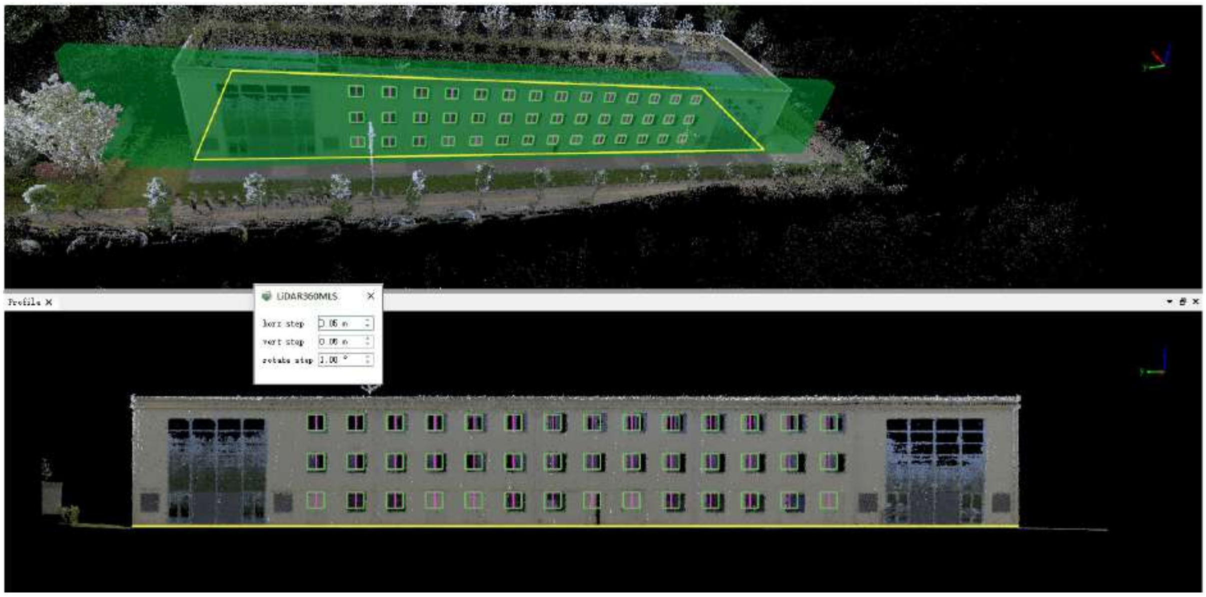

Profile

Faced Move Rotate

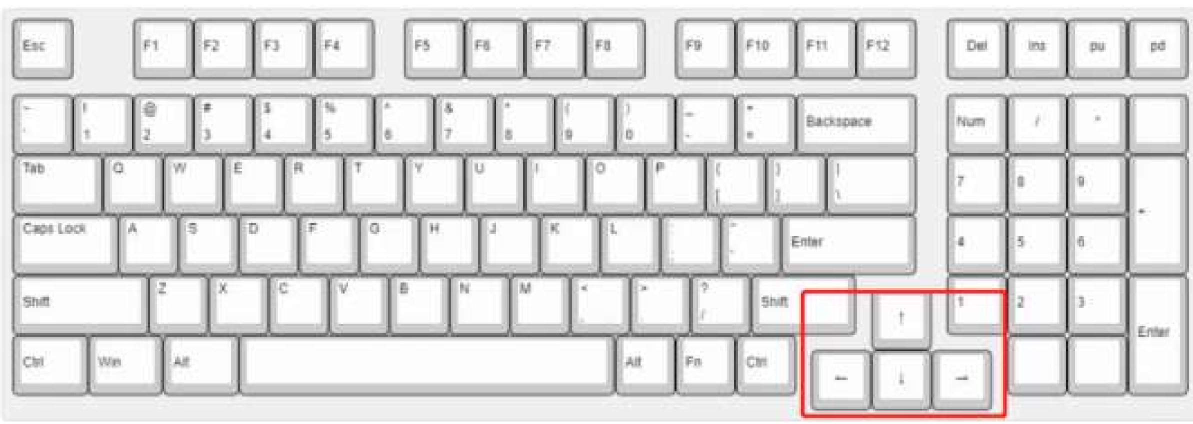
Function Description： If some elements are not drawn accurately, you can activate the elevation movement and rotation tool.

Steps

1.Activate elevation "Faced Move Rotate" , select the elements that need to be fine-tuned with the left click, adjust the movement and 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, it is shown in the figure:

Copy Vector

Function Description：You can copy the drawn elements to any position of the current elevation

Steps

1.Click **Copy Vector**  in editing activate the vector copy 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 a copy operation. The left mouse button can always be used to click and place.

5.After all copies are completed, right-click to exit copying.

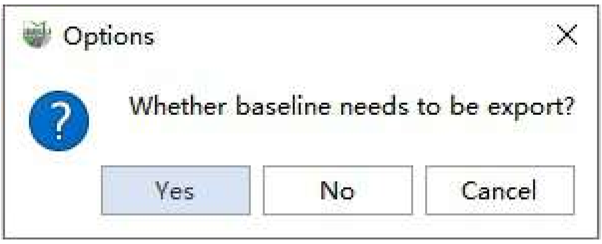
Export

Function Description: The export tool can export the engineering data into various formats as the input of third-party software for data display or further data processing.

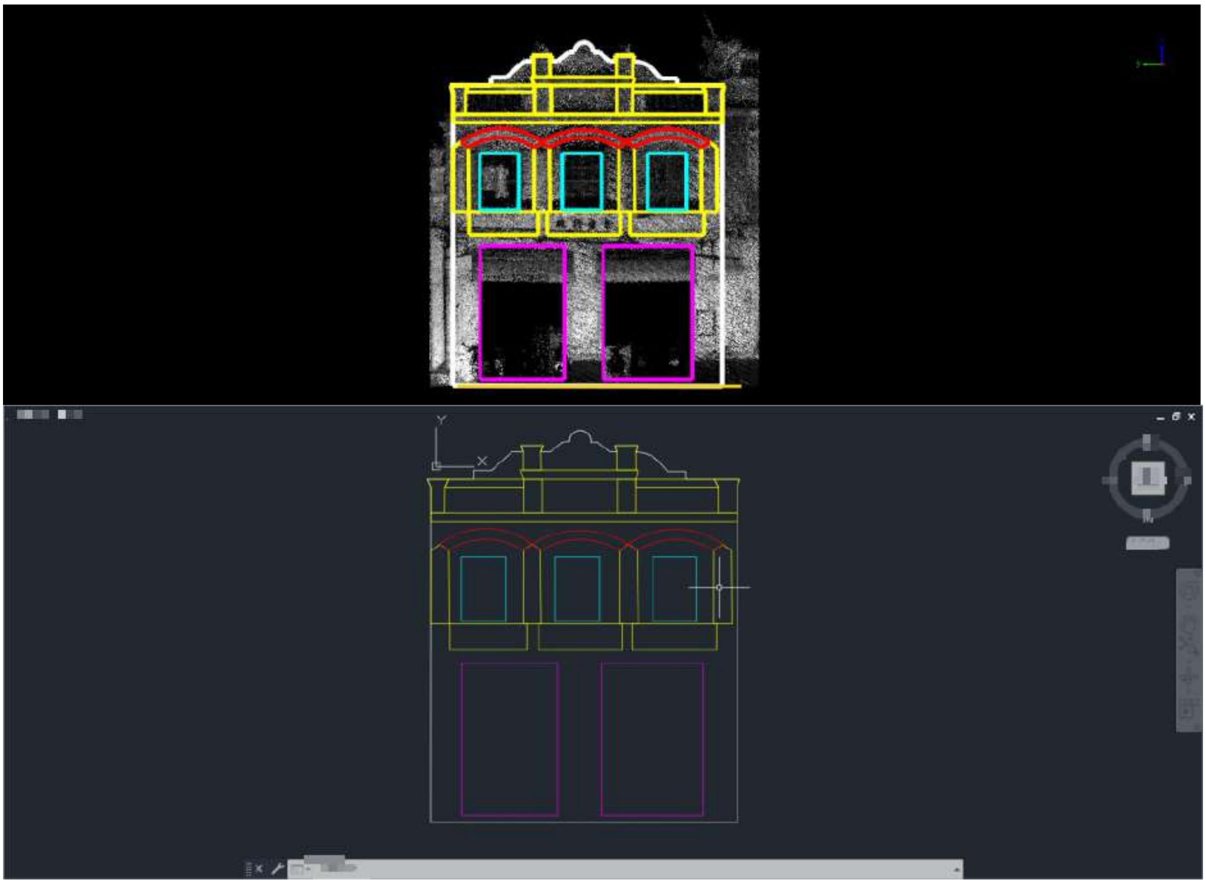
Steps

Export DXF

1.Click the **export DXF** function, and you will be prompted whether to save the baseline.

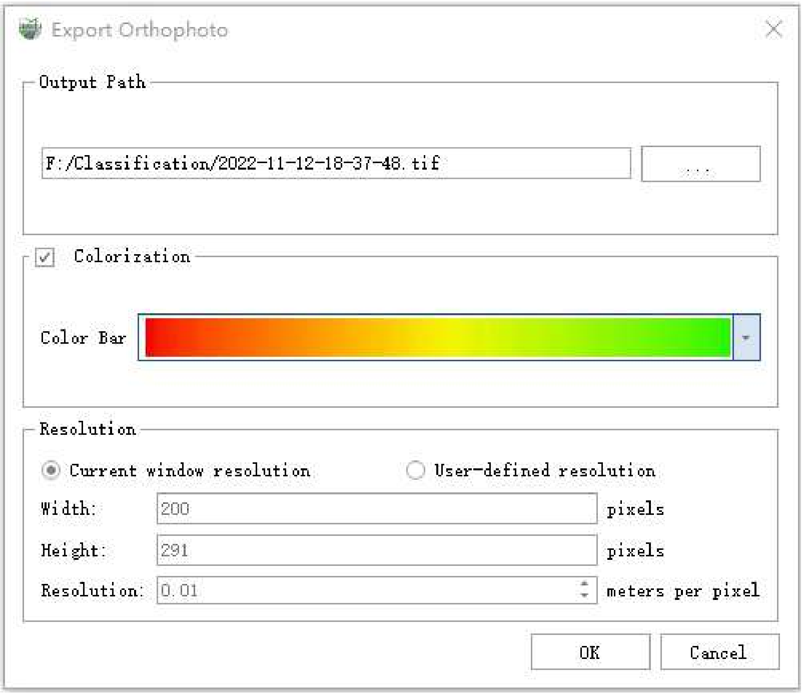


2.If you click Yes, the datum line and range line (if any) corresponding to the elevation will be saved together, and the exported DXF can be opened with CAD.



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 rendering 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**： Then 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.

Appearance

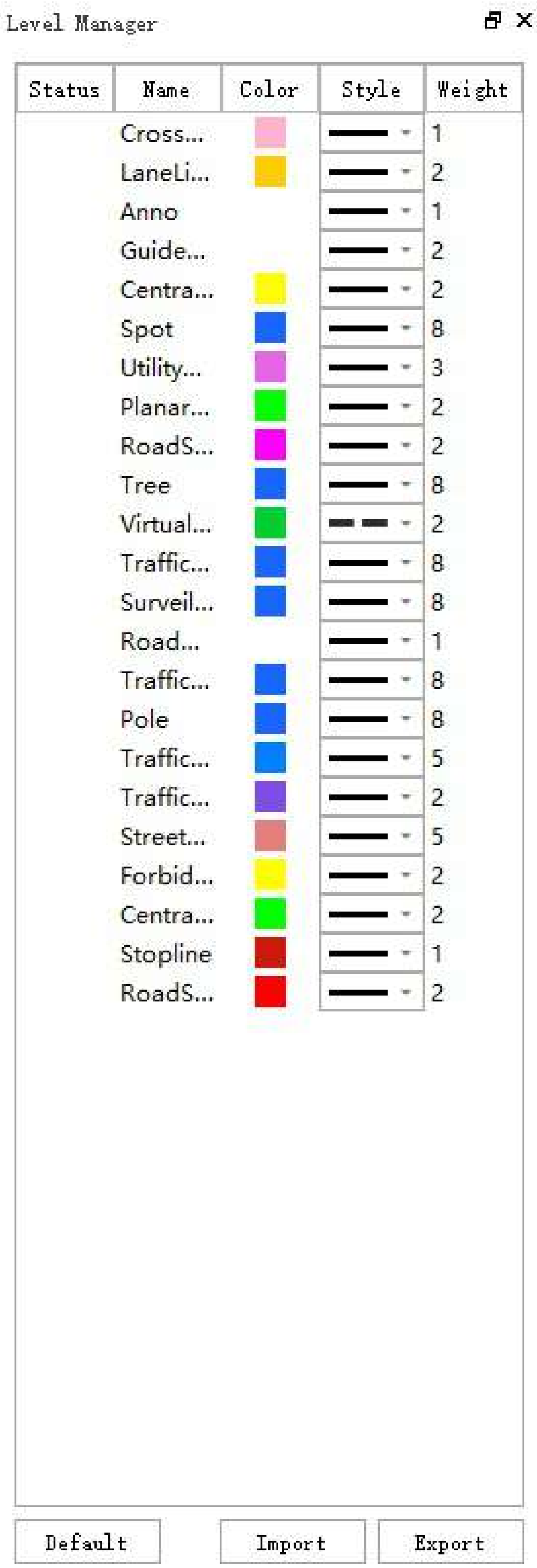
- Layers Manager
- Symbol
- Annotation
- Layer Layers

Layers Manager

Feature description：This function can modify the color and line type of the layer, and supports operations such as importing and exporting layers that have already been set.

Steps

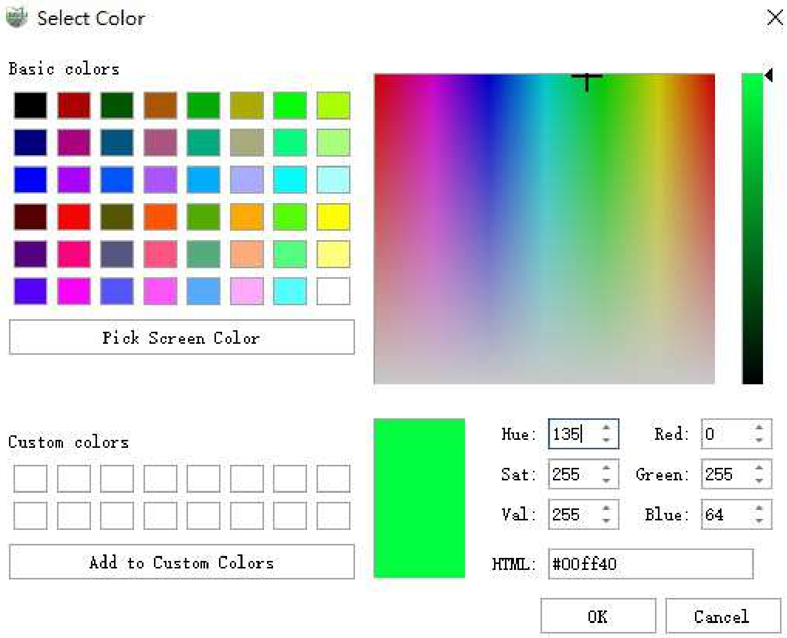
1.click **Layers Manager** Button, pop up **Layer Manager dialog box**



Parameter description:

- **Name**: Displays the name of each vector.

- **Color**：You can modify the color of the layer
  - Double-click to bring up the settings dialog.



- **Style**：You can modify the line type of a layer.
- **Weight**：You can set the width of the line.
- **Import**：Import**.ini**configuration file.
- **Export**：Export the set layer and save it as**.ini**. A configuration file ending in a suffix.
- **Default**：Click this button to restore the default values of all parameters.

Symbol

Show Symbols

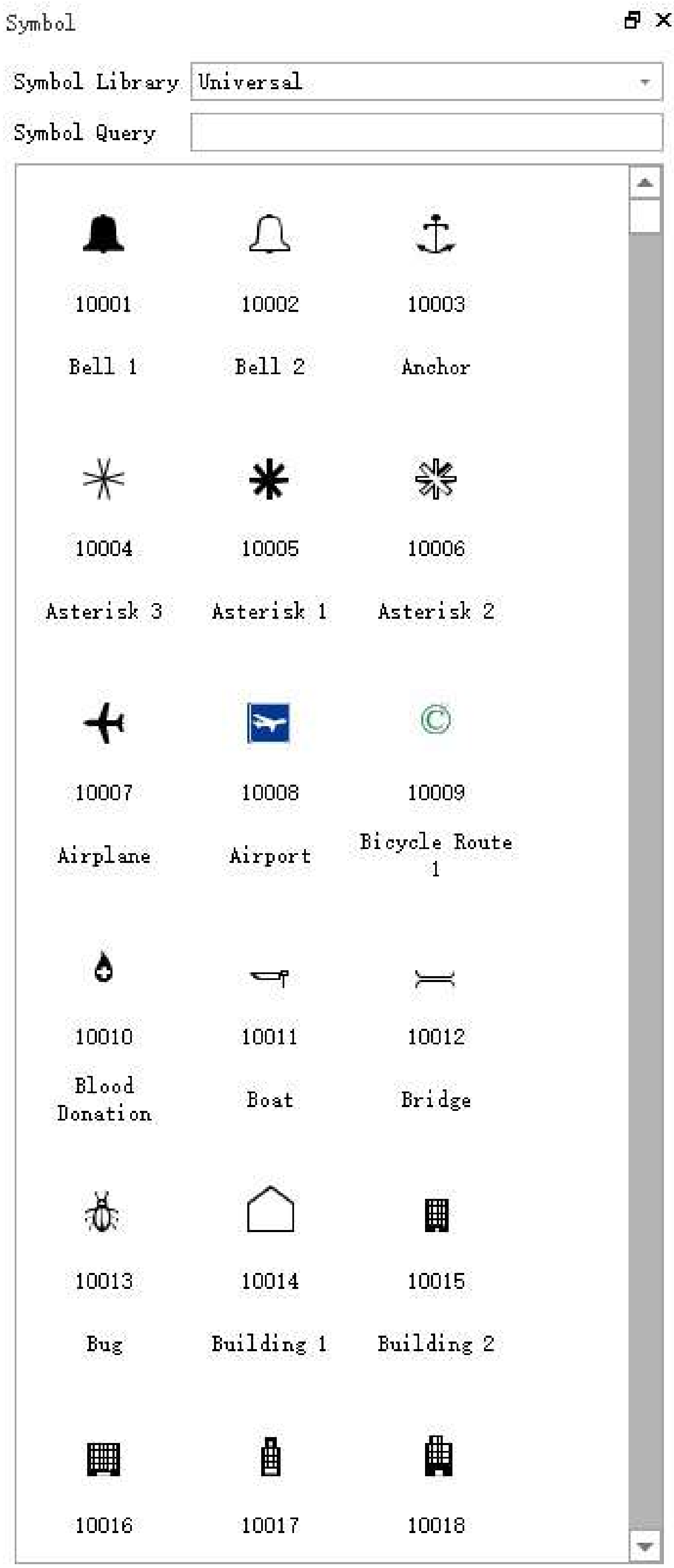
Feature Description：If point features in your project have symbols set, you can use this feature to control the display and visibiliity of symbols

Code Table

Feature Description：Open the built-in symbol library that the software already has

Steps

1.Click **Code Table** button, pop up **Symbol dialog box**.


Symbol
Symbol Library
Universal
Symbol Query
10001
Bell 1
10002
Bell 2
10003
Anchor
10004
Asterisk 3
10005
Asterisk 1
10006
Asterisk 2
10007
Airplane
10008
Airport
10009
Bicycle Route 1
10010
Blood Donation
10011
Boat
10012
Bridge
10013
Bug
10014
Building 1
10015
Building 2
10016
10017
10018

Parameter description:

- **Symbol Library**: Select Standard or Universal Mode
- **Symbol Query**: Enter via the text box to find the symbol you want.

Add Feature

Feature Description: This feature is used to collect point features directly with symbols.

Steps

1.Click **Add Feature** button, pop up **Code table**。

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.

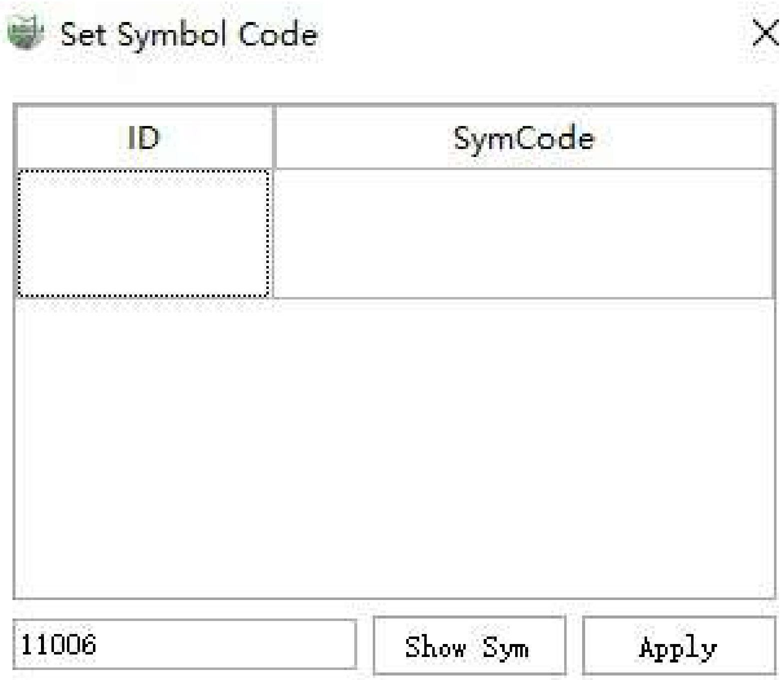
> Click**Show Symbol**，You can display the icon of the corresponding symbol.

Set Code

Feature Description: This function can modify existing symbols or set corresponding symbols for point features without symbols.

Steps

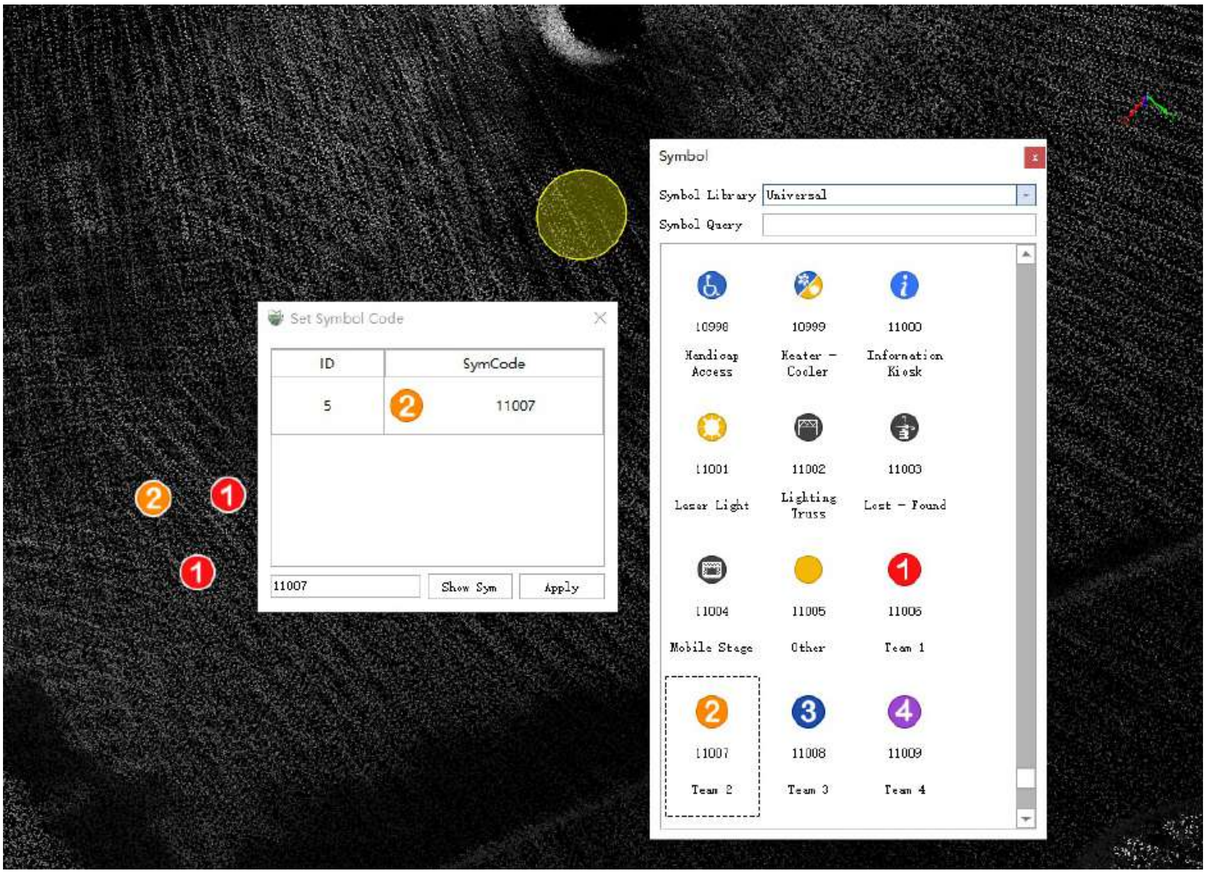
1.Click **Set Code** button,pop up**Set Code dialog box**.



Parameter description：

- **Show Sym**：You can display a symbol table and modify the current symbol.
- **Apply**：Click Apply to modify the symbol.

2.Click the point feature on the point cloud to modify/set the symbol. The set Encoding 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 Encoding dialog box. Click Apply to complete the symbol modification or add new settings.



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
Set Symbol Code
ID
SymCode
5
11007
11007
Show Sym
Apply

Annotation

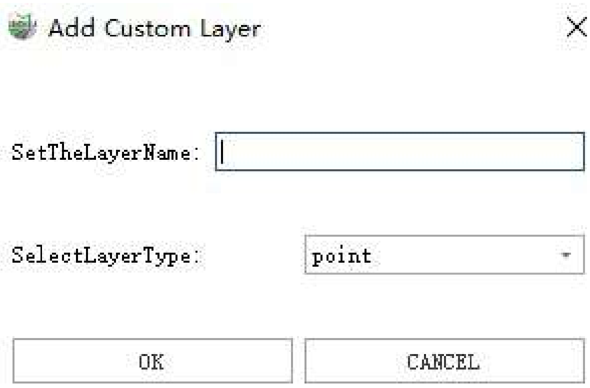
Add text annotation

Function description：This function can annotate the nodes/edges of point clouds, vectors, or vector objects

> This function must create a new annotation layer.

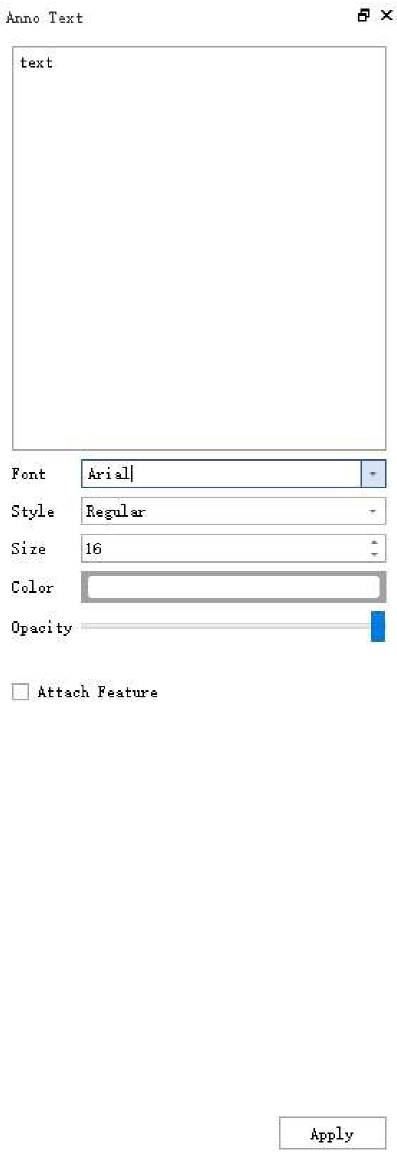
Steps

1.Custom vector >Right-click to add a layer and pop up**Add Custom Vector**。



2.Set the layer name as required, select the layer geometry type as**Anno**，And click OK

3.click **Add text annotation** Button, pop up**Anno Text dialog box**。



Parameter description:

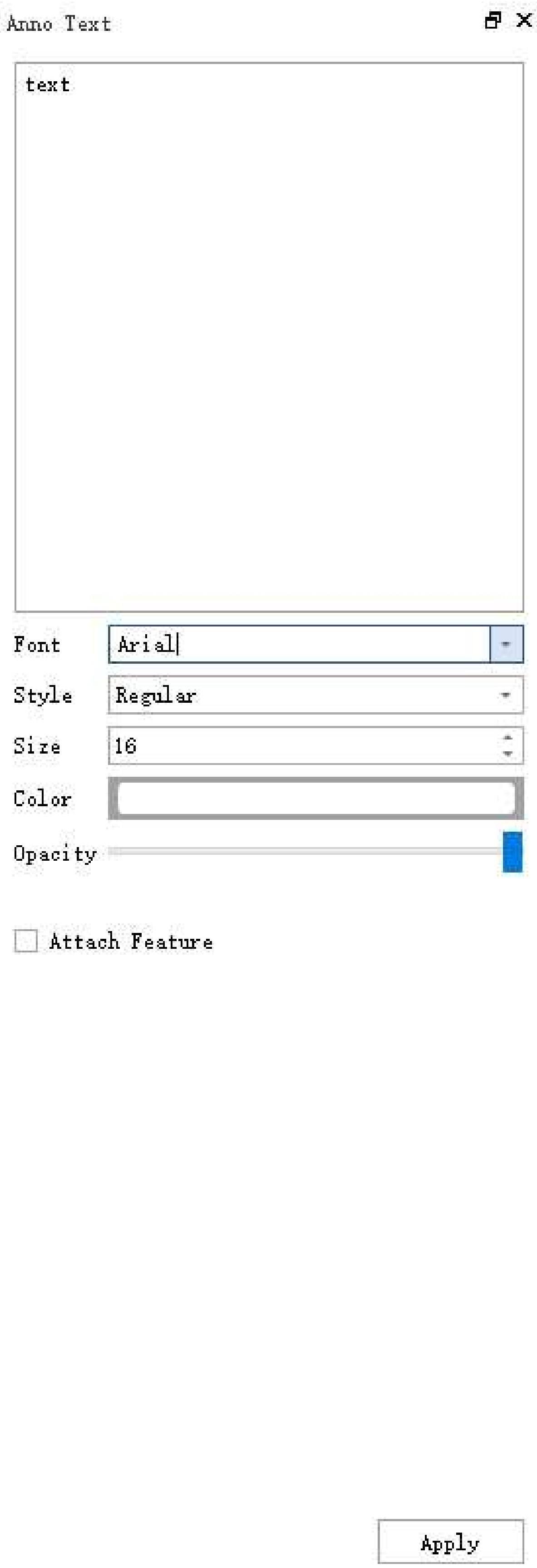
- **Font**: Set the font.
- **style**: The style changes with the font.
- **Color**: You can set the font color by yourself according to your needs.
- **Opaoity**: Text annotation indicates the intensity setting displayed on the point cloud.
- **Attach Feature**: Check or uncheck additional elements.
- **Apply**: Click Apply to save the current settings.

Modify Annotation

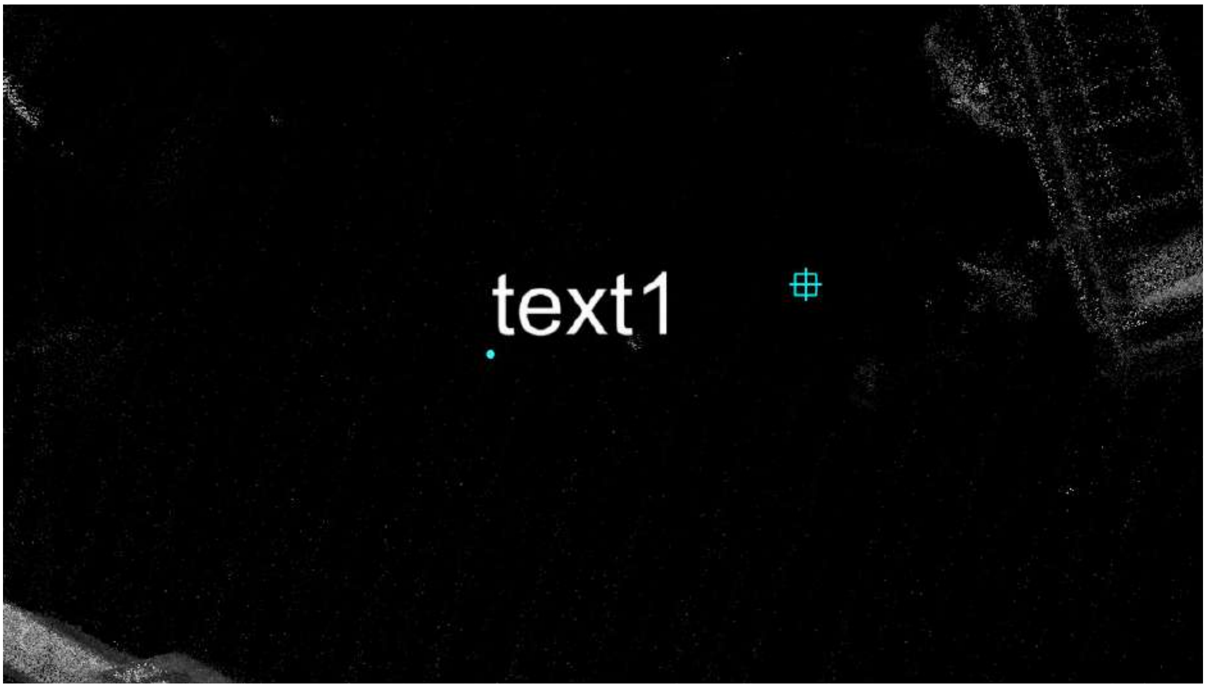
Function description：This function can modify existing annotations

Steps

1.Click **Modify annotation**Button, click the annotation you want to modify, and the**Text annotation**dialog box will pop up.



2. Re-edit in the text editing box, set the font, color, transparency, etc., click Apply, you can save the current settings.


text1

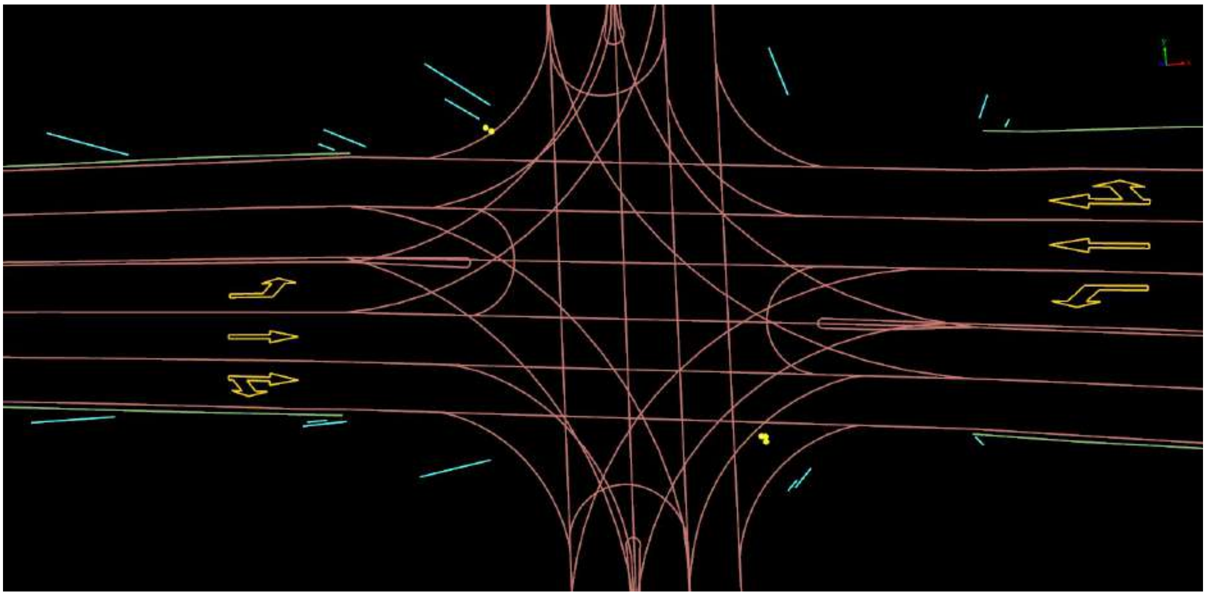
Label Layers

Function description：This function can set the font, size, color and other styles of the layer label, as well as the direction of the label and other operations.

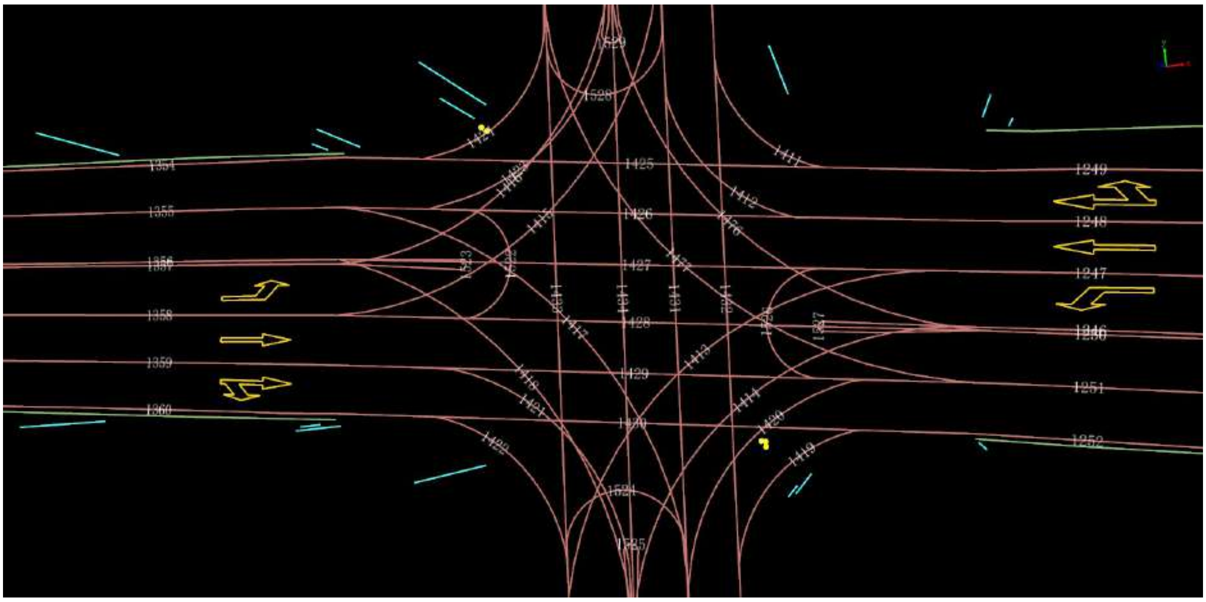
> Note: This function is used to ensure that there is vector result data in the project and that the attribute table field already has a value.

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**：Show or hide labels.
- **Label Direction**：You can choose a view or scene.
- **Font**：Choose the style of the font.
- **Size**：Set the size of the font.
- **Color**：Set the color of the font.
- **Bold**：Set the font to be bold.

> Note: Currently, label styles cannot be set by layer

Road analysis

Mainly used for road-related analysis work

Mainly includes：

- Road condition analysis
  - Pavement Damage Detection
  - Clearance Analysis
  - 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 Damage Detection
- Clearance Analysis
- Observation Observation Analysis
- Target Observation Analysis

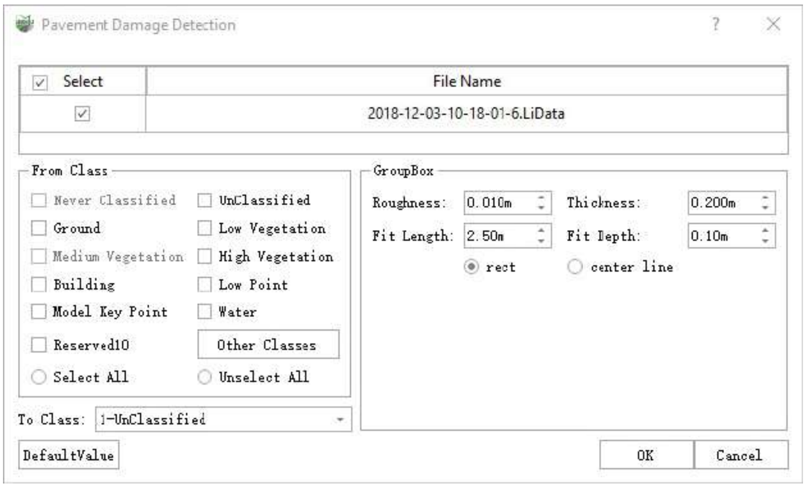
Road damage detection

Function description: 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 road damage detection function is only applicable to point cloud data scanned by high-precision lasers such as Riegl.

steps

1.Click **Road Damage 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 poor and thick, 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 a point cloud with a certain thickness to participate in the calculation, which can prevent some noise points from participating in the calculation.
  - Fitting length: used to 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: used to set the width of the point cloud used by the reference line for calculating the depth

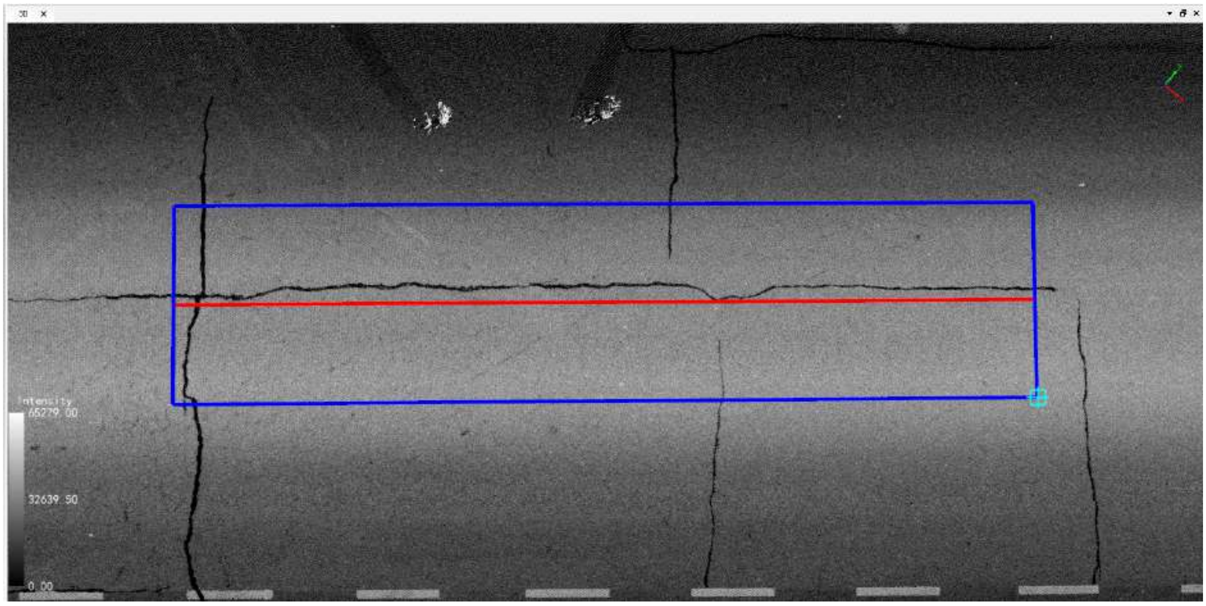
of damage, the default is 0.1 meters

> Length along the advance direction; width 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: used to 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: used to set the width of the point cloud used by the reference line for calculating the depth of damage, the default is 0.1 meters

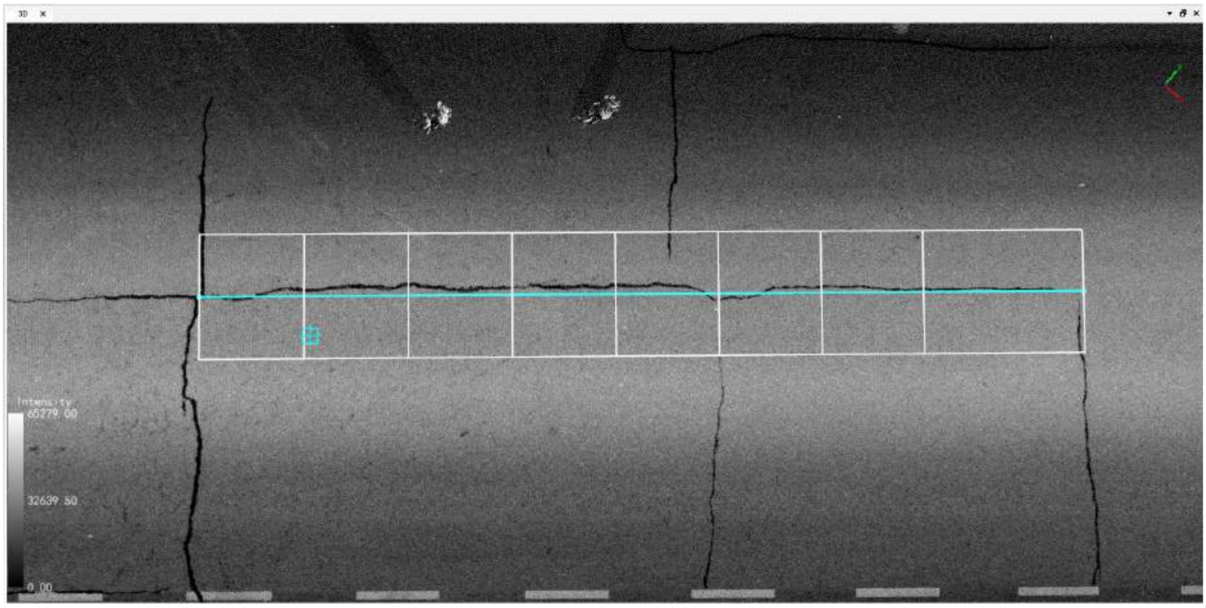
> Length along the advance direction; width perpendicular to the advance direction. In the detection process, the method of block detection is adopted, and the length and width of the block generally use the default parameters, unless the damaged position is larger than the default fitting length along 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.



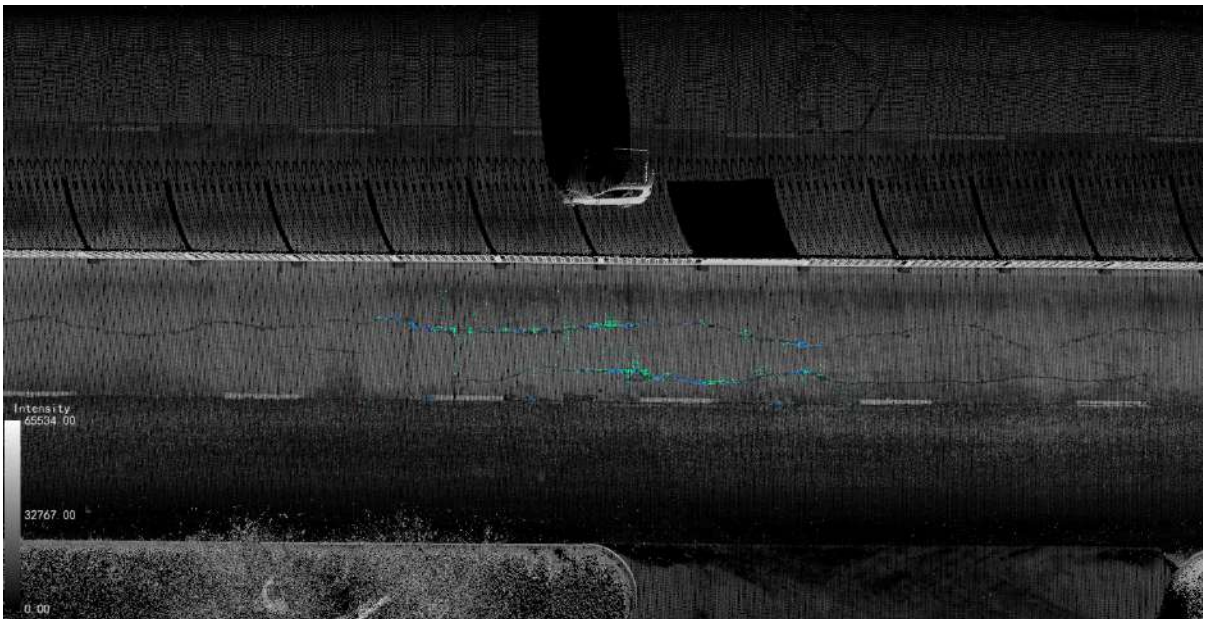
Rectangular Mode

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.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.

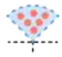


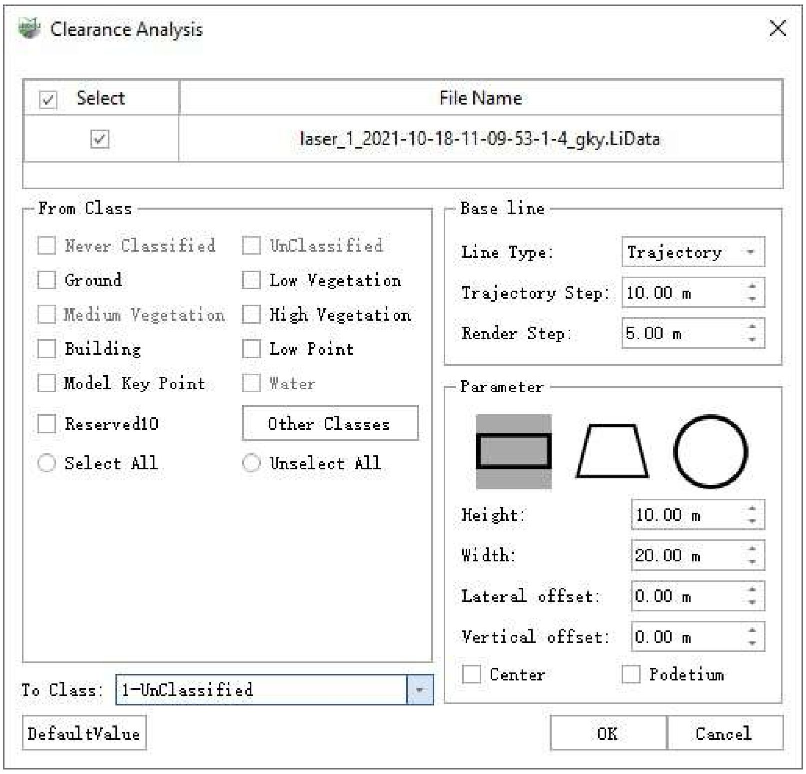
Rectangular Mode

Clearance Analysis

Function description: It is used to calculate whether there are occlusion points on the selected path, select the template and analyze the path, and analyze the points in the clearance area as the target category, which will be described in detail below.

Steps

1.Click **Clearance Analysis**  button, pops up **Clearance analysis dialog**



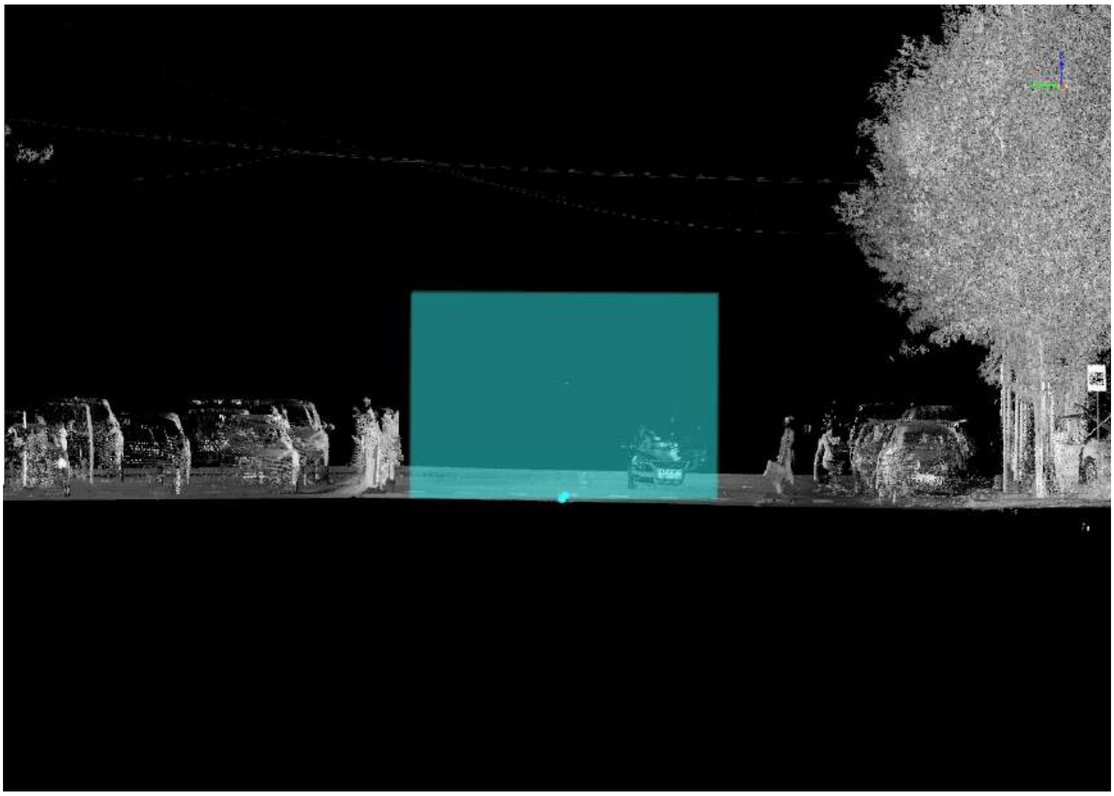
Parameter Description:

- **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

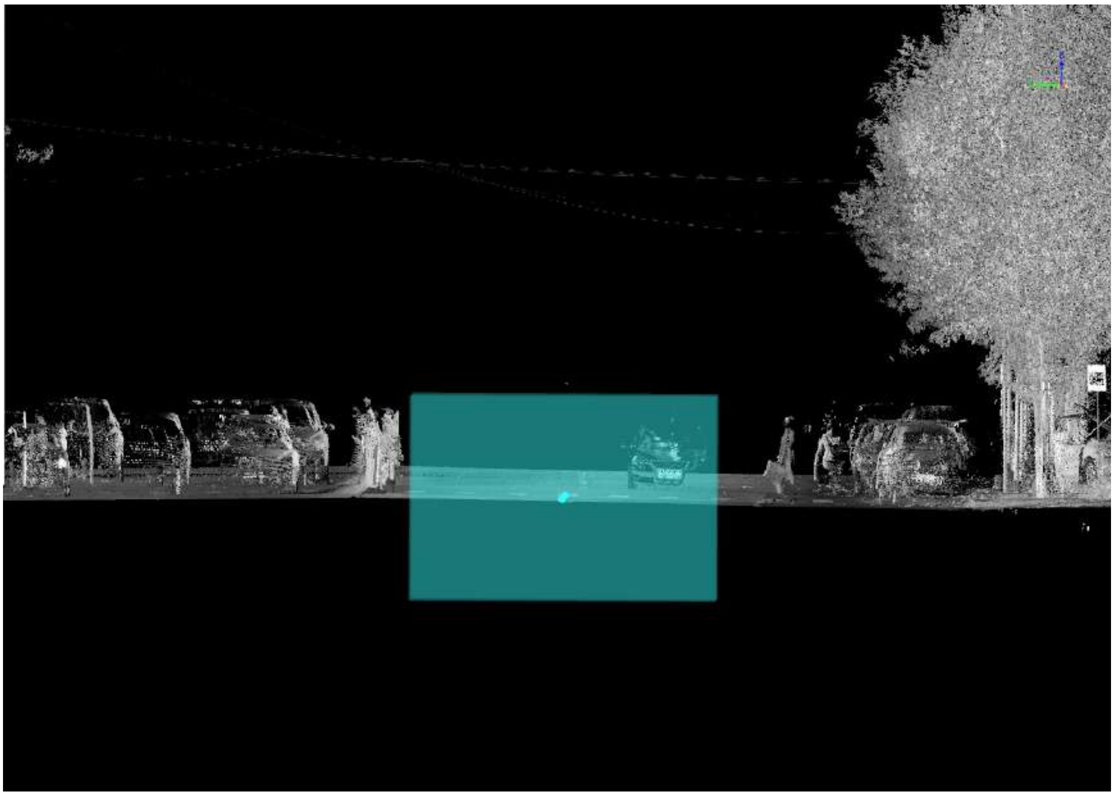
> Note: The slice position is not the actual vector node position, it is for display only.

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



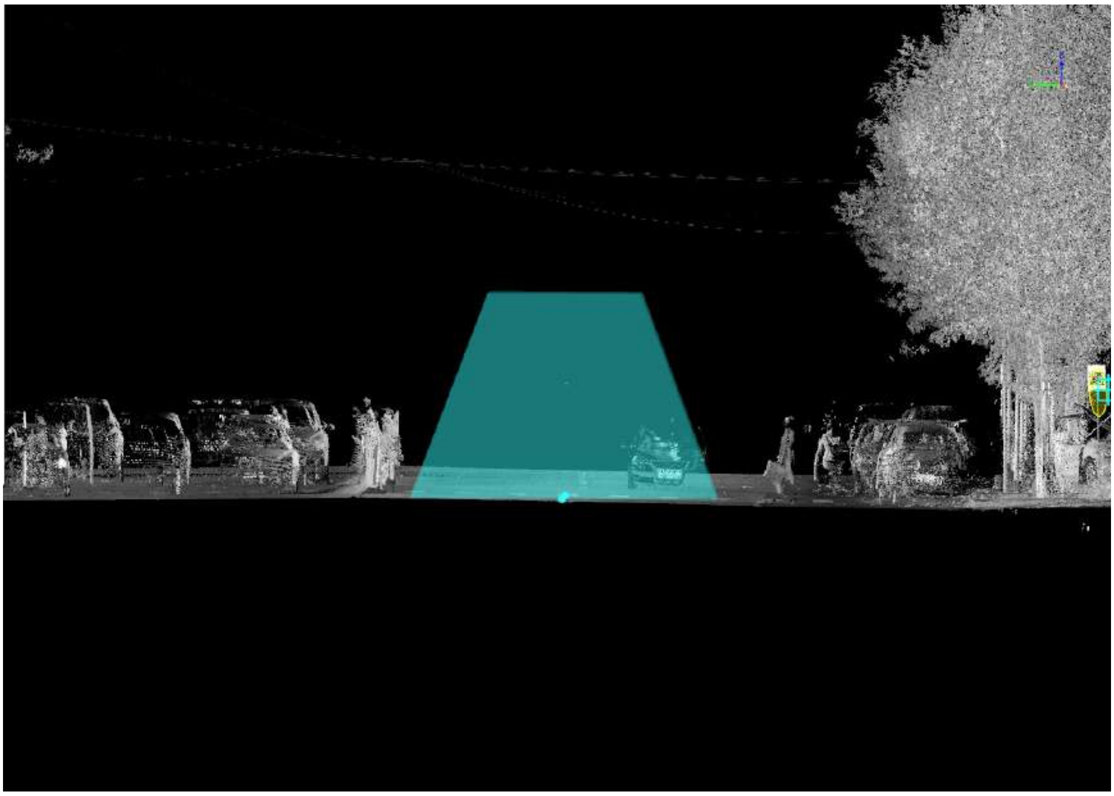
- Check the center: the geometric center position of the rectangle, which is the designated center position



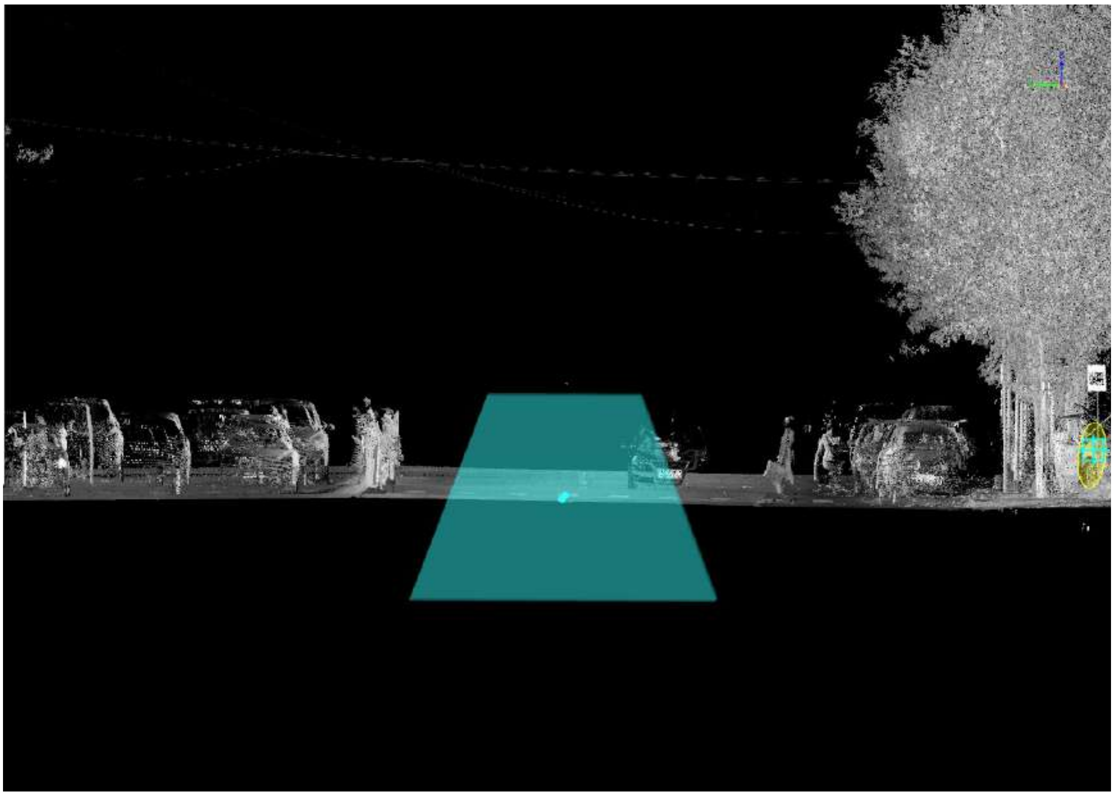
- **Height**: used to set the height of the analysis area rectangle
- **Width**: used to set the width of the analysis area rectangle
- **Lateral offset**: Take the selected baseline as the reference line, the horizontal distance between the specified center position of the rectangle and the reference line, the left side of the reference line advancing direction is a negative value, and the right side is a positive value

- **Vertical offset**: Taking the selected baseline as the reference line, the vertical distance between the specified center position of the rectangle and the reference line, the upper part of the reference line (Z axis) is a positive value, and the lower part 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

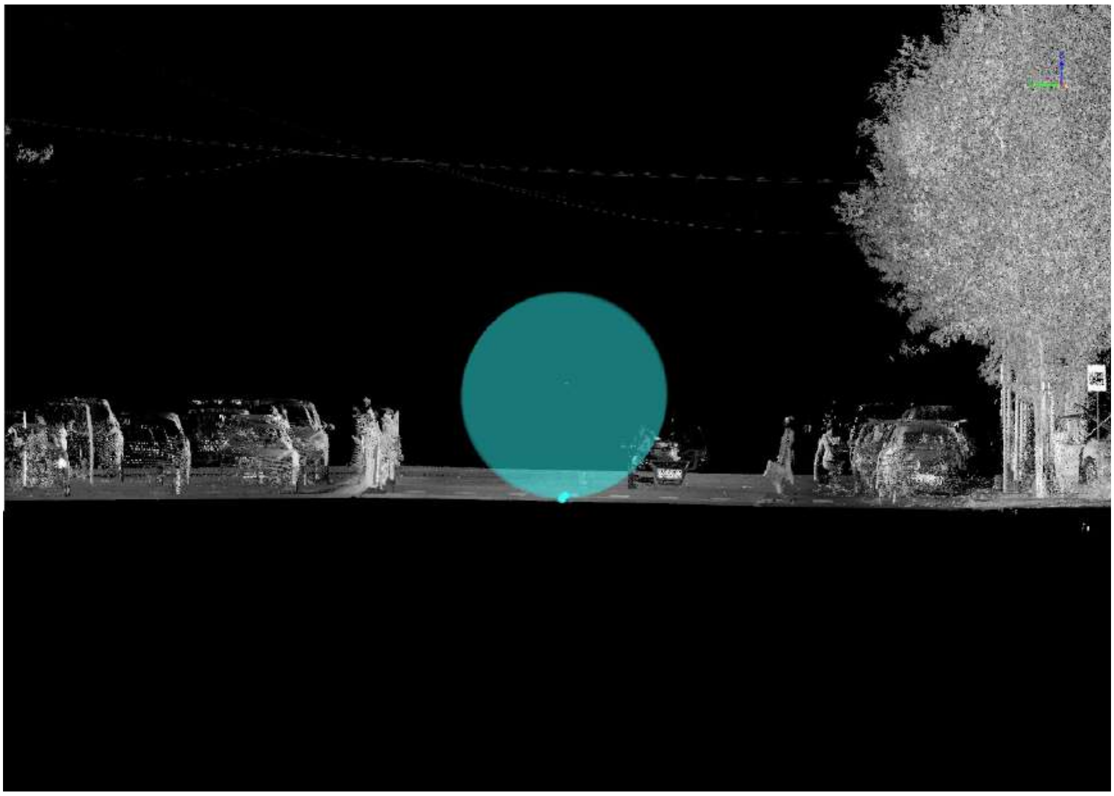
    

  - Check the center: the geometric center position of the trapezoid, which is the designated center position

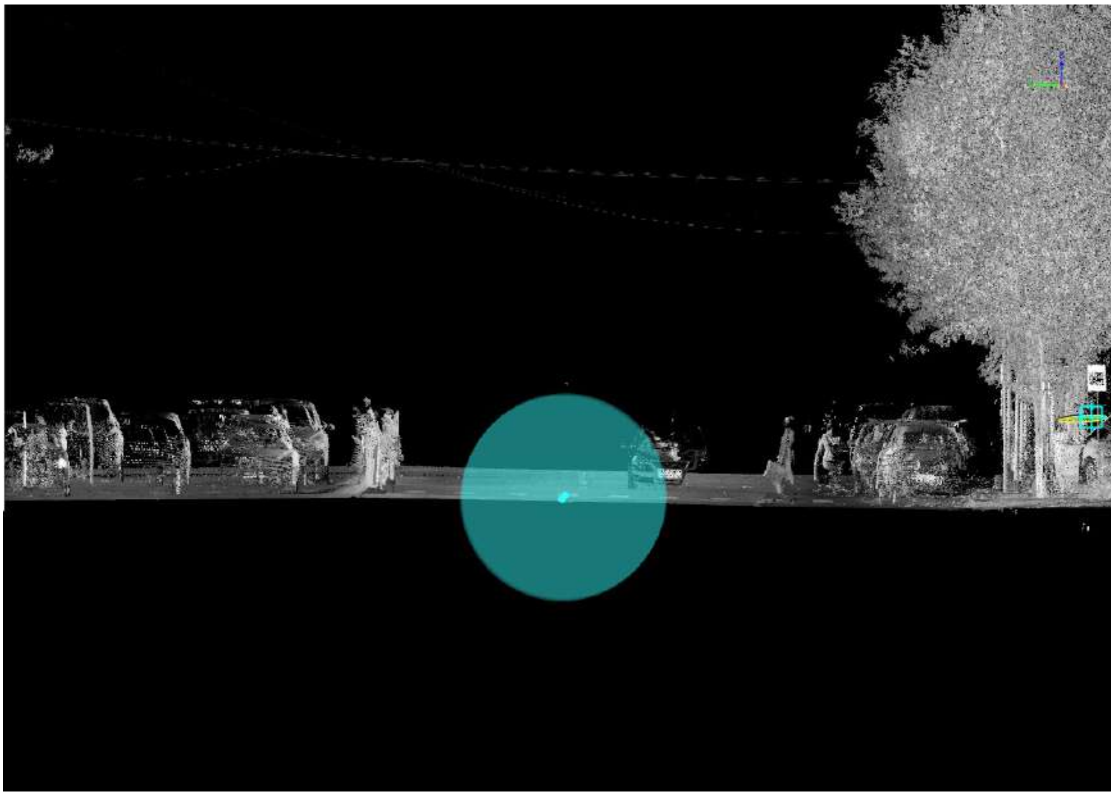


- **Height**: same rectangle
- **Upper base length**: used to set the width of the analysis area rectangle
- **Bottom base Length**: used to set the width of the analysis area rectangle
- **Lateral offset**: same rectangle
- **Vertical offset**: same rectangle

- **Circle Mode**:
  - Uncheck the center: the bottom position of the circle is the designated center position



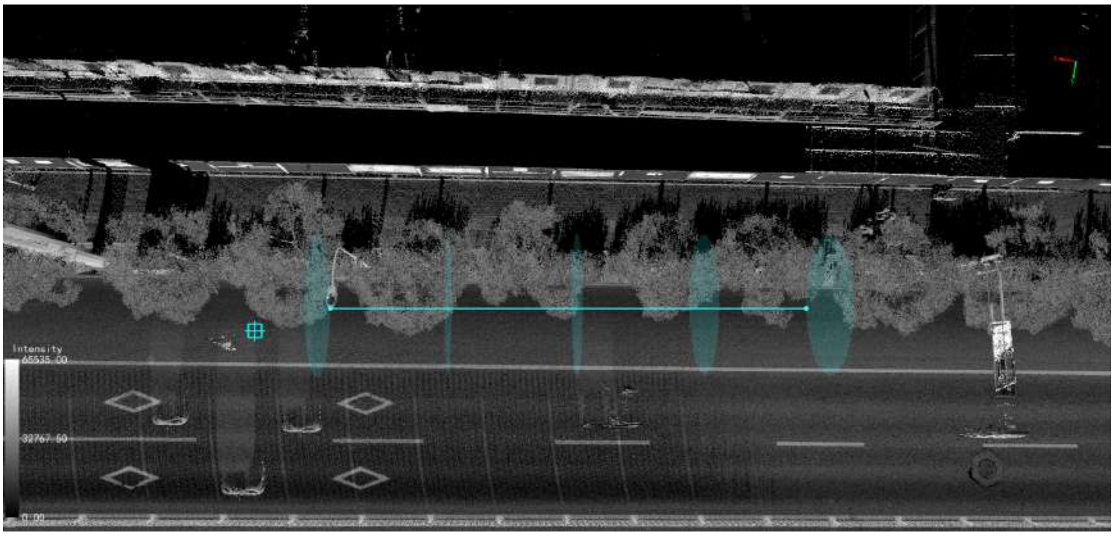
- Check the center: the center position of the circle, which is the designated center position



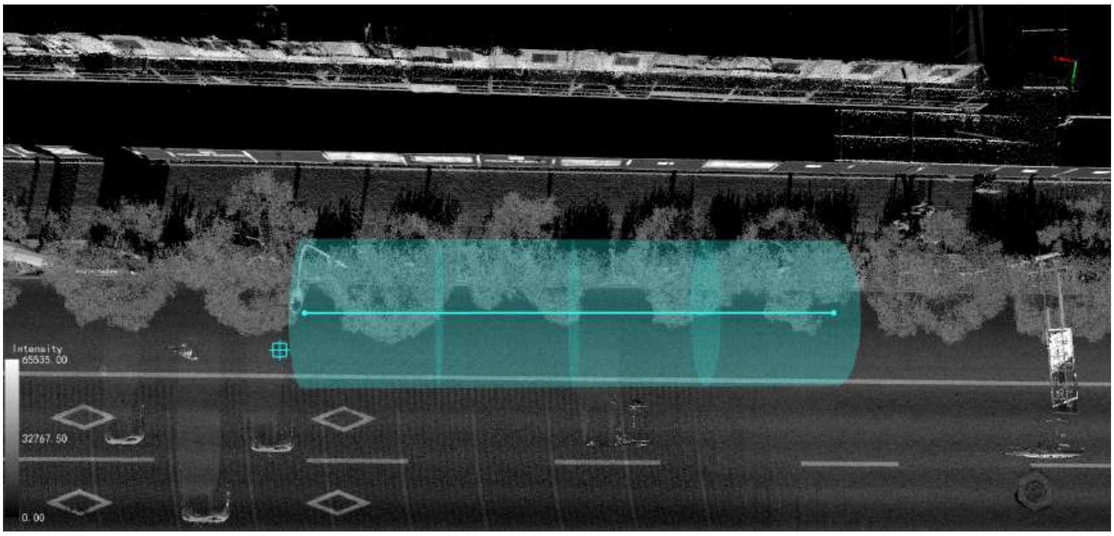
- **Radius**: The size of the radius of the circular parting
- **Lateral offset**: same rectangle
- **Vertical offset**: same rectangle

- **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



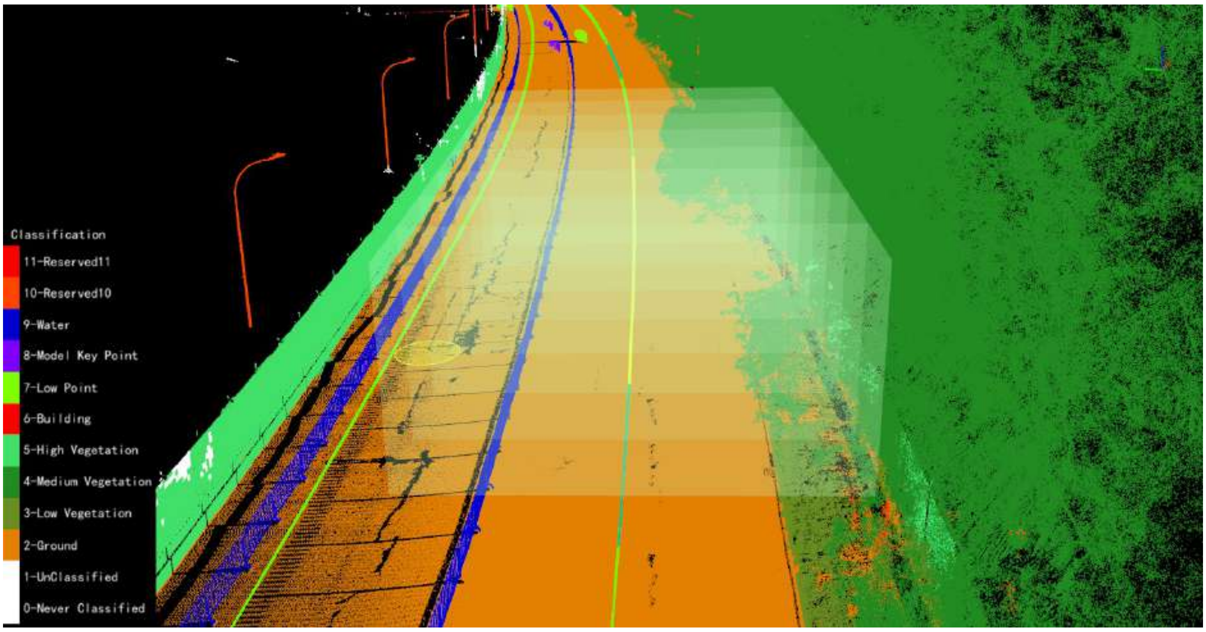
    - checked



- **Default Values**: Click this button to restore all parameter default values.

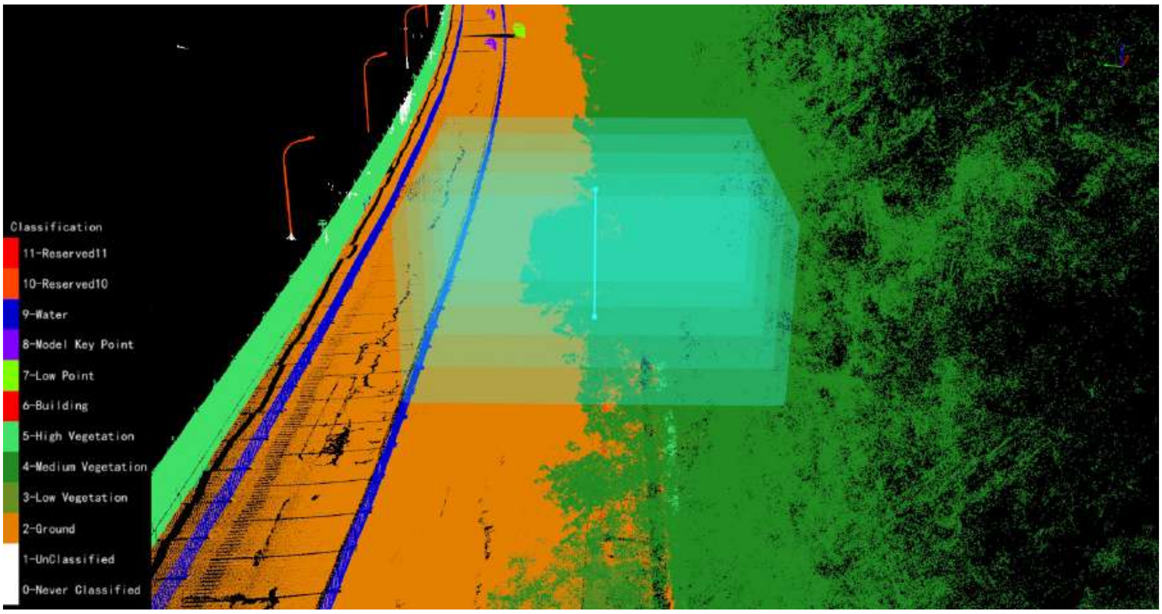
2.Select the line type (trace, vector line, custom line).

- **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 to select A certain track is used as a reference line.



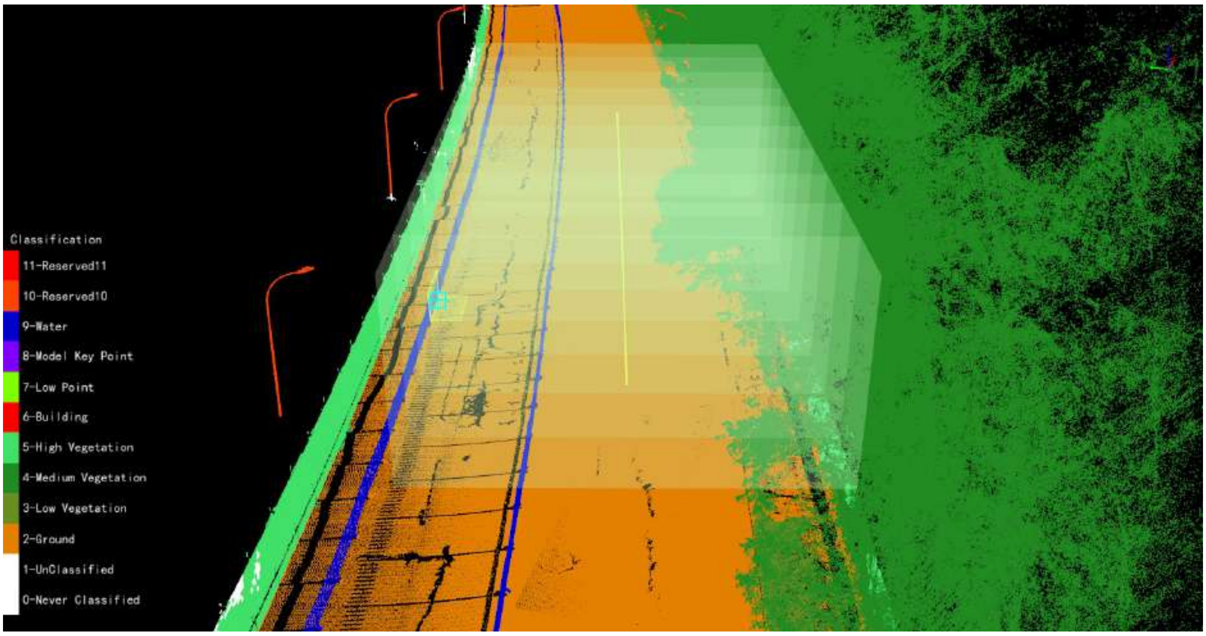
Observation point selection track

- **Vector line**: Select an existing vector line as a reference line



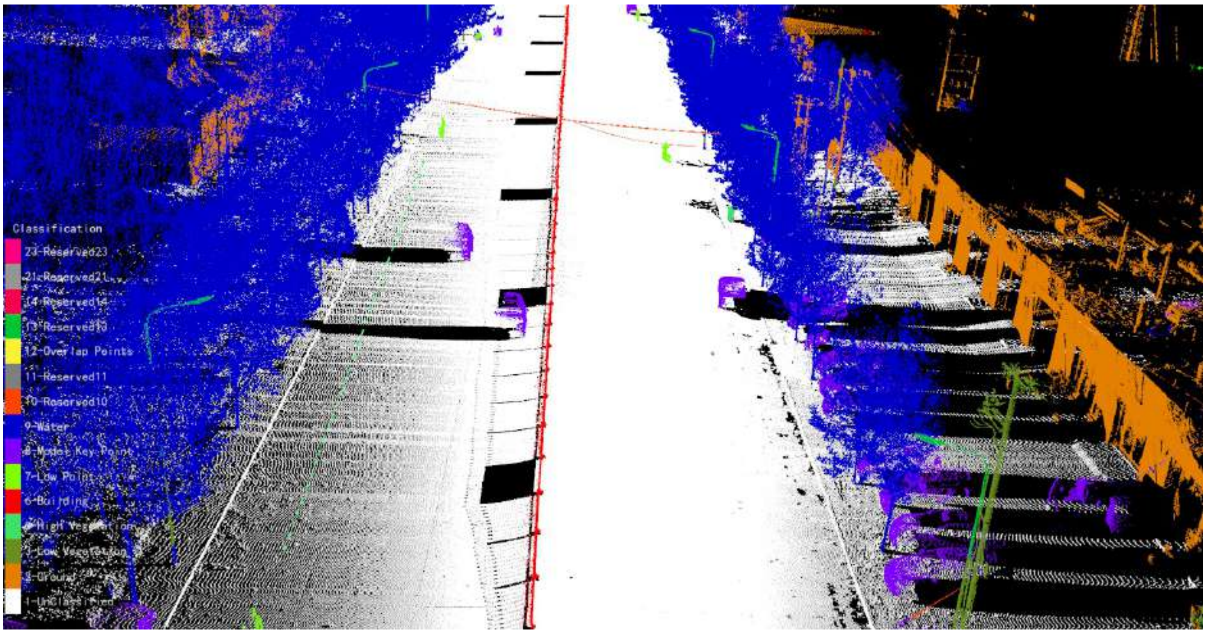
Observation point selection vector line

- **Custom Line**: After selecting a custom line, you can draw a temporary vector line at the position to be analyzed as a reference line

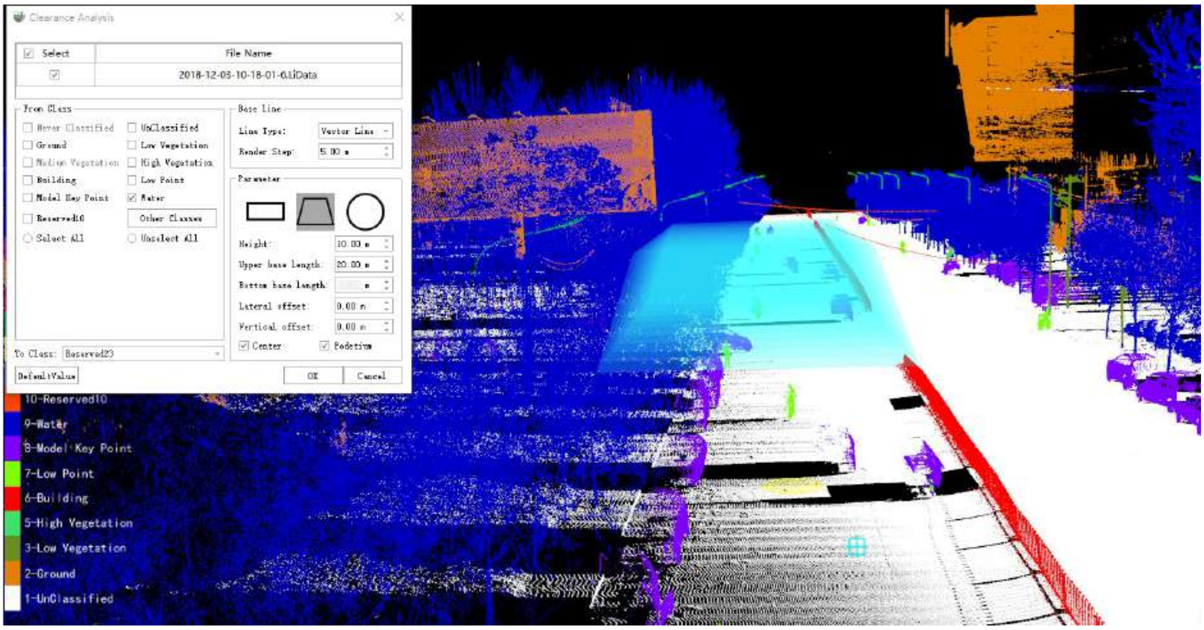


Observation point selection custom line

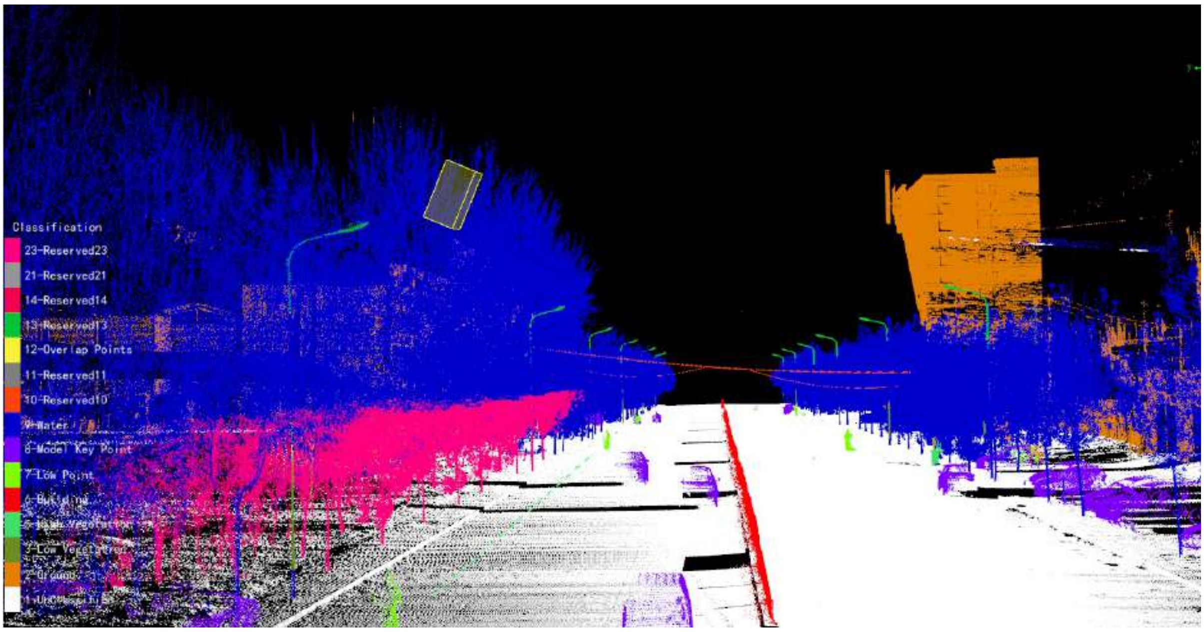
3.Select the source category, target category, and select the template (rectangle, trapezoid, circle) according to the actual needs, and set the parameters, then click OK.



Before opening headroom analysis



Select the template, after setting the parameters



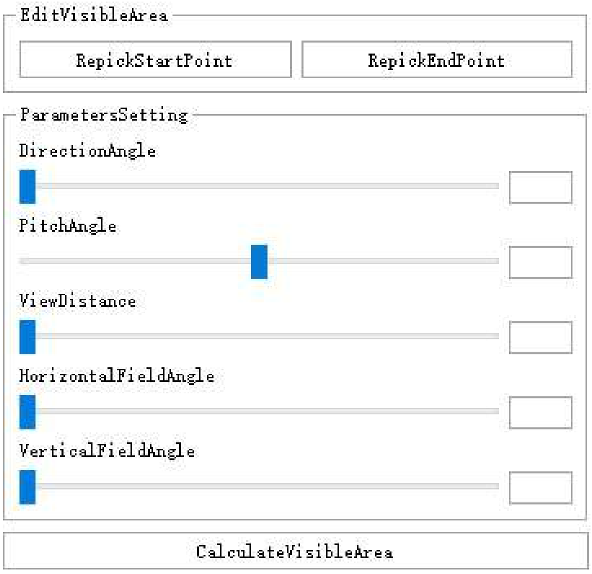
Results after analysis

Observation point visual field analysis

Function description : This function is used to originate from the observation point and analyze the occlusion of the target point from the observation point.

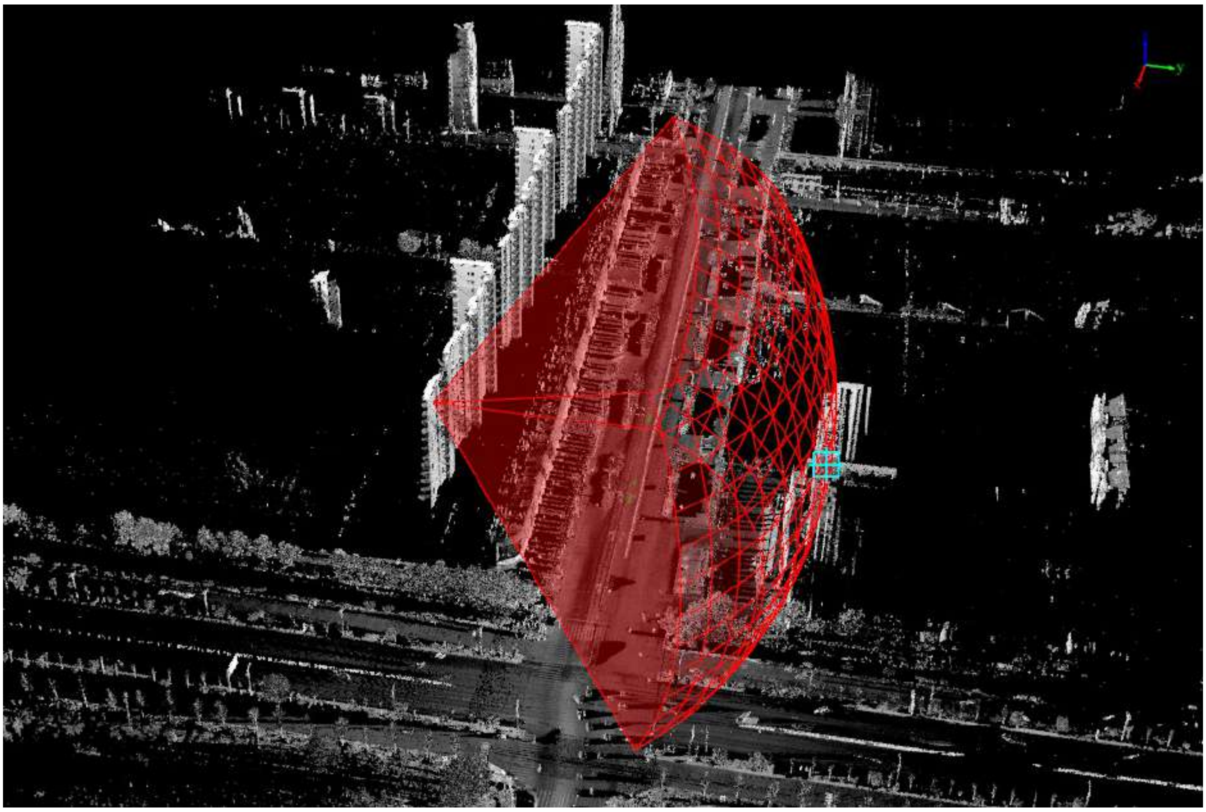
Step

1.In the road analysis menu bar, click **Observation point visual area analysis** button, the visual field editing settings page pops up



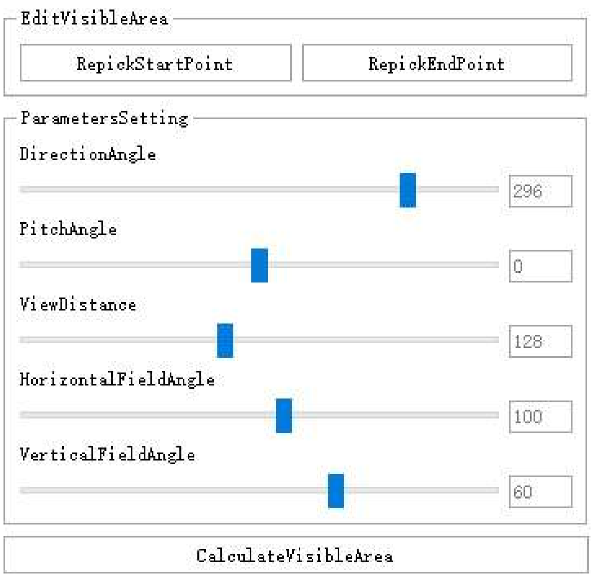
Visual field edit

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 cone.At this time, the optic vertebrae will no longer change with the mouse.



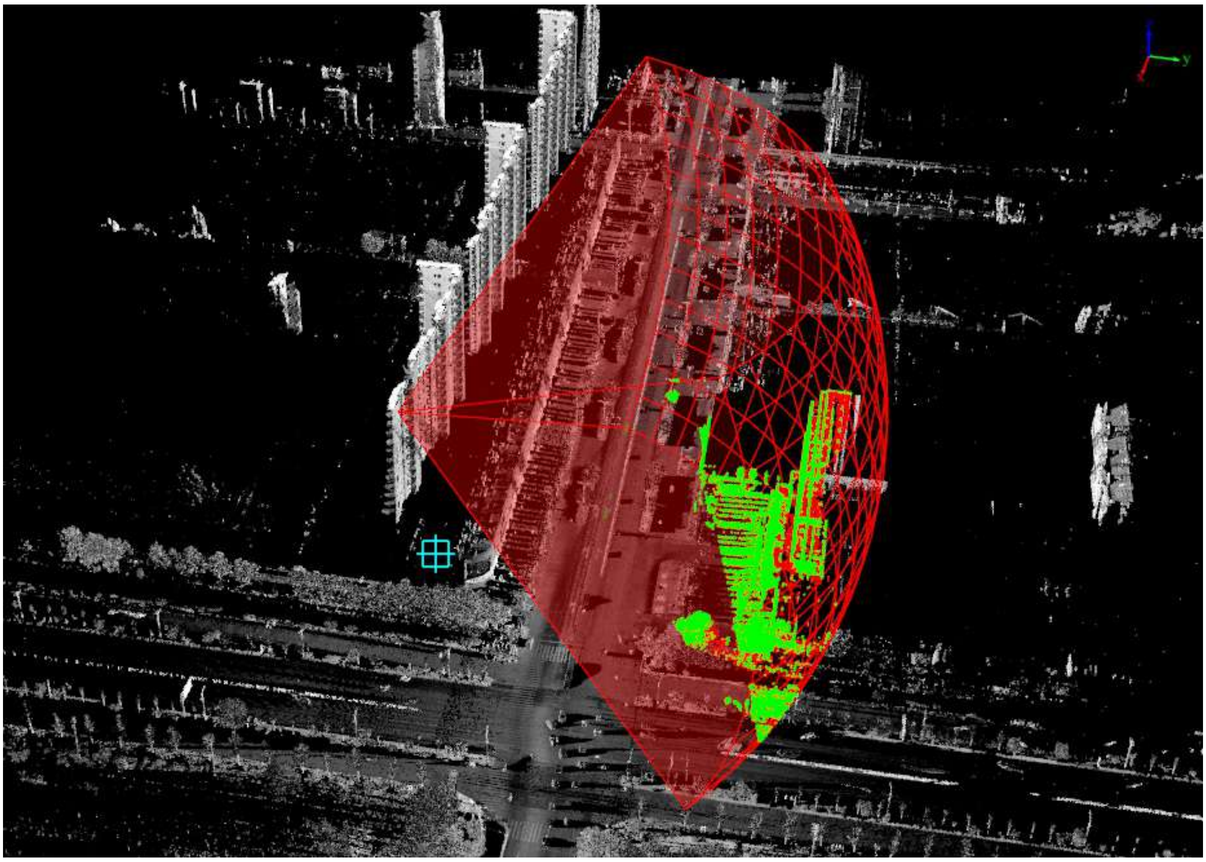
After the starting point is selected, the angle is displayed

4.In the settings pop-up 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, that is, you can adjust the horizontal observation range.
    - 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**, 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 : Calculate the visibility between the target point cloud and the observation point, which will be described in detail below.

Step

1.Click on the road analysis page**Target** button, the settings page will pop up on the software page at this time:

Parameter adjustment pop-up window

Parameter description:

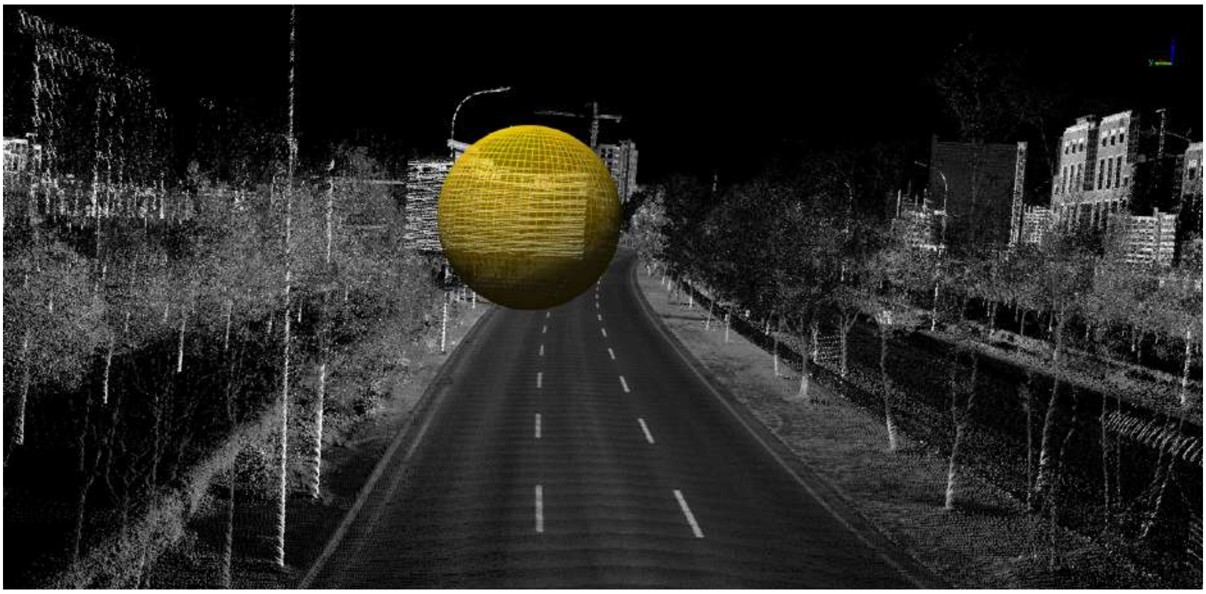
- **Select the target point**: Select the cluster of points to be observed through the ball selection.
- **Select observation point**: Select observation point data, including trajectory segment selection, existing vector line selection, and custom line drawing three modes.
- **Visual parameters**: Adjust the visual parameters to obtain different visual effects.
    - **Track step size**: When the track segmentation mode is selected, set the observation point to take the point step size on the track.
    - **Voxel size**: The smaller the voxel, the finer the analysis result and the longer the calculation time.
    - **Masking point threshold**: When the number of masking points between the target point and the

observation point is less than the threshold value, it is considered unobstructed

    - **Masking rate range**: The masking 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**: You can 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.

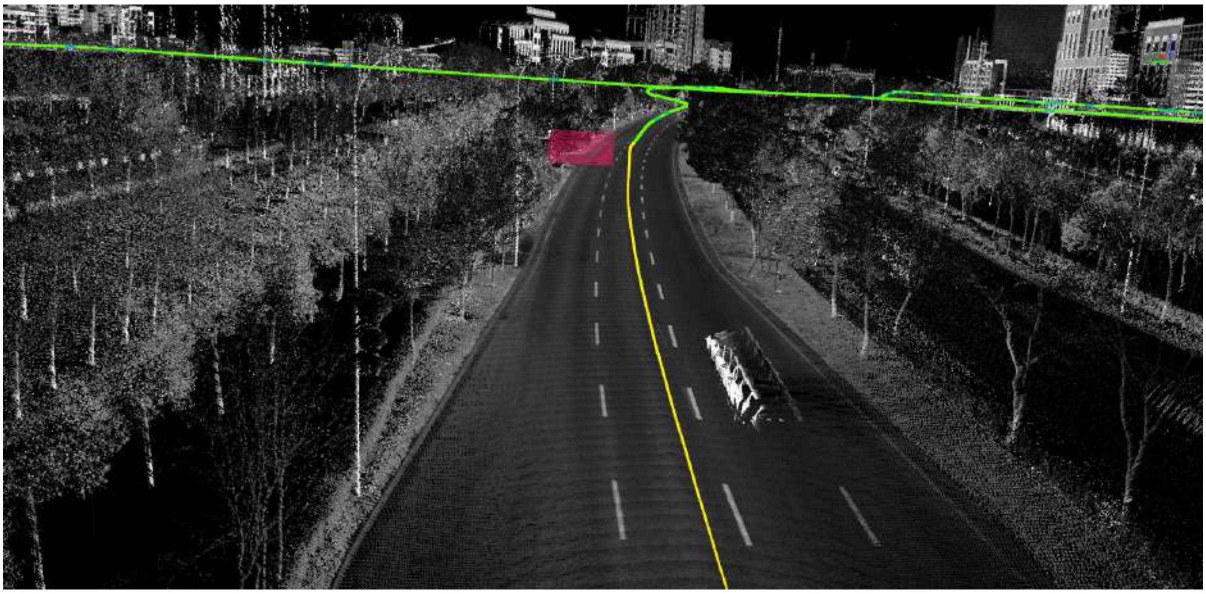


Before the target point is selected

After the target point is selected

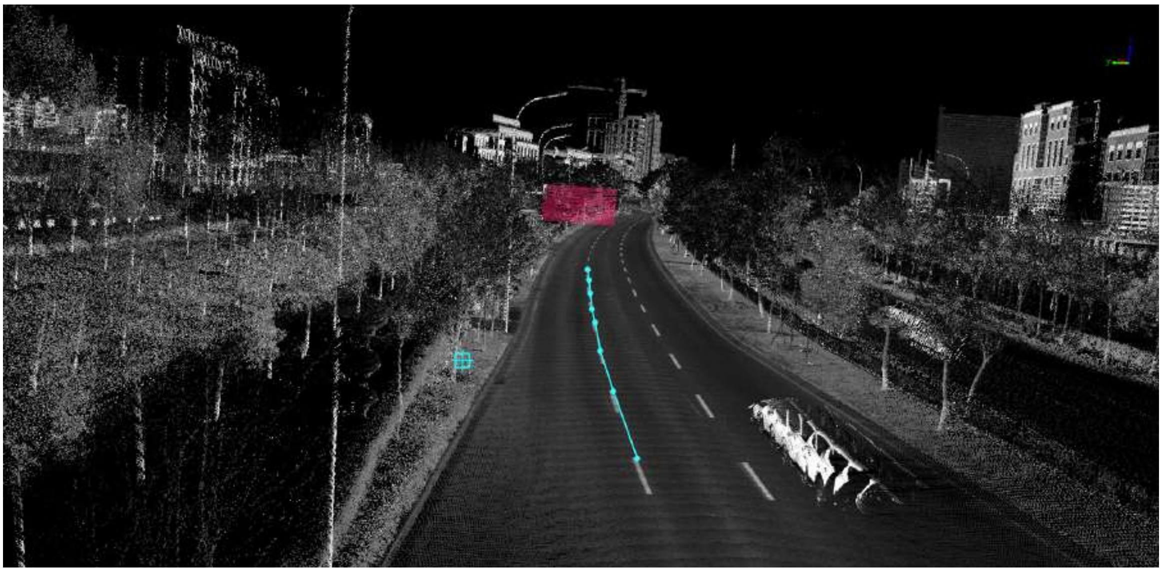
3.Click the Select observation point button to activate the observation point type drop-down box

- **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, you can select a certain track as the observation point.



Observation point selection trajectory

- **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.

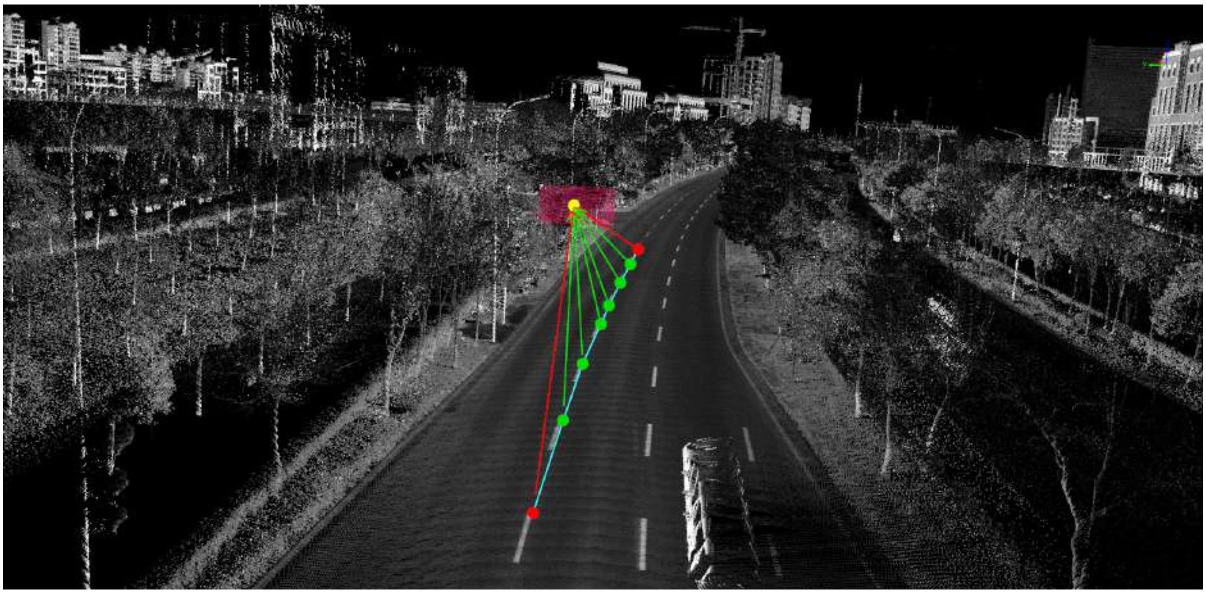


Select vector line at observation point

- **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 a custom line at the observation point

4.After clicking OK, perform visual domain analysis.



Analysis results

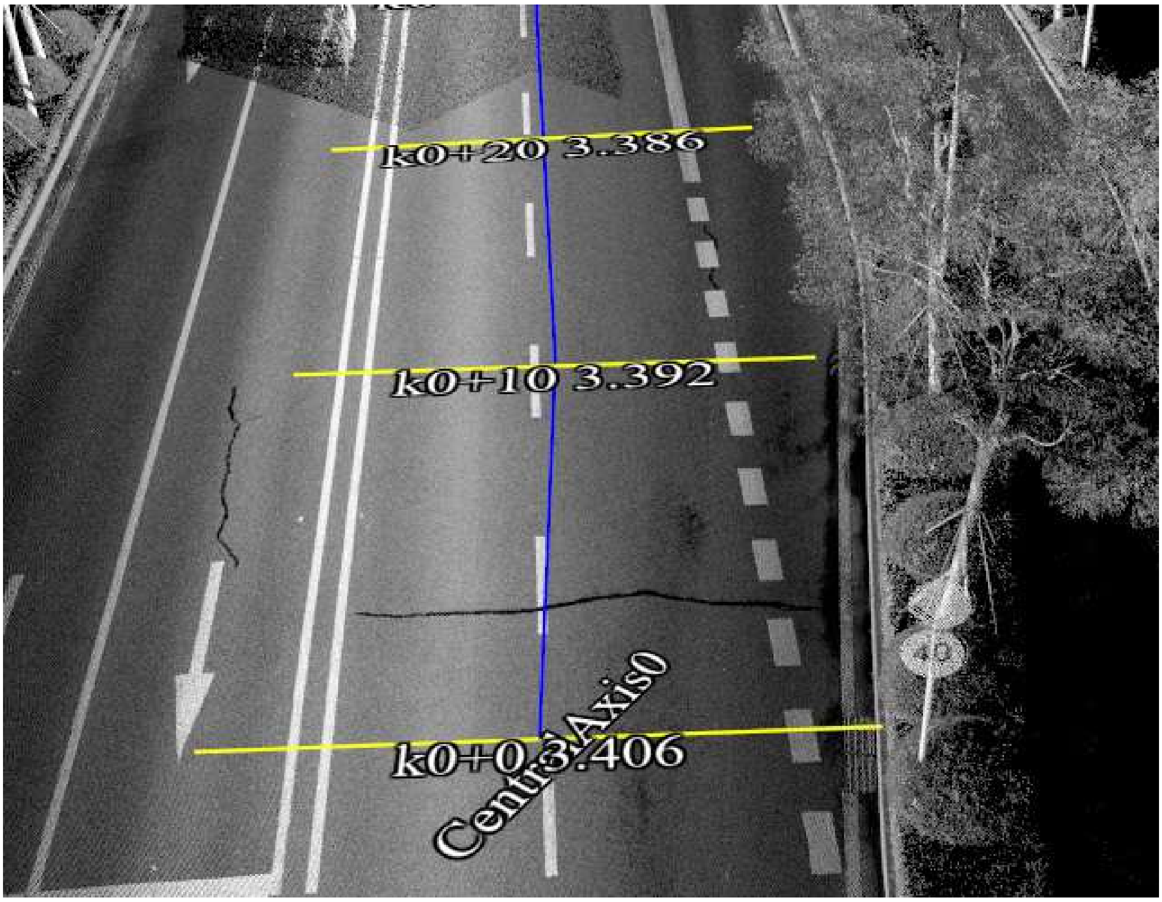
5.The analysis results can be exported by clicking the save button*.vis file, you can also reload the file to display and view in the future.

Road section analysis

Road reference line setting

Noun explanation

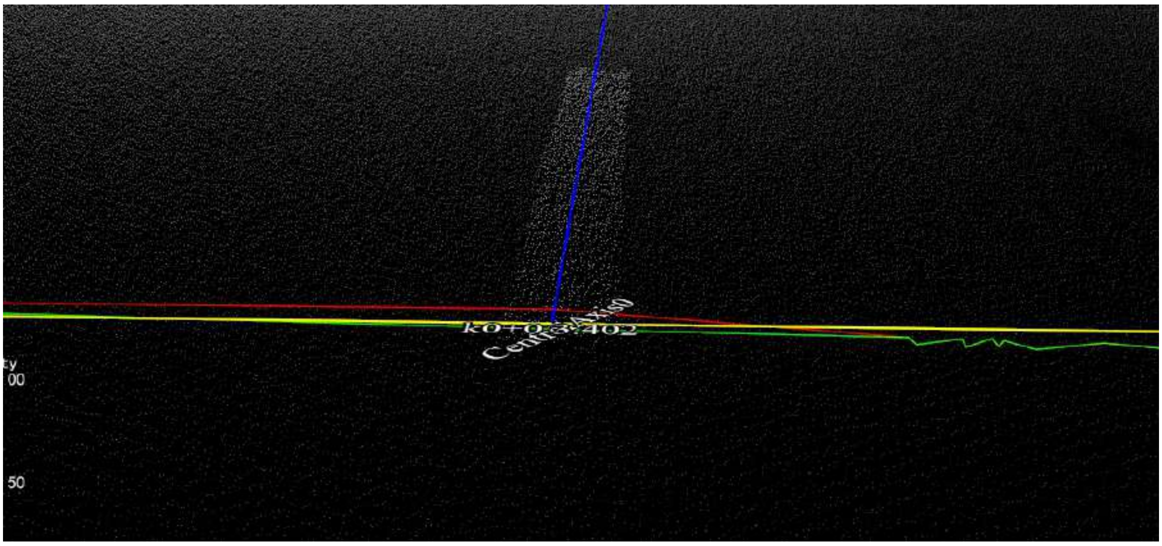
- **Vertical reference line**: Represents the direction of the road and is a virtual reference line, as shown in the blue line in the figure below.
- **Cross-section reference line**: Perpendicular to the direction of the road, it is a virtual reference line and the basic reference line for cross-section analysis. The yellow line in the figure below.



- **Design section**: The section data 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.The green line in the figure below.

- **Measurement section**: The section data generated by the current data, as shown in the red line in the figure below.

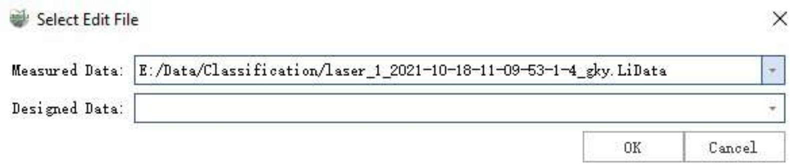
> In order to effectively distinguish, the data is manually exaggerated and displayed



Editor

Function description : 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, you can not choose.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

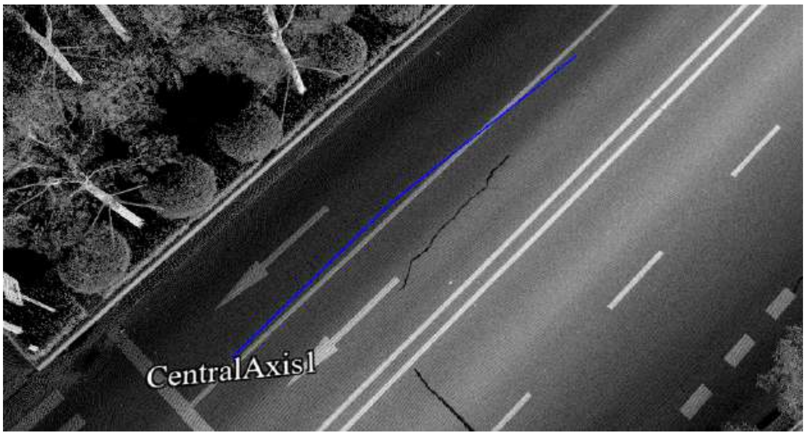
Exit the overall function of road cross-section analysis and eliminate various vector data generated during the use of the function.

> 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 the left mouse button in turn.



Dotted line dialog box

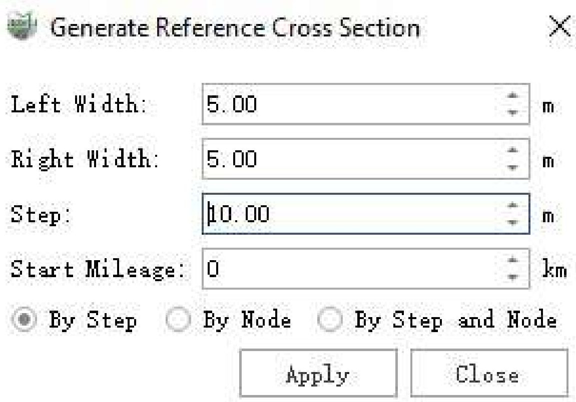
2.Double-click the left button to end the creation.

Select reference line

Function description : Click with the left mouse button to select an existing line object as the vertical reference line.

Generate Orthogonal Section Automatically

Function description : According to the set parameters, the reference cross-section is automatically generated.
Parameter dialog box



Parameter dialog box

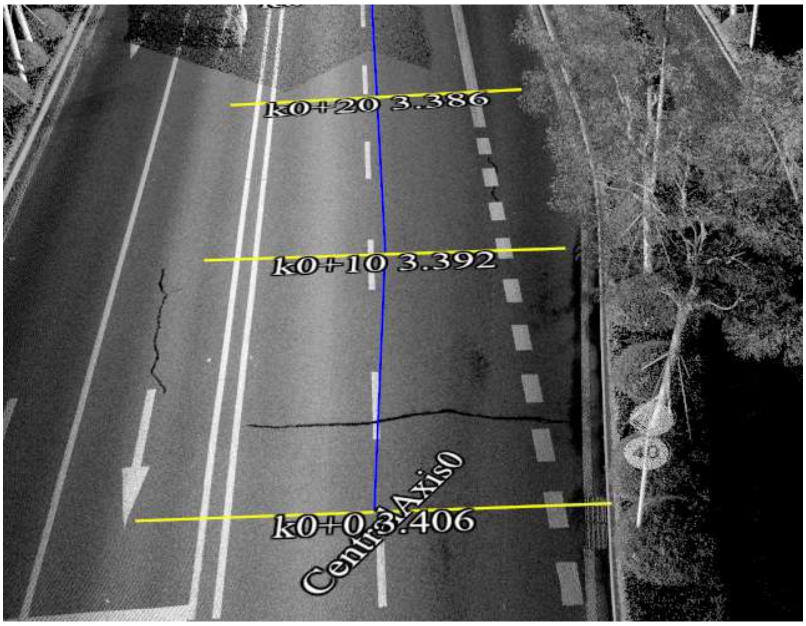
Parameter description

- **Left side 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 side 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 size**: 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 size**: 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 size and node**: Generate a reference cross-section by a specific step size, and process the nodes at the same time.

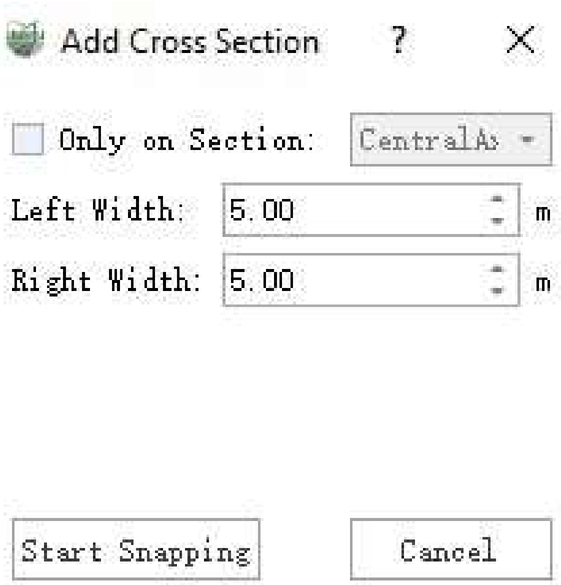
The blue vector line is a manually drawn longitudinal reference line, the name is CentroAxis0, where 0 is the line number, and the yellow line is a cross-section reference line automatically generated according to the step size, the name is K0 +0 3.406, where K0 is the pile number, + 0, +10, etc. Are 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 : According to the set parameters, the reference cross-section is automatically generated.



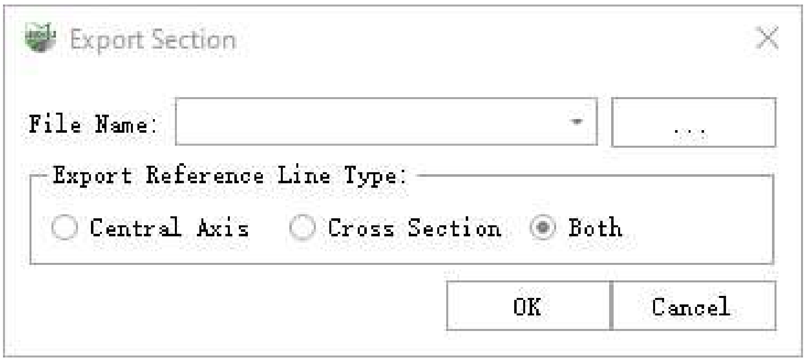
Parameter dialog box

Parameter description

- **Only add on a certain centerline**: Only add on a specific centerline (vertical reference line), which is invalid when other centerlines (vertical reference line) are captured.
- **Width on the left side**: The default is 5.0 meters, the distance from the leftmost side of the cross-section reference line to the longitudinal reference line.
- **Width on the right side**: The default is 5.0 meters, the distance from the rightmost side of the cross-section reference line to the longitudinal reference line.
- **Start capture**: 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 : Save the reference line and cross-section line in the form of a file.



Parameter dialog box

- **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.Support for saving results.dxf,.shp,.There are three formats of txt.
- **Export reference line type**:
  - Central axis: Only the longitudinal reference line is saved.
  - Cross section: Only the cross section reference line is saved.

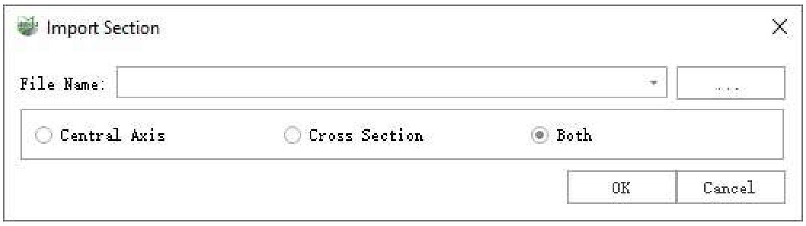
- Axis and cross-section: Save the longitudinal reference line and the cross-section reference line at the same time.

Import

Function description : External reading of existing section data, including reference lines, section analysis files, and design sections.

Import reference line

Function description : External read-in of reference line data saved by the save reference line function.
Parameter dialog box



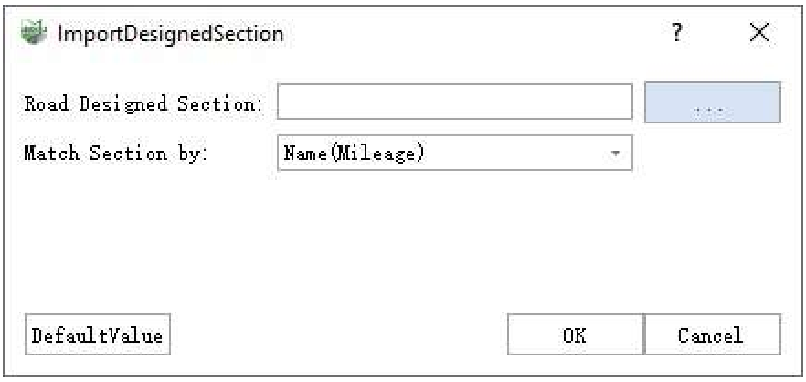
Parameter dialog box

Parameter description

- **File name**: Combine the buttons at the back of the edit box to select the file to be imported.support.dxf,.shp,.There are three formats of txt.
- **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 : 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

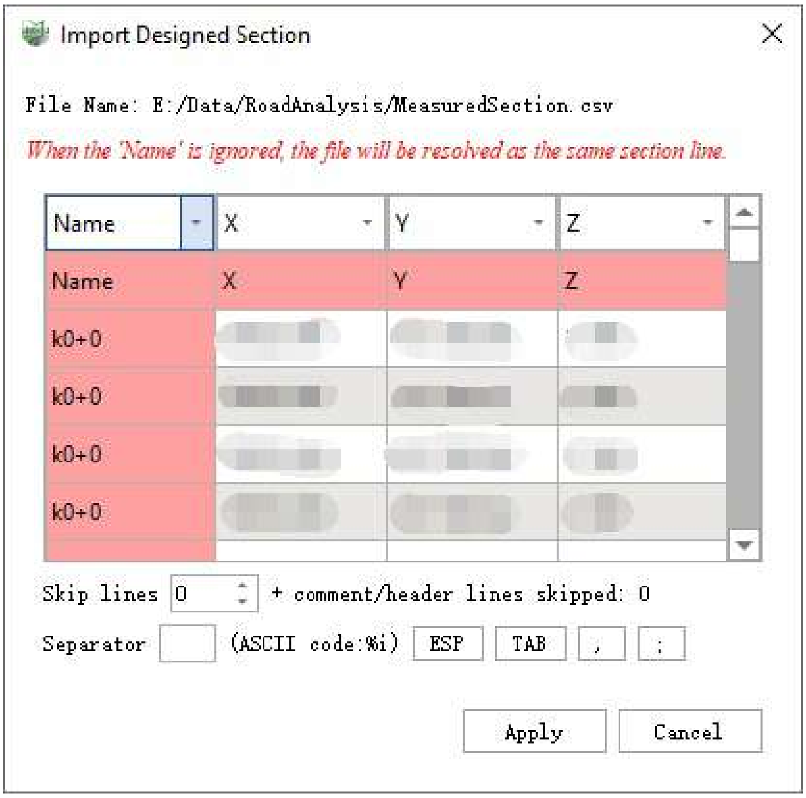
- **Road design section**: Combine the buttons at the back of the edit box to select the design section to be imported.csv file.

- **Matching section**: 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 can it be matched.

Steps

1.Select the design section you want to import.

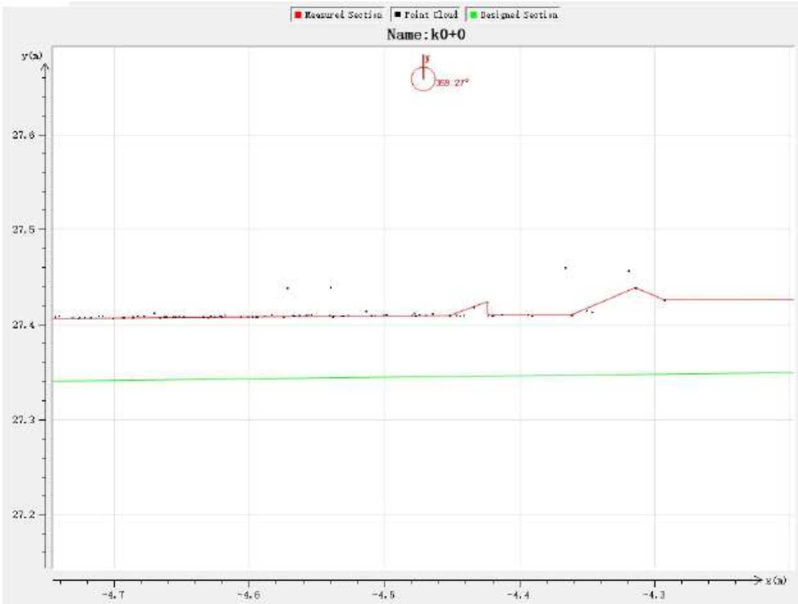
2.Confirm and adjust the format of the read data.



Parameter dialog box

3.Result view

The green line is the design 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, the generation method refers to Section Analysis Window Control

- **Generate Tunnel Section Views**

Generate Tunnel Section Views of the tunnel, refer to Section Analysis Window Control for the generation method

Hidden

Function description : Control the concealment of cross-section correlation vector lines.

Show spindle

Function description : Control the concealment of the vertical reference line.

Show horizontal line

Function description : Control the concealment of the cross-section reference line.

Display measurement section

Function description : Control the concealment of the measurement section.

Display design section

Function description : Control the concealment of the design section.

Show all

Function description : Control the display and concealment of all reference lines.

Clear section

Function description : 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 : Clear the data related to the vertical reference line.

Clear Cross Sections and its Measured Section

Function description : Clear the horizontal reference line and the corresponding measurement section.

Clear Measured Sections

Function description : Clear the measurement section.

Clear All Sections

Function description : Clear all data, including vertical reference lines, horizontal reference lines, measurement sections, and design sections.

Section analysis window control

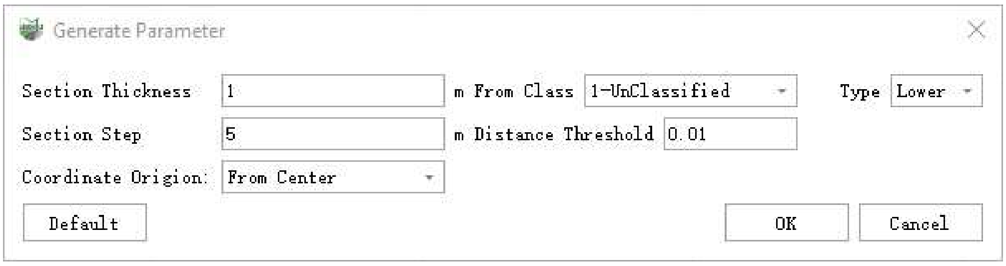
Function description : Calculate, modify, save and export cross-section related operations.

> 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.

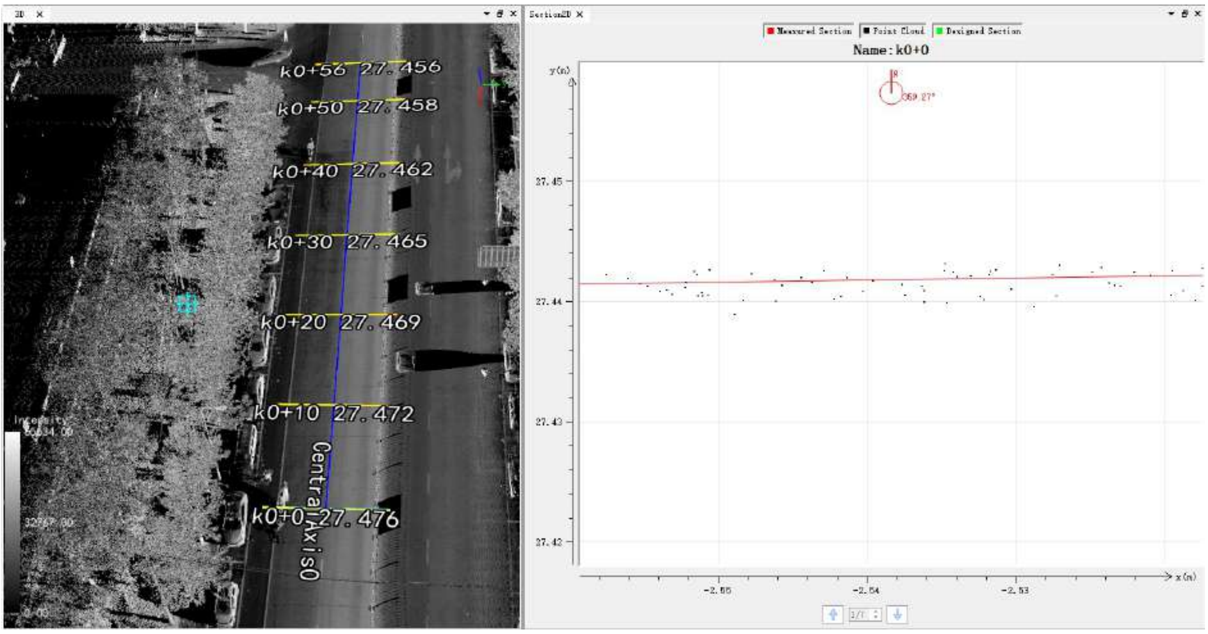


Parameter dialog box

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 origion**: 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

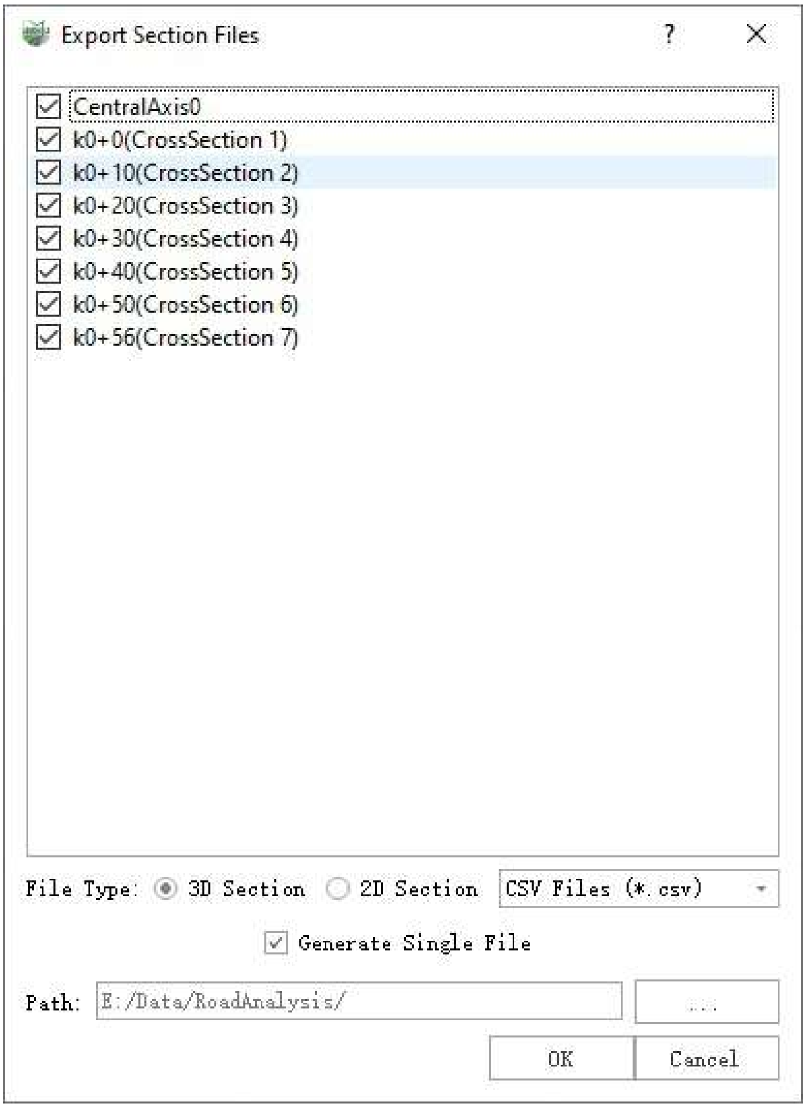


Parameter dialog box

Export section

You can export 2D or 3D cross-section lines into a variety of ways, and you can save them into one or more files. It currently supports “.csv",“.hdm",“.shp",“.dxf" and other formats.

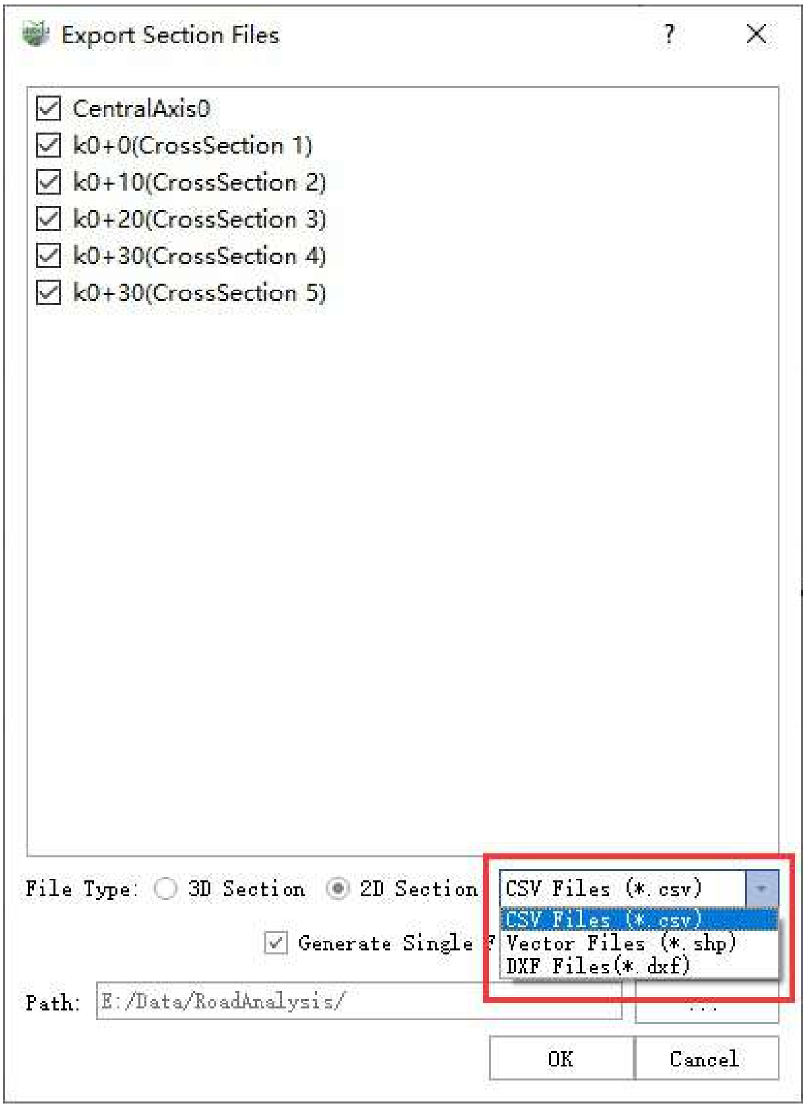
1.Click **Export section**  button, the parameter dialog box pops up.



Parameter dialog box

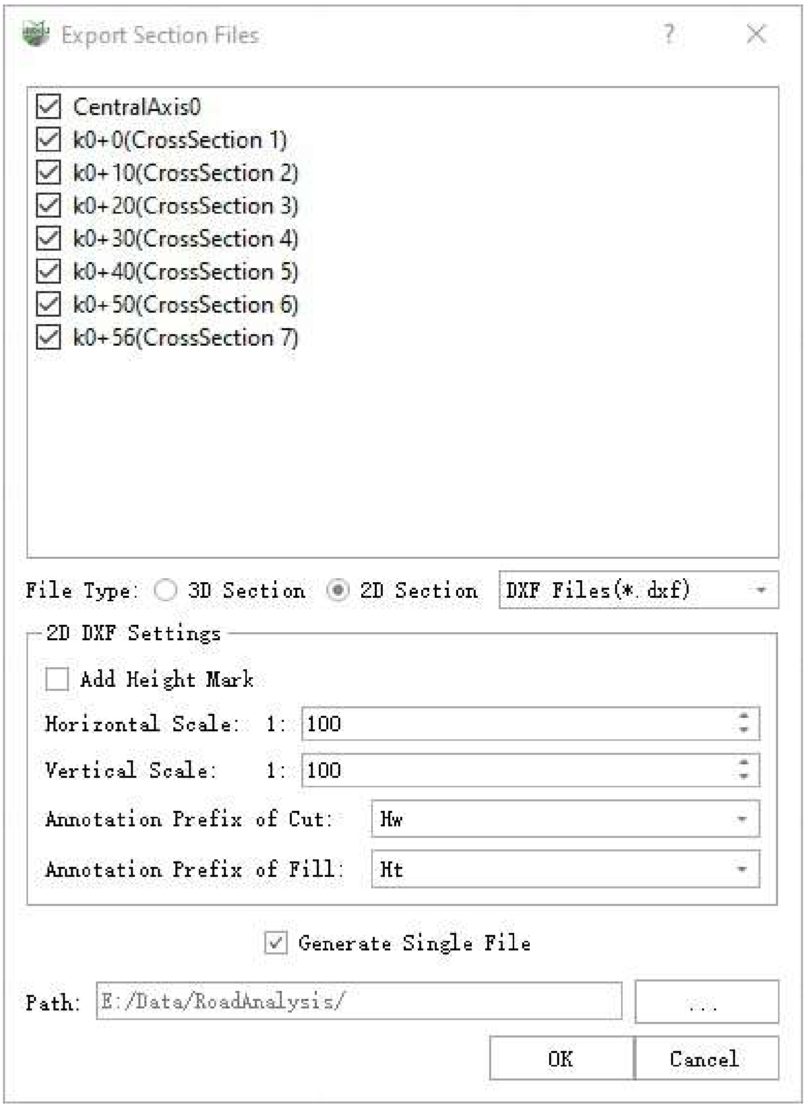
Parameter description

- csv is a common format used internally by the software, and it is read when importing the design section.Cross-section data in csv format
- Files in hdm format, support 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,shp two kinds of vector files, support the import of third-party software, such as AutoCAD to view the cross-section results and related data, here to generate.Take a file in dxf format as an example. The parameter settings are shown in the figure below. Set the output path and click OK.

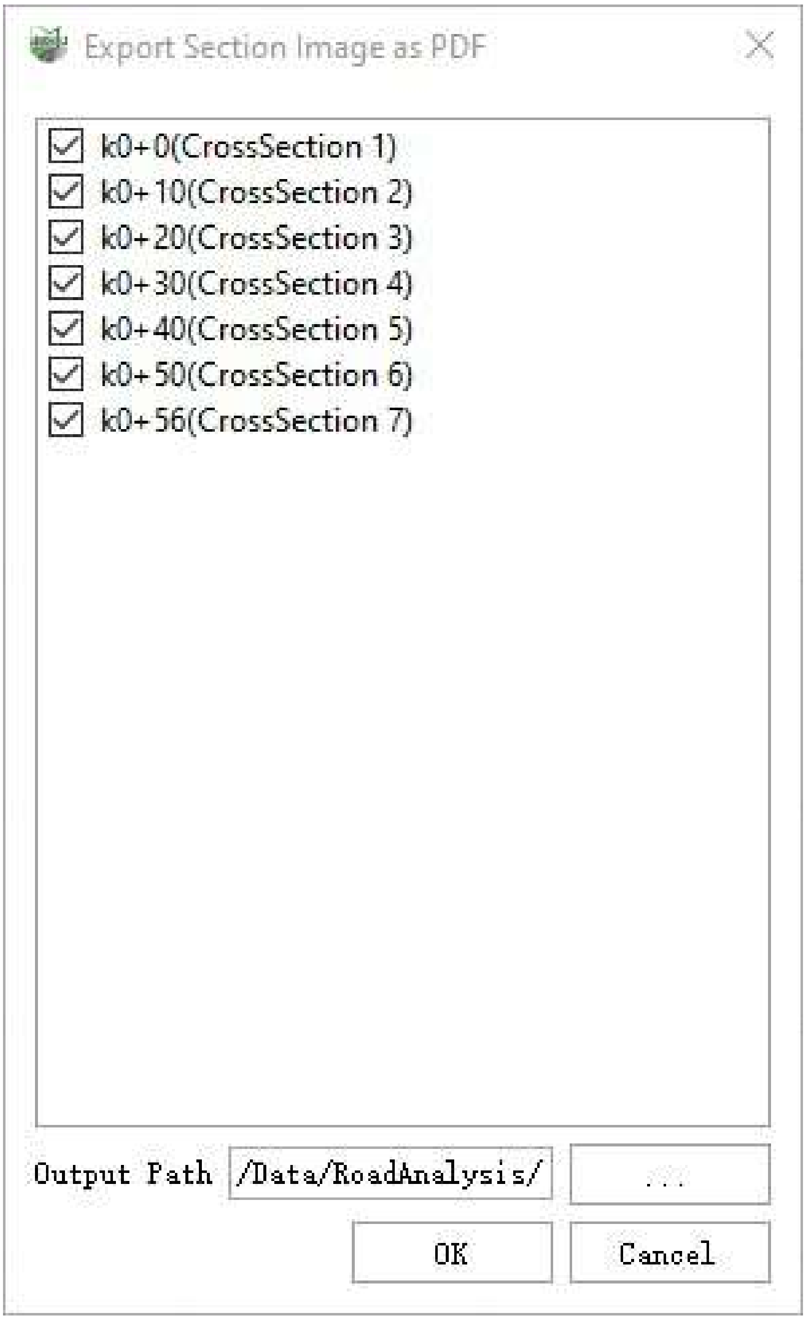


Parameter dialog box

Export pdf

1.Click **Export pdf** button, the parameter dialog box pops up.

2.Select the export path to complete the section information export



Parameter dialog box

Edit section node

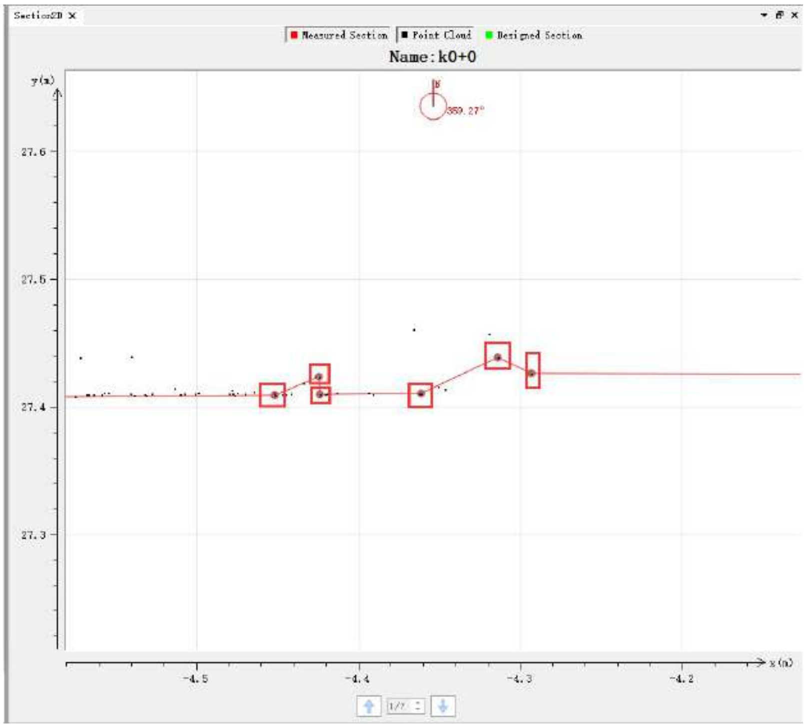
The function of editing cross-section nodes can drag and drop nodes and add nodes.Used to edit incorrect or noisy tunnel boundaries.

- **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.

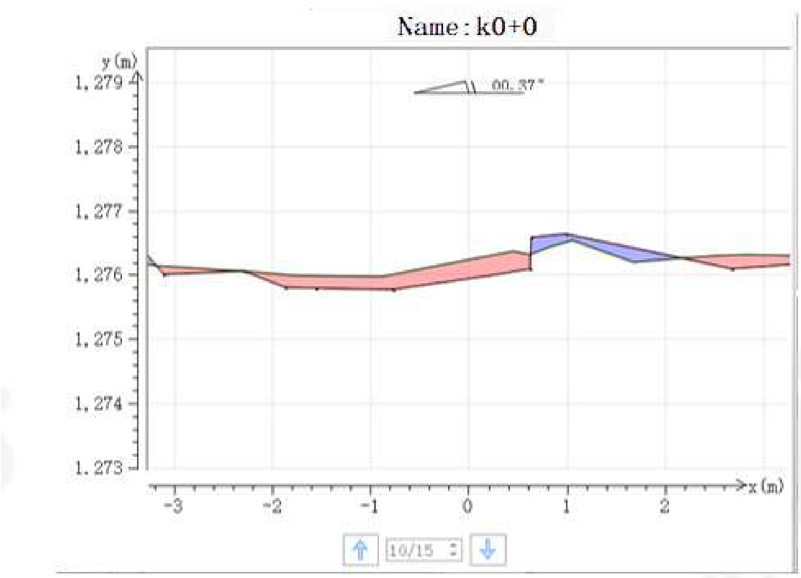


Edit node

Section Compare

Compare the generated road reference section with the road design section and calculate 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



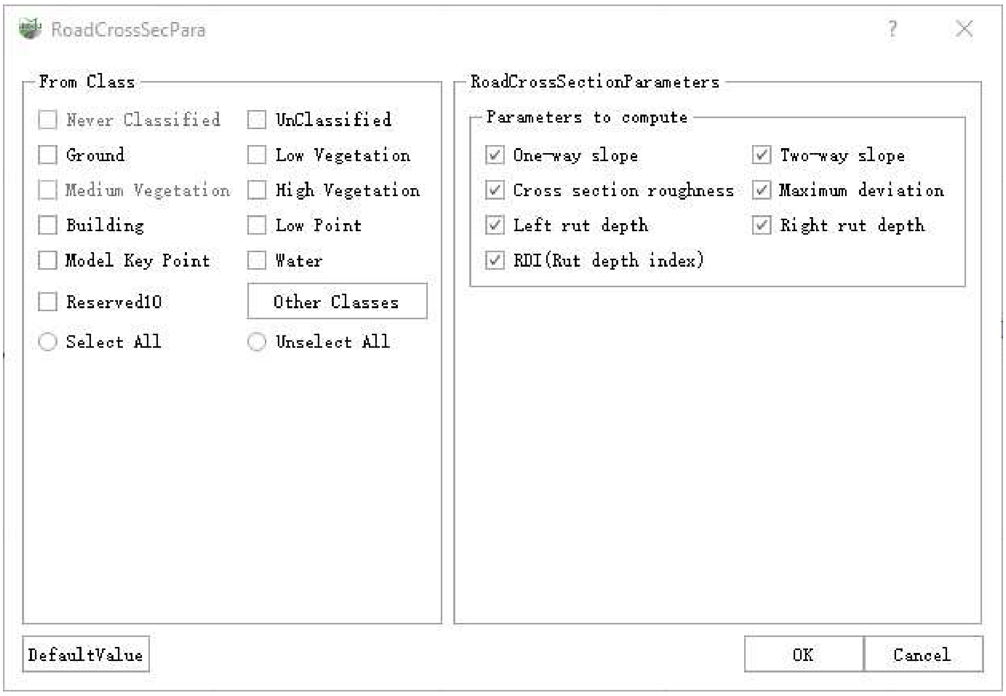
Break-face ratio

Export compare report

Click The button exports the cross-sectional view and cross-sectional information on the current canvas into a report and outputs it to the specified folder.

Calculate road parameters

Calculate the relevant road parameters based on the horizontal reference line.</div>



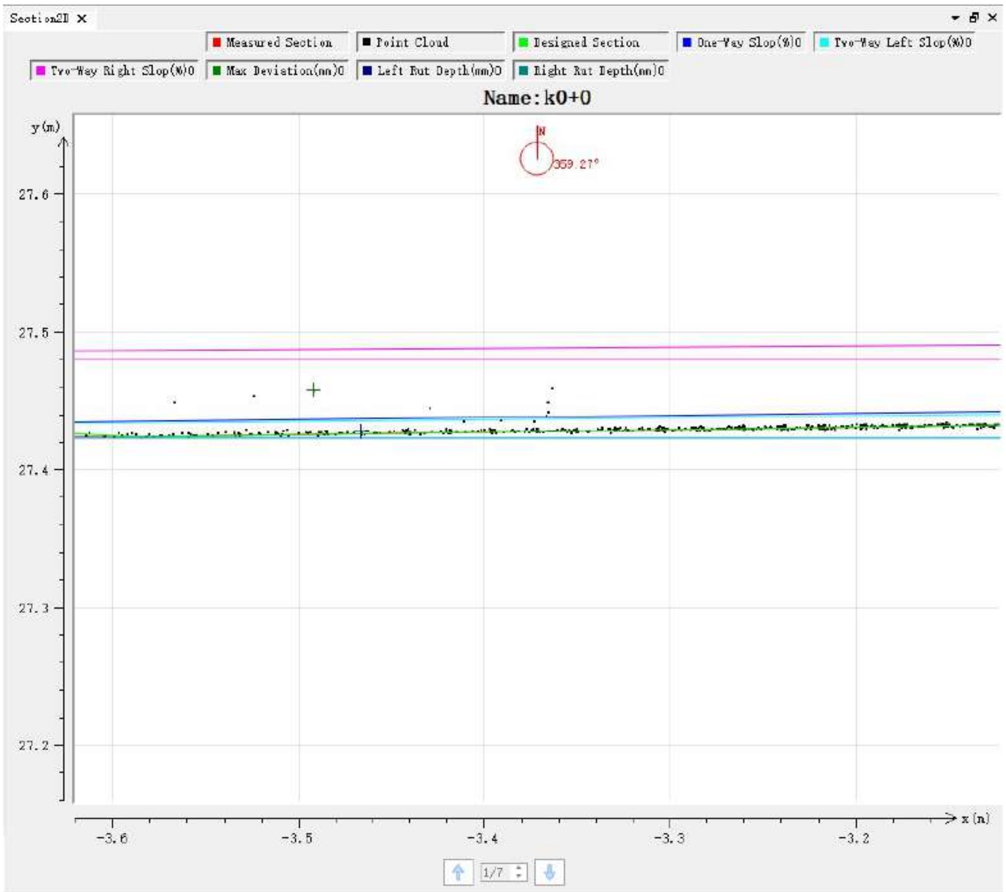
Parameter dialog box

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**:

  > 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: the 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)
  - Road rut depth index: RDI



Section measurement

Measure 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.

Set up

According to user habits, set the color of each information 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

Displays each default view of the currently active view.

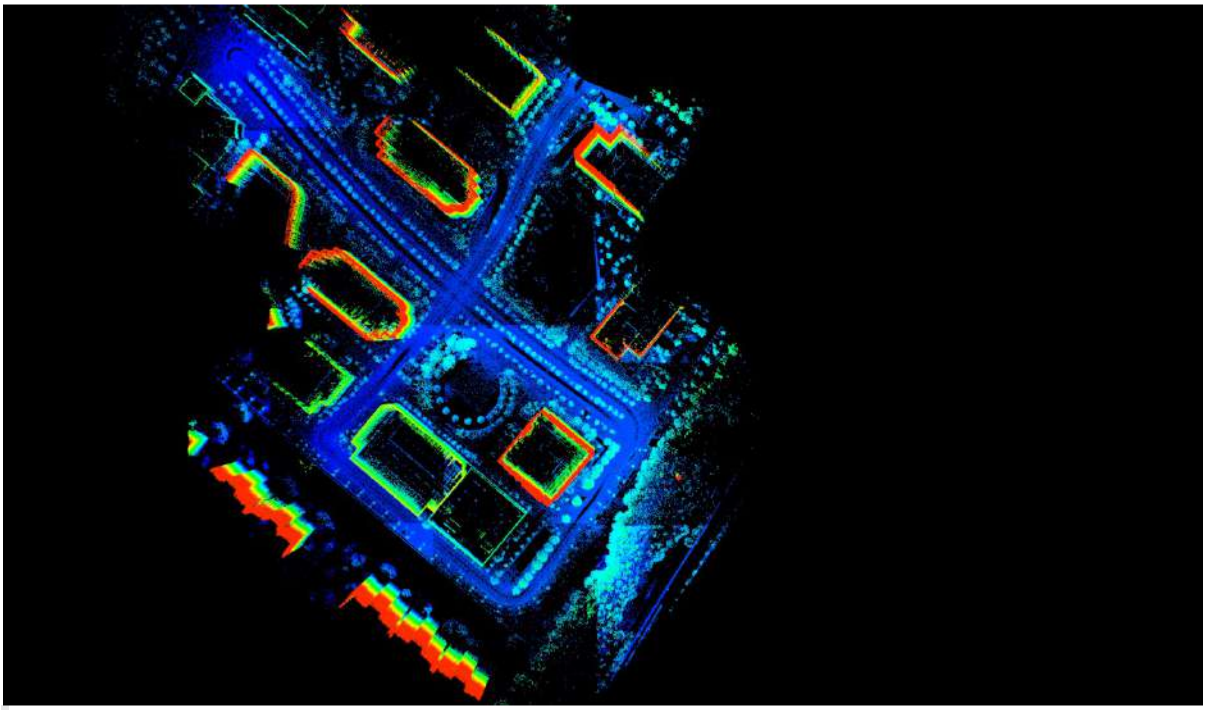
- Top view
- Bottom view
- Left view
- Right view
- Front view
- Rear view
- Isometric front view
- Isometric back view
- Global display
- Set projection mode
- Set point size and type
- Screenshot

Top view

Function description: Set the camera position to view the top view, that is, from +z to-z to view the three-dimensional data, the plane is the x-y plane.

Step

Click **Top view**  button, the current activation form is displayed as shown in the figure：



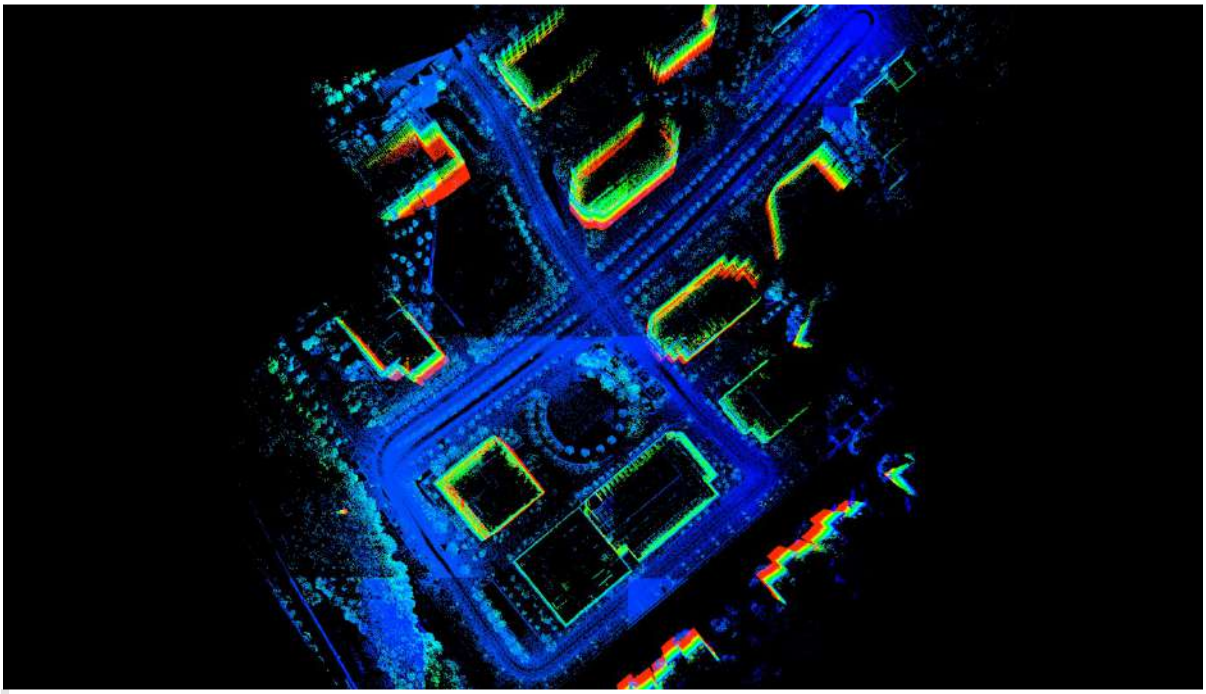
> Note: This function is only for views displayed in 3D; this function does not reset the center position of the viewpoint. If you need to reset to the default viewpoint, please click Global display.

Bottom view

Function description: Set the camera position to view the bottom view, that is, from-z to +z to view the three-dimensional data, the plane is the x-y plane.

Step

Click **bottom view**  button, the current activation form is displayed as shown in the figure：



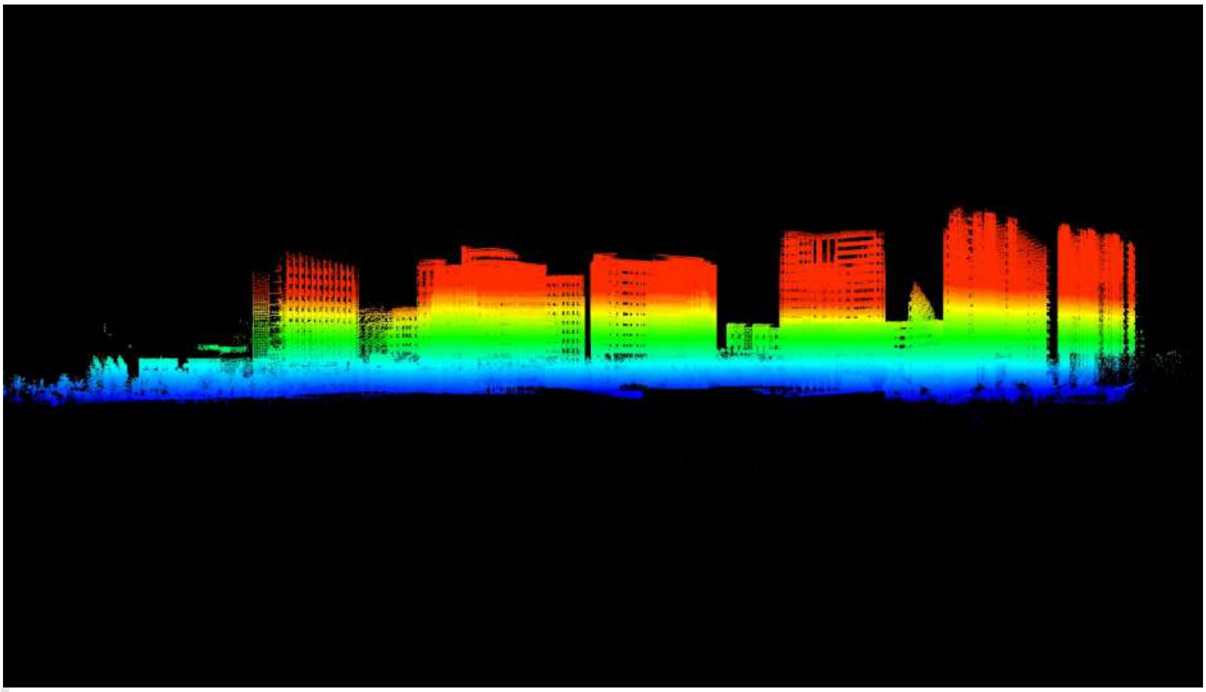
> Note: This function is only for views displayed in 3D; this function does not reset the center position of the viewpoint. If you need to reset to the default viewpoint, please click Global display.

Left view

Function description: Set the camera position to view the left view, that is, from -x to +x to view the three-dimensional data, the plane is the y-z plane.

Step

Click **Left view**  button, the current activation form is displayed as shown in the figure：



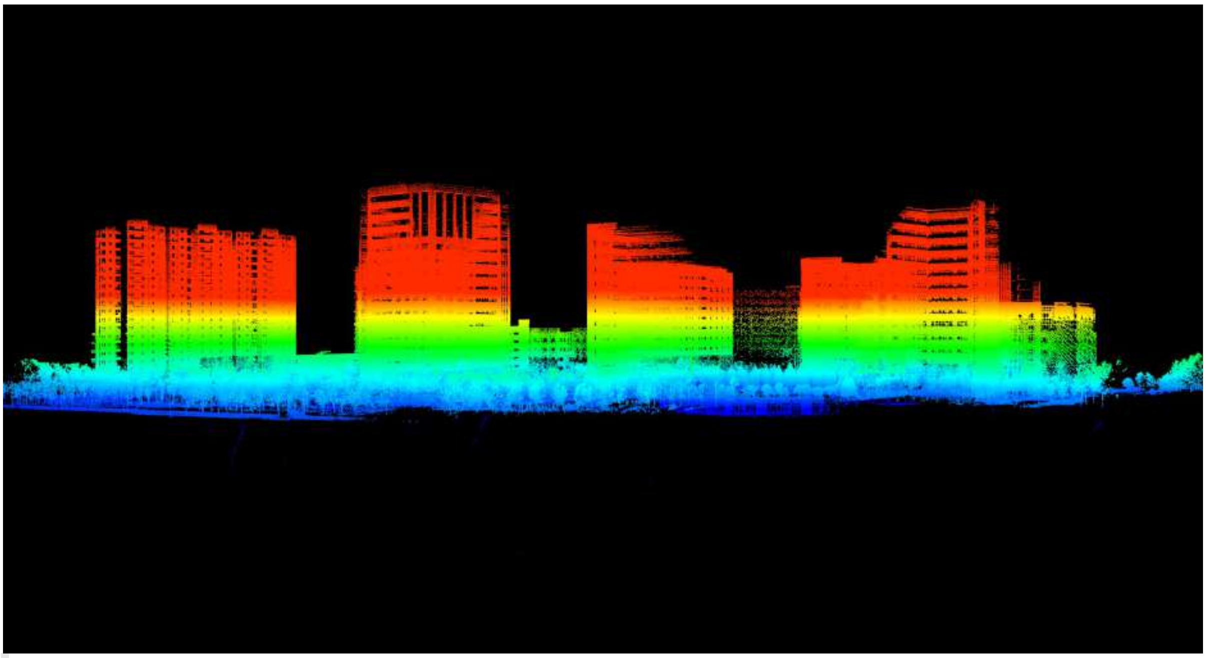
> Note: This function is only for views displayed in 3D; this function does not reset the center position of the viewpoint. If you need to reset to the default viewpoint, please click Global display.

Right view

Function description: Set the camera position to the right view, that is, view the three-dimensional data from +x to-x direction, and the plane is the y-z plane.

Step

Click **Right view** button, the current activation form is displayed as shown in the figure：



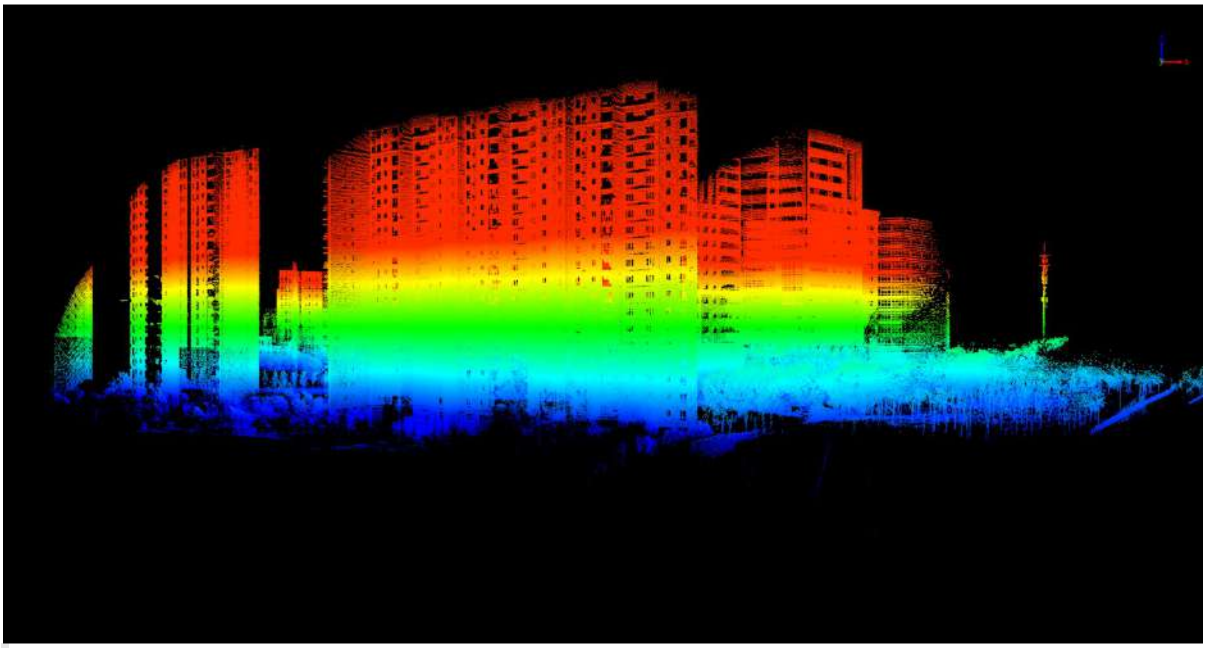
> Note: This function is only for views displayed in 3D; this function does not reset the center position of the viewpoint. If you need to reset to the default viewpoint, please click Global display.

Front view

Function description: Set the camera position to the front view, that is, view the three-dimensional data from the-y to +y direction, and the plane is the x-z plane.

Step

Click **Front view**  button, the current activation form is displayed as shown in the figure：



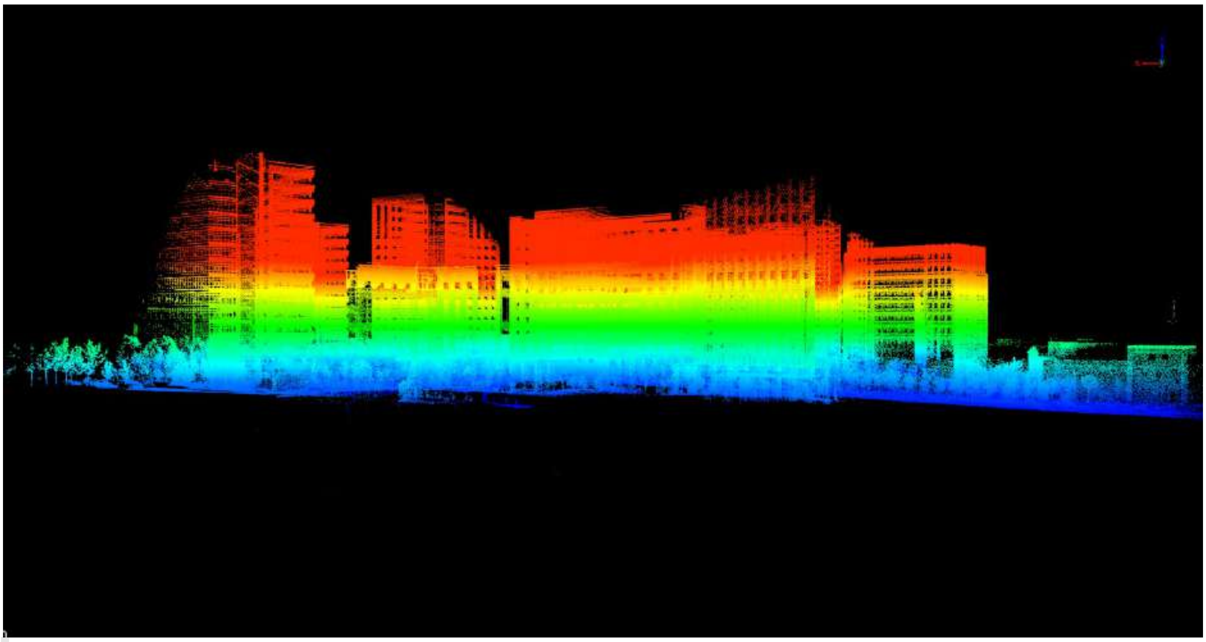
> Note: This function is only for views displayed in 3D; this function does not reset the center position of the viewpoint. If you need to reset to the default viewpoint, please click Global display.

Back view

Function description: Set the camera position to the rear view, that is, view the three-dimensional data from +y to-y direction, and the plane is the x-z plane.

Step

Click **Back view**  button, the current activation form is displayed as shown in the figure：



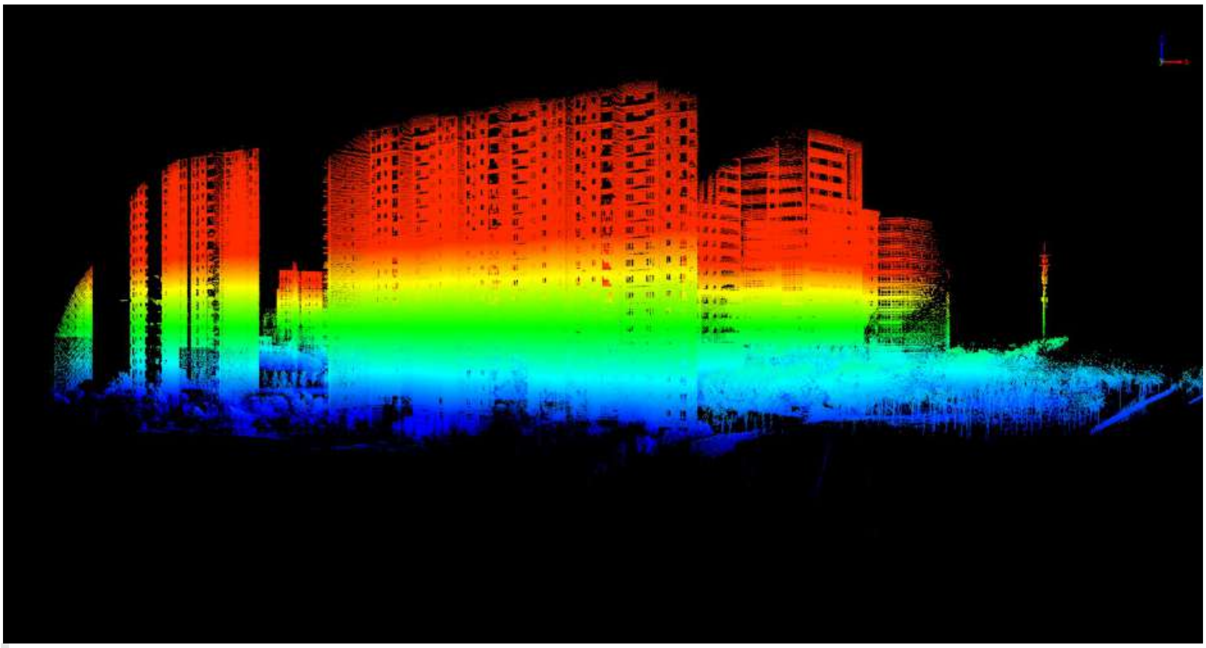
> Note: This function is only for views displayed in 3D; this function does not reset the center position of the viewpoint. If you need to reset to the default viewpoint, please click Global display.

Isometric front view

Function description: Set the camera position to tilt 45° in front of x-Y.

Step

Click **equidistant front view**  button, the current activation form is displayed as shown in the figure：



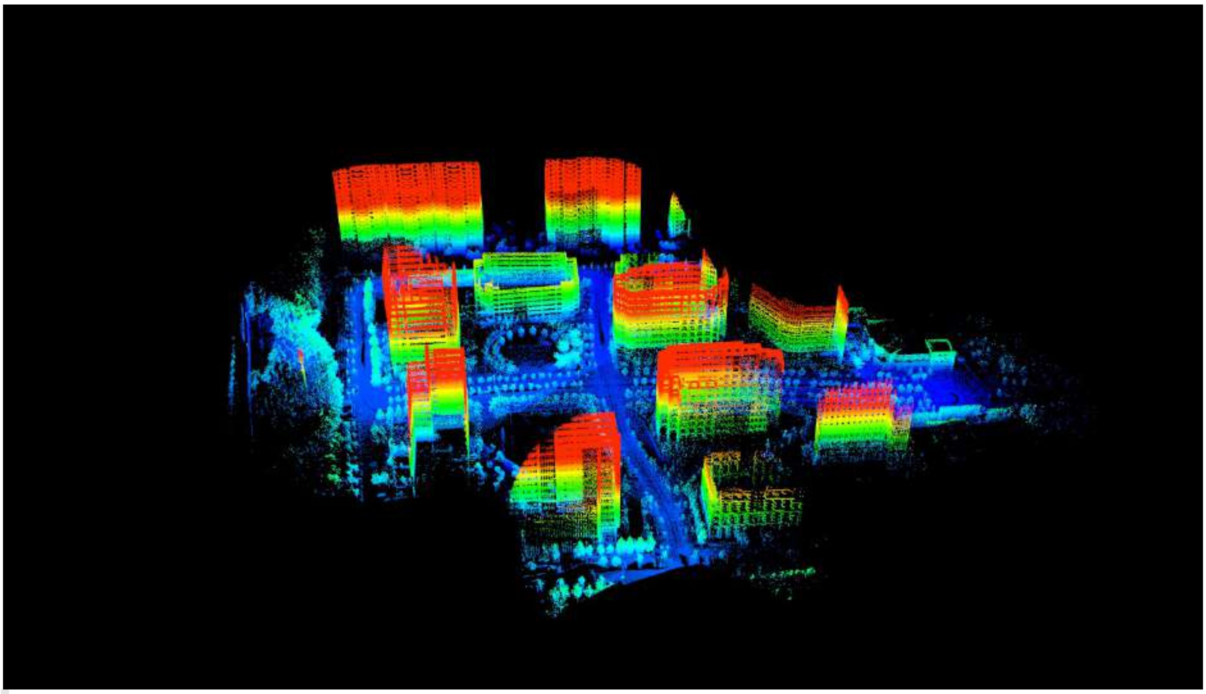
> Note: This function is only for views displayed in 3D; this function does not reset the center position of the viewpoint. If you need to reset to the default viewpoint, please click Global display.

Isometric Back view

Function description: Set the camera position to tilt 45° backward on the x-y surface.

Step

Click **equidistant rear view**  button, the current activation form is displayed as shown in the figure：



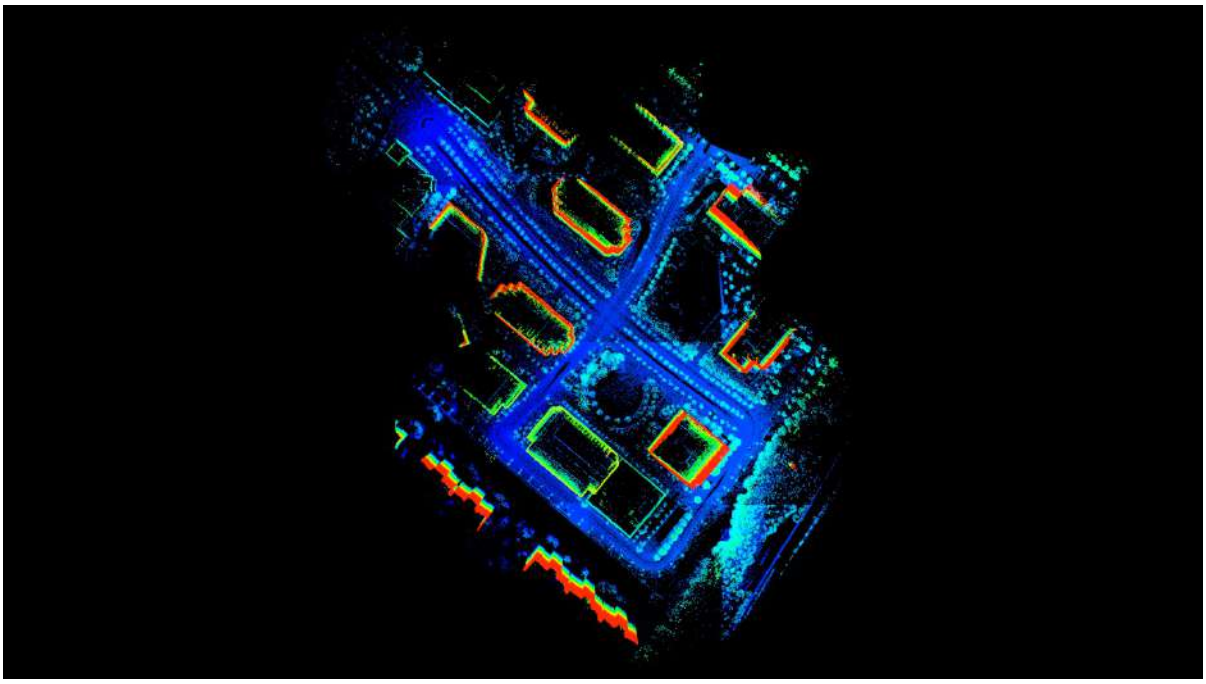
> Note: This function is only for views displayed in 3D; this function does not reset the center position of the viewpoint. If you need to reset to the default viewpoint, please click Global display.

Global display

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.

Step

Click on the toolbar **Global display**  Button, the data in the 3D window will be automatically scaled to cover the entire window, as shown in the figure：



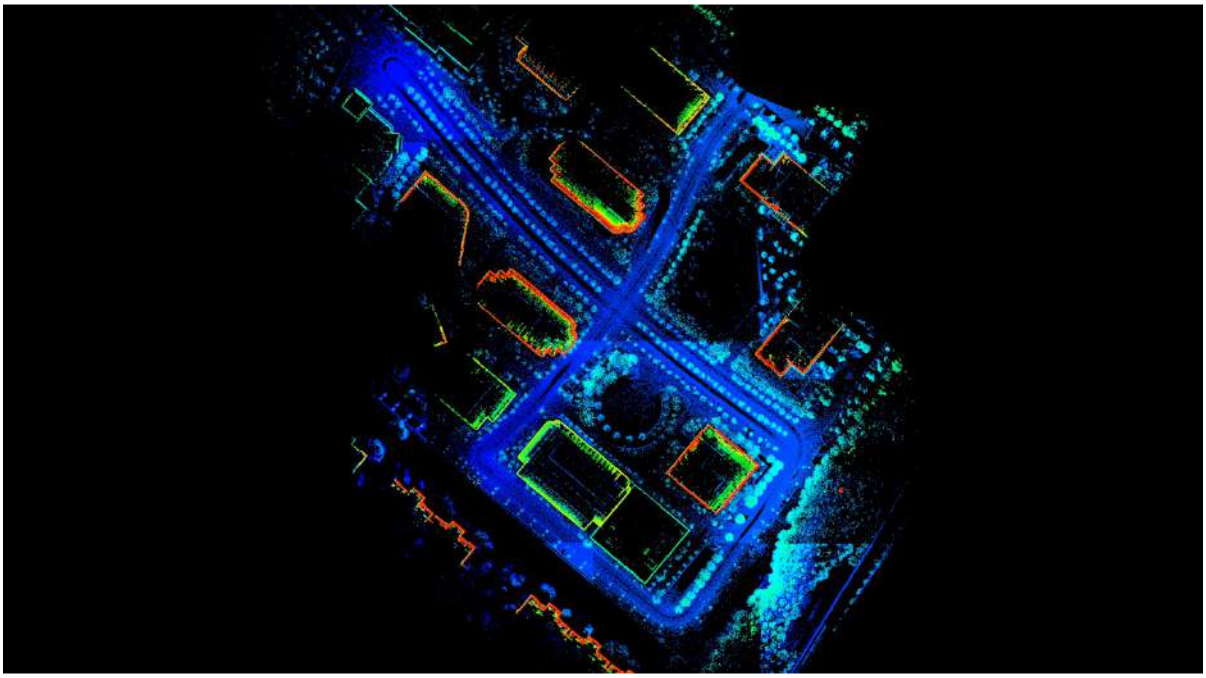
Set projection mode

Function description: Change the projection method of the view, the view supports two projection methods: orthogonal and perspective.

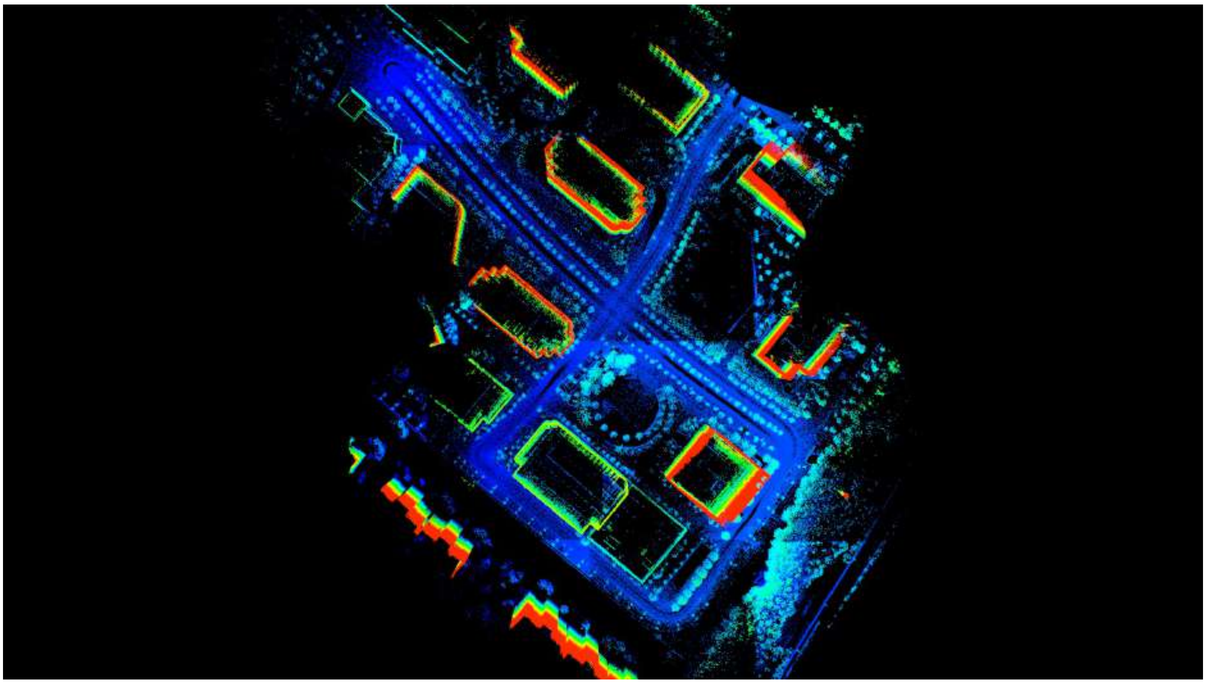
Step

1.Click **To set the projection mode**  button, pop up the option to select the projection method.

2.If you select orthophoto projection, the point cloud window is orthophoto projection.



3.If perspective projection is selected, the point cloud window is perspective projection.

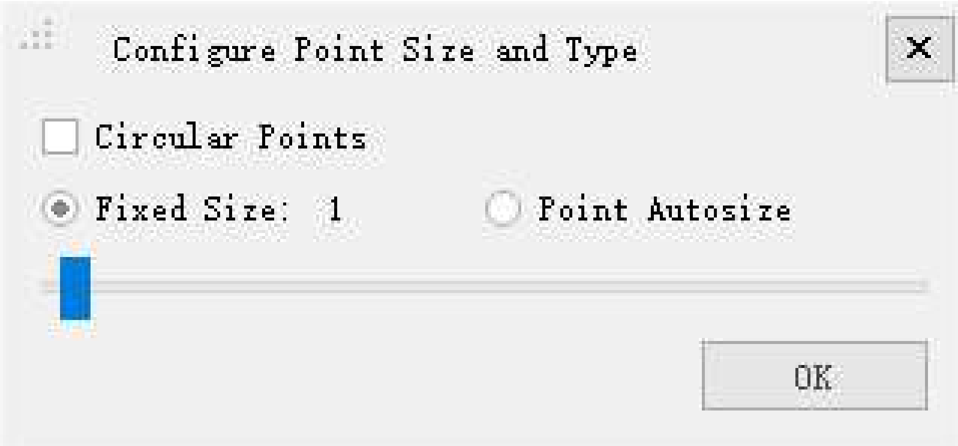


Set point size and type

Function description: Set the size and type of points in the three-dimensional point cloud in the entire software system.

Step

1.Click **To configure the size of the point** button, the interface shown in the figure below pops up：



Parameter setting

- **Circle (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 midpoint 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.
- **Adaptive Size (Auto Size)**: This parameter defines the depth display of the visual vertebral body in the midpoint adaptive form of the software system.

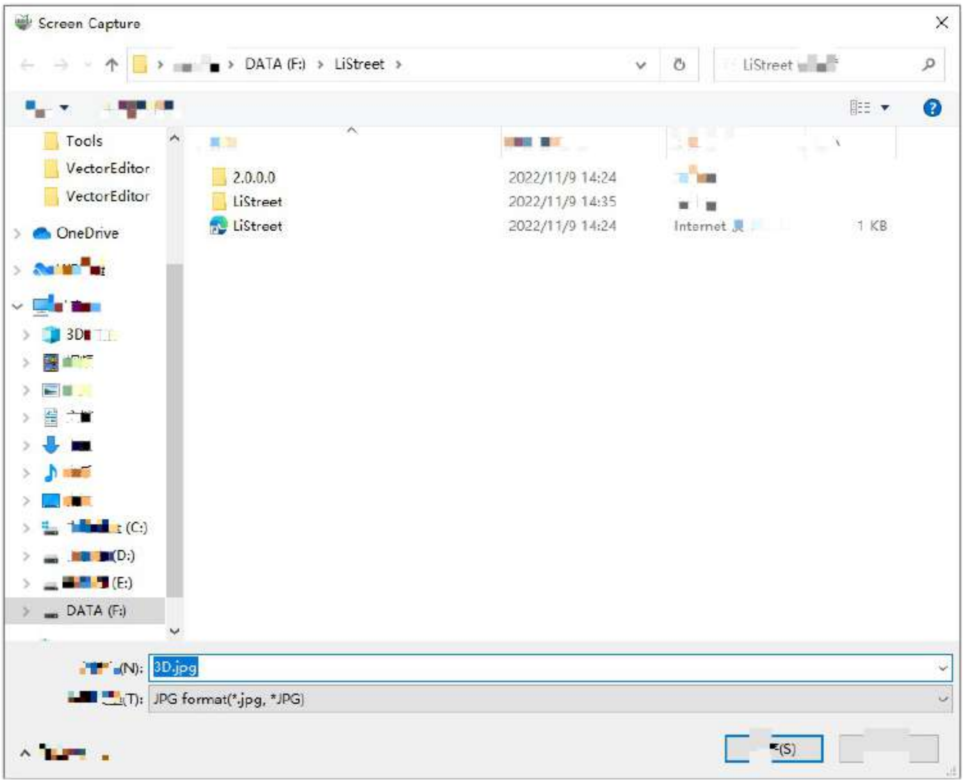
2.Set the size and display type of the point, click OK；

Screenshot

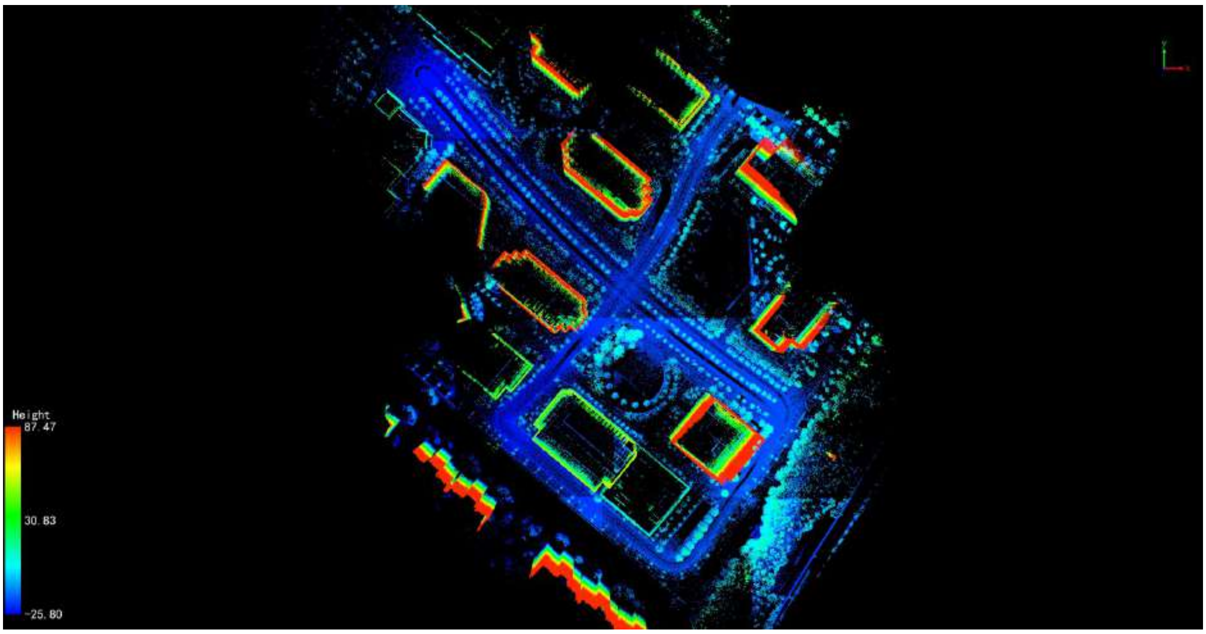
Function description: Directly take a screenshot of the point cloud that the user has customized or set a certain viewing angle in the 3D window.

Step

1.Click **screenshot**  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 of massive point cloud data, and the best 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 elevation
- Display by intensity
- Display by category
- Display by RGB
- Display by number of echoes
- Display by GPS time
- Mixed display
- Display by combination
- Display by selected color
- Display by user data
- Display by point source ID
- Display by tree ID
- EDL display

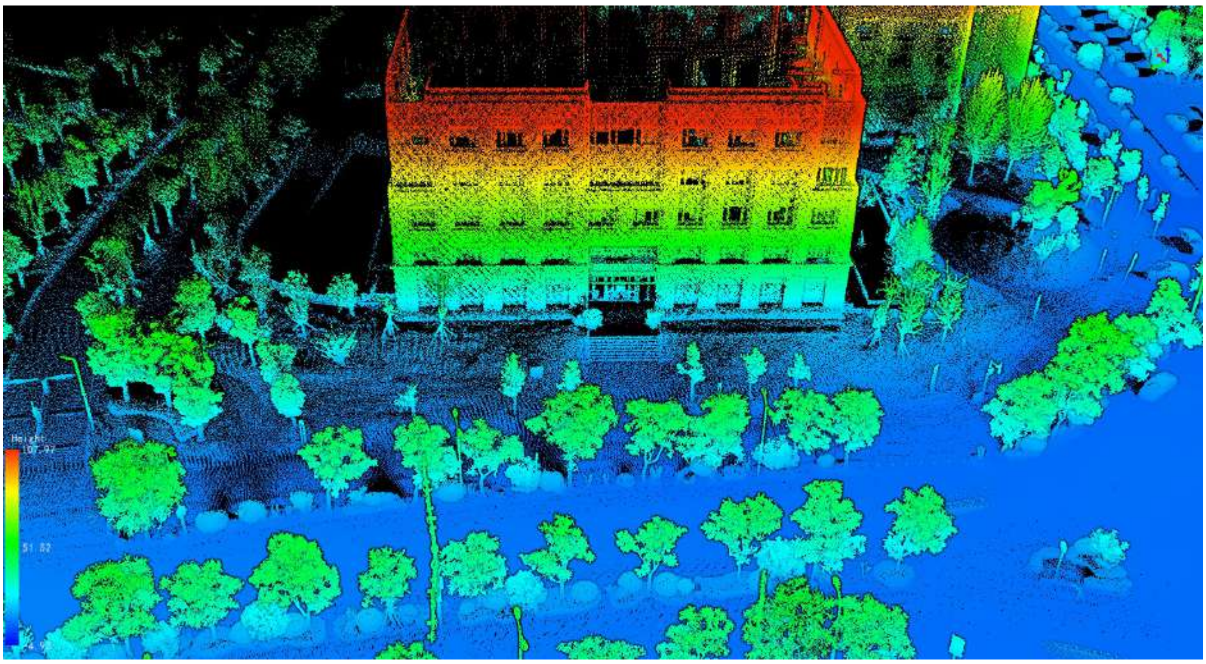
Display by Height

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, the display dialog box by elevation pops 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 better with EDL display.



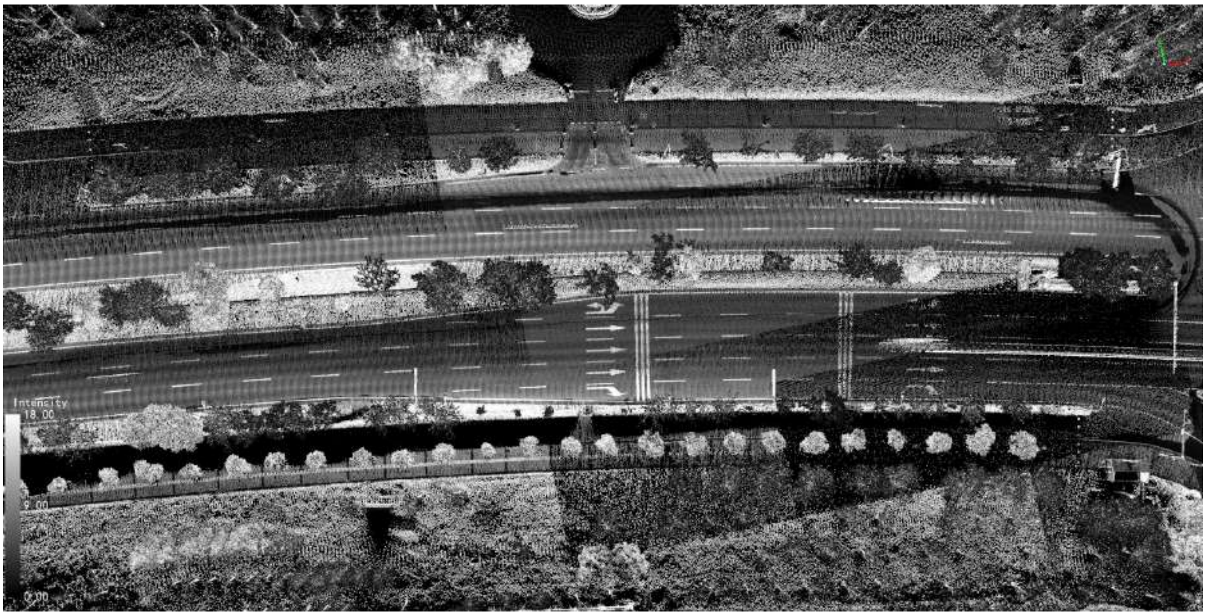
> Note: This feature only works on point cloud data.

Display by intensity

Function description : It 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.



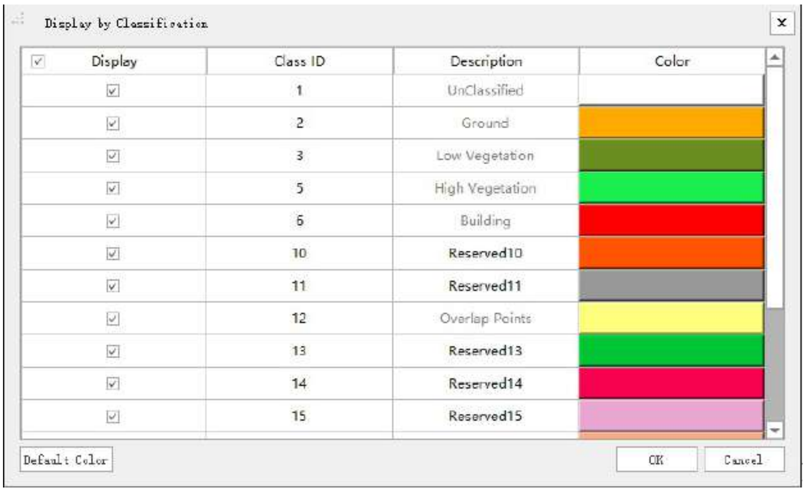
> Note: This feature only works on point cloud data.

Display by category

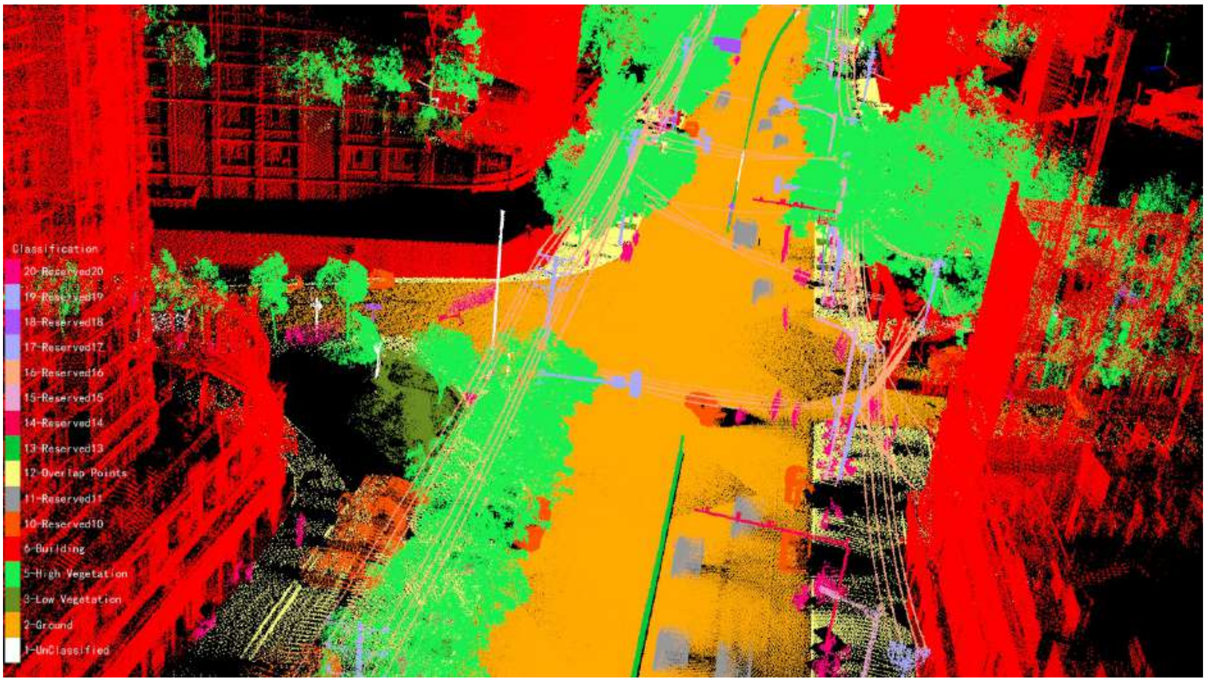
Function description : It can be used for the display of point cloud data, mapping the category attributes of point cloud data to different color values, and more intuitively distinguishing different categories of point cloud data.

Step

1.Click  button, a dialog box to display by category pops 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 better with EDL display.



> Note: This feature only works on point cloud data.

Display by RGB

Function description : It can be used for the display of point cloud data, and the point cloud data is drawn with the RGB color attributes of the point cloud data itself.

Step

1.Click Button, the midpoint cloud data in the scene is displayed according to its own RGB value, the effect is shown in the figure.

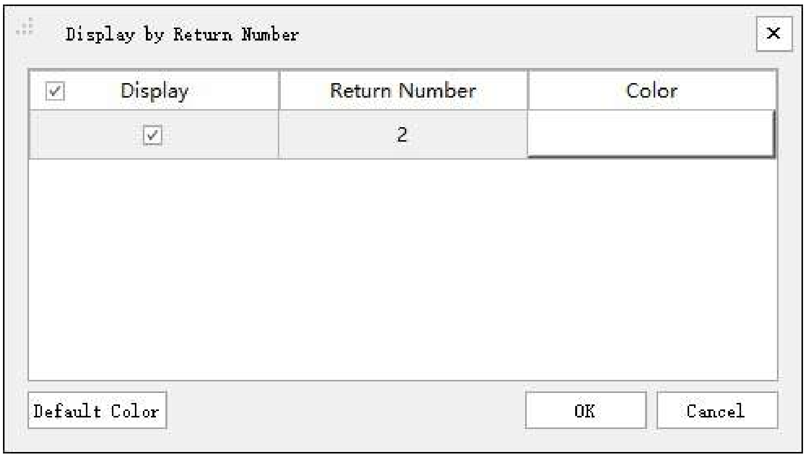
> Note: This function only works on point cloud data that contains RGB attributes.

Press Return to display

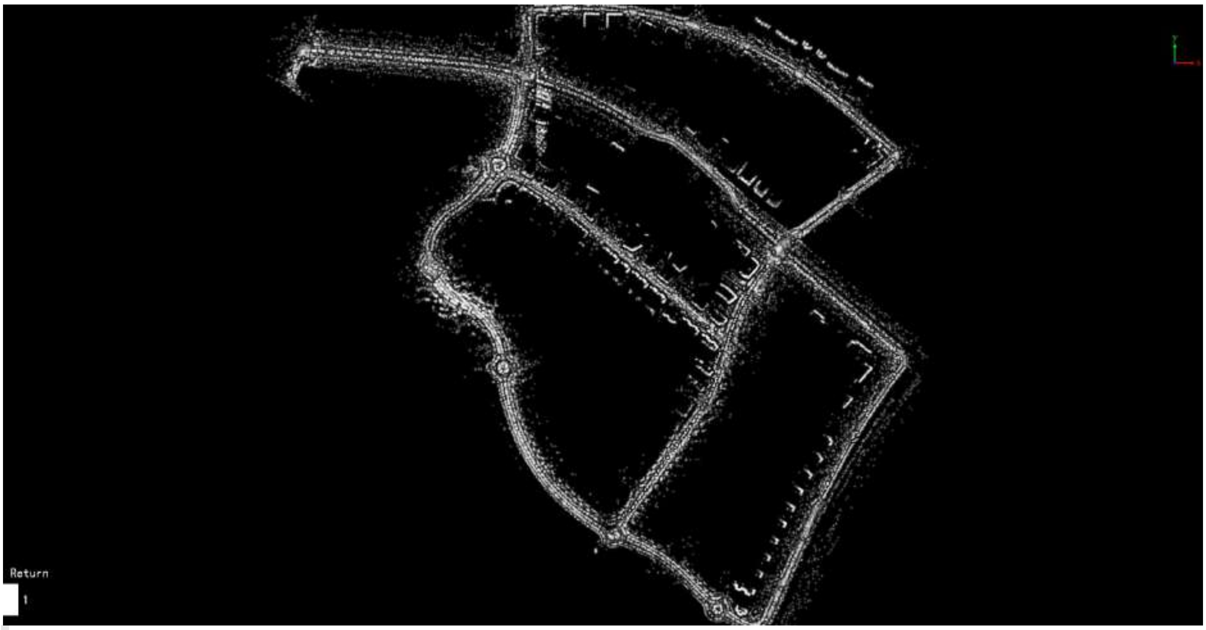
Function description : It can be used for the display of point cloud data, mapping the return number attributes of point cloud data to different color values, and more intuitively distinguishing point cloud data as the first echo.

Steps

1.Click  button, a dialog box will pop up to display the number of times the echo is pressed, as shown in the figure.



2.Select different colors for different echo times, click the OK button, and the color indicator in the lower left corner of the window automatically maps the different echo times 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 number of echoes, which is better with EDL display, the effect is as shown in the figure.



> Note: This feature only works on point cloud data.

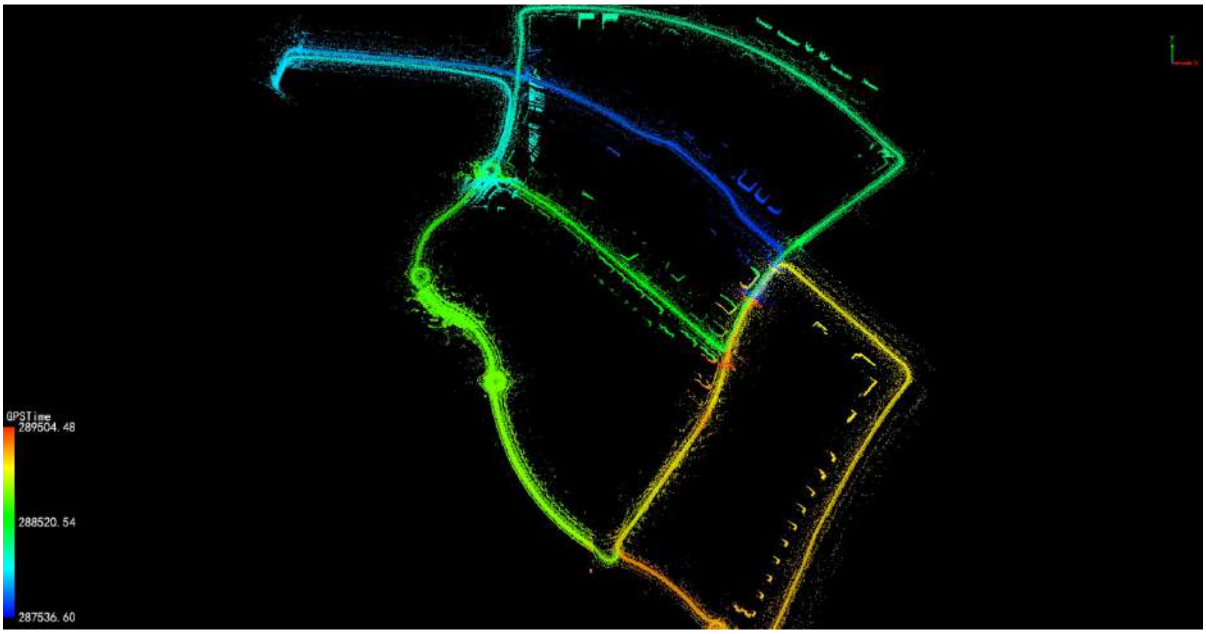
Display by GPS time

Function description : It 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.

Step

1.Click  button, pop up the display dialog box by GPS time.

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 feature only works on point cloud data.

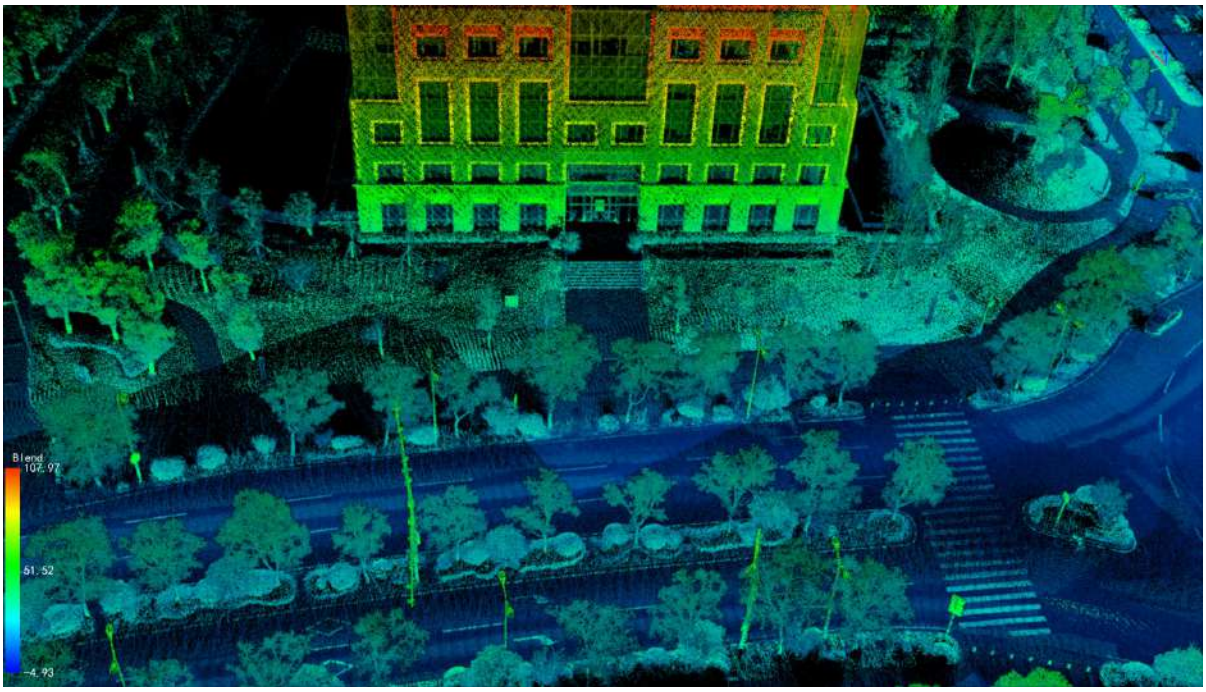
Mixed display

Function description : 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.

Step

1.Click  button, a mixed display dialog box pops 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 better with EDL display.



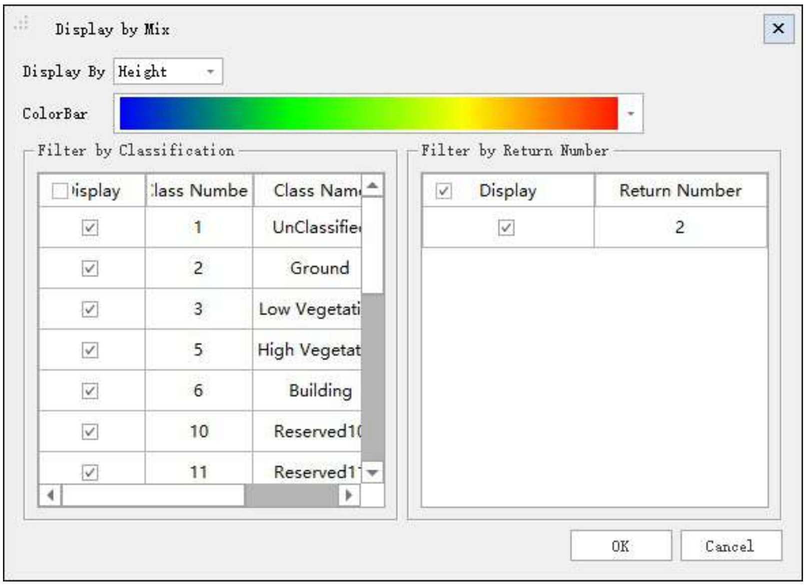
> Note: This function only works on point cloud data, and the mixed display effect is better after PCV processing of point cloud data.

Display by combination

Function description: It 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.

Step

1.Click  button, a dialog box will pop up to display by combination, 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 category**: Lists the category values used to select the filtered point cloud data.
- **Filter by number of echoes**: Lists the values of the number of echoes used to select the filtered point cloud data.

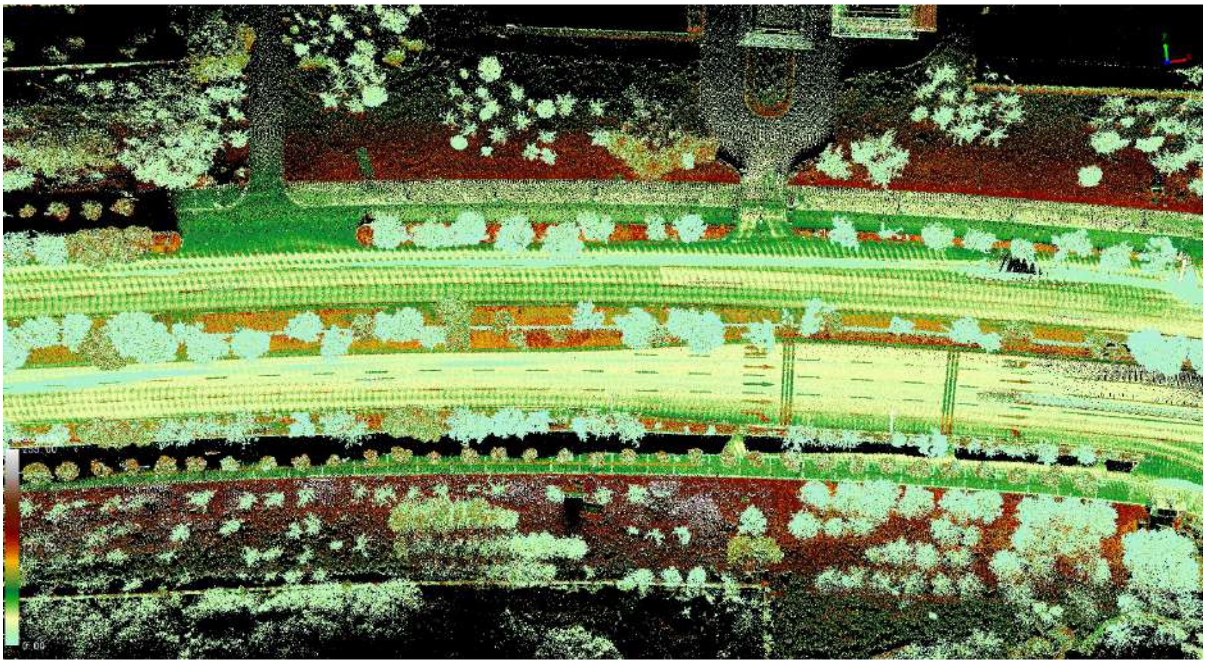
2.Select the attributes to display.

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 better 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:



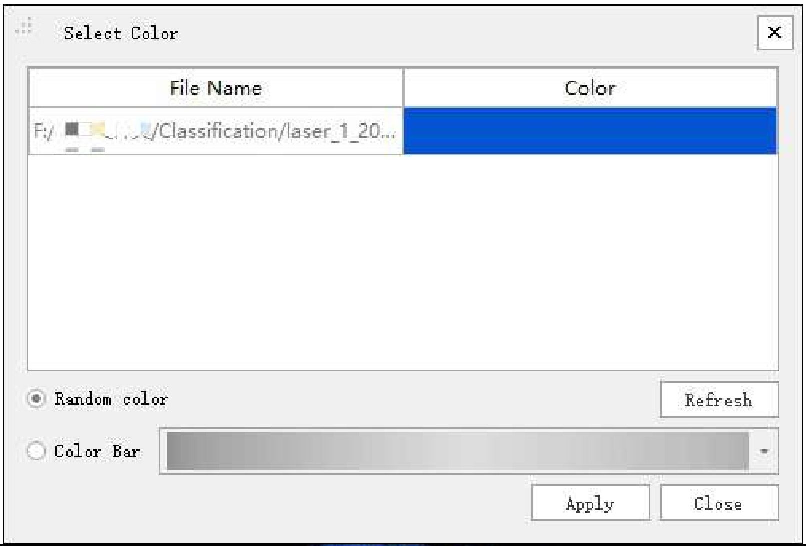
> Note: This feature only works on point cloud data.

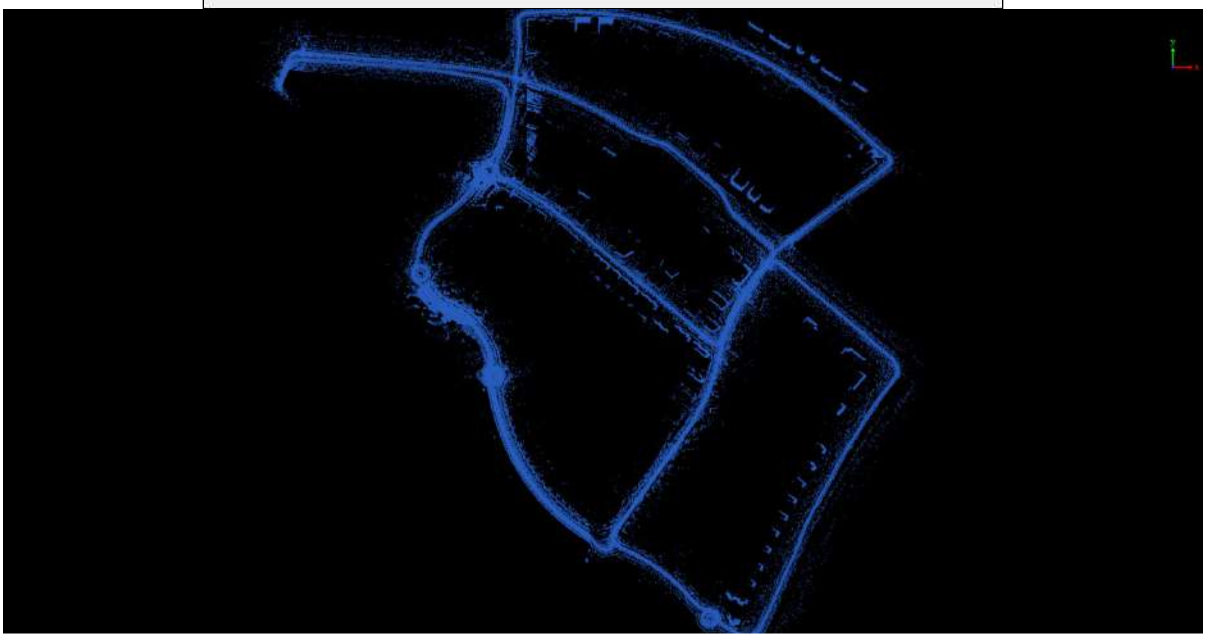
Display by selected color

Function description : Each point cloud data is displayed in the specified color.

Step

1.Click  button, click the color button corresponding to each point cloud to display the point cloud as the specified color.



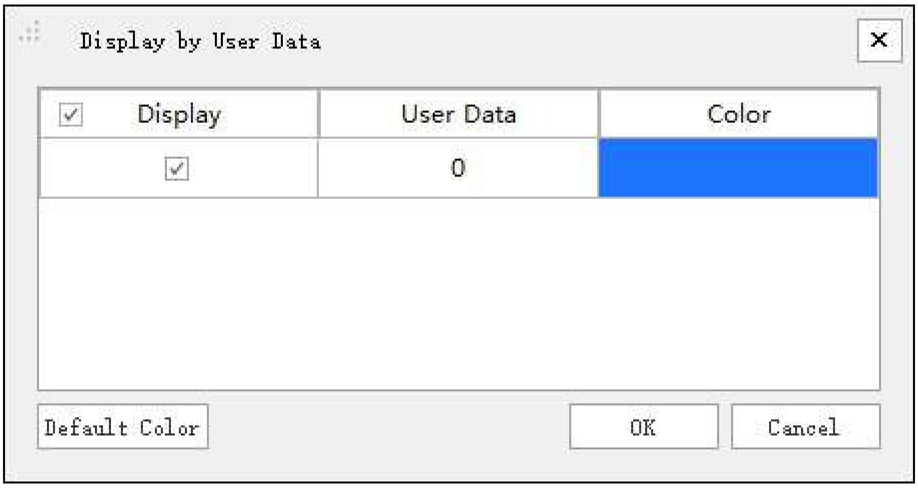


Display by user data

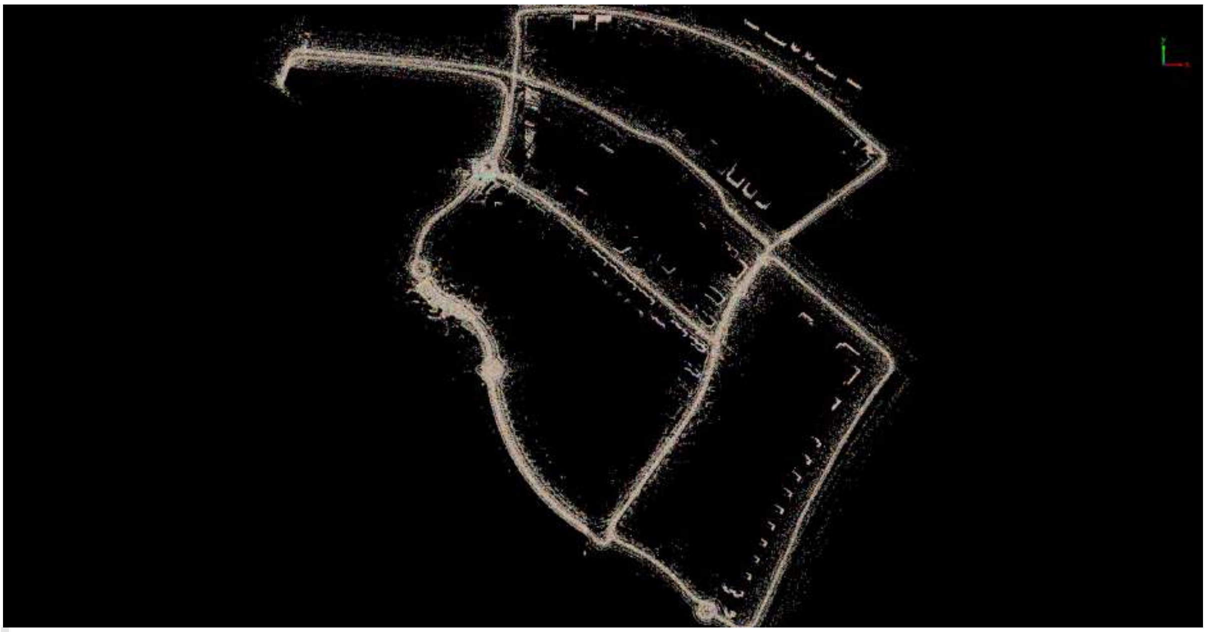
Function description : It can be used for the display of point cloud data, mapping the user data attributes of point cloud data to different color values, and more intuitively distinguishing the point cloud data of different user data.

Step

1.Click  button, pop up the display dialog box by user data, 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 better with EDL, and the effect is shown in the figure.



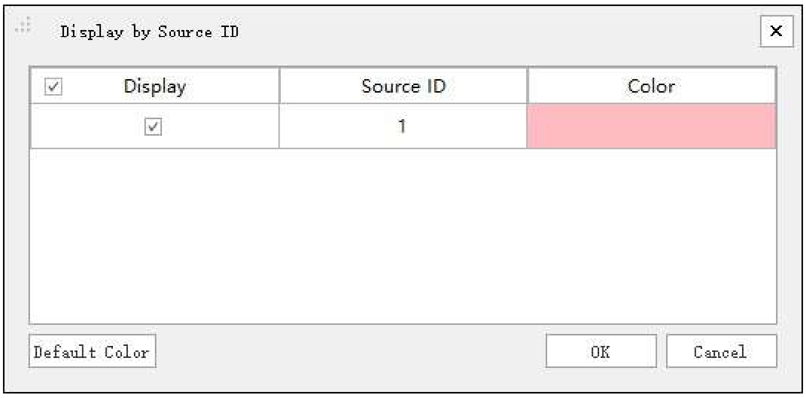
> Note: This feature only works on point cloud data.

Display by point source ID

Function description : It 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 with different point source IDs.

Steps

1.Click  button, pop up the display dialog box by point source ID, 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 better with EDL display, and the effect is as shown in the figure.

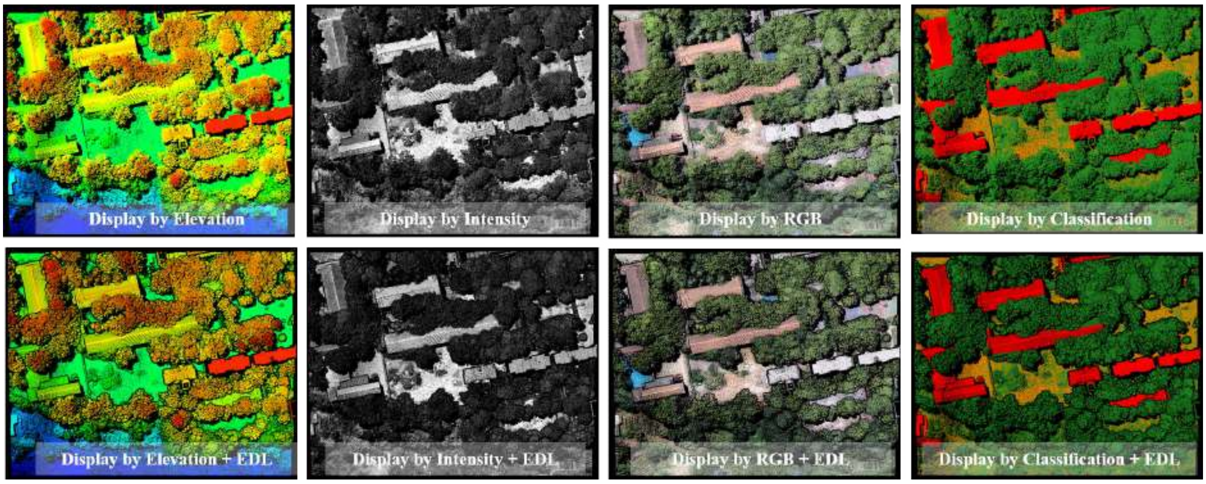
> Note: This feature only works on point cloud data.

EDL display

Function description : It 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 information of point cloud objects.

Step

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.



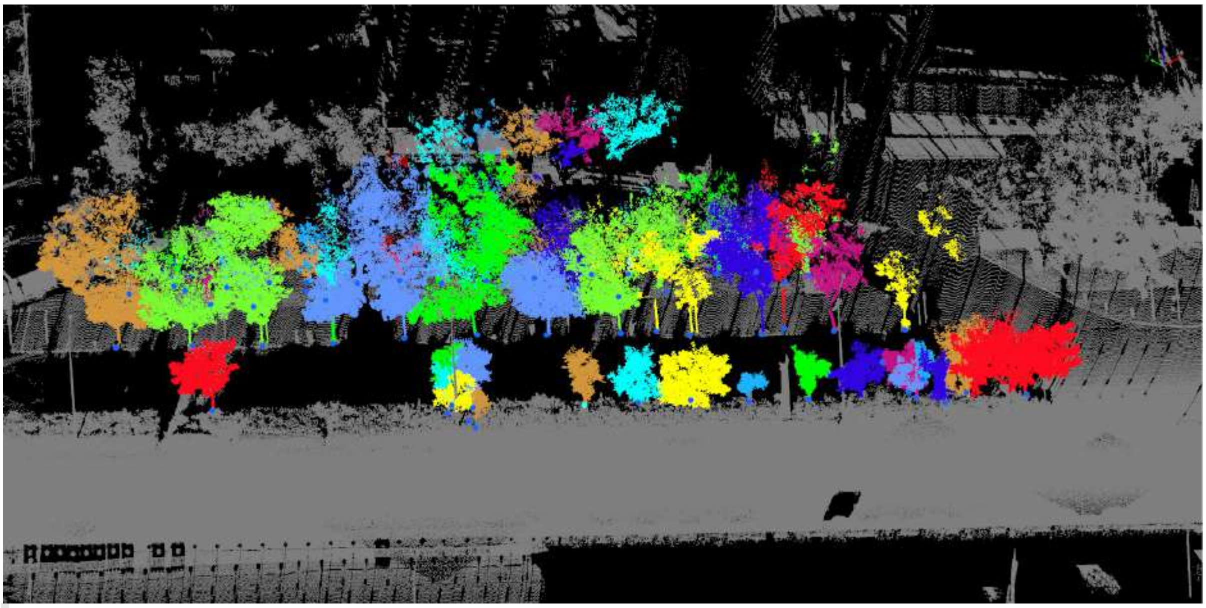
> Note: This feature only works on point cloud data.

Display by tree ID

Function description : It 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 with different tree IDs.

Step

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 feature only works on point cloud data.

Project Management

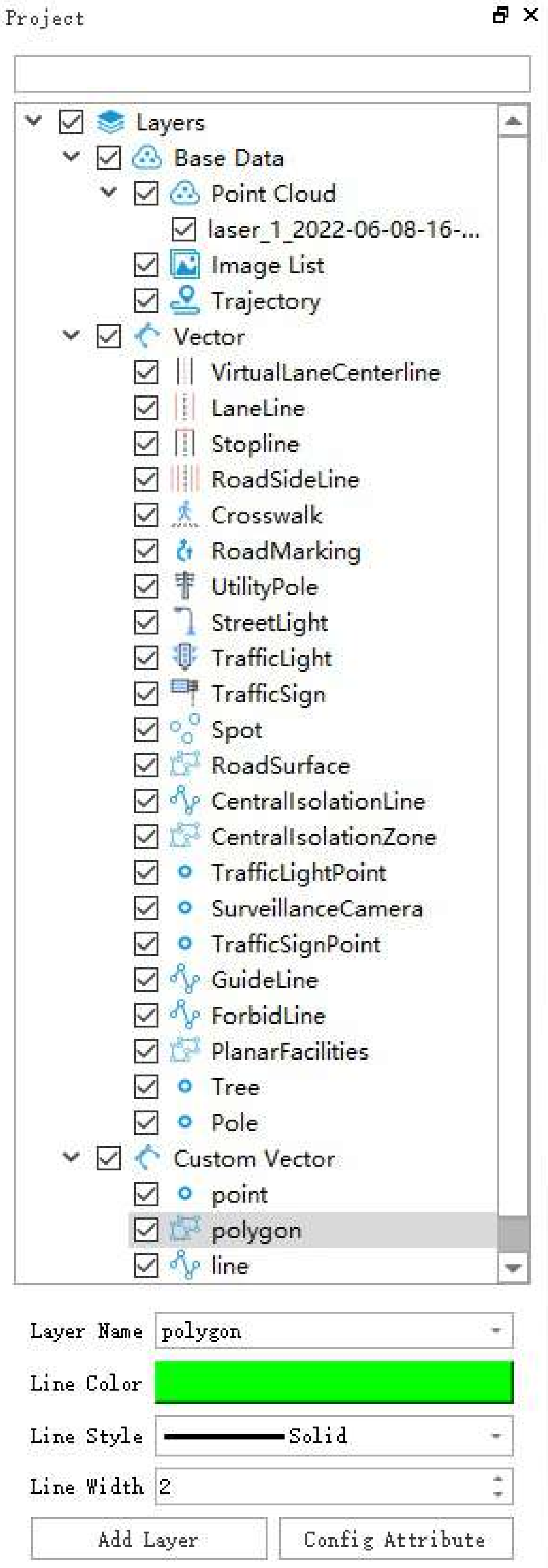
The project management includes five parts: layer management, layer setting, window management, display mode and element attribute.

Layer management

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 the checkbox of the tree node, you can control the display and hiding of 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 and vectors) varies.

The layer management window is open by default. If the window is closed, you can click on it in the shortcut toolbar. 

The button displays a list of layers, as shown in the figure below：

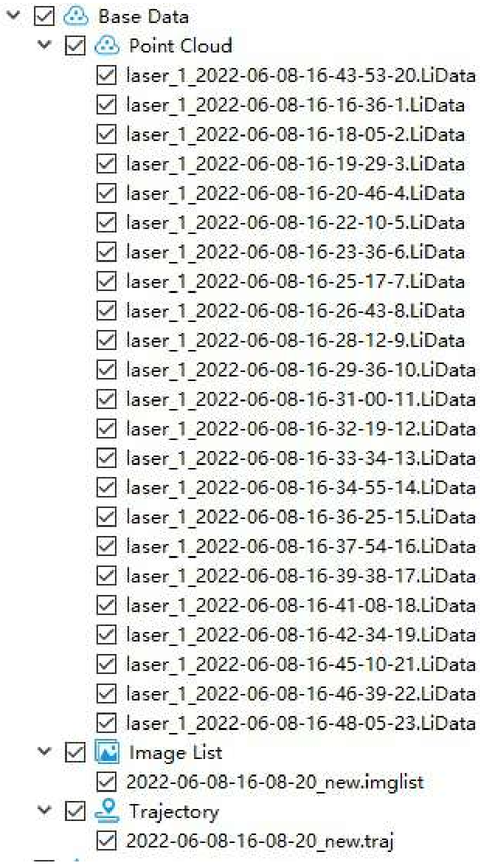


According to the data source, the layer can be divided into two parts: the basic data layer and the vector data layer. The vector data layer includes software built-in layers, custom vector layers, and table data layers：

- Basic data
- Vector
- Custom vector
- Tables

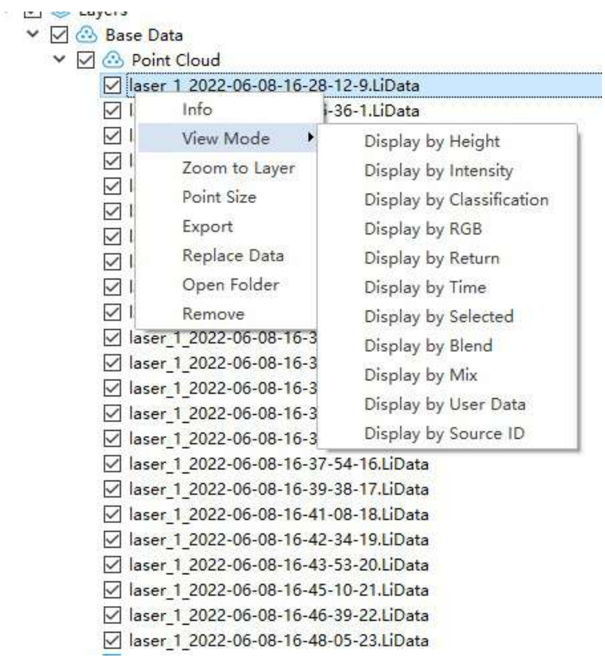
Basic data layer

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

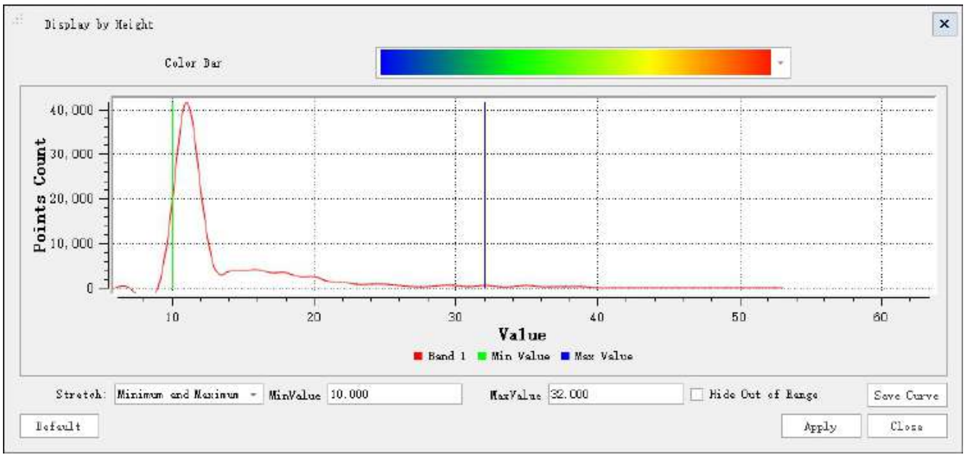
The right-click menu of point cloud contains the content shown in the figure below：



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.**Display**：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：

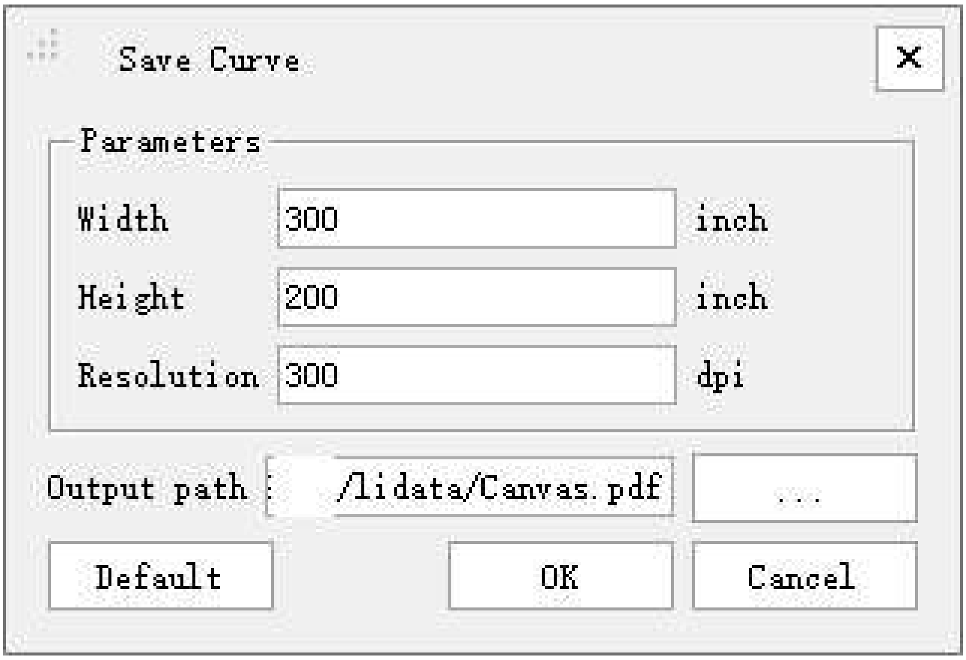
- **Color Bar**：A color bar used to map the elevation properties of a point cloud.
- **Stretch**：Set the stretching method of the histogram.
- **Minimum maximum value (default)**：Linear stretching is applied based on the minimum and maximum pixel values, with the minimum and maximum pixel values as the 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 beyond the 2nd standard deviation 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：

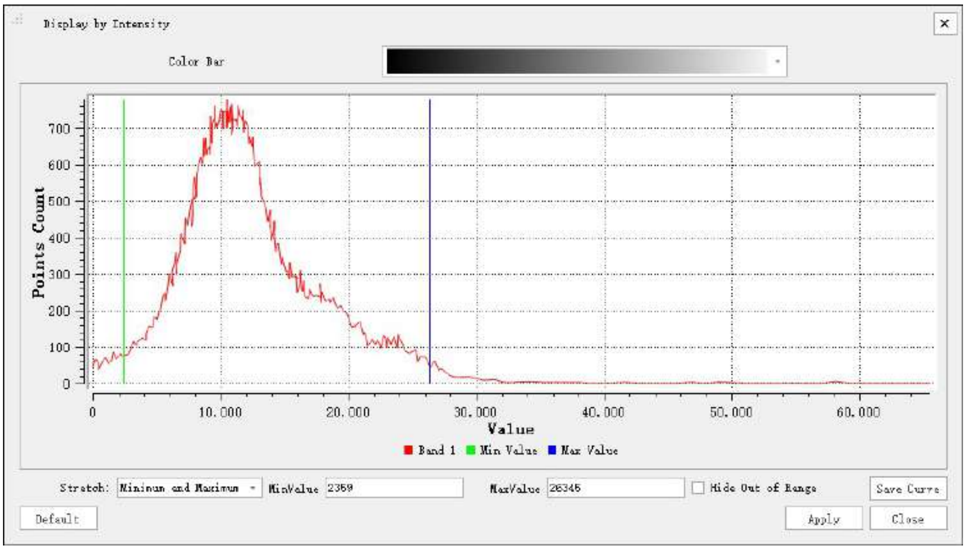
- **Width**：Save the pixel width of the curve.
- **Height**：Save the pixel height of the curve.
- **Resolution**：Save the resolution of the curve.
- **Output Path**：Save the output path of the curve.

Select the width, height, and resolution of the exported curve, select the 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.



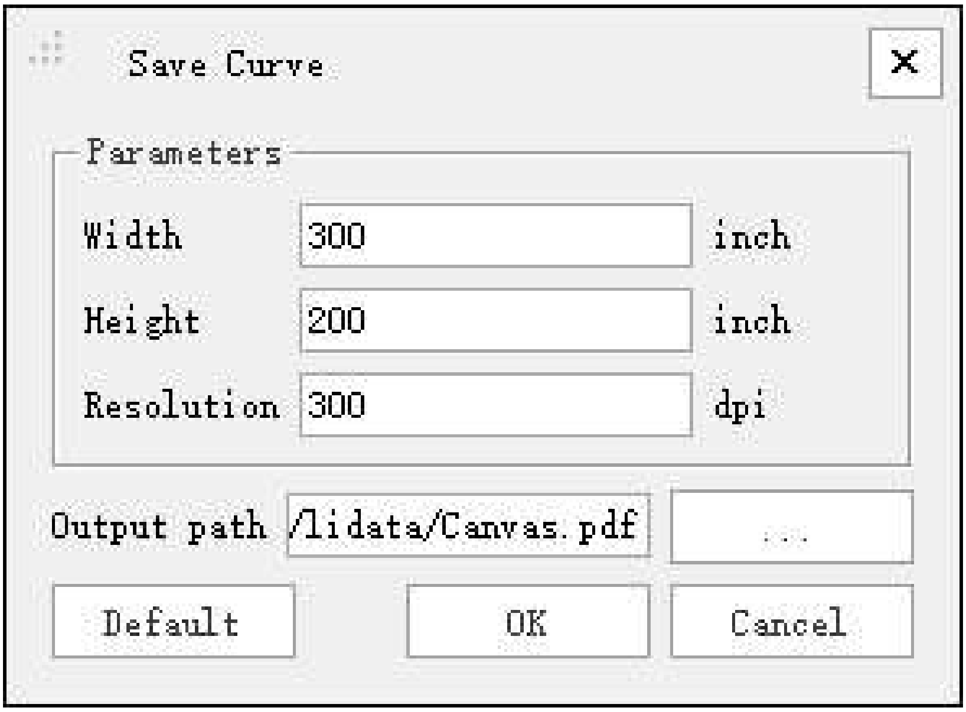
Parameter setting

- **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.

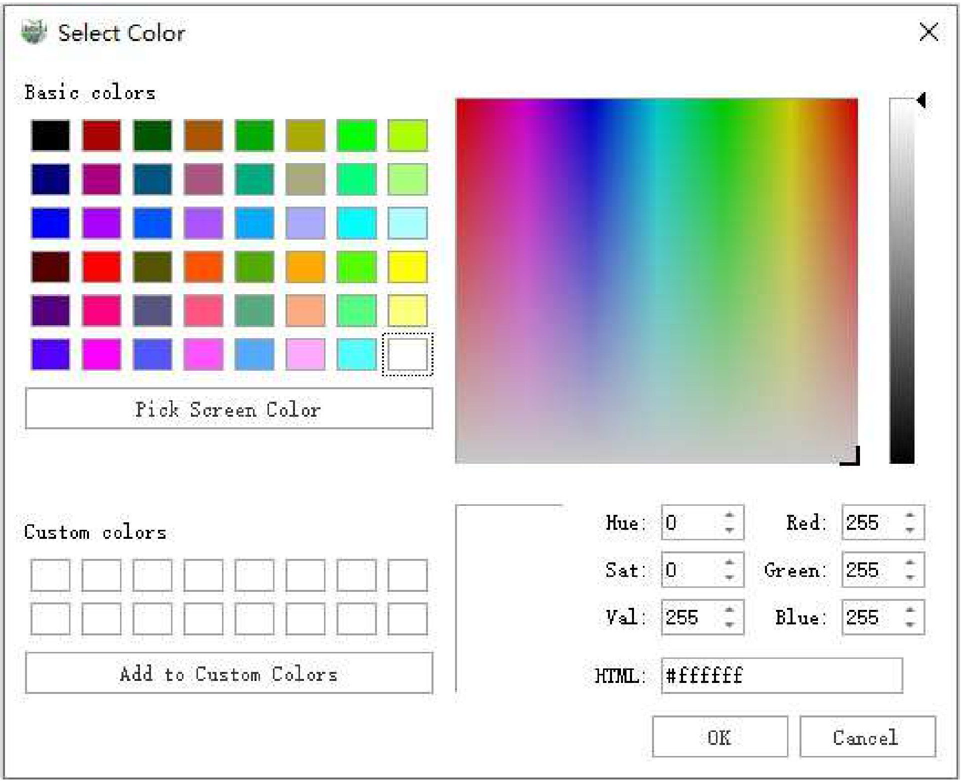
- **Save Curve**：

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 pop up the "Save Curve" dialog box. 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**: 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.

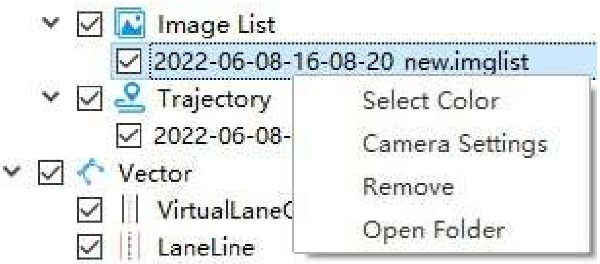
5.**Export**: Export a single point cloud data as LAS (*.las, *.laz) format.

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

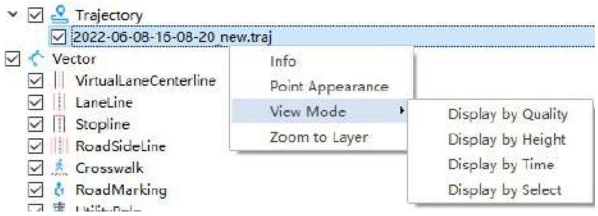


1.**Select Color**: Set the display color of the frame.

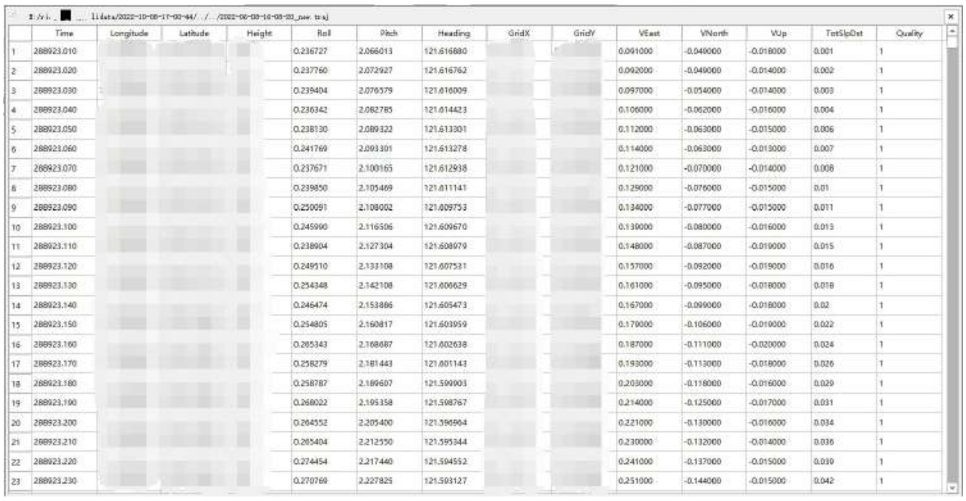
2.**Remove**: Removes the current image data.

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.



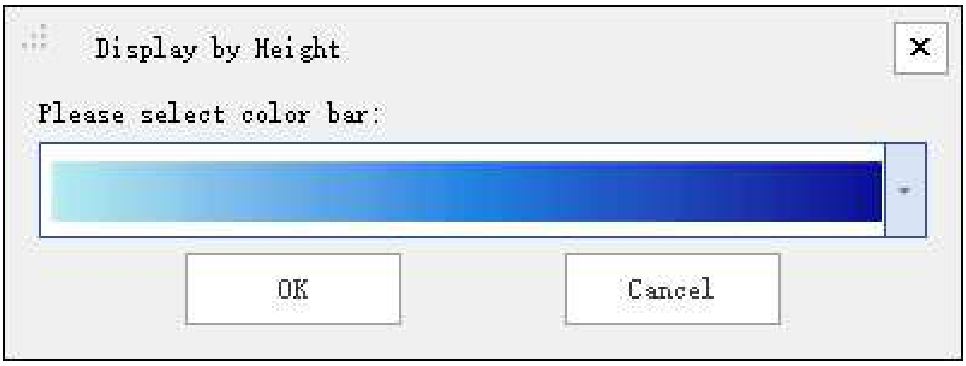
	Time	Longitude	Latitude	Height	Roll	Pitch	Heading	GridX	GridY	VEast	VNorth	VUp	TotSlpDst	Quality
1	288923.010				0.236727	2.066013	121.616880			0.091000	-0.049000	-0.018000	0.001	1
2	288923.020				0.237760	2.072927	121.616762			0.092000	-0.049000	-0.014000	0.002	1
3	288923.030				0.239404	2.076579	121.616009			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	2.108002	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	121.607531			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：

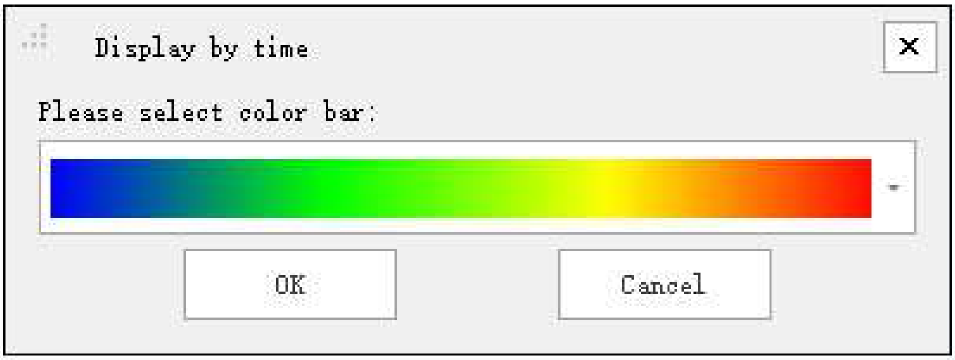
3.**View Mode**: Set the display mode of the track, including the following types：

- **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.

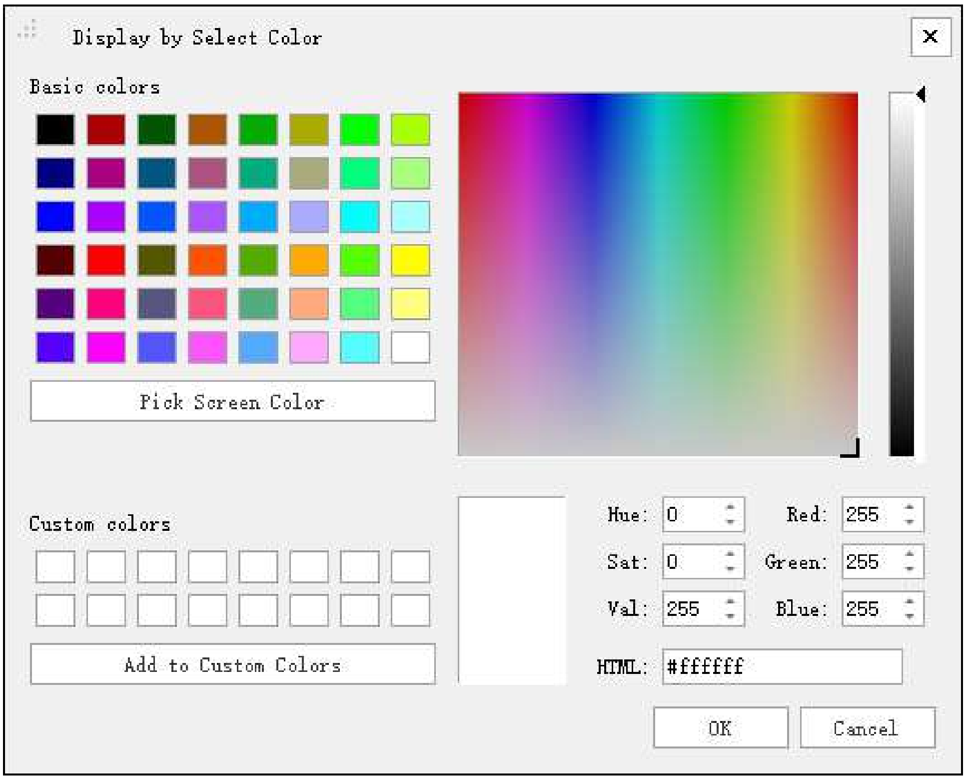
- **Display by Height**: The interface as shown in the figure pops up, and the track can be displayed by selecting different color bars.



- **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 Selected**: Each track data is displayed in the specified color.

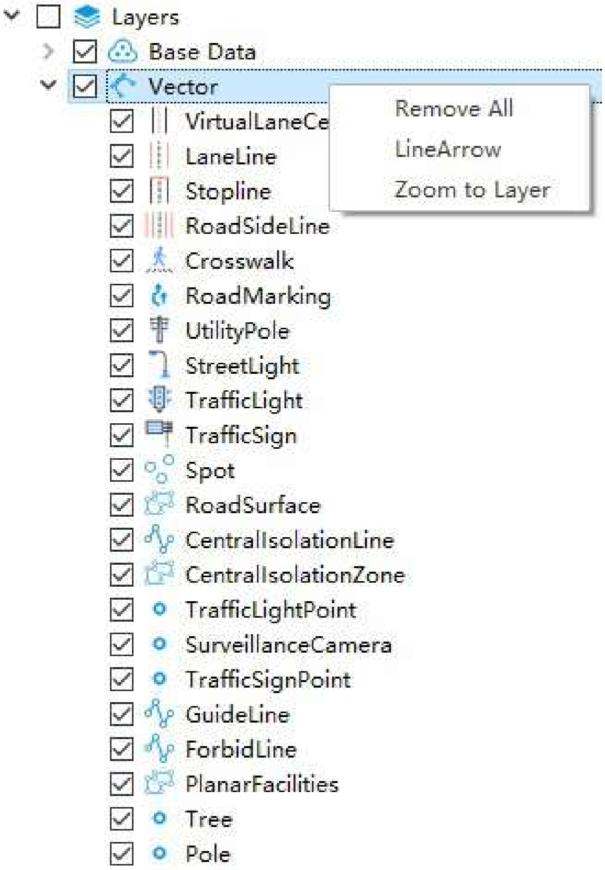


4.**Zoom to Layer**：Center by current track range.

> 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.

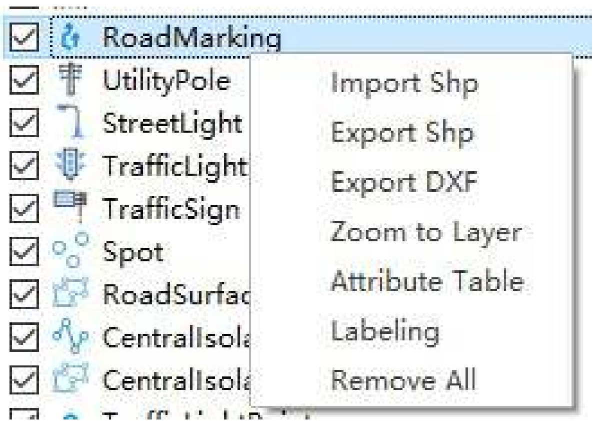


Description

- **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：

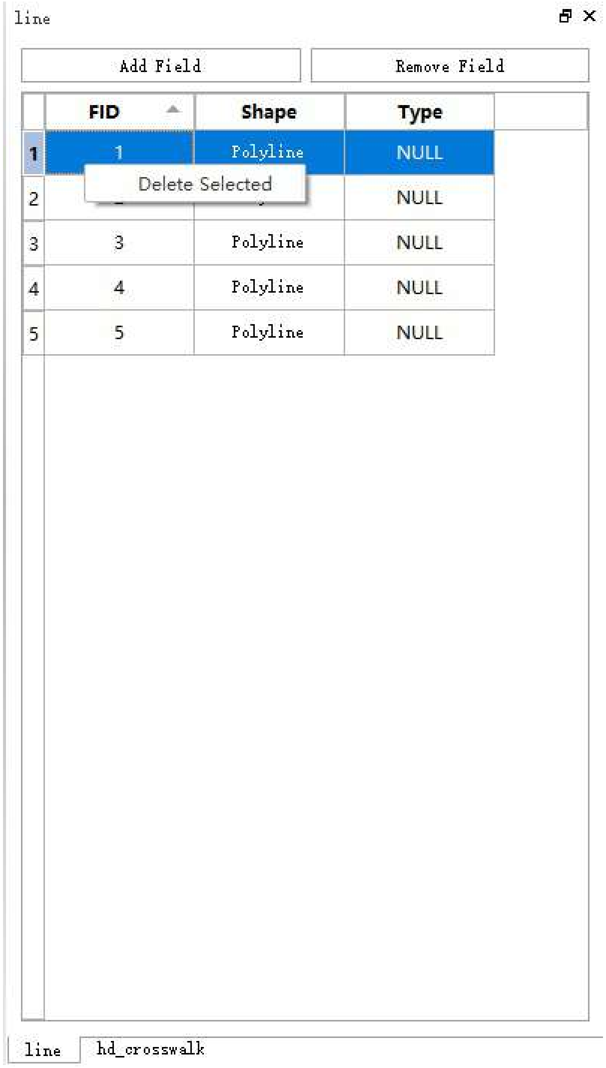


Surface layer

- **Import SHP**: Import external polygon layer SHP files separately into one of the project's polygon layers.

> Note: Only geometry is imported, not attributes

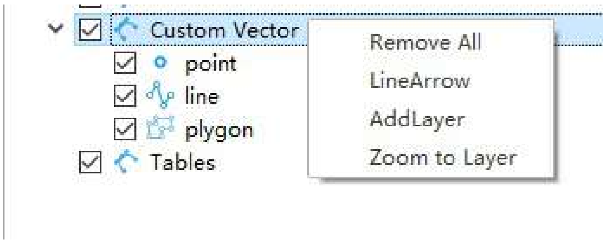
- **Export SHP**: Export the map feature layer to an SHP file separately.
- **Export Dxf**: Export this vector layer to a Dxf file separately.
- **Scale to Layer**: Displays vector data for the layer that was selected first to the middle of the window.
- **Attribute Table**: View the properties of the 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 pop up "Delete Selection" to delete it. At the same time, hold down the Ctrl key to make multiple selections and delete.



- **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 data layer

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. The right-click menu of the four layers is the same, which will be described in detail below.

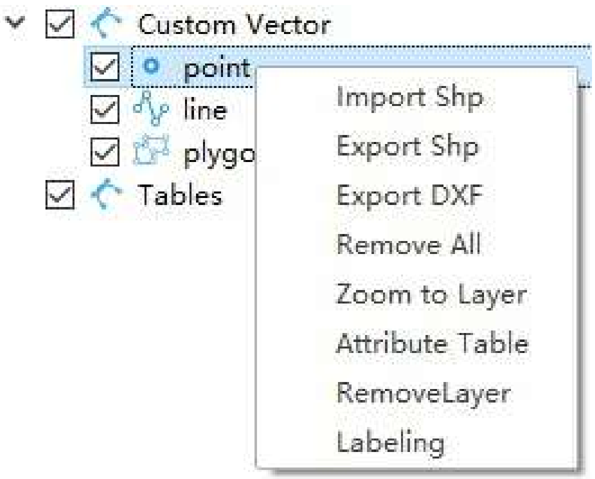


Description

- **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：

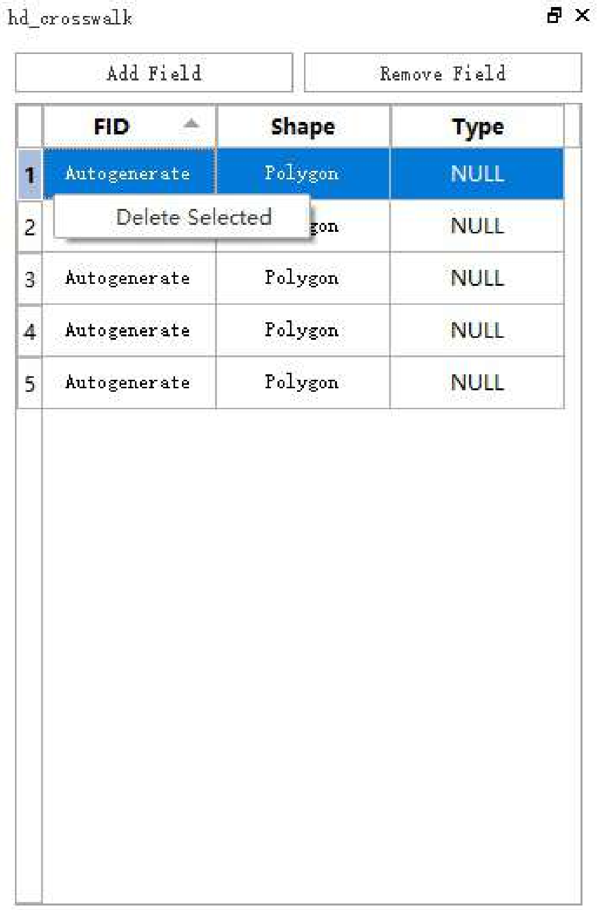


- **Import shp**: Import the shp file of an external point layer into a point layer of the project separately.

  > Note: Only geometric information is imported, not attribute information is imported.

- **Export shp**: Export the vector layer to a separate shp file.
- **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 pop up “Delete Selected content” to perform the delete operation.At the same time, hold down the Ctrl key to multi-select and delete.



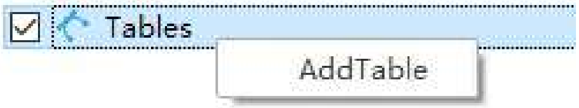
- **Remove layer**: Activate the remove layer function, and a pop-up box will pop up to choose whether to remove the file.
- **Labeling**: Activate the label function, the different attribute values of the layer have been displayed.

List

Function Description : Manage custom lists.

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;



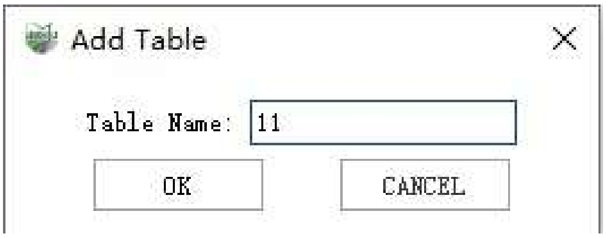
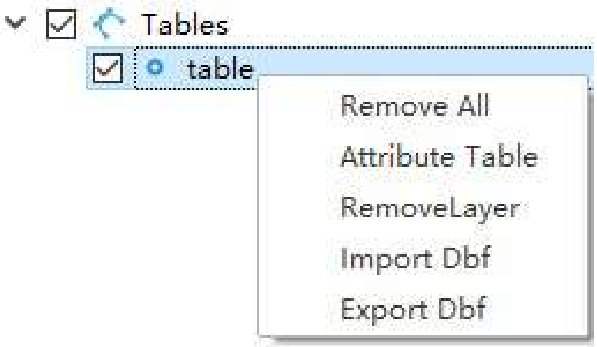


Table right-click operation



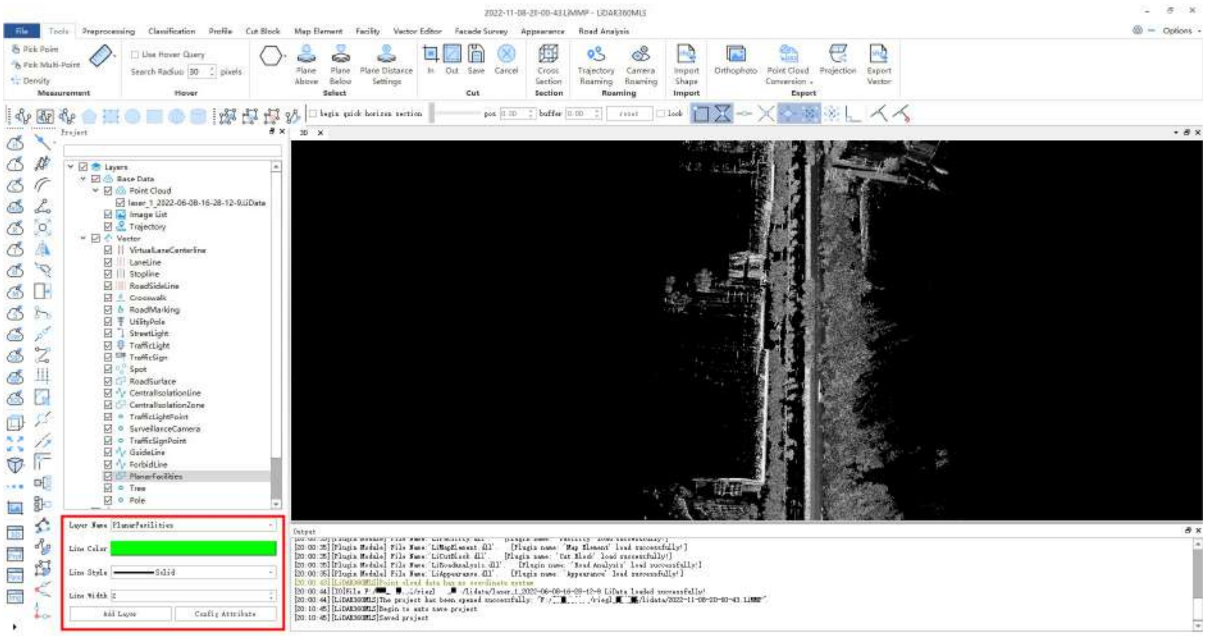
- **Remove all**: Select Remove all to clear all the contents in the list;
- **Attribute table**: Select the attribute table and the attribute table interface will pop up. Users can edit the fields and content of the attribute table, add/delete fields or add a row of records.;
- **Remove layer**: Select Remove layer to remove the current table from the list tree;
- **Import Dbf**: Choose to import Dbf, you can import Dbf files;
- **Export Dbf**: Select Export Dbf, you can export the current table into a Dbf file;

Layer settings

Layer explanation : Used to manage different types of vectorization results.According to the geometry, it is mainly divided into three categories: dot-like layers, linear layers, and planar layers. For example, layers such as lane centerline, lane line, and stop line belong to linear layers, traffic lights, camera points, etc. Belong to dot-like layers, and no-stop lines, pedestrian crossings, etc. Belong to planar 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.

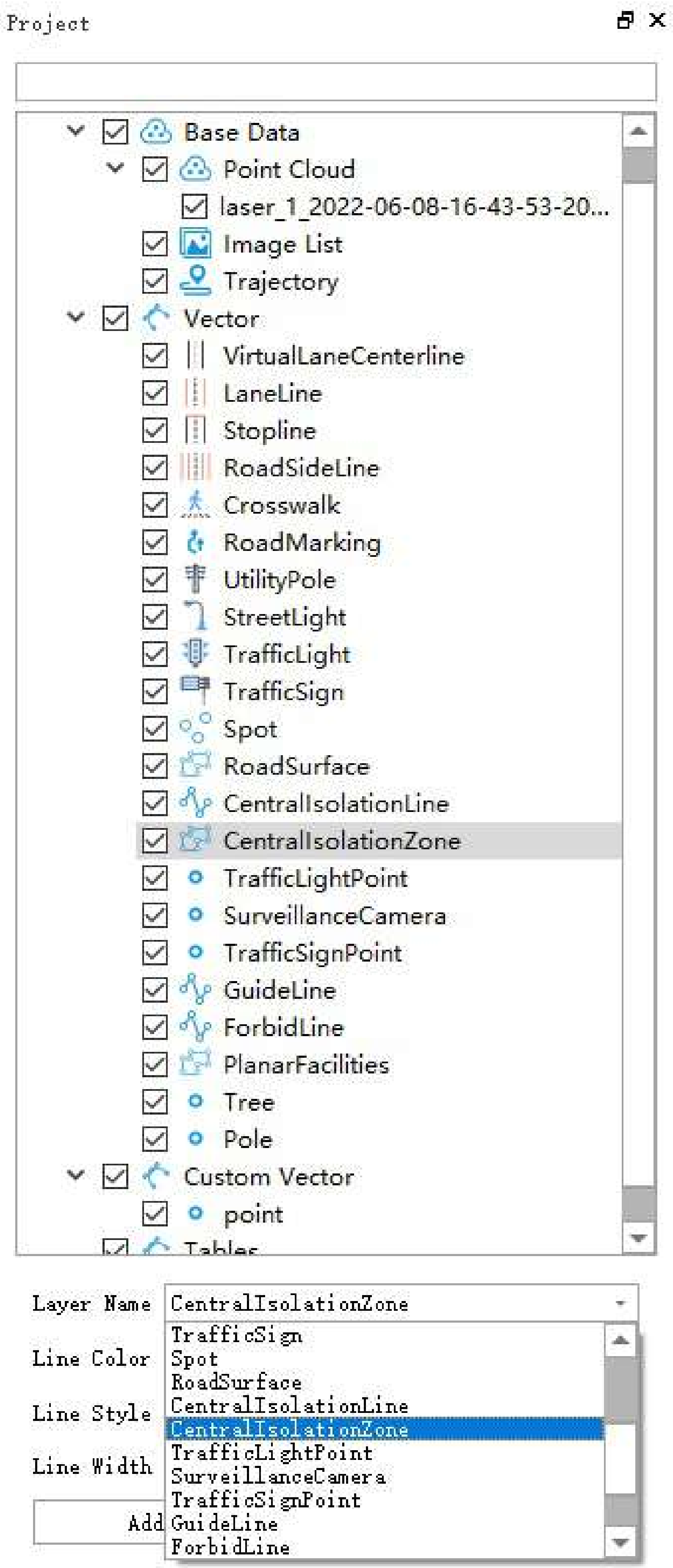
Function description : The role of the layer management module is to select the currently activated layer, set the line type, line width, and color of the currently selected layer, and also add custom layers and configure the selected layer attribute table functions.You can also switch the current layer by clicking the vector layer tree in window management with the left mouse button.



Layer management interface

Description

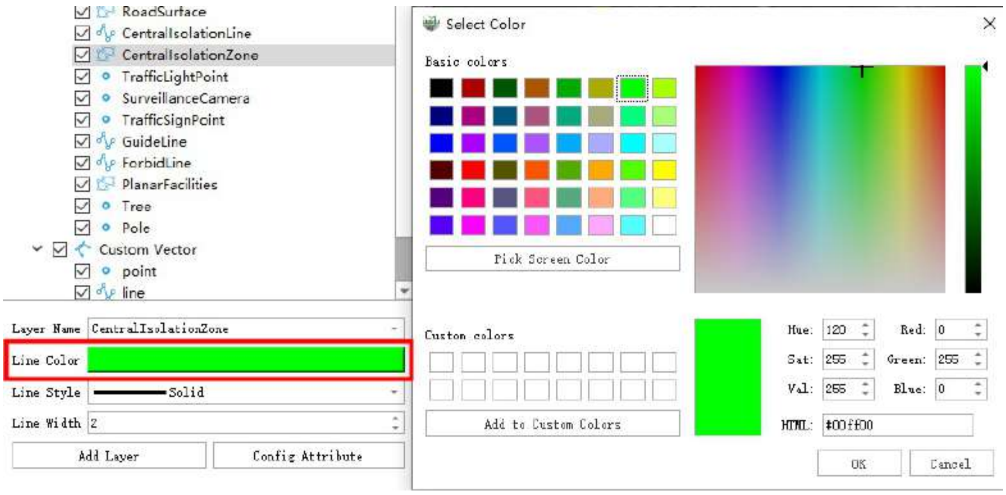
1.**Layer selection**: Click the drop-down arrow under the layer name in the layer manager to select the layer you want to switch, or click the layer on the node tree with the left mouse button to achieve the same effect.


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
Layer Name
CentralIsolationZone
Line Color
Line Style
Line Width
Add
TrafficSign
Spot
RoadSurface
CentralIsolationLine
CentralIsolationZone
TrafficLightPoint
SurveillanceCamera
TrafficSignPoint
GuideLine
ForbidLine

Layer switching interface

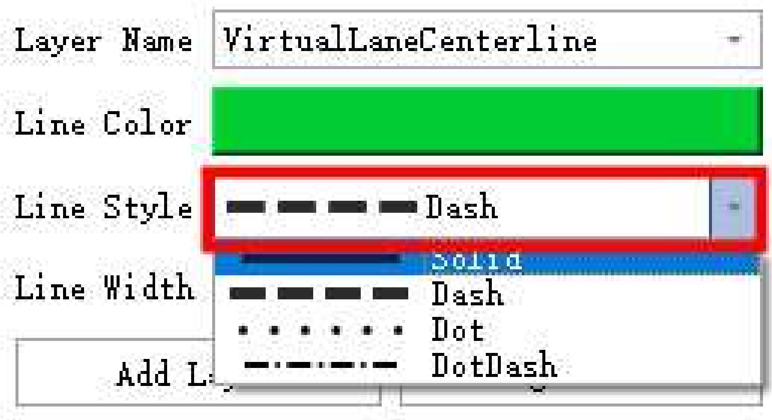
2.**Layer property settings**：

- **Vector color settings**: Click on the color of the midline in the layer manager to pop up the display palette. Users can choose existing colors or customize the various colors they want.



Layer color selection setting interface

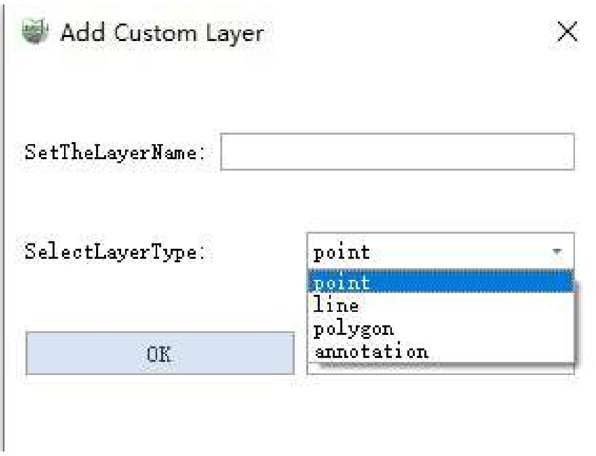
- **Vector line type setting**: Click the line type drop-down arrow to switch the desired line type and the width of the line.



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**: Click the Add Layer button to pop up the Add user-defined layer dialog box

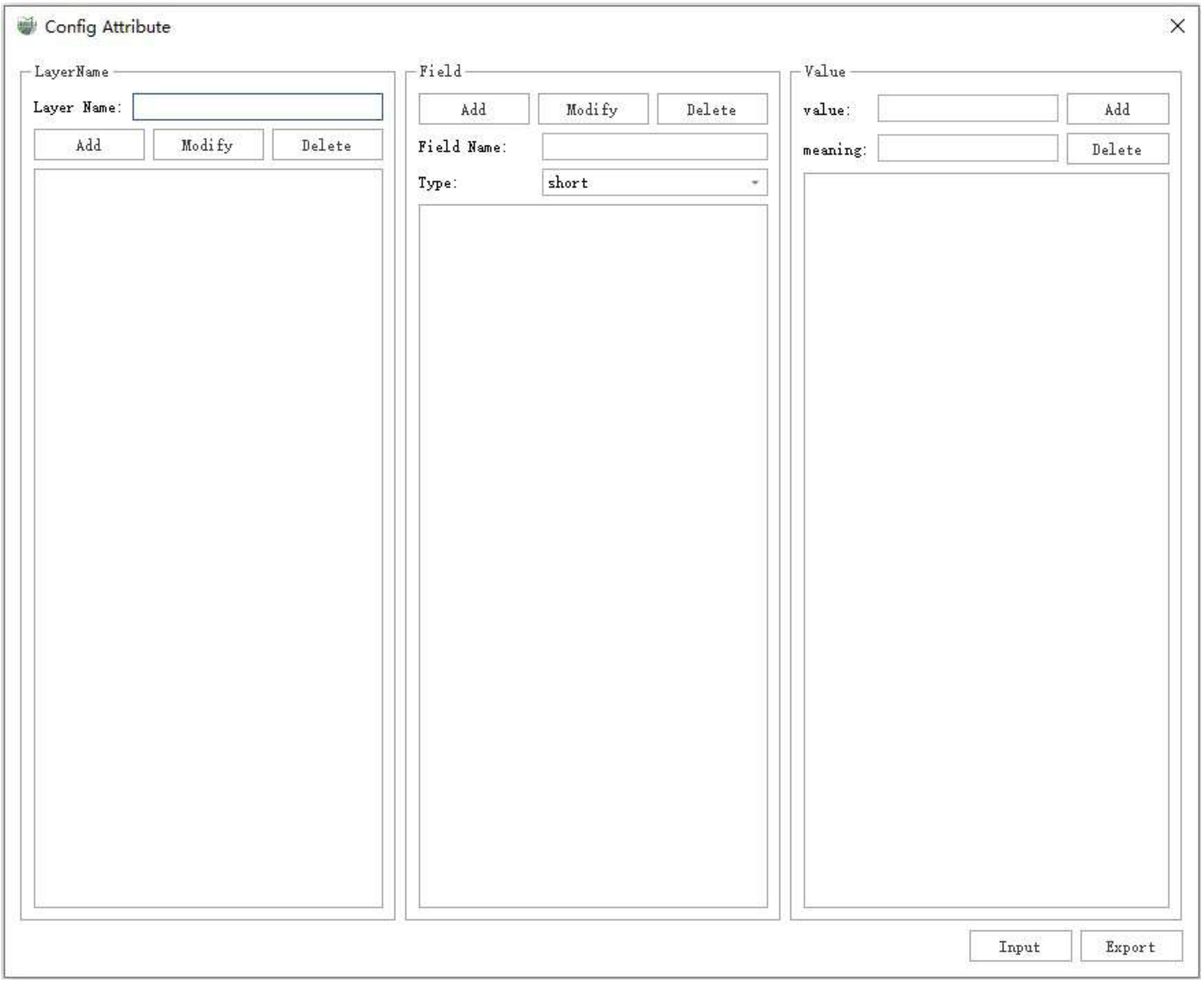


Add layer interface

Parameter setting:

- **Set the layer name**: Customize the layer name.
- **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.

4.**Attribute table configuration**: Click the attribute table configuration button to display the attribute table configuration dialog box in a pop-up window.



Attribute table configuration interface

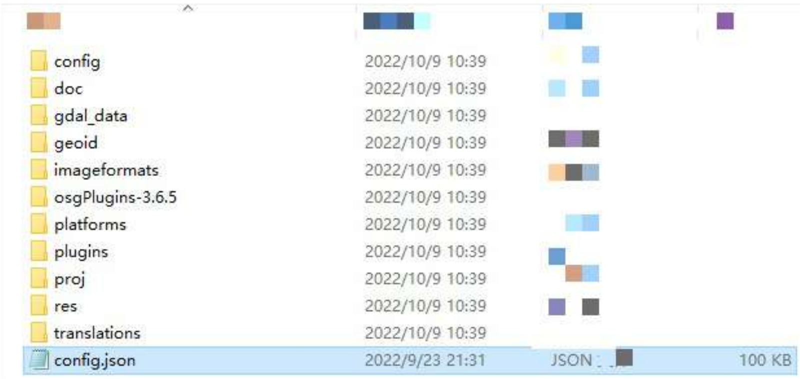
4.1**Parameter setting:**

- **Layers**: In the layers tab, you can add layers that need to be configured, or modify or delete layers that have been set.
- **Fields**: In the field tab, you can add attribute fields that need to be configured for the layer, and set the type and accuracy of the fields.
- **Value**: The value tab can set the value of the new field, and note the meaning of the current value at the same time.
- **Import**: The import button can import the json file generated by the previous configuration and continue to edit the configuration.
- **Export**: The export button can export the current configuration to generate a json file and save it for next use.

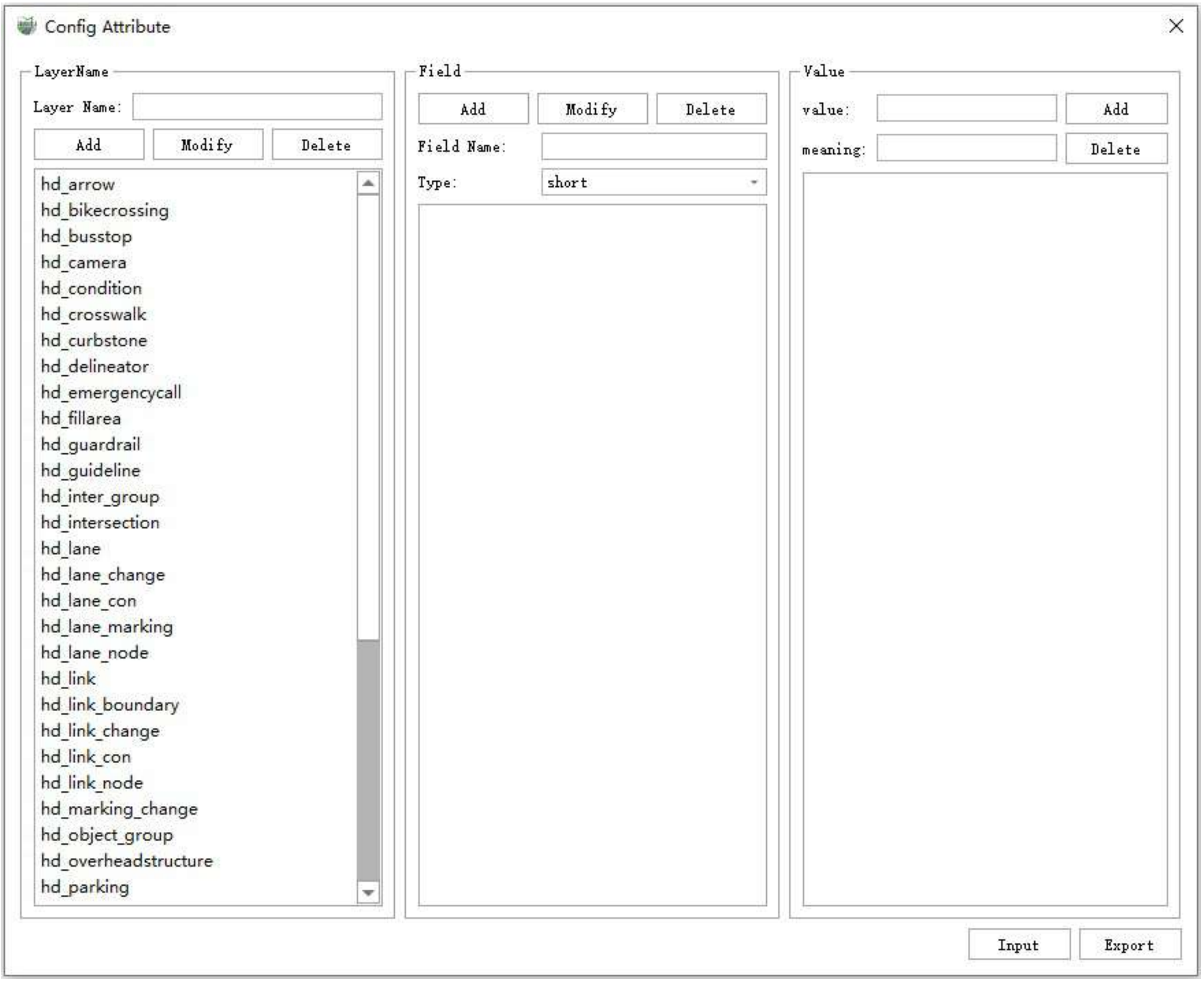
4.2**Operation instructions**

1.**Import data**: The software has a default json file built-in, named config.json, located in the installation directory, click the import button of the attribute table configuration interface, and load config.json file.As shown in the figure below

> json file description: Used to save the software's built-in or user-defined layer properties, including the values of each field and field inside the layer.Among them, the json file has some layer attribute settings built in by default, see the figure below for details.



config.json

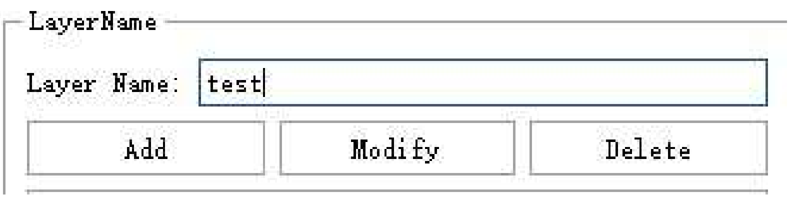


Import the built-in json file of the software

2.**Layer paging settings**: In the layer paging, click the layer name in the list with the left mouse button to modify and delete the currently selected layer, and it can also be used to add custom layers.After selecting the operation layer, you can set the fields and field values separately in the field paging and value paging.

> Note: The layer name in the layer tab needs to be the same as the name in the layer node tree for the settings saved in the json file to take effect

- **Add a layer**: Enter the name of the layer you want to create in the layer name in the layer tab, and click the Add button to complete the layer addition.After the layer is added, you can set the fields and field values. For details, see the "Modify fields and Modify Values" section later.



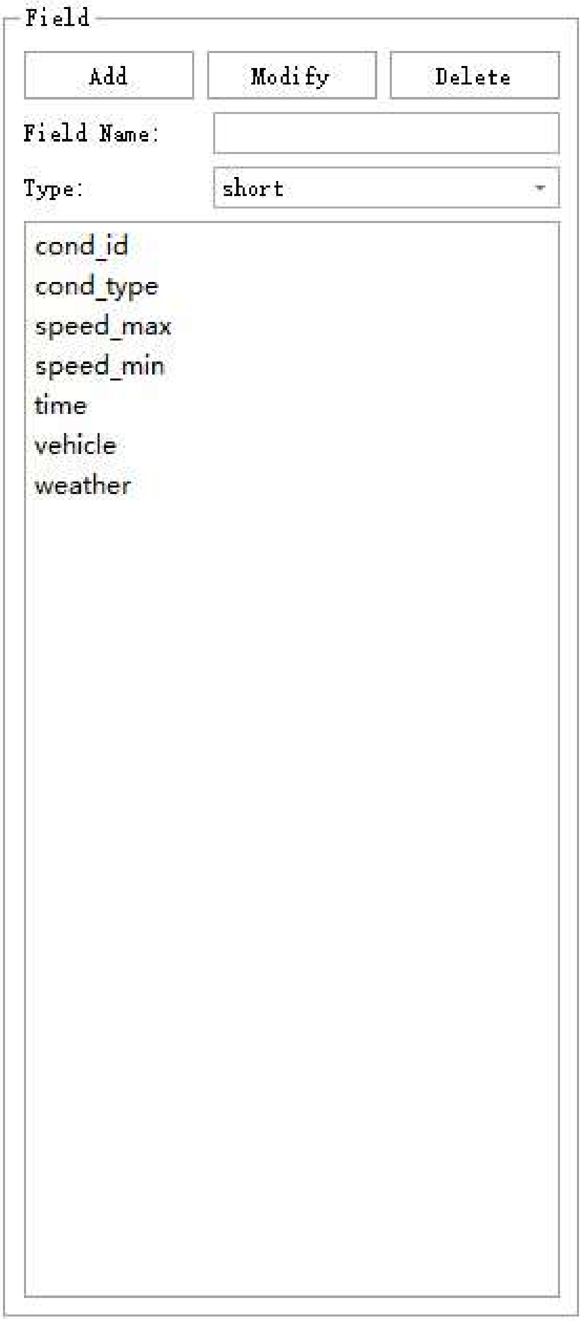
Enter the layer name

- **Modify layer**: Click the left mouse button to select the layer, enter the name you want to modify in the layer name, and click the Modify button to modify the layer name

Modify layer

- **Delete layer**: Click the selected layer with the left mouse button and click the Delete button to delete the selected layer

3.**Field paging settings**: After selecting the layer to be modified in the layer paging, you can set the fields in the selected layer in the field paging, including adding fields, modifying fields, and deleting fields.



Field paging

Parameter description

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

> Note: You need to set the field name, type, width, and precision before you can add it successfully.

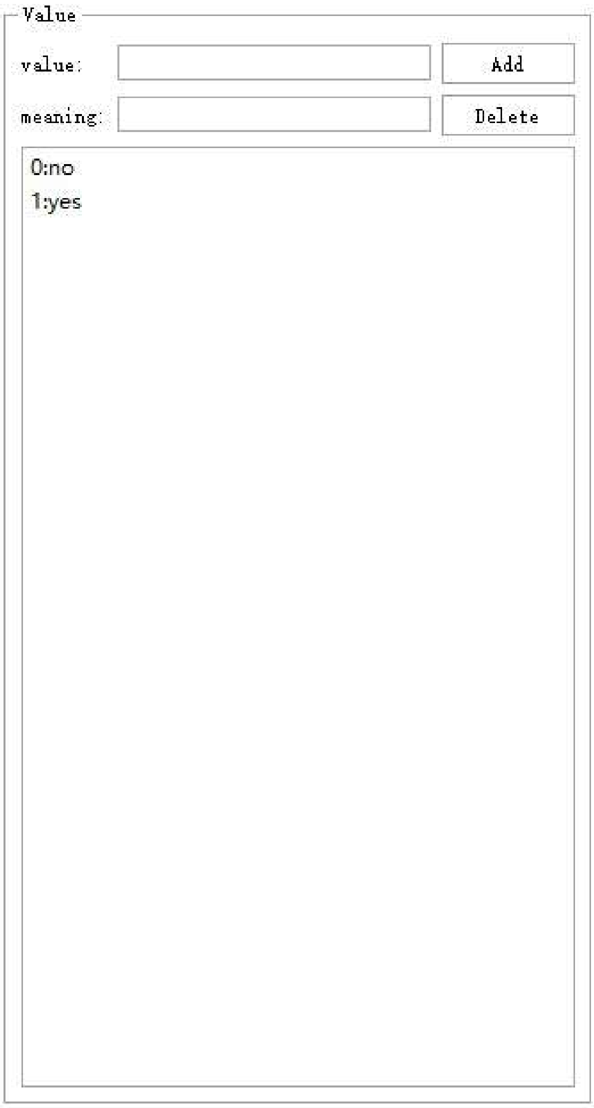
- **Modify**: Modify the existing fields in the selected layer

> Note: Modify the properties of the field itself (field name) here. To modify the field value, refer to the field value settings later.

- **Delete**: Delete existing fields in the selected layer
- **Field name**: One of the field attributes

- **Type**: One of the field attributes, which identifies the type of the current field stored. The options are as follows：
  - short: Identifies that the current field is stored as an integer type with a small range of values
  - long: Identifies that the current field is stored as an integer type with a large range of values
  - float: Identifies that the current field is stored as a decimal type, the data accuracy is small, and it is not suitable for storing values with high accuracy requirements such as coordinates.
  - double: Identifies that the current field is stored as a decimal type, with high data accuracy, and can be used to store coordinate data
  - txt: Identifies the current field stored as a text type
  - date: Identifies the current field stored as the time type
- **Width**: Identifies the number of valid digits of the current field, such as a field of type short, set the width to 2, and its value range is 0~99
- **Precision**: Identifies the number of digits after the decimal point of the current field, such as a field of type double, and the true value is 0.3333333. After setting the precision to 3, the value stored in the attribute table is 0.333

4.**Field value setting**: After selecting the layer and field, you can set the value of the field and the corresponding explanation in the value tab.



Value paging

Parameter description

- **Value**: The value of the selected field
- **Meaning**: The corresponding meaning of the selected field
- **Add**: Add a new value

> Note: After the value and meaning are set, the addition can be successful.

- **Delete**: Delete existing values

5.**Export data**: After completing the layer settings, click the export button on the lower right side of the attribute table configuration interface to pop up the json save folder location selection dialog box.

> Note: The json file name cannot be changed, the default setting is config.json, the save path must be stored in the config file of the software installation directory.

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.

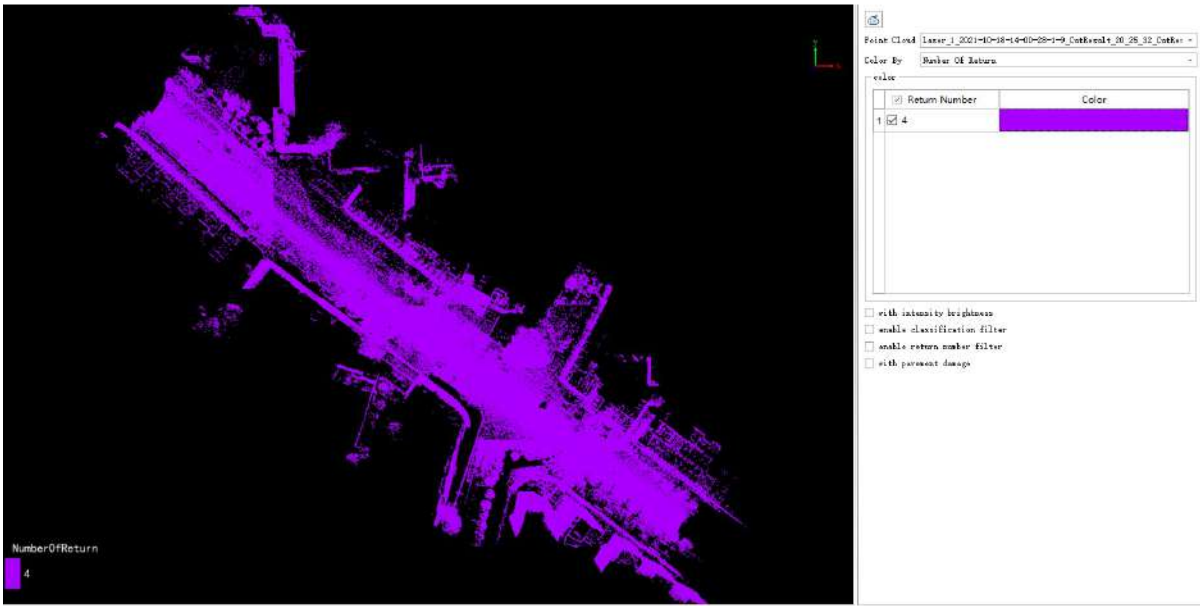
5. Display by RGB: For specific display effects, please refer to Display by RGB.

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.

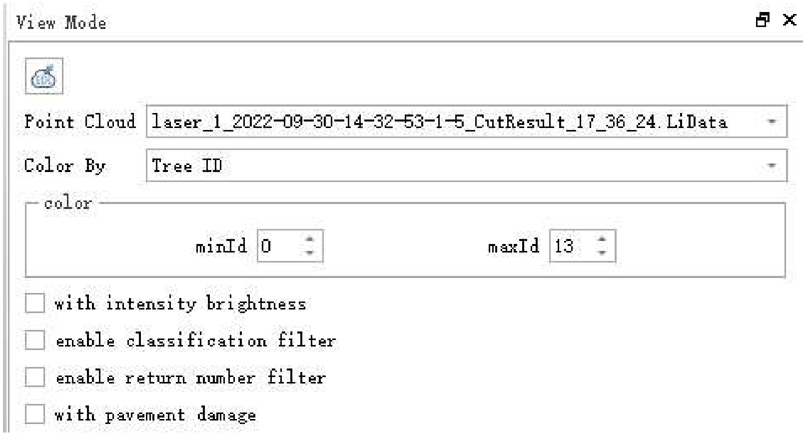
7.1 Click the number of echoes in the drop-down box, and a dialog box will pop up to display the number of echoes.

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, see 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 point cloud data records data source information, it can be displayed and rendered according to this attribute

12.Display by Scan Angle Rank: If the scan angle attribute is recorded in the point cloud data, 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.

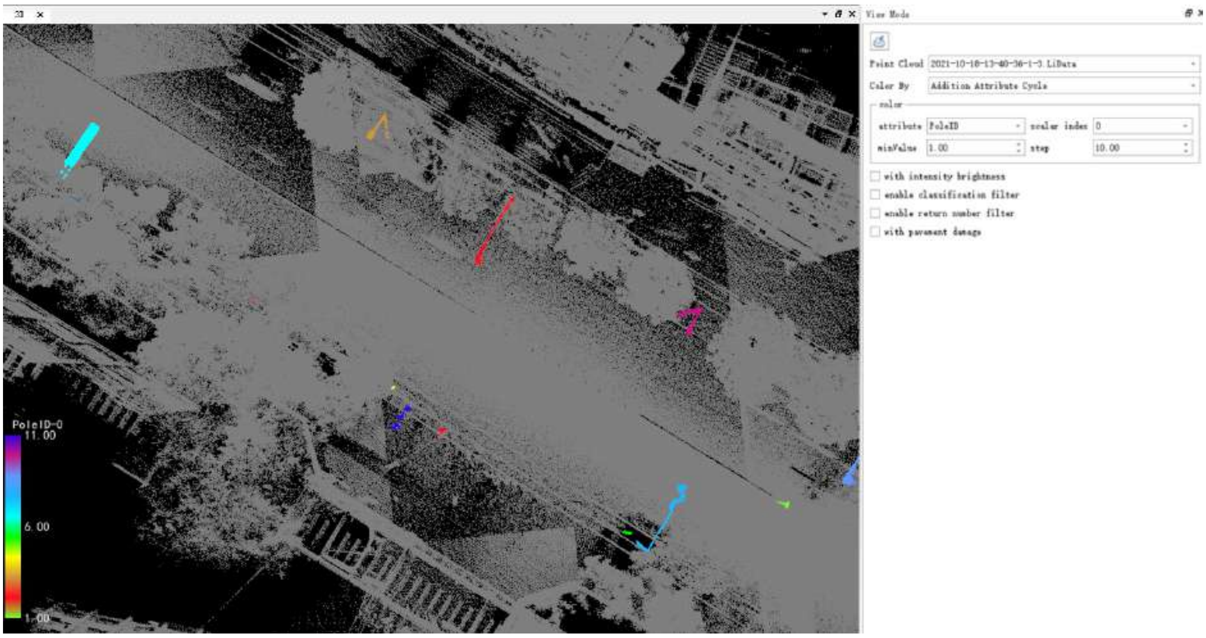
14.Display by Scan Channel: If the point cloud data records the scan channel information, it can be displayed and rendered according to this attribute.

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: If there are additional attributes in the point cloud data, as shown in the figure below, there is a single feature ID (PoleID), the color cycle can be displayed 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, you can set a ribbon to render incremental rendering 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 according to a certain additional attribute value for rendering

Multi-attribute combination display

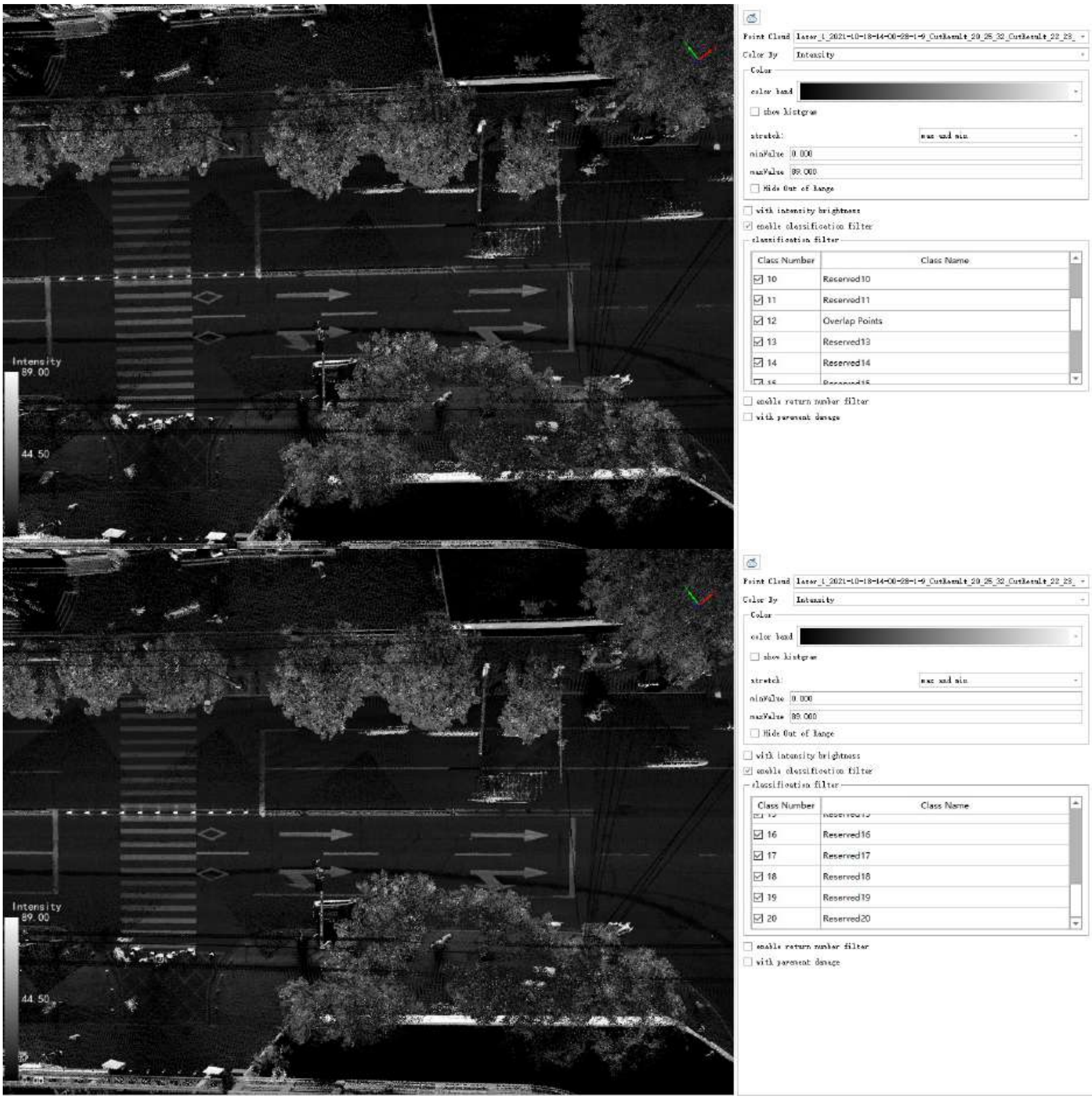
In the display mode window, you can set the categories that need to be displayed in the color display, and in the check box at the bottom of the window, check the categories that need to be superimposed.

Description

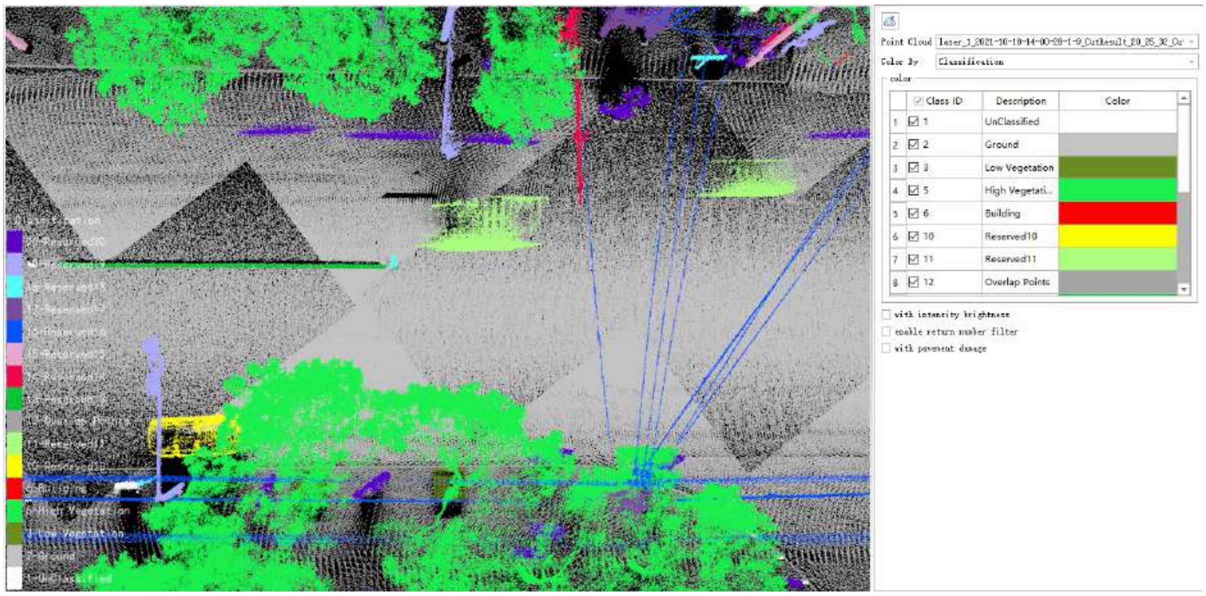
- **Turn on intensity filtering**: After checking, the intensity filtering attributes of the point cloud data are mapped to different color values to more intuitively distinguish between different intensity filtering point cloud data.
- **Enable category filtering**: After checking, filter the categories 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.

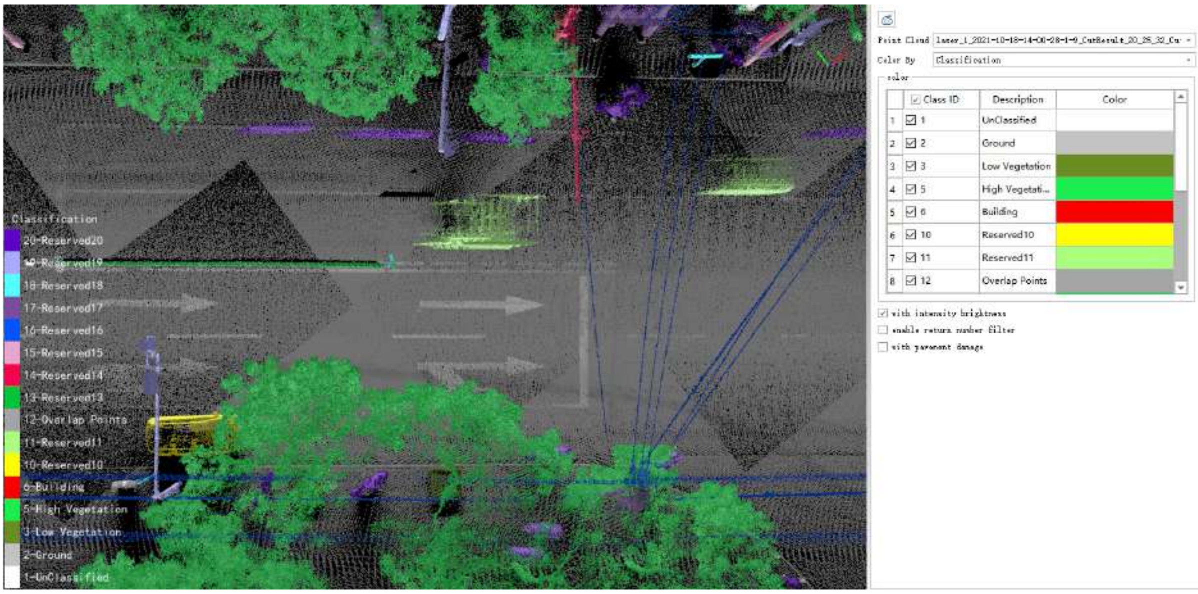
The commonly used combination display modes are：

(1) Intensity superposition Classification filtering：

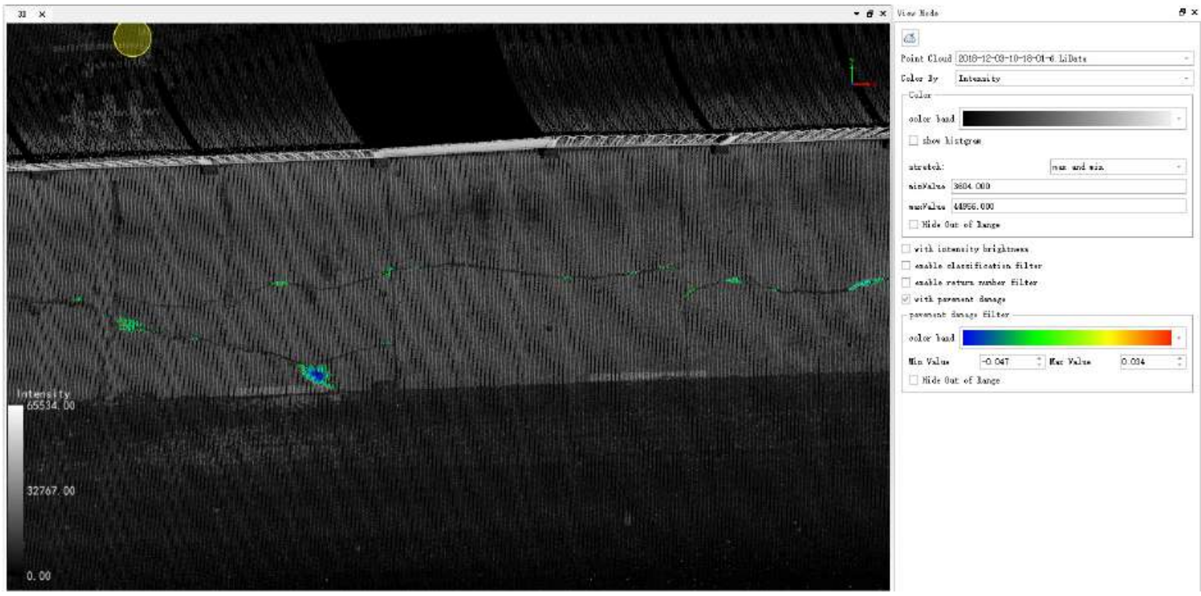


(2) Classification with intensity brightness：





(3) Intensity with pavement damage：

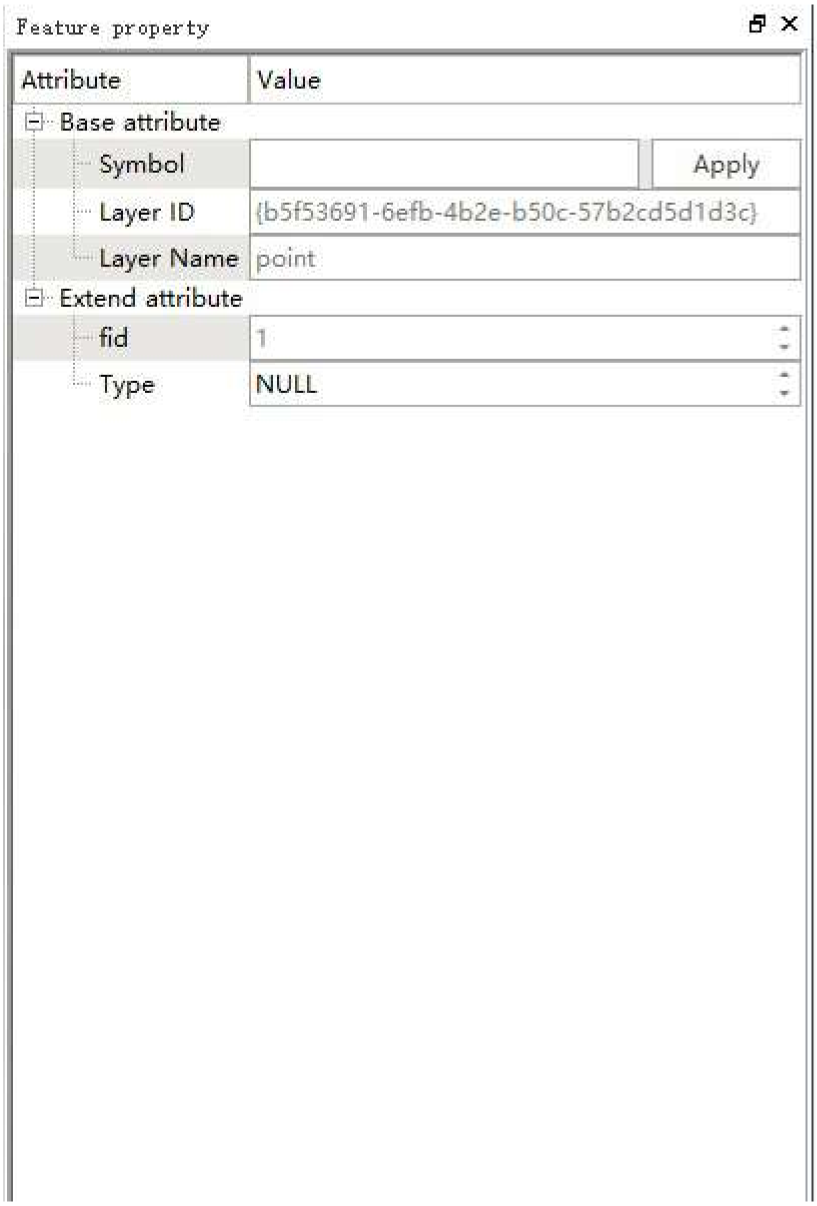


Feature Attributes

Function description：Display vector-related information. When modifying vector information, the feature attributes correspond to it. When the layer attribute is a point layer, the added symbol can be displayed.

Description

1.**Feature attributes**： Select a vector to display vector-related information.



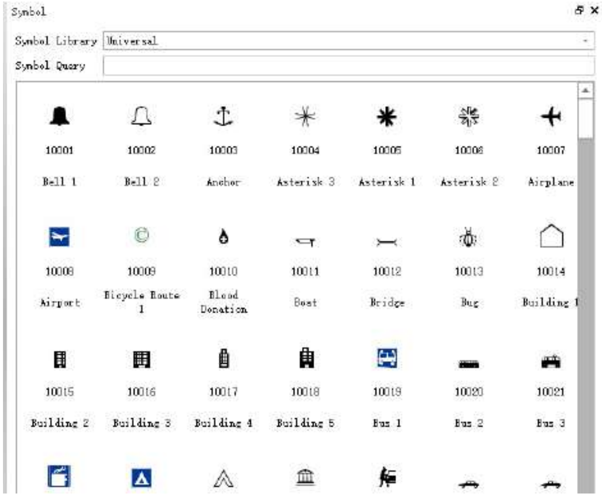
Feature attributes

Basic attributes

- **Symbol**：You can add the desired symbols on the point layer.
- **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 pop up the symbol table:

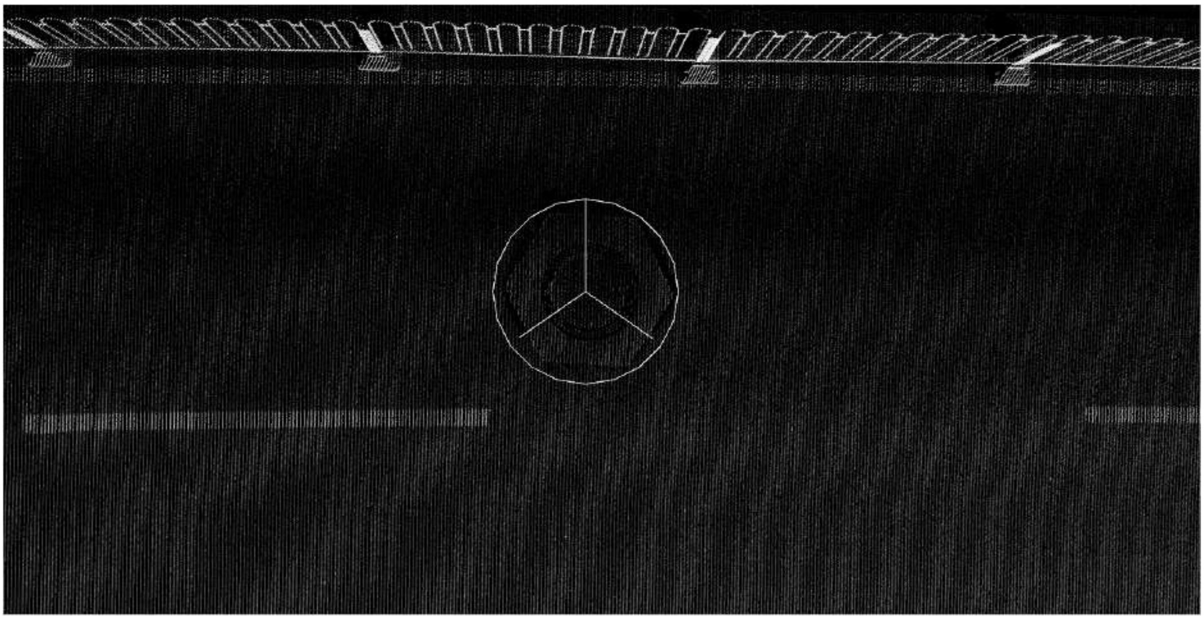


Symbol table

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, click "Apply".Then click in the appearance function Display symbol Viewable.


Symbol table

Modify attribute values

You can directly click the edit control in the second column of the extended attribute to modify it.

Index

As shown in the figure below, the file page contains the following functions:

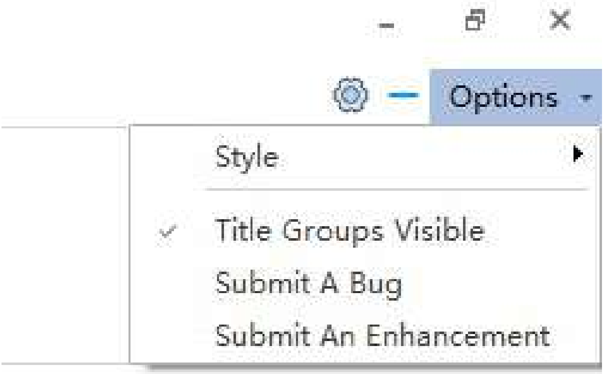
- Options
- Display Options
- High Graphics Performance Mode
- FAQ
- Version Update Subscription

Options

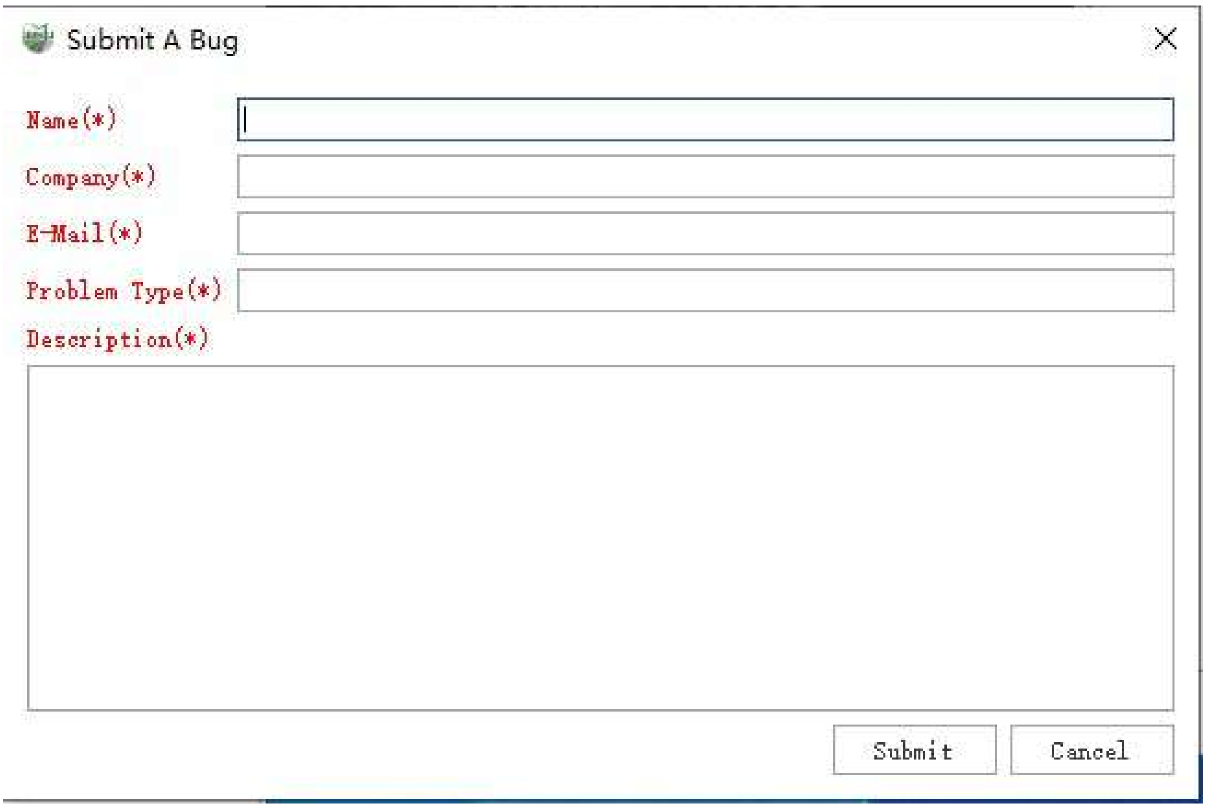
Function description：The software main interface option function provides auxiliary operations such as style setting, group name display and hidden, bug and demand feedback.

Steps

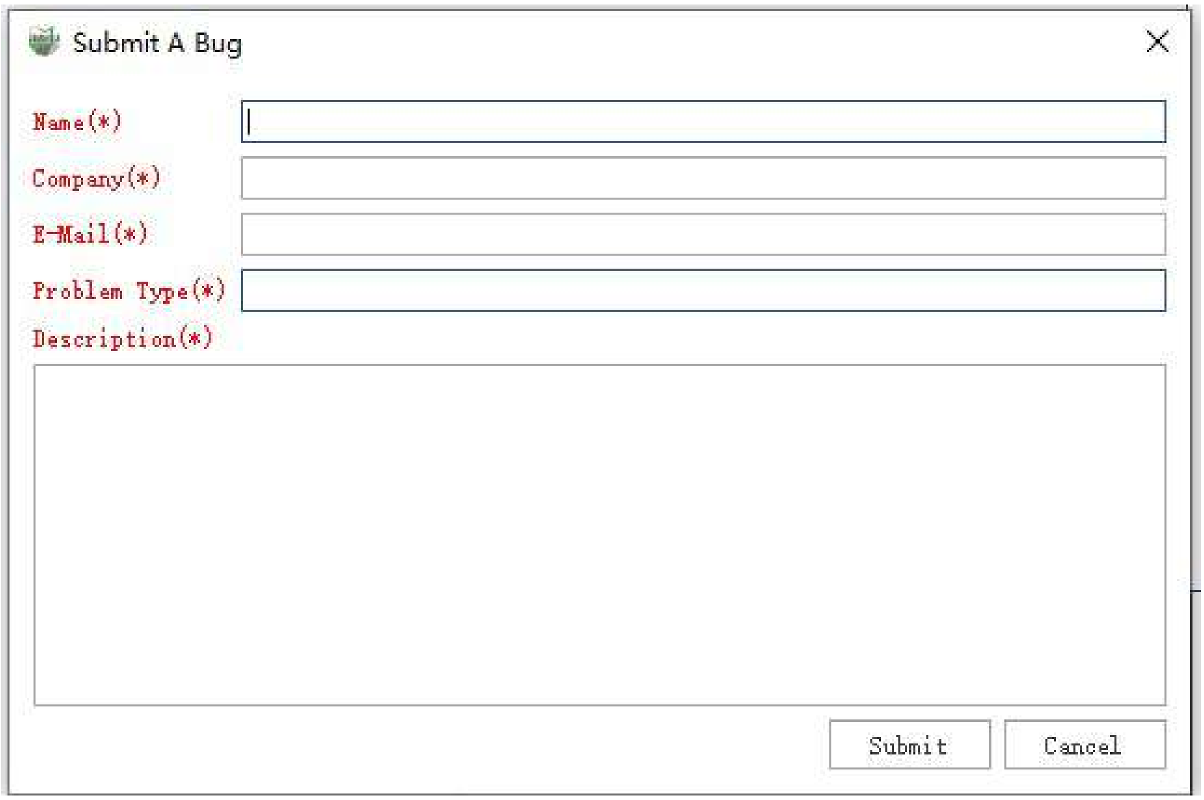
1.In the upper right corner of the main interface, click "Options", where you can modify the interface style and show or hide the menu bar group names.



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 requirements: Click "Submit Requirements", 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.

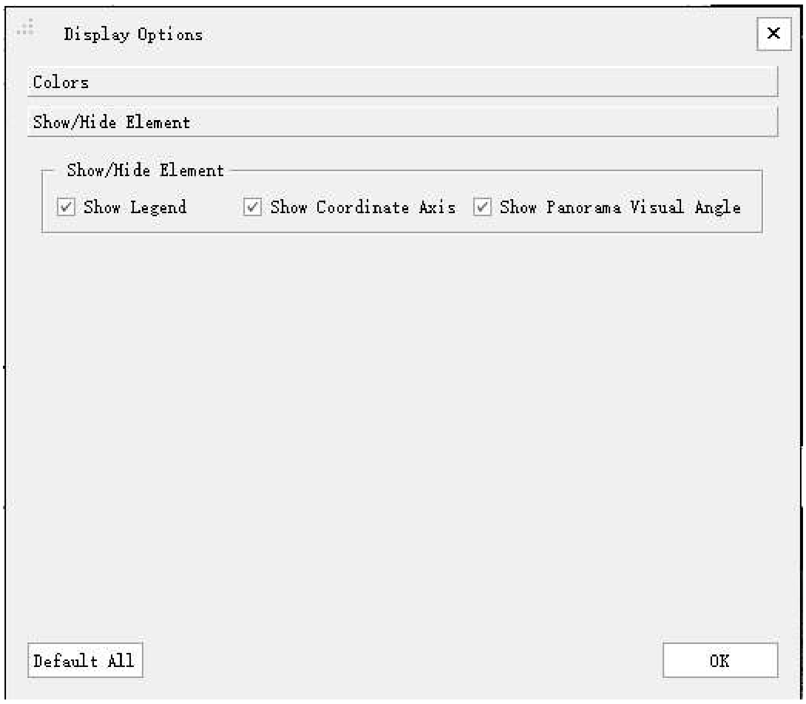


Display Options

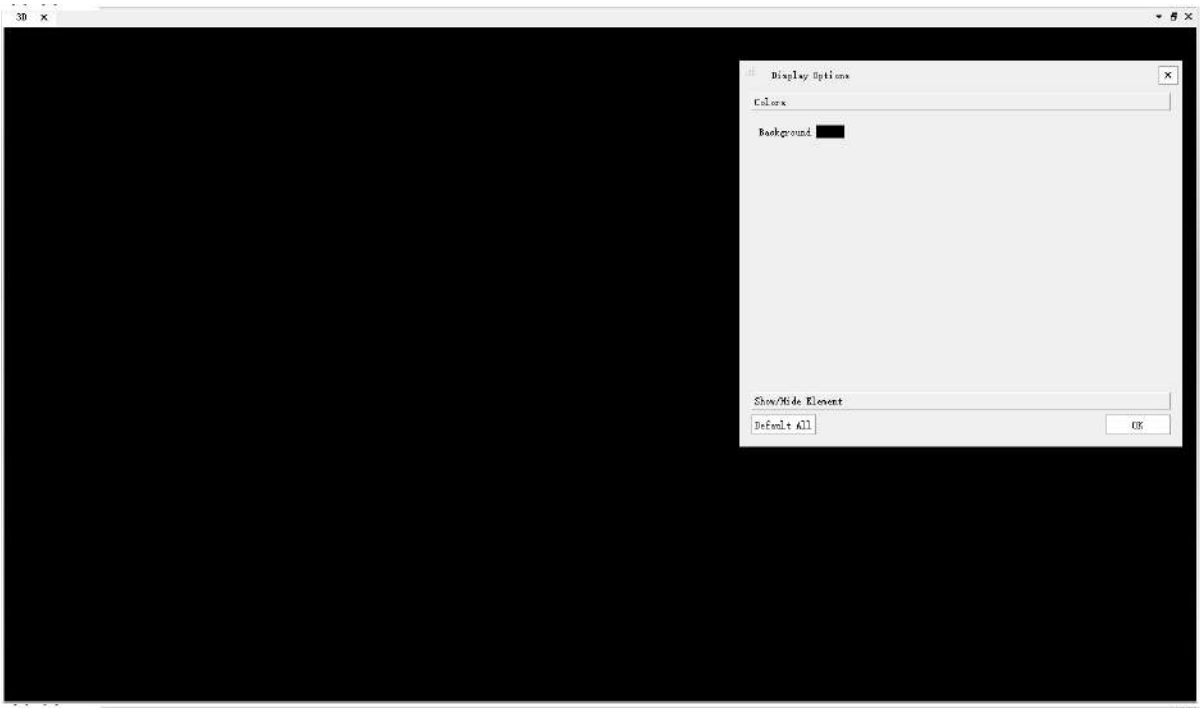
Function description: The display options function of the main interface of the software provides settings for options such as background color, element display and hide, camera height and orientation.

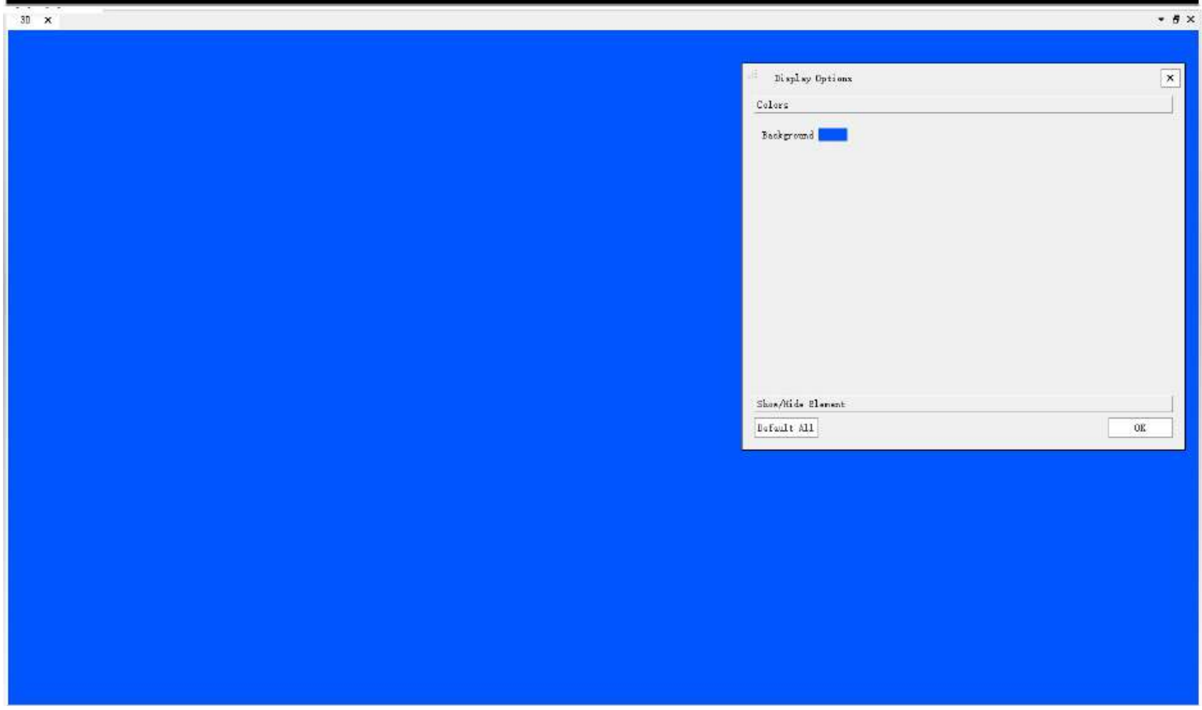
steps

1.Click the display option setting  button in the upper right corner of the software to pop up display options Settings dialog.



2.Color: 3D window background color can be set





3.Show/Hide Elements: Whether to show/hide the legend, coordinate axis and panoramic view of the 3D window

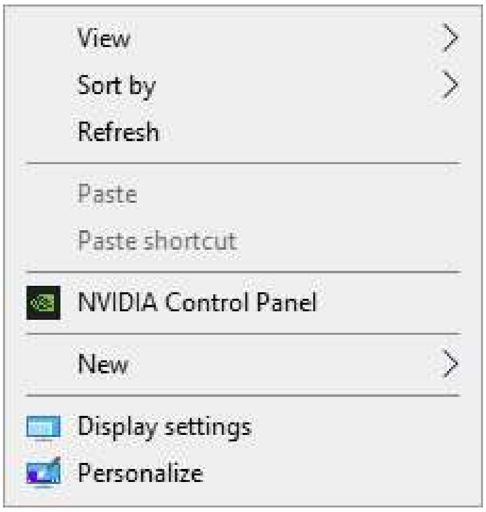

3D
Display Options
Colors
Show/Hide Element
Show/Hide Element
Show Legend
Show Coordinate Axis
Show Panorama Visual Angle
Default All
OK


3D
Display Options
Colors
Show/Hide Element
Show/Hide Element
Show Legend
Show Coordinate Axis
Show Panorama Visual Angle
Default All
OK

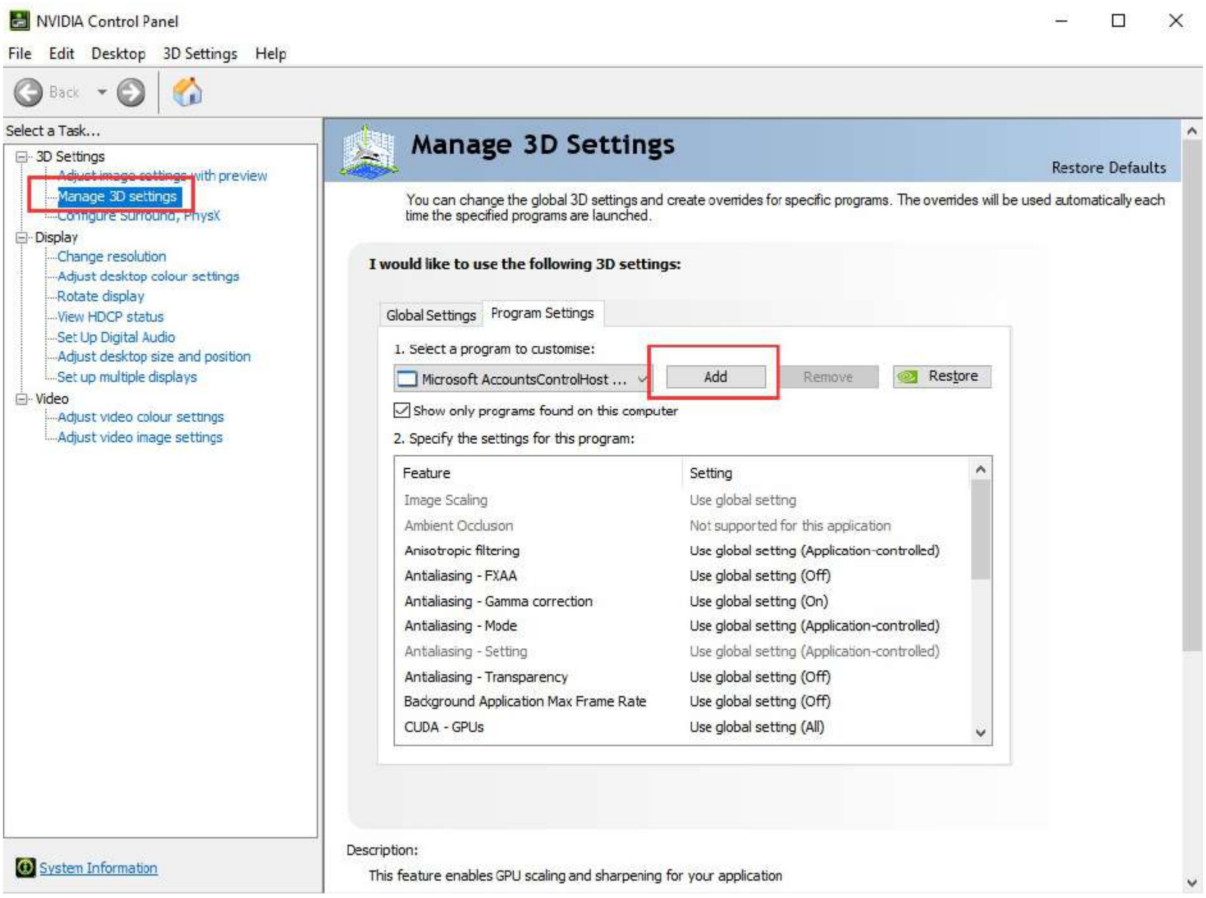
Adjusting to High-Performance Display Mode

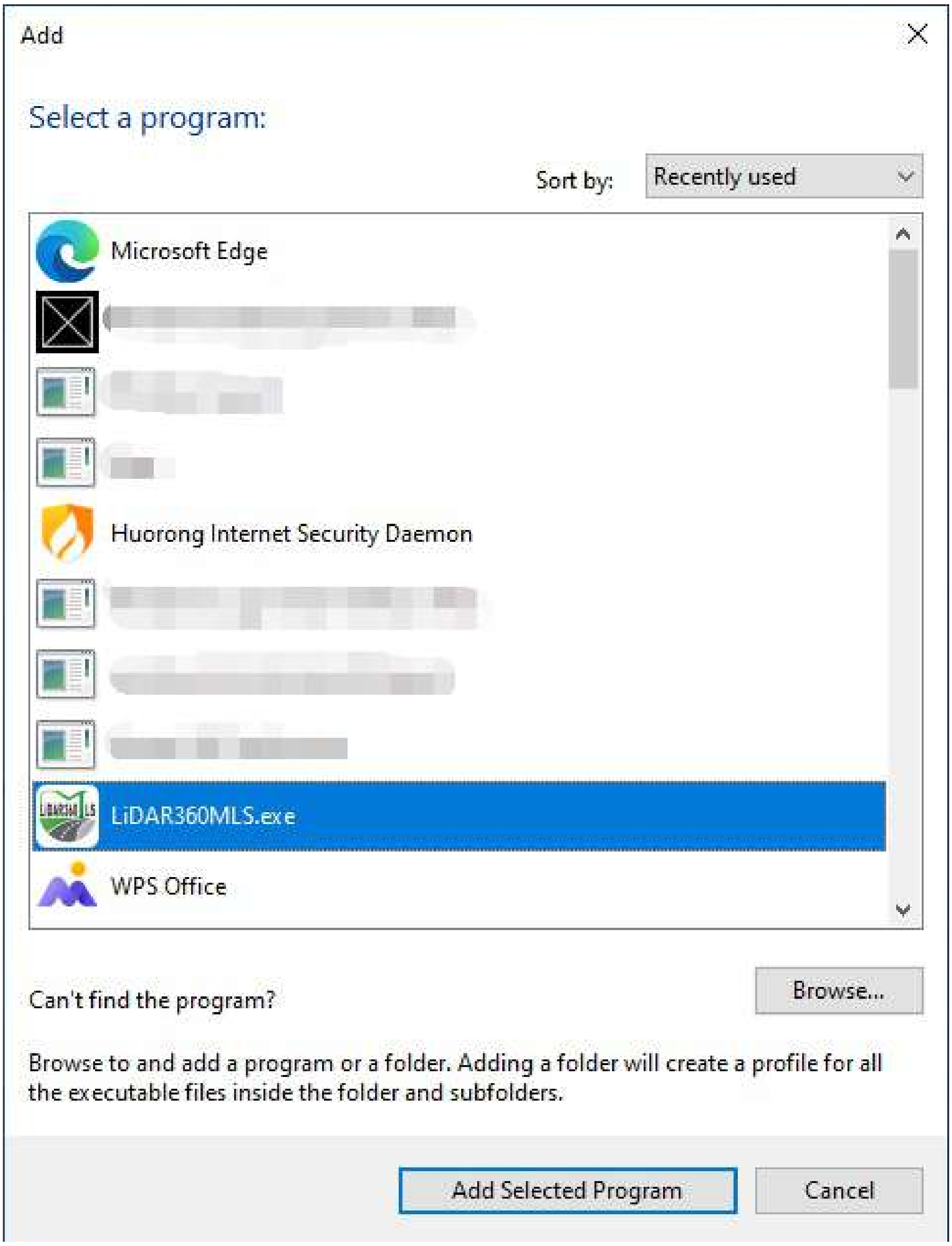
Follow the steps below to optimize the graphics mode of LiDAR360MLS.exe (for NVIDIA graphics cards).

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

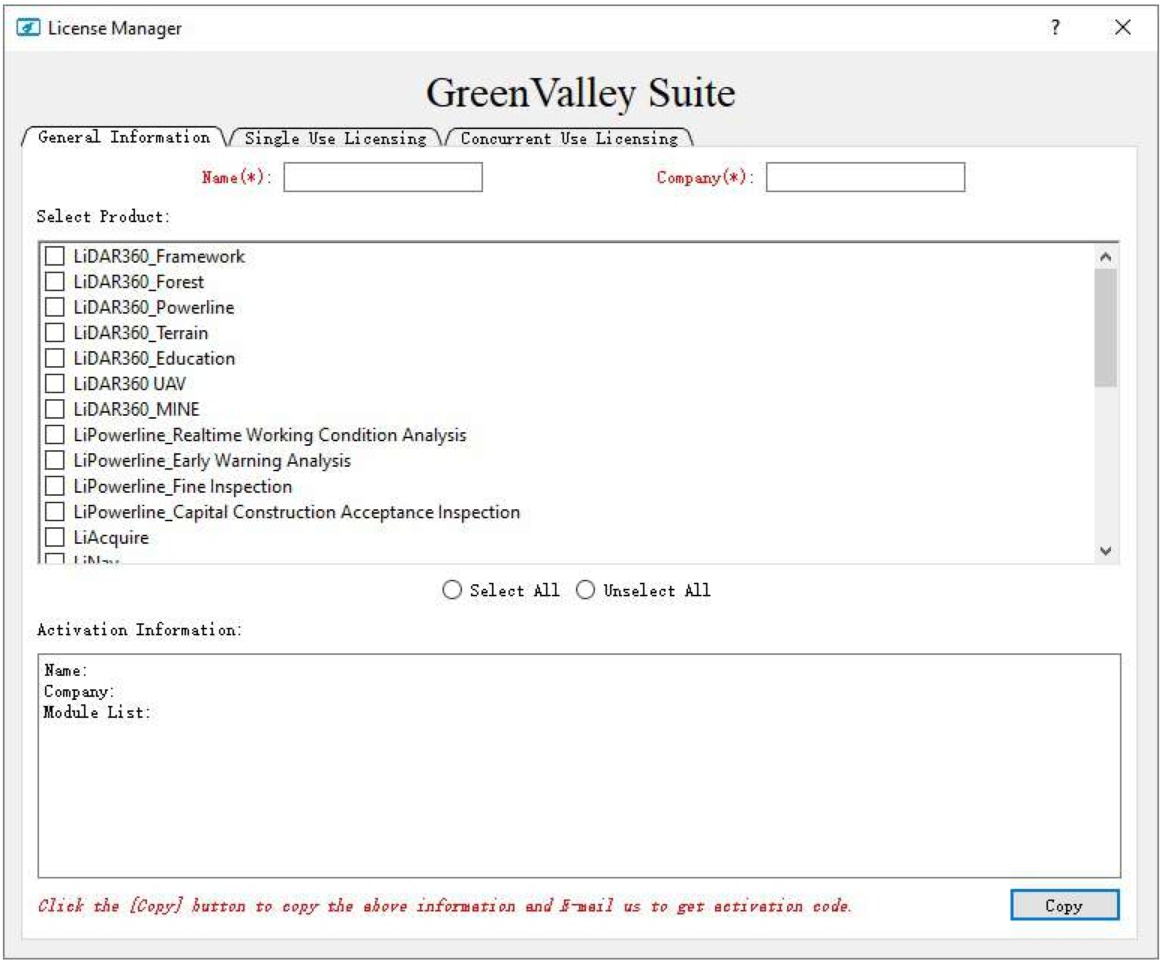
Function description: This chapter introduces common problems and solutions during the use of LiDAR360 MLS software.

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 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?

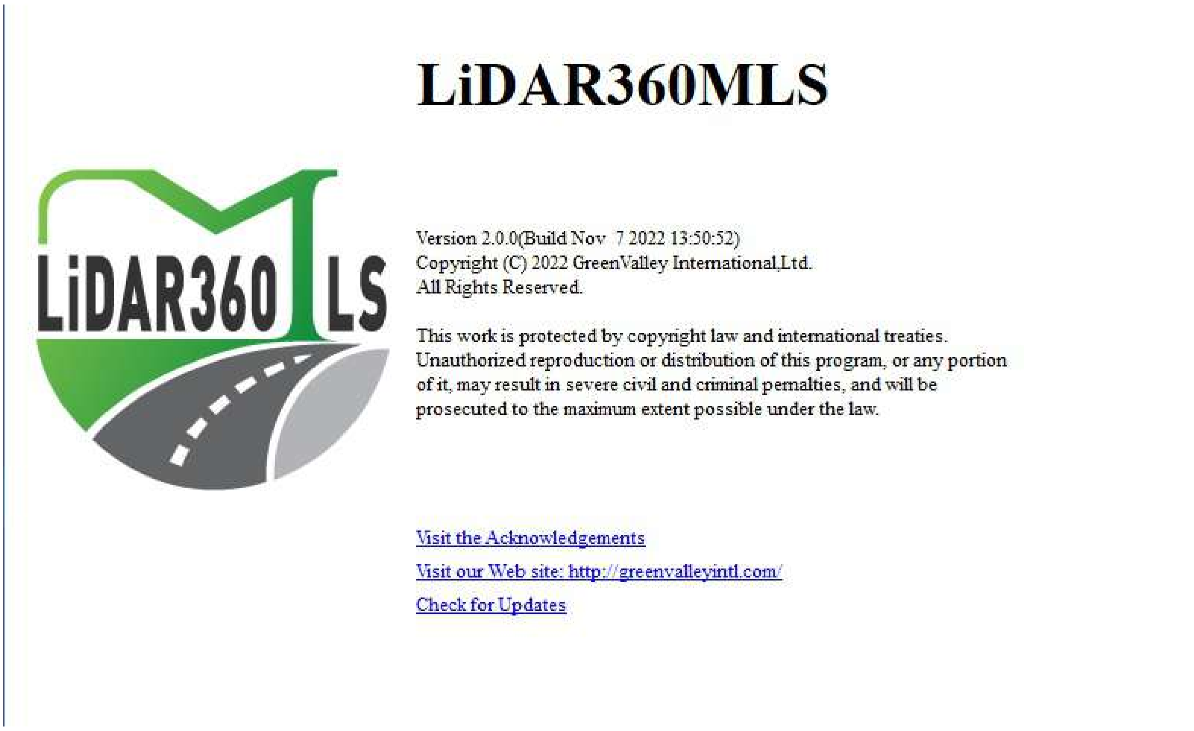
- Click the Help button in the upper right corner of the software interface.

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 2 2022 14:14:38 is the compilation date.



How do I 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).

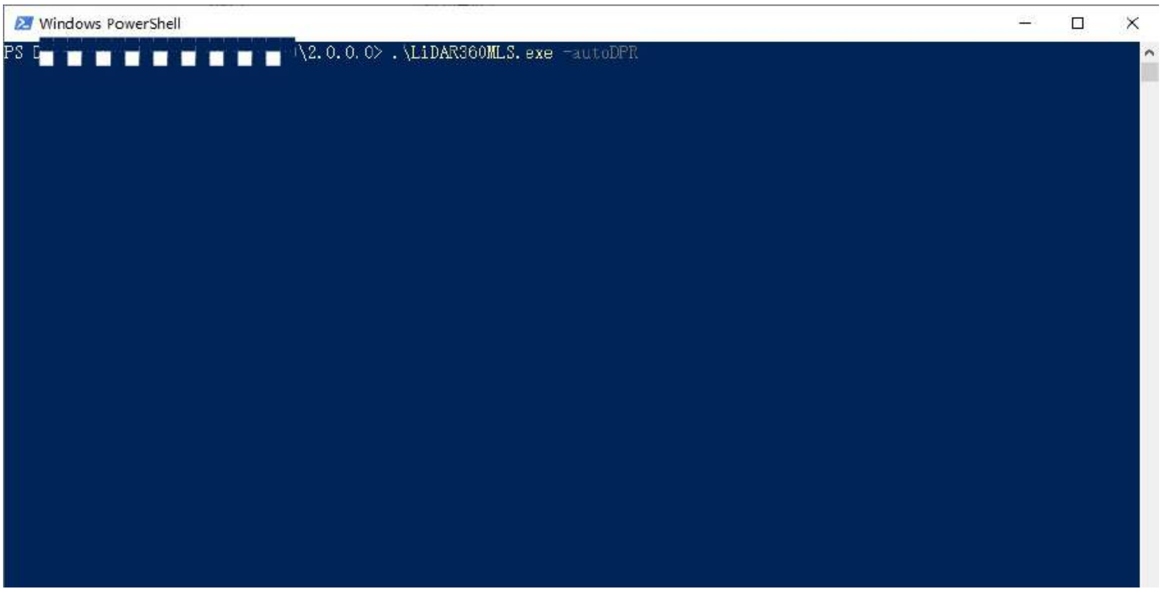
- Vector: Vector data (*.shp,* .dxf).

LiDAR360 MLS software crashes after startup?

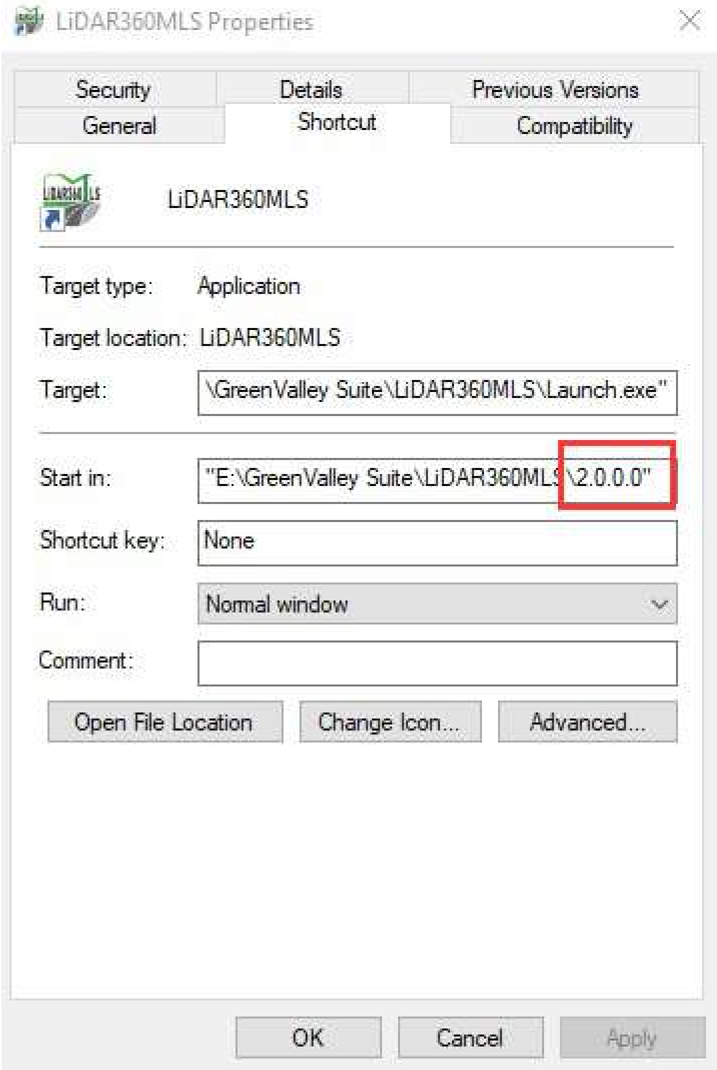
- Please check whether the computer screen is plugged in correctly. Normally, it should be plugged into the position of the graphics card connector of the host computer.

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


LiDAR360MLS Properties
Security
Details
Previous Versions
General
Shortcut
Compatibility
LiDAR360MLS
Target type: Application
Target location: LiDAR360MLS
Target: \GreenValley Suite\LiDAR360MLS\Launch.exe"
Start in: "E:\GreenValley Suite\LiDAR360MLS\2.0.0.0"
Shortcut key: None
Run: Normal window
Comment:
Open File Location
Change Icon...
Advanced...
OK
Cancel
Apply

LiDAR360 MLS version update log

V2.0

- Added modules and features
  - New Construction
    - Support drag and drop point cloud to create a new project, and save as to the specified path
    - Support for importing original projects of LiFuser-BP and LiGeo
    - Supports point cloud and panoramic data entry software for more than 90% of mobile surveying devices on the market in formats such as Trimble MX9, Leica Pegasus, Orbit Pos, etc.
  - Database
    - Support database for vector storage
    - Supports adding, deleting, modifying and querying layers in the database
  - Basic Platform
    - Added point cloud and vector projection conversion function
    - Support point cloud format conversion
    - Support basic tools such as point cloud selection and cropping
    - Support vector data format conversion
    - Support category display 0-255 categories
    - Supports cross-sectional export of color orthophotos
    - Support 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
      - Support domestic standard road facility symbol library
      - Support for common symbol libraries
      - Support symbol number storage
      - Support to set symbols for collected point features
      - Support for modification of symbols
    - 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 3D, elevation annotation
      - Support annotation style modification
    - Added layer label function
      - Support 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 template extraction function of strip features
    - Added single segmentation function
    - Added individual editing function
    - Added single 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
        - Support point cloud deep learning classification
        - Support polygon-based point cloud classification
        - Supports point cloud classification based on vector lines
        - Support air noise classification
        - Supports subsurface classification
        - Support above ground point classification
        - Support for separating lows
        - Support outlier classification
        - Support proximity point classification
        - Support ground point classification
        - Support classification by attribute
        - Support classify by cluster size
    - Point cloud extraction
        - Support extraction by category
        - Support extraction by elevation
        - Support extraction by intensity
        - Support to extract by GPS time
        - Support extraction by echo times
- 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
    - Support trajectory segmentation
    - Supports laser boresight
    - Support track quality check and repair
    - Added control point correction function
    - Added strip adjust
- 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
    - Support vertical vertical
    - Support horizontal guide line drawing
    - Support range line drawing
    - Supports taking a section based on a reference/extent line and making adjustments
    - Support the drawing of the basic elements of the facade
    - Support for construction line drawing
    - Support array drawing
    - Support translation and rotation
    - Support 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:
  - Add data block module
    - Support track segmentation
    - Support area block node editing
    - Support region block merging
    - Support point cloud segmentation based on block
    - Support area block selective display
    - Support display and hide of area blocks and their labels
      - Add more road feature template symbol library
    - Support all types of arrow reticle drawing
    - Support for manhole covers, parking spaces, and drainage grate to be drawn according to templates
    - Support traffic signs to be drawn according to templates
    - Support Chinese and English character drawing
    - Support 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
    - Support for modifying the shape of line features
    - Supports tracing other vectors when drawing vectors
    - Support point break line, line break line, line break surface
    - Support vector stickers
    - Precise selection when supporting 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
    - Add 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