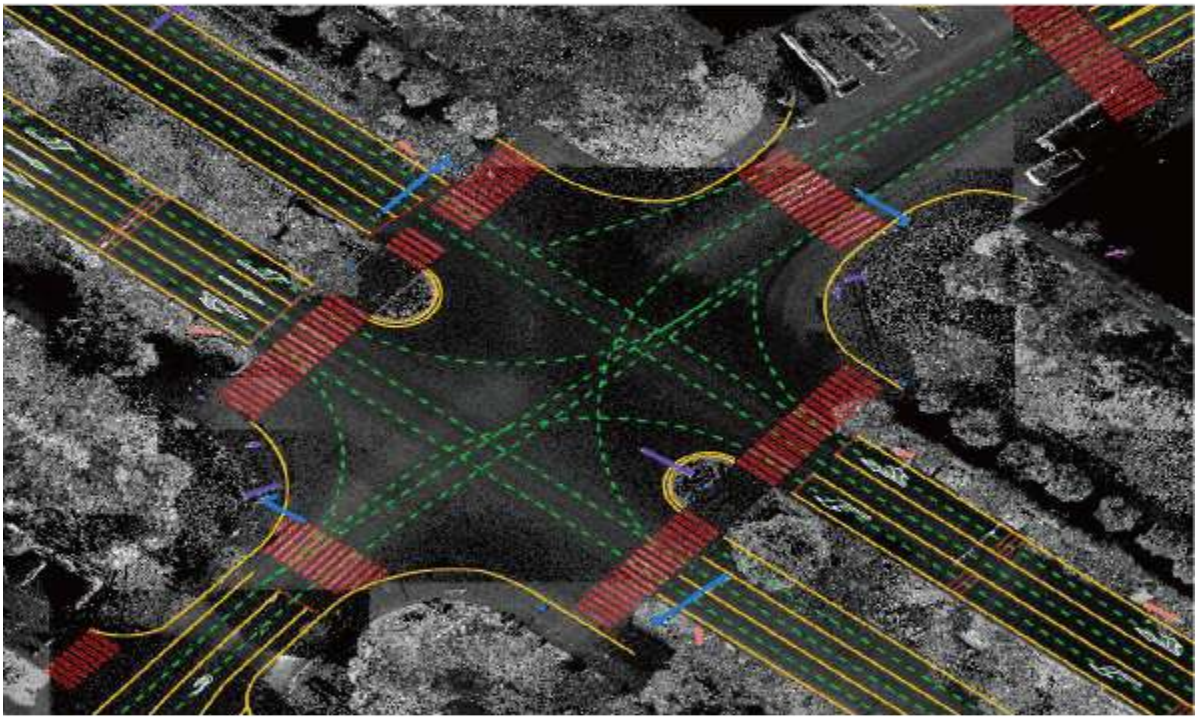
GreenValley International

LiDAR360 MLS User Guide

Point Cloud Feature Intelligent Extraction and Analysis Software



http://www.greenvalleyintl.com

Table of Contents

User Guide 1.1
Copyright Information 1.2
Introduction 1.3
Software Installation and Activation 1.4
File 1.5
    New Project 1.5.1
        Convert Data 1.5.1.1
        New mapping Project 1.5.1.2
        New SLAM Project 1.5.1.3
    New From Template 1.5.2
    Open Project 1.5.3
    Modify Project 1.5.4
    Batch Processing 1.5.5
    Save Project 1.5.6
    Save As Project 1.5.7
    Close Project 1.5.8
    Option 1.5.9
    About 1.5.10
    License 1.5.11
    Help 1.5.12
    Exit 1.5.13
Tools 1.6
    3D Measurement Tools 1.6.1
        Pick Point 1.6.1.1
        MultiPick Point 1.6.1.2
        Density Measurement 1.6.1.3
        Length Measurement 1.6.1.4
        Area Measurement 1.6.1.5
        Angle Measurement 1.6.1.6
        Gradient Measurement 1.6.1.7
        Height Measurement 1.6.1.8
        Volume Measurement 1.6.1.9
        Two Stage Volume 1.6.1.10
        Two Stage VolReport 1.6.1.11
        Hover 1.6.1.12

Select and Clipping 1.6.2
Polygon Selection 1.6.2.1
Rectangle Selection 1.6.2.2
Sphere Selection 1.6.2.3
Circle Selection 1.6.2.4
LineAbove Selection 1.6.2.5
LineBelow Selection 1.6.2.6
Plane 1.6.2.7
Subtract 1.6.2.8
Plane Above 1.6.2.9
Plane Below 1.6.2.10
Plane Inside 1.6.2.11
Plane Distance Settings 1.6.2.12
InCut 1.6.2.13
OutCut 1.6.2.14
Save And Cancel 1.6.2.15
Clip By Polygon 1.6.2.16
Cross Section 1.6.3
Cross Section 1.6.3.1
Horizon Section 1.6.3.2
Roaming 1.6.4
Import 1.6.5
Import Vector 1.6.5.1
Import TXT 1.6.5.2
Import GPKG 1.6.5.3
Export 1.6.6
Orthophoto 1.6.6.1
Point Cloud Convert 1.6.6.2
Export Vector 1.6.6.3
Unroll 1.6.7
Unroll Polygonal Section 1.6.7.1
Unroll Curved Section 1.6.7.2
Unroll Cylindrical Section 1.6.7.3
Catch 1.6.8
Point Cloud Tools 1.7
Point Cloud 1.7.1
SOR Filter 1.7.1.1
Noise Filter 1.7.1.2
SubSample 1.7.1.3

Merge Data 1.7.1.4
Normalization By Ground. 1.7.1.5
DeNormalization 1.7.1.6
Compute Normals 1.7.1.7
Transform GPS Time 1.7.1.8
Brightness Contrast 1.7.1.9
Define Projection 1.7.1.10
Geoid Model 1.7.1.10.1
Projection 1.7.1.11
Transform 1.7.1.12
Creating Conversion Relationships 1.7.1.12.1
Extract 1.7.2
Extract By Elevation 1.7.2.1
Extract By Intensity 1.7.2.2
Extract By Return 1.7.2.3
Extract By Time 1.7.2.4
Extract By Class 1.7.2.5
Extract By Additional Attributes 1.7.2.6
Point Cloud Classification 1.8
Classify by Attribute 1.8.1
Classify Ground Points 1.8.2
Classify AirPoints 1.8.3
Classify by Cover 1.8.4
Classify by ClusterSize 1.8.5
Classify Below Surface Points 1.8.6
Classify by Height Above 1.8.7
Classify Low Points 1.8.8
Classify Isolated Points 1.8.9
Classify Closeby Points 1.8.10
Classify by Polygons 1.8.11
Classify by Polylines 1.8.12
Cylinder Detection 1.8.13
Classify by RegionGrow 1.8.14
Classify by Deeplearning 1.8.15
Classify 1.8.16
Custom Deep Learning Classification 1.8.17
Install 1.8.17.1
Point Cloud Full Supervised Labeling And Weak Supervised Labeling Process 1.8.17.2

Prepare Point Cloud Training Data 1.8.17.3
Train Point Cloud Classification Model 1.8.17.4
Classify Point Cloud Using Trained Model 1.8.17.5
Report 1.8.17.6
Profile 1.9
Select Profile Region 1.9.1
Fixed Buffer 1.9.1.1
Measure Tools 1.9.2
Move Selected Section 1.9.2.1
Rotate Selected Section 1.9.2.2
Expand Selected Section 1.9.2.3
Switch View 1.9.2.4
Manual Classification 1.9.2.5
Panoramic Measurement Tools 1.10
Point Cloud and Panoramic Display Roaming 1.10.1
Panoramic Measurement Based on Depth Interpolation 1.10.2
Pick Point 1.10.2.1
Pick MultiPoint 1.10.2.2
Measure Length 1.10.2.3
Measure Height 1.10.2.4
Measure Angle 1.10.2.5
Panoramic Measurement Based on Forward Intersection of Images 1.10.3
Pick Point 1.10.3.1
Measure Length 1.10.3.2
Result Panel 1.10.4
Setting Dialog 1.10.5
Update EXIF 1.10.6
BlurPlates And Face 1.10.7
Planar Images 1.11
Point Cloud and Planar Images Roaming 1.11.1
Planar Measurement 1.11.2
Pick Point 1.11.2.1
Pick MultiPoint 1.11.2.2
Measure Length 1.11.2.3
Measure Height 1.11.2.4
Measure Angle 1.11.2.5
Image Undistort 1.11.3
Setting Dialog 1.11.4
Update EXIF 1.11.5

- BlurPlates And Face 1.11.6
- Preprocessing 1.12
  - Trajectory 1.12.1
    - Trajectory Segmentation 1.12.1.1
    - Segment Table 1.12.1.2
    - Trajectory Graph 1.12.1.3
    - Trajectory Quality Datection 1.12.1.4
    - Trajectory Burst Repair 1.12.1.5
  - Segmentation 1.12.2
    - Colorize By Segments 1.12.2.1
    - Spli By Segments 1.12.2.2
  - Calibration 1.12.3
    - Boresight 1.12.3.1
  - GCP Adjustment 1.12.4
  - Strip Adjust 1.12.5
  - Colorization 1.12.6
- CutBlock 1.13
  - Subdivision 1.13.1
    - Point Cloud Subdivision 1.13.1.1
    - Point Cloud Cut Block 1.13.1.2
    - Create Along Trajectory 1.13.1.3
    - Create Along Center Line 1.13.1.4
  - Cut Block 1.13.2
    - Cut Block Edit 1.13.2.1
    - Split Point Cloud 1.13.2.2
  - Block Display 1.13.3
- Map Element 1.14
  - Road Shape 1.14.1
    - Add Virtual Lane Centerline 1.14.1.1
    - Add Lane SolidLines 1.14.1.2
    - Add Lane DashLines 1.14.1.3
    - Add StopLine 1.14.1.4
    - Add Road Side Line 1.14.1.5
    - Generate Virtual Lane Centerline 1.14.1.6
    - Solid Lane Detecter 1.14.1.7
    - Dash Lane Detecter 1.14.1.8
    - Road Side Detecter 1.14.1.9
  - RoadSurface 1.14.2

Straight Arrow 1.14.2.1
Left Arrow 1.14.2.2
Right Arrow 1.14.2.3
Left And Right Arrow 1.14.2.4
Straight and Left Arrow 1.14.2.5
Straight and Right Arrow 1.14.2.6
Straight and Left and Right Arrow 1.14.2.7
Turn Around Arrow 1.14.2.8
Straight and Turn Around Arrow 1.14.2.9
Left and Turn Around Arrow 1.14.2.10
Left Merge Arrow 1.14.2.11
Right Merge Arrow 1.14.2.12
Right and Turn Round 1.14.2.13
Straight and Right and Turn Round Arrow 1.14.2.14
Left and Right and Turn Round Arrow 1.14.2.15
CrossWalk 1.14.2.16
Longitudinal Speed Reduction 1.14.2.17
Speed Reduce Mark 1.14.2.18
CrossWalk Warning 1.14.2.19
NoU-turn 1.14.2.20
Parking Area 1.14.2.21
BicycleLane 1.14.2.22
Add Font 1.14.2.23
Template Matching 1.14.3
Template Matching Parameter 1.14.3.1
Template Matching Tables 1.14.3.2
Template Matching Auto Match 1.14.3.3
Template Matching Manual 1.14.3.4
Template Matching Data Base 1.14.3.5
New Template 1.14.3.6
Extraction by DeepLearning 1.14.4
Road Marking 1.14.4.1
Road Vector 1.14.4.2
Merge To Lane 1.14.4.3
Facility 1.15
Road Structure 1.15.1
Pole Struct Detection 1.15.1.1
One Point Mode 1.15.1.1.1
Two Point Mode 1.15.1.1.2

Polygon Model 1.15.1.1.3
Grip To Ground 1.15.1.1.4
PowerLine Detection 1.15.1.2
Add Traffic Sign 1.15.1.3
Add Parking Spot 1.15.1.4
Add Rain Strainer 1.15.1.5
Add Manhole 1.15.1.6
Road Facility Detection 1.15.2
Linear Facility Extraction 1.15.3
RailWay Detection 1.15.4
Pole Seed Editor 1.15.5
Object Editor 1.15.6
Segmentation 1.15.6.1
Editor 1.15.6.2
Measurement 1.15.6.3
Vector Editor 1.16
Draw Map 1.16.1
Point 1.16.1.1
MutiPoints 1.16.1.2
Line 1.16.1.3
SegLine 1.16.1.4
Arc 1.16.1.5
Polygon 1.16.1.6
Rect 1.16.1.7
Parallelogram 1.16.1.8
Circle 1.16.1.9
Hexagon 1.16.1.10
Octagon 1.16.1.11
EditElement 1.16.2
Select 1.16.2.1
Pickup Vector 1.16.2.1.1
Rect Pickup Vector 1.16.2.1.2
Pickup by Line 1.16.2.1.3
Pickup by Polygon 1.16.2.1.4
Pickup by Lasso 1.16.2.1.5
Pickup by Circle 1.16.2.1.6
Pickup by Box 1.16.2.1.7
Pickup by Sphere 1.16.2.1.8

Pickup by Cylinder 1.16.2.1.9
Interrupt 1.16.2.2
Point Split Line 1.16.2.2.1
Line Split Line 1.16.2.2.2
Line Split Polygon 1.16.2.2.3
Line Split Objects 1.16.2.2.4
Layer Split Layer 1.16.2.2.5
Alignment 1.16.2.3
Move Vector 1.16.2.3.1
Parallel Copy Vector 1.16.2.3.2
Rotate Vector 1.16.2.3.3
Scale Vector 1.16.2.3.4
Mirror Vector 1.16.2.3.5
Reshape 1.16.2.4
ReShape Line 1.16.2.4.1
Reshape Rect 1.16.2.4.2
Merge Lines 1.16.2.4.3
Break Line 1.16.2.4.4
Split Lines 1.16.2.4.5
Extend or Trim 1.16.2.4.6
Split and Merge Polygon 1.16.2.4.7
Hook Lines 1.16.2.4.8
Extend Lines 1.16.2.4.9
Fillet 1.16.2.4.10
Combination Polygons 1.16.2.4.11
Smooth 1.16.2.4.12
Point To Line 1.16.2.4.13
Line To Points 1.16.2.4.14
Construct Polygon 1.16.2.4.15
Intersection Line 1.16.2.4.16
Multipart To Singleparts 1.16.2.4.17
Auxiliary 1.16.2.5
Grip To Ground 1.16.2.5.1
Line To Polygon 1.16.2.5.2
Polygon To Line 1.16.2.5.3
Draw By Intersection 1.16.2.5.4
Perpendicular Draw 1.16.2.5.5
Edit Vertices 1.16.3
Auxiliary Line 1.16.4

- Add H/V Lines 1.16.4.1
- Add Gridlines 1.16.4.2
- Add Lines 1.16.4.3
- Pick Lines 1.16.4.4
- Quality Check 1.16.5
  - Attribute Check 1.16.5.1
  - Geometry Quality Check 1.16.5.2
- Floor Plan 1.17
  - Auto Vectorization 1.17.1
  - Measurement 1.17.2
  - Vector Editor 1.17.3
    - Line 1.17.3.1
    - Draw By Intersection 1.17.3.2
    - Perpendicular Draw 1.17.3.3
    - Merge Lines 1.17.3.4
    - Break Lines 1.17.3.5
    - Line Intersection 1.17.3.6
    - Move Vector 1.17.3.7
  - Node Editor 1.17.4
  - Close Line 1.17.5
  - DXF 1.17.6
  - BluePrint 1.17.7
- FacadeSurvey 1.18
  - Set Vertical 1.18.1
  - Base line 1.18.2
  - Boundary Line 1.18.3
  - Profile 1.18.4
  - Drawing 1.18.5
  - TectonicCollect 1.18.6
  - Array 1.18.7
  - Edit 1.18.8
  - Export 1.18.9
- Appearance 1.19
  - LabelStyle 1.19.1
  - Symbol 1.19.2
  - Annotation 1.19.3
  - LabelLayers 1.19.4
- Road Analysis 1.20

- Road Condition Analysis 1.20.1
  - Clearance Analysis 1.20.1.1
  - Clearance Analysis Data Manager 1.20.1.2
  - Export Report 1.20.1.3
  - Bridge Clearance 1.20.1.4
  - Export Report 1.20.1.5
  - Height Limit Analysis 1.20.1.6
  - Export Report 1.20.1.7
- Pavement Damage Analysis 1.20.2
  - Detect Damage 1.20.2.1
  - Pavement Condition Index 1.20.2.2
  - Export Report 1.20.2.3
- Viewshed Analysis 1.20.3
  - Observation 1.20.3.1
  - Target 1.20.3.2
- Road Section Analysis 1.20.4
  - Road Reference Line 1.20.4.1
  - Generate Section Views 1.20.4.2
  - Show Or Hide 1.20.4.3
  - Clear Sections 1.20.4.4
  - Section 3D Windows Control 1.20.4.5

Road Surface 1.21

- Classification Tools 1.21.1
  - Classify By Attribute 1.21.1.1
  - Classify Ground Points 1.21.1.2
  - Classify by Deeplearning 1.21.1.3
- Manual Classification 1.21.2
- Terrain Creation 1.21.3
  - Road Surface Key Points By Grid 1.21.3.1
  - Create TIN 1.21.3.2
- Display 1.21.4
- Vehicle Passability Analysis 1.21.5
  - Vehicle Passability Analysis 1.21.5.1
  - Vehicle Designer 1.21.5.2

Urban Forestry 1.22

- Classification Tools 1.22.1
  - Classify By Attribute 1.22.1.1
  - Classify by Deeplearning 1.22.1.2
- Manual Classification 1.22.2

Normalization By Ground 1.22.3
Seed Editor 1.22.4
Seed Point Conversion 1.22.4.1
Update 1.22.4.2
Editor 1.22.4.3
Calculator 1.22.5
DBH And Height Calculation 1.22.5.1
Azimuth 1.22.5.2
Tree Editor 1.22.6
Segmentation 1.22.6.1
Editor 1.22.6.2
Measurement 1.22.6.3
Export Report 1.22.6.4
OtherTools 1.22.6.5
Settings 1.22.6.6
Display 1.22.6.7
DeNormalization 1.22.7
Ecological Landscape Analysis 1.22.8
Thematic Map 1.22.8.1
Export Report 1.22.8.2
Visual Analysis 1.22.8.3
Viewing Tools 1.23
Top View 1.23.1
Bottom View 1.23.2
Left View 1.23.3
Right View 1.23.4
Front View 1.23.5
Back View 1.23.6
Front Isometric View 1.23.7
Back Isometric View 1.23.8
Full Extent 1.23.9
Set View Mode 1.23.10
Configure Point Size And Type 1.23.11
Screen Capture 1.23.12
Color Tools 1.24
Display By Height 1.24.1
Display By Intensity 1.24.2
Display By Classification 1.24.3

Display By RGB 1.24.4
Display By Return Number 1.24.5
Display By GPSTime 1.24.6
Display By EDL 1.24.7
Display By Xray 1.24.8
Attributes 1.25
Start Page 1.25.1
Layer Management 1.25.2
Point Cloud 1.25.2.1
Image List 1.25.2.2
Trajectory 1.25.2.3
Vector 1.25.2.4
Table 1.25.2.5
Raster 1.25.2.6
Layer Manager 1.25.3
Window Management 1.25.4
View Mode 1.25.5
Feature Property 1.25.6
Attribute Table 1.25.7
Status Bar 1.25.7.1
Field Calculator 1.25.7.2
Export Image 1.25.7.3
Appendix 1.26
Option 1.26.1
Display Options 1.26.2
Appendix High Performance Graphics Mode Adjustment 1.26.3
FAQ 1.26.4
Version Update Description 1.26.5

Copyright

Greenvalley International Co., Ltd.

LiDAR360MLS Point Cloud Feature Intelligent Extraction and Analysis Software

User Guide

Imprint and Version

Document Version 7.2

Copyright © GreenValley International, Ltd. All rights reserved. This document may be copied and printed only in accordance with the terms of the Frame License Agreement for End Users of the related LiDAR360MLS software.

Published by:

GreenValley International, Ltd.

Website: https://greenvalleyintl.com

Dear Users,

Thank you for using LiDAR360MLS Point Cloud Intelligent 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

GreenValley® and LiDAR360MLS® are the registered trademarks of GreenValley International. All other product names, company names, and brand names mentioned in this document may be trademark properties of their respective holders.

All rights reserved.

© 2022-2024 Greenvalley International Co., Ltd.

Date printed: 26th Mar 2024

Introduction

LiDAR360MLS is a Point Cloud Feature Intelligent Extraction and Analysis Software developed by Beijing Digital Green Earth Technology Co., Ltd.

The main functions of the software include：

- File
- Tools
- Point Cloud Tools
- Classification
- Preprocessing
- CutBlock
- Map Element
- Floor Plan
- Facility
- Vector Editor
- Facade Survey
- Road Analysis
- Road Surface
- Urban Forestry
- 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.

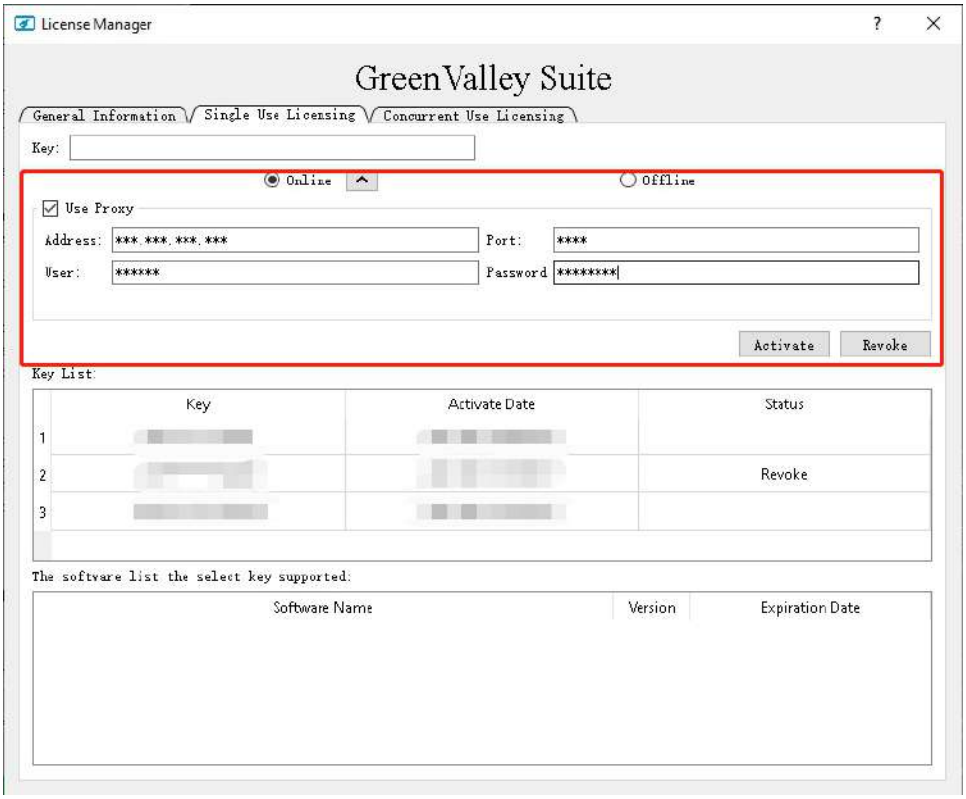
2）License key

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.

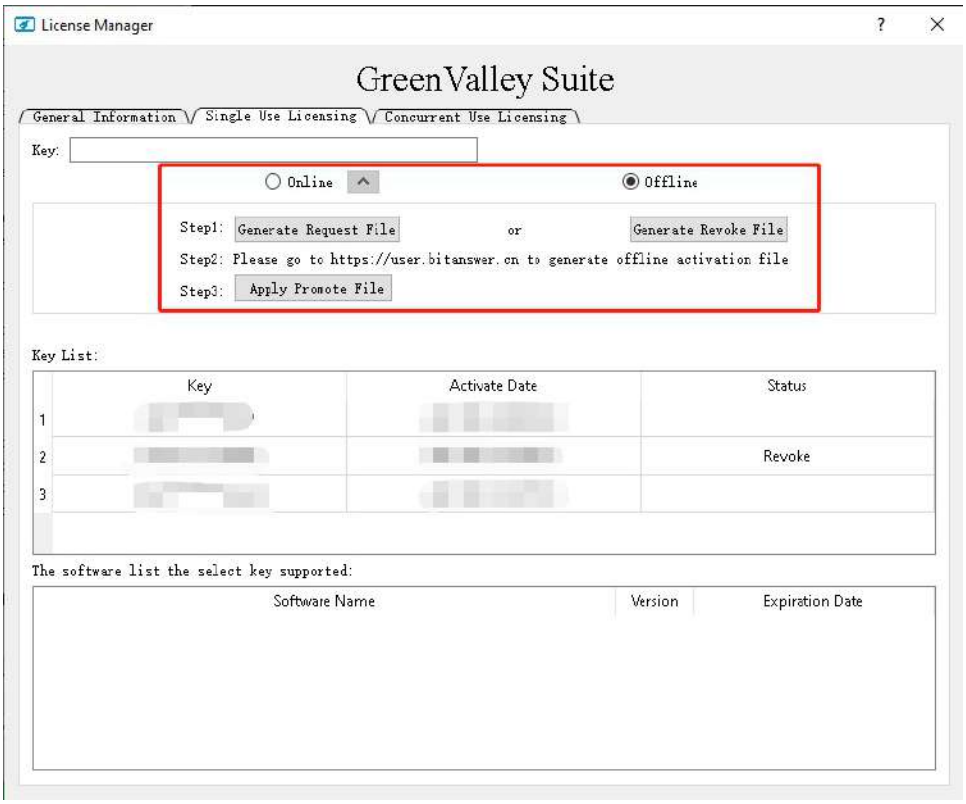
1. Run LiDAR360 MLS, the license management dialog box will appear.
2. Fill in the name and company name, select the module that needs to be activated, and then click "Copy".
3. Paste the copied information into the body of the email and send it to info@lidar360.com.
4. The software authorization mode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



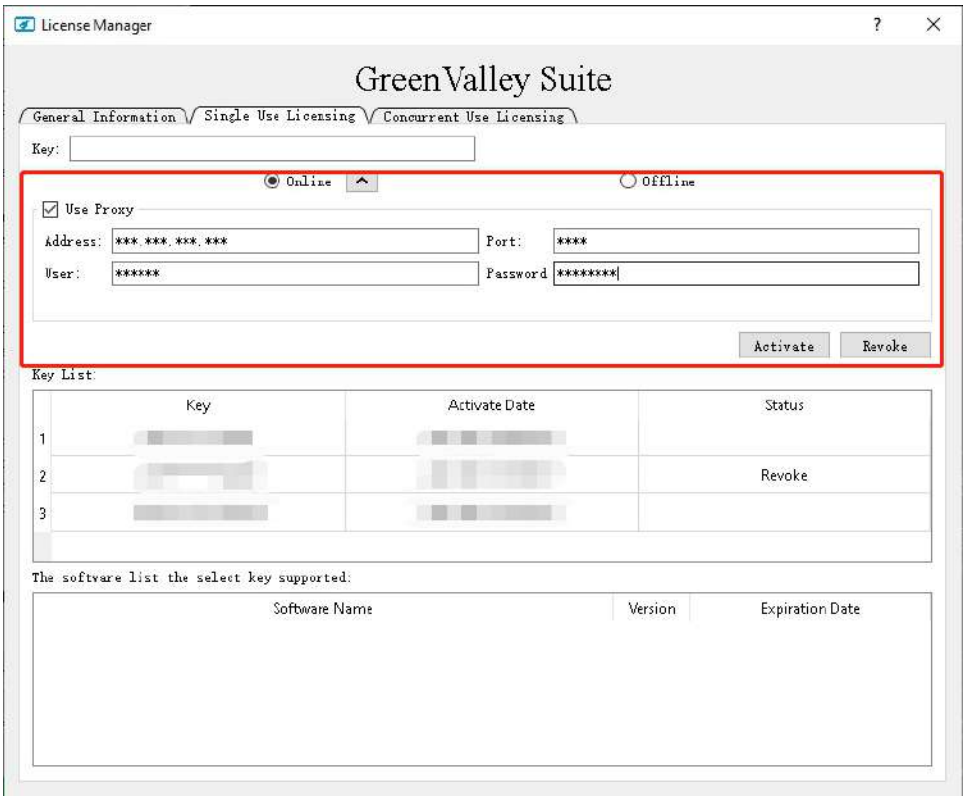
Offline Activatin/Update: Enter the key under Single Use License tab, select "Offline", and click"Generate request file"(.req). Use a computern with internet access, type https://user.bitanswer.cn in an internet browser, enter the license key to log in and click "Offline update". Upload .req file, and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



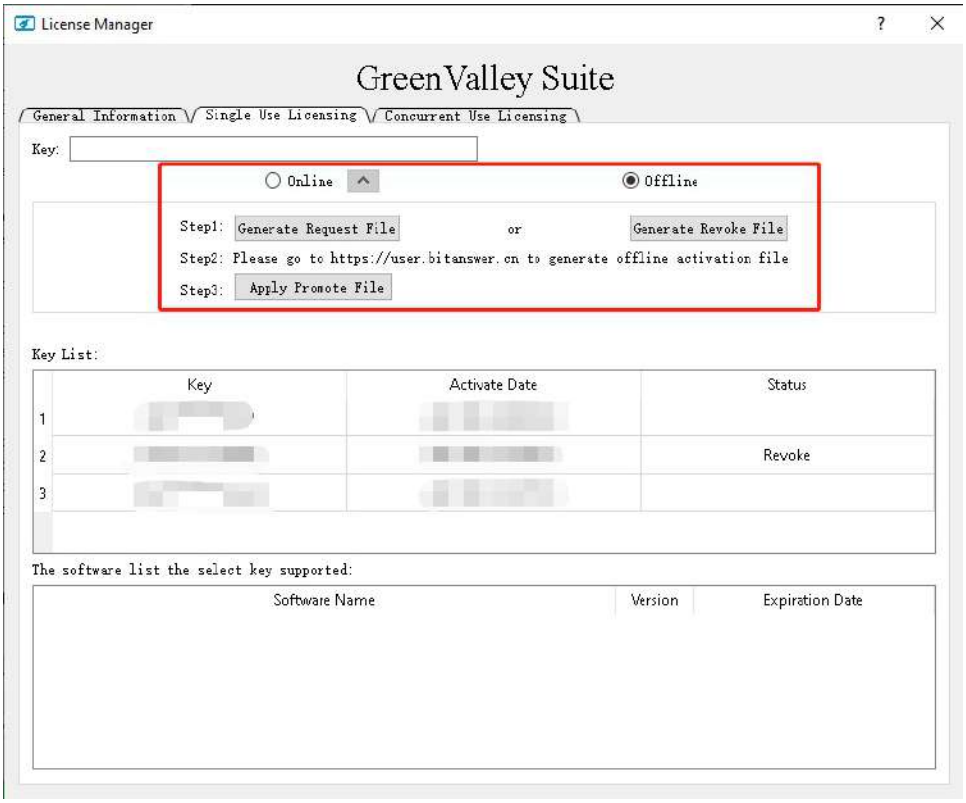
- Revoke

  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.

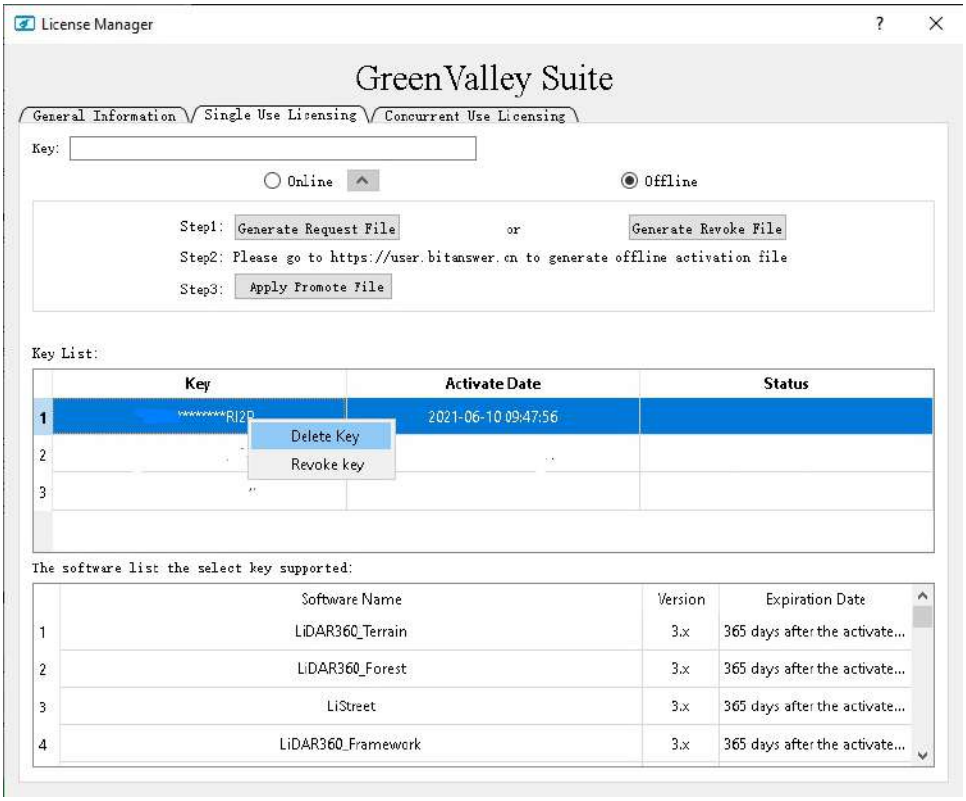


Revoke offline: Enter the authorization code, select "Offline", click "Generate Move Out File" to generate move out request file (.req). With the help of a computer that can be connected to the Internet, enter https://user.bitanswer.cn in the browser, enter the authorization code to log in, click Offline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



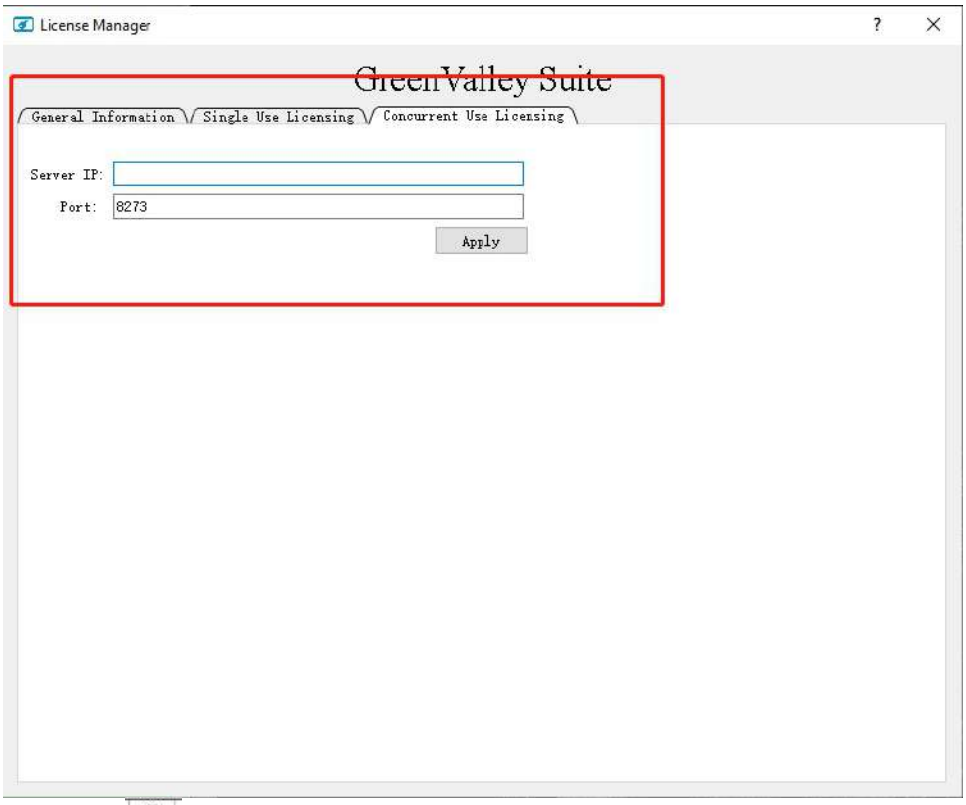
- Delete

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:

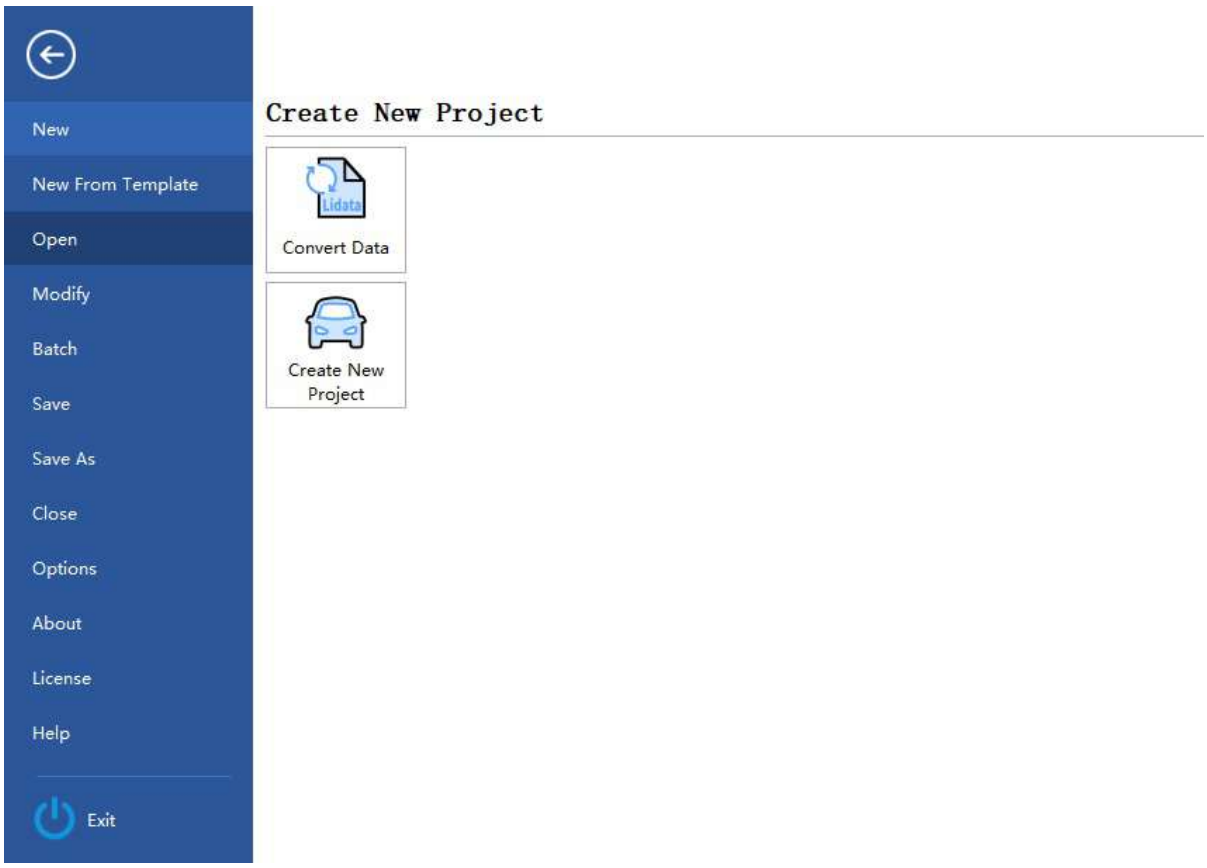
1.Click **Display > Language > English, Chinese**.

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

The File page includes the following functionalities:

- New
  - Convert Data
  - New mapping Project
  - New SLAM Project
- New from Template
- Open
- Modify
- Batch Processing
- Save
- Save As
- Close
- Options
- About
- License
- Help
- Exit

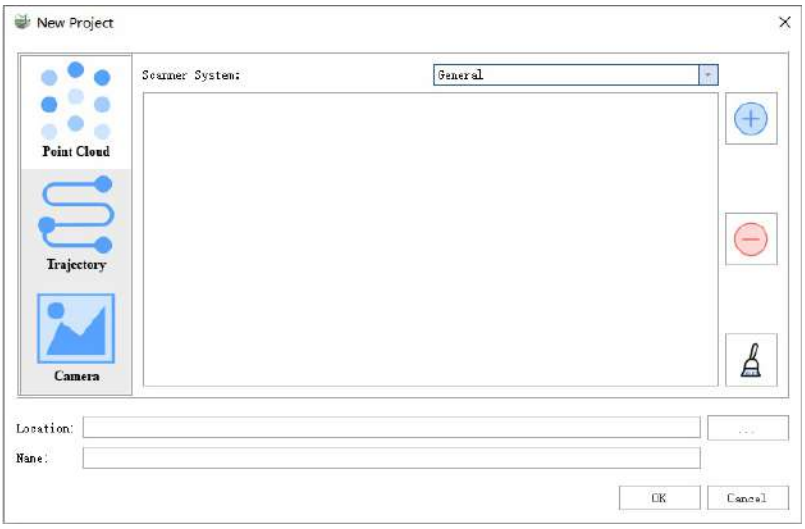


New Project

Feature Descriptiown: Create a LiDAR360MLS project based on the point cloud, trajectory, image, and other data.

Steps

After starting the software, click the **New Project** button to access the New Project Wizard to configure original data page:



> Before starting to process the data, it's recommended to gather all collected data into a fixed directory on the local disk of the computer. The directory name should be in English or in numerical characters, such as "LiDARData". The New Project Wizard mainly includes adding point cloud data, trajectory data (optional), panoramic/flat image data (optional), and setting the project file path.

Scanning System

> Scanning systems supported: General, Trimble, Lecia, Orbit, Riegl, LIDARUSA, Navis. Once the scanning system is set, the file format filter will be applied according to the scanning system set when selecting the camera file path.

Set Point Cloud File Path

> The software supports ***.LiData**, ***.las**, ***.laz**, ***.e57** point cloud files.

- **Add**: Add point cloud data files, multiple selections are supported
- **Delete**: Deletes the point cloud file path selected on the left
- **Clear**: Clears all point cloud file paths

Set Trajectory File Path

> The software supports ***.traj**, ***.pos**, ***.txt**, ***.csv**, ***.asc**, ***.xyz**, ***.pts**, ***.out** trajectory files

- **Trajectory File**: You can directly edit the box to input the trajectory path, or use the browse button on the right to choose the trajectory file.

- **Trajectory Format Selection**: You can choose to convert the trajectory format, including:
  - **None**: No format conversion.
  - **Original System and Target System**: Select the time format of the input trajectory file and the output time format, and choose the measurement date for conversion.
  - **Multiplication and Addition Constants**: Use a formula to linearly transform the time, output time = a * input time + b.
  - **Heading Angle Adjustment**: Perform meridian convergence angle conversion on the 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. Users can also set the coordinate system themselves.
- **Conversion Options**: Conversion options for the trajectory coordinate system, you can choose whether to use the seven parameters.

Set Camera File Path

- **Image Information**: Support for panoramic images and flat images.

Panoramic Image

- **Camera File**: Input panoramic image file

  > The software supports ***.imglist**, Leica Pegasus File ***.csv**, Trimble MX9 File ***.csv**, Riegl MMS File ***.csv**, LIDARUSA SNOOPY File ***.txt**, OrbitPos ***.txt** ***.csv** files.

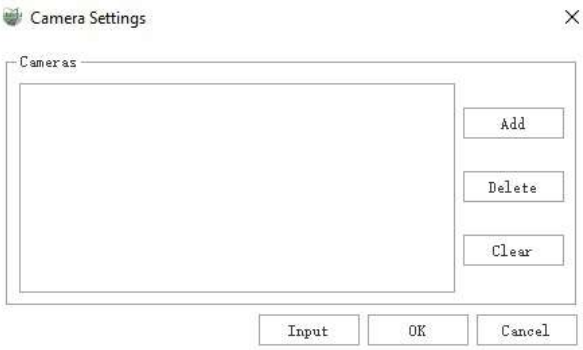
Flat Image

- **Camera File**: Input flat image file, supports suffix ***.imglist**, Leica Pegasus File ***.csv**, Trimble MX9 File ***.csv**, Riegl MMS File ***.csv**, LIDARUSA SNOOPY File ***.txt**, OrbitPos ***.txt** ***.csv** files.
- **Camera Parameters**: You can choose our company's cal calibration file as the parameter import, or you can click the add button in the camera parameters to manually add the calibration parameters of the flat camera, you need to fill in the camera ID, the internal parameters of the camera, the axis angle of the installation structure, and calibration parameters
  - **Camera ID**: When the device has multiple cameras, the camera ID is cumulative, corresponding to the CameraName field in the .imglist file
  - **Camera Internal Parameters**: f is the focal length; cx, cy are the offsets of the optical axis to the coordinate center of the projection plane; k1, k2, k3 are the coefficients of the Taylor series expansion, used to correct the radial distortion caused by the lens; p1, p2 are used to correct the tangential distortion caused by the lens; b1 and b2 are generally not required to be filled in.
  - **Installation Structure (Axis)**: The degree value of the camera rotating around the z, x, and y-axis to the coordinate system where the IMU is located.
  - **Calibration**: X, Y, and Z are the boom values of the camera, and roll, pitch, and heading are the camera installation angles.

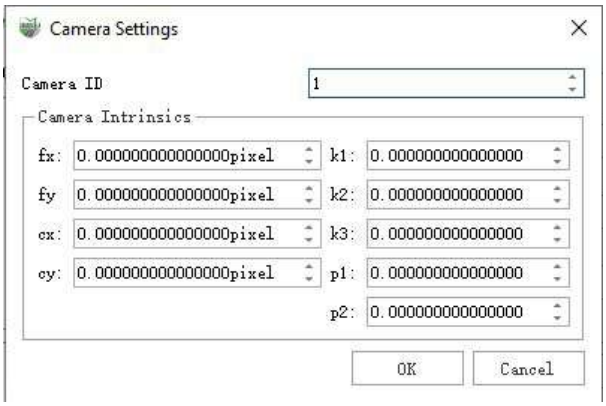
Steps

1.Select the flat image, and add the image file.

2.Click settings, and the **Camera Settings** dialog box pops up.



3.You can click the "Import" button to directly import the cal calibration file or the saved .ini file, or manually add the calibration parameters of the flat camera.



Manually add the calibration parameters of the flat camera

4.Click the OK button after the setting is completed.

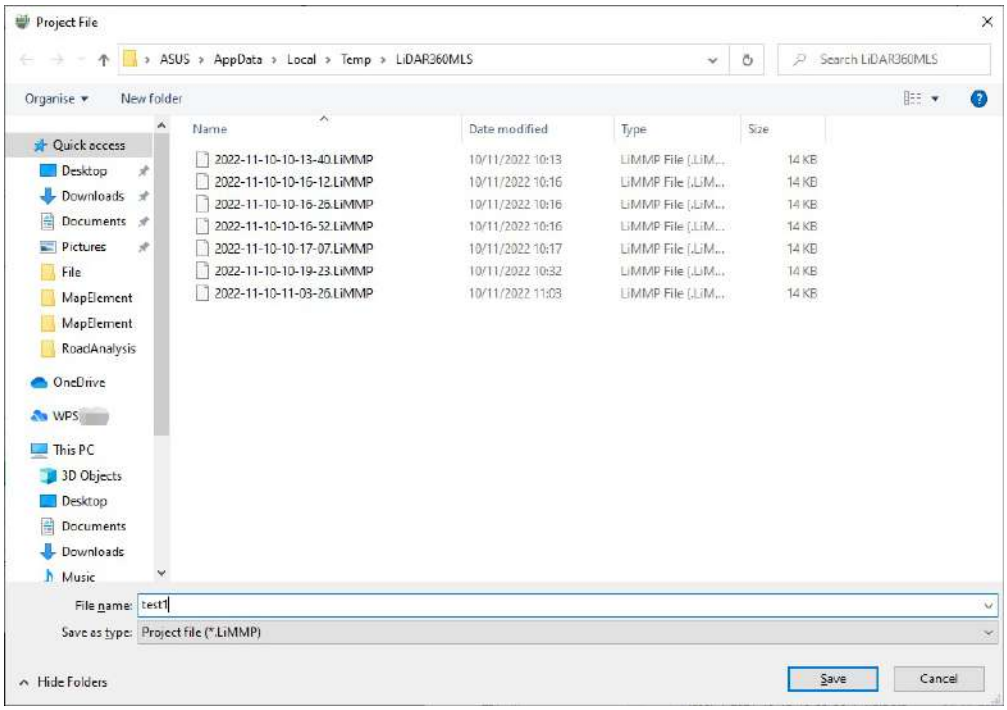
Set Project File Path

- **Location**: Set the folder where the project file is stored, which will be in the same level directory as the point cloud by default, and the path can also be changed.
- **Name**: Set the name of the project file (suffix .LiMMP), named by the current time by default.

Drag and Drop Point Cloud

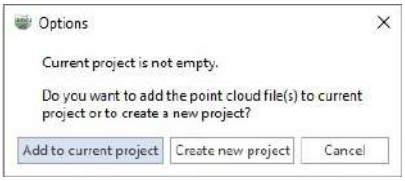
Feature Description: Complete the creation or modification of the project, by dragging and dropping several point cloud files onto the main window of the software and releasing the left mouse button, the relevant operation can be completed.

1.If the current project is empty, the software will automatically create a project in the temporary path based on the current project. During subsequent project saving, a save dialog box will pop up to set the official save path of the project. Once set, all subsequent operations are saved in the set path.



Project File

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



Options Dialog Box

- **Add to Current Project**: The software will add the selected point cloud files to the current project. This process will automatically filter out the point cloud files that have already been loaded into the current project;
- **Create 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 operations, and the dialog box automatically closes.

> Note: After the cut operation, the project cannot drag and drop point clouds into the project.

Data Conversion

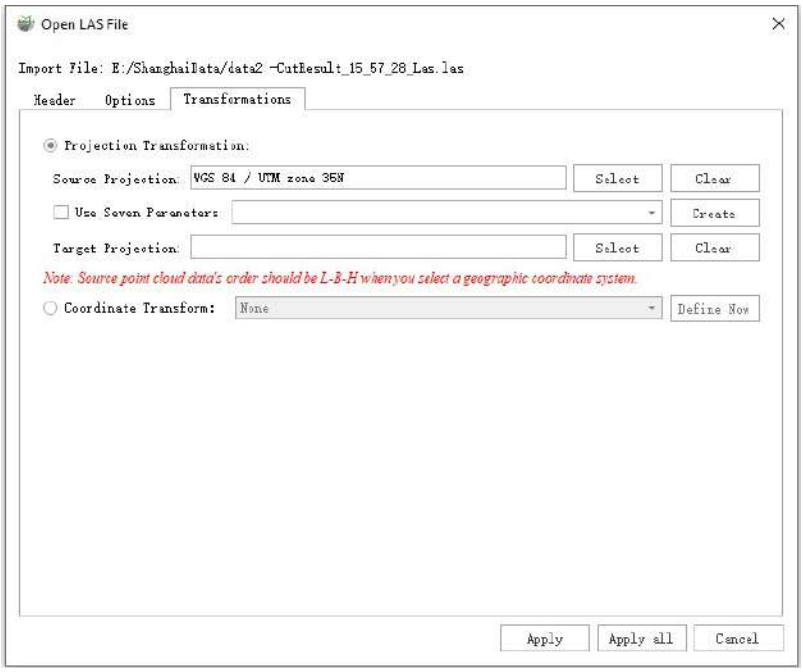
Feature Description: Convert point cloud data in *.las, *.ply, *.e57 formats into *.LiData format

Steps

1.After starting the software, click the **Data Conversion** button to access the data conversion page:

- **Add**: Add point cloud data files, multiple selections are supported.
- **Delete**: Deletes the point cloud file path selected on the left.
- **Clear**: Clears all point cloud file paths.

2.After clicking **OK**, a parameter setting interface for importing data will pop up:



- **Source Projection**：The software automatically reads the coordinate system of point cloud and vector files and displays it in the output box. Users can also manually select it by clicking the browse button.
- **Target Projection**：Click the browse button on the right, choose the target coordinate system to which you want to project, and you can set both horizontal and vertical coordinate systems.

- **Seven Parameters**: Add seven parameters to the projection relationship.
- **Coordinate Transform**: You can refer to Creating Conversion Relationships

New SLAM Project

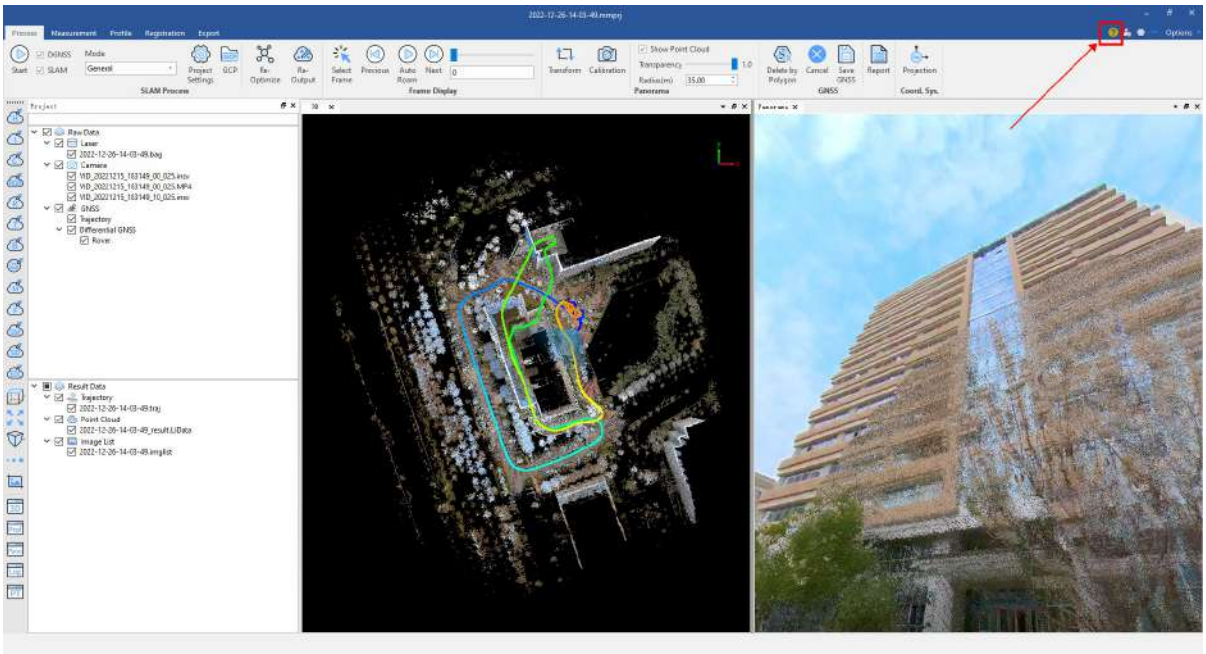
Feature Descriptiown：Based on the data collected by the LiGrip and LiBackpack devices, we perform data solving and splicing operations.

Steps

1.After launching the software, you can find the button related to SLAM new creation on the startup page or under the New group of files.

- New SLAM Project ：New single SLAM project can be created for data solving.
- New Merge Project ：Possibility to create new splicing projects by solving multiple projects
- New Betch Project ： New batch project for single-project solving.

2.Specific instructions on the different buttons can be found in the user manual when the SLAM project is opened.



Create a Project from a Template

Function Description：This function enables you to create a new LiDAR360MLS project based on a pre-configured layer template.

Steps

1.Configure the Template:

1.1 After launching the software, start by creating a project using the **New Project** button and configure it according to the guide.

1.2 Once the project has been successfully created and you've entered the software interface, click the **Layer Configuration** feature in the bottom left corner.

1.3 In the pop-up **layer configuration** dialog, configure the template layer, fields, and enumerated values. The specific operations are as follows:

Attribute table configuration interface

（1）**Parameter Settings:**

- **Layer**：The Layer tab allows you to modify the name of the built-in layers, and add/modify/delete custom layers.
- **Field**：The Field tab allows you to add attribute fields to the selected layer in the left-side layer tab and set the type of field values.
- **Value**：The Value tab allows you to set the values for the added fields, and simultaneously annotate the meaning of the current value.
- **Save**：Save the configured layer as a template, which is convenient for creating new projects based on this template in the future.
- **Export**：Export the configured layer, which is convenient for directly importing in the future, or creating a new project based on this template.

(2)**Operation Instructions:**

(2.1)**Layer Tab Settings**：In the Layer tab, **left-click** on the layer name in the list to modify and delete the **currently selected** layer and also add custom layers.

After selecting the operation layer, you can set fields and field values in the Field and Value tabs respectively.

- **Add Layer**: Input the name of the desired layer to be created in the Layer Name input box in the Layer tab, select the layer type from the Layer Type dropdown menu, and click the Add button to complete the addition of the layer. The added layer will subsequently be displayed as a custom layer in the directory tree. After adding the layer, you can set the field and field values (see later section on **Modify Fields & Modify Values**).
- **Modify Layer**: Left-click to select the layer, input the modified name in the Layer Name input box, select the modified layer type from the Layer Type dropdown menu, and click the Modify button to complete the layer modification. If you need to modify the color, line type, line width, etc., double-click the corresponding cell to modify it.

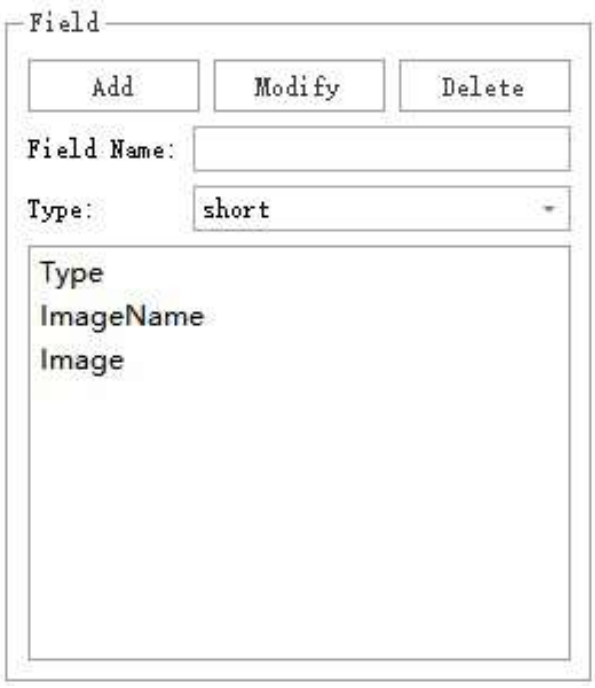
> Note:
>
> - Inherent layers cannot modify the layer type

- **Delete Layer**: Left-click to select the layer, then click the Delete button to delete the selected custom layer.

> Note:
>
> - Inherent layers cannot be deleted

(2.1)**Field Tab Settings**: After selecting the layer to be configured in the Layer tab, you can set the fields within the selected layer in the Field tab, including adding fields, modifying fields, and deleting fields.

> Note:
>
> - Inbuilt layer fields cannot be deleted or modified, they can only be added



Field tab

Parameter Instructions

- **Add**: Select the layer in the Layer tab where you want to add fields, enter the field name in the Field Name input box, set the field type in the Type dropdown menu, and click the **Add** button.

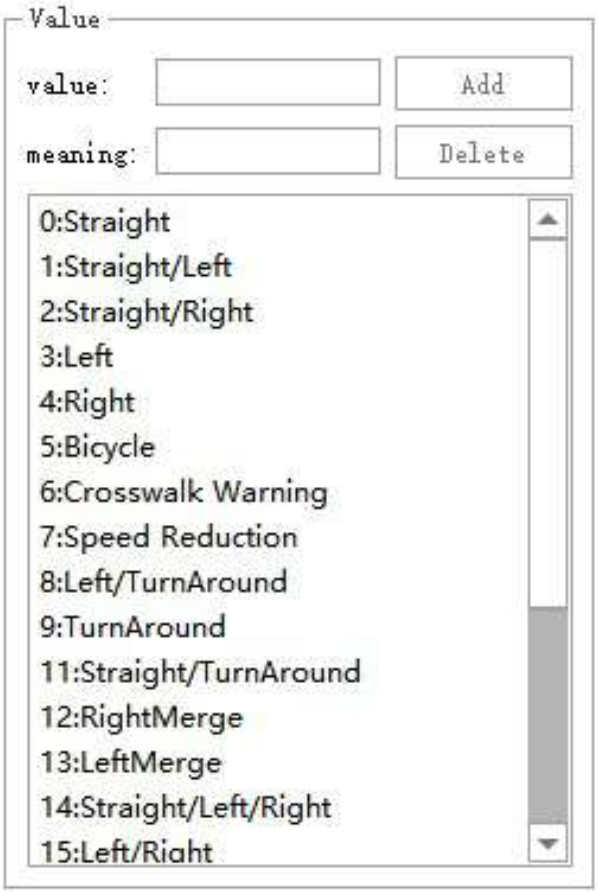
> Note:
>
> - The field name and type must be fully set before you can successfully add them.
> - **Modify**: Select the layer in the Layer tab where you want to modify the fields, then select the field name you want to modify in the field list, make the necessary modifications to the field name, field type, and field length, and click the **Modify** button.
>
> Note:
>
> - This modification pertains to the field's own properties (field name). If you want to modify the field value, refer to the Field Value Settings section.
> - Inbuilt layer fields are not modifiable

- **Delete**: Select the layer in the Layer tab where you want to delete fields, then select the field name you want to delete in the field list and click the **Delete** button.

> Note:
>
> - The delete button is disabled when trying to delete fields from an inherent layer, as they cannot be deleted.

- **Field Name**: The name of the field.
- **Type**: One of the field's attributes, indicating the storage type of the current field. The options include:
  - **short**: Indicates that the current field is stored as an integer type with a small value range.
  - **long**: Indicates that the current field is stored as an integer type with a large value range.
  - **float**: Indicates that the current field is stored as a decimal type, the data precision is small and is not suitable for storing coordinates and other high-precision values.
  - **double**: Indicates that the current field is stored as a decimal type, with high data precision, suitable for storing coordinate data.
  - **text**: Indicates that the current field is stored as a text type.
  - **date**: Indicates that the current field is stored as a date type.

（2.3）**Field Value Settings**: After selecting the layer and fields, you can set the field value and the corresponding interpretation in the Value tab, which is used to configure the enumerated values of a field.



Value tab

Parameter Instructions

- **Value**: The value of the selected field.
- **Meaning**: The meaning of the selected field value.
- **Add**: Enter a number in the Value text box, enter the meaning of the corresponding number in the Meaning text box below, then click the **Add** button on the right.

> Note:
>
> - You can only successfully add the value when both Value and Meaning have been set.
> - Once you have configured the enumerated value, any subsequent projects created based on the current template can directly populate fields according to the enumerated values and meanings. The value will be stored in the database.
> - If no field is selected in the Field tab, the Add button is disabled.

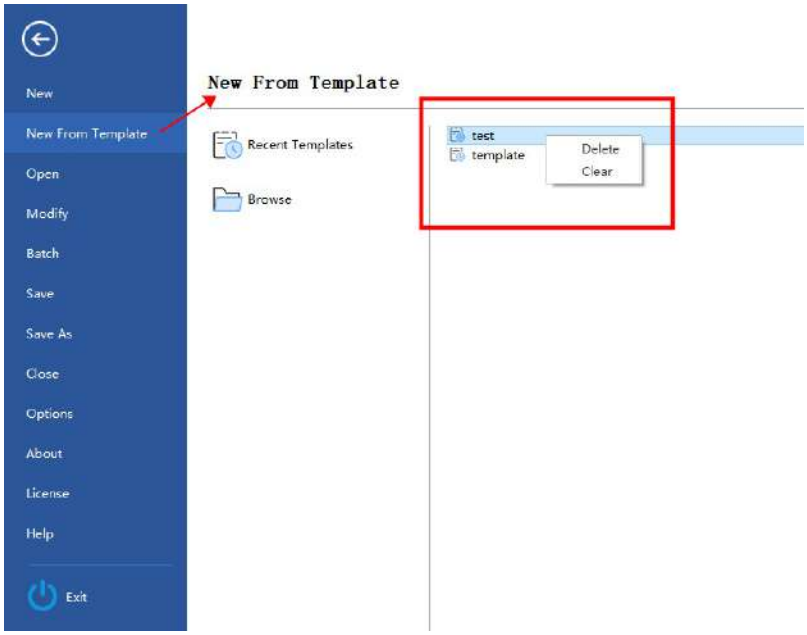
- **Delete**: Select a row record in the list below, then click **Delete** to delete the corresponding value.

(3)**Save**: Click the **Save** button in the lower right corner, select the desired save path in the pop-up browser dialog, and click **OK** to save the current configuration template.

> - To create a new project based on the currently saved template, go to File > Create from Template, click Browse, and select the template file saved last time.

2.Create From Template:

Once the template is set and saved, switch to **Create From Template** in the File menu page, click the template you need to use, and then configure the point cloud, trajectory, and image data for the new project according to the guide. The specific configuration can refer to the [New Project] operation.



Create a Project from a Template

Parameter Description:

- **Delete**: Delete the selected module.
- **Clear**: Clear all current templates.c

Clear all current templates

The most recently used templates are listed on the right side of the vertical line. Double-click on the template name to enter the subsequent project creation process.

Browse

After clicking the button, select the location of the template file in the pop-up file browsing dialog box to enter the project creation process based on the selected template.

> This operation automatically adds the currently selected template to the list of recent templates.

Open Project

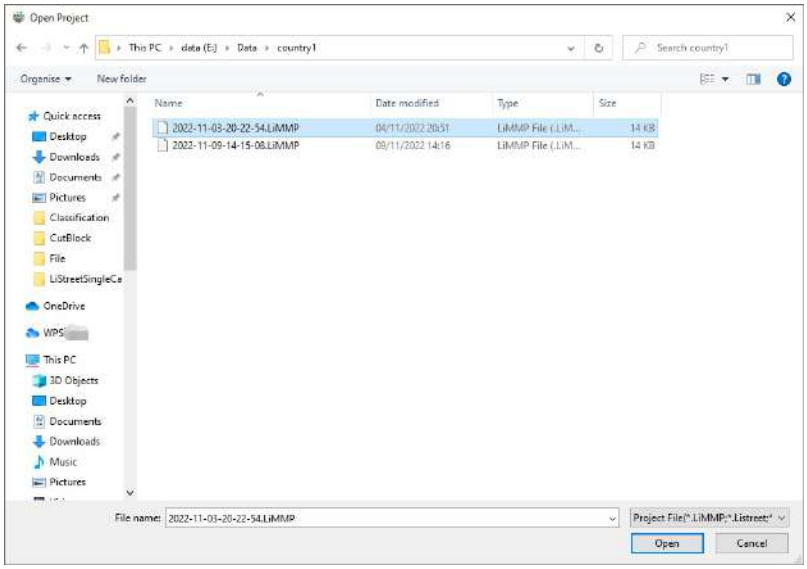
Function Description：Allows the user to launch the software. Click "Open", then select a recently saved project or open a project saved at a specific directory.

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

Steps

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

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



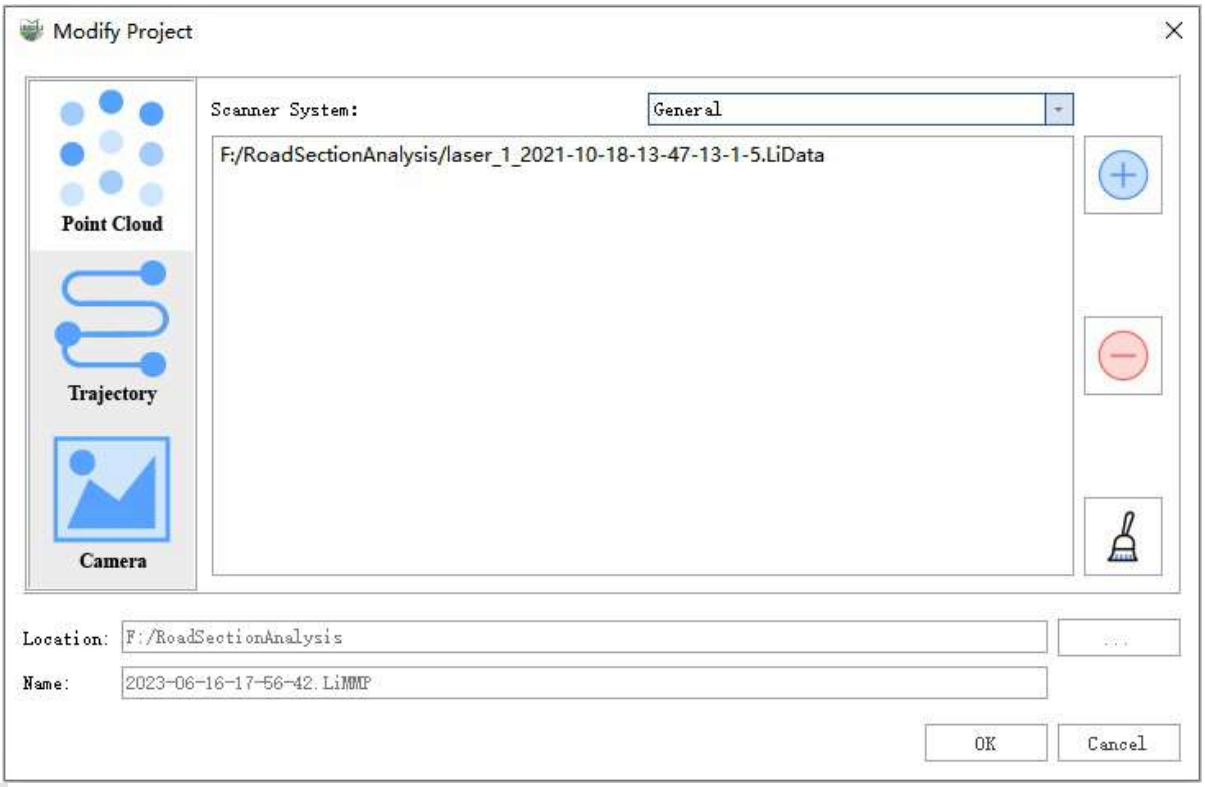
Modify/Organize Project

Function Description: By clicking on "Modify Project" in an opened project, you can modify the point cloud, trajectory, and image information of the current project.

Modify Project

1.Launch the software and open a project.

2.To make adjustments to the project information, click on "File" in the menu bar, then click on **Modify Project** on the right side. This allows you to add, delete, or clear the point cloud; add, delete, or adjust coordinate systems for trajectory files; and modify camera files and image information.



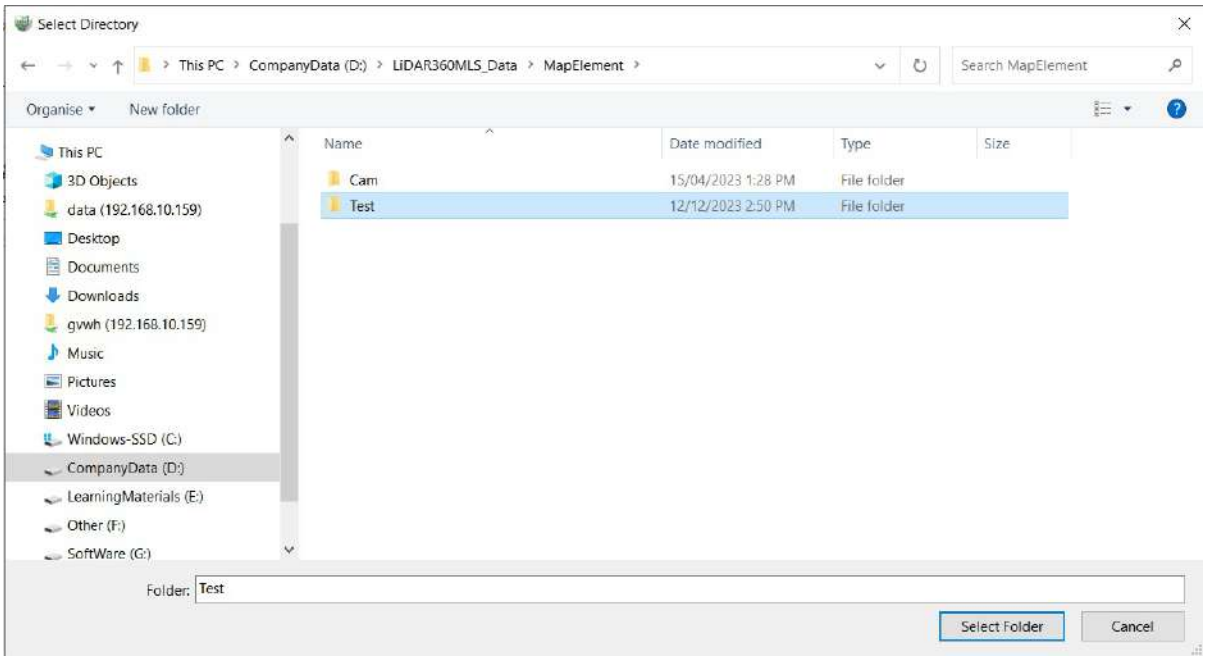
Note:

- Currently, modifying projects that have been divided into tiles is not supported.
- Although the project supports clearing the point cloud with a single click, if you clear the point cloud without adding any new points, the modified project cannot be saved and you will be prompted to "Please check the laser data path.

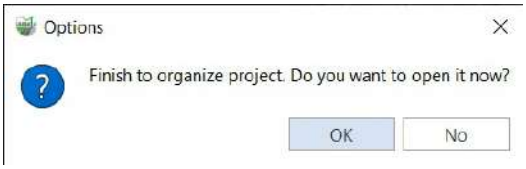
Organize Project

1.Launch the software and open a project.

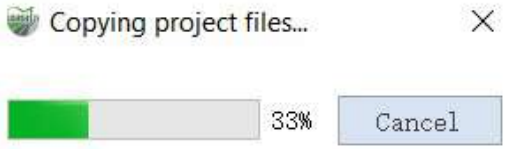
2.Click the 'Organize Project' button, and a dialog box will appear for selecting a path. Choose a path where you want to save the project.



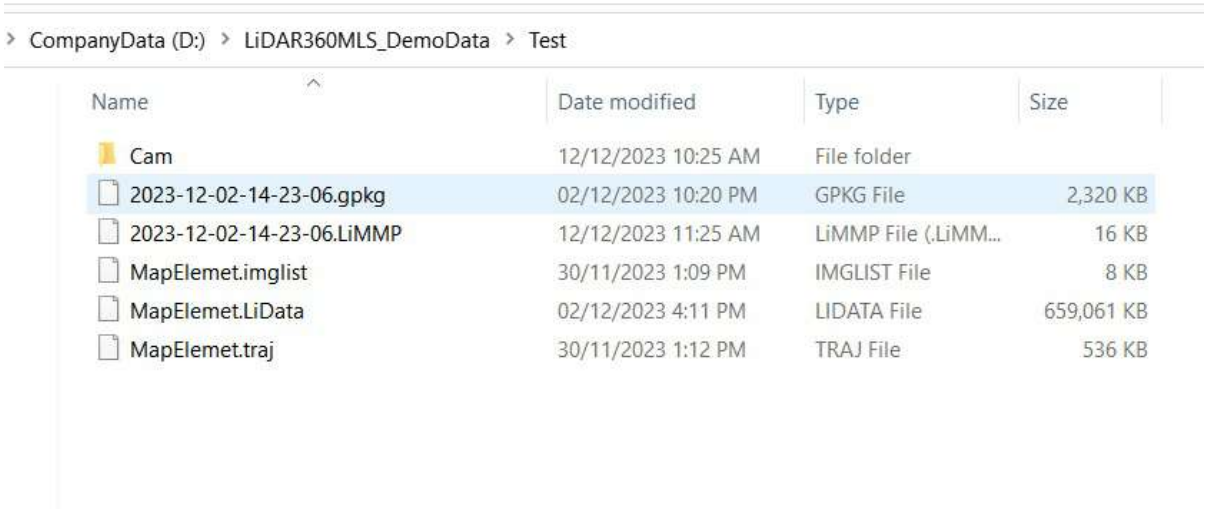
3.After clicking 'OK,' the software will automatically copy all associated source data and result data from the currently open project to the newly selected path.



4.After the copying is complete, the software will prompt whether to open the project in the new path. You can choose 'Yes' or 'No' based on your preferences.



When point cloud, trajectory, and image files are configured in the original project, the organized project folder will appear as shown in the following image:


CompanyData (D:) › LiDAR360MLS_DemoData › Test
Name
Date modified
Type
Size
Cam
12/12/2023 10:25 AM
File folder
2023-12-02-14-23-06.gpkg
02/12/2023 10:20 PM
GPKG File
2,320 KB
2023-12-02-14-23-06.LiMMP
12/12/2023 11:25 AM
LiMMP File (.LiMM...
16 KB
MapElemet.imglist
30/11/2023 1:09 PM
IMGLIST File
8 KB
MapElemet.LiData
02/12/2023 4:11 PM
LIDATA File
659,061 KB
MapElemet.traj
30/11/2023 1:12 PM
TRAJ File
536 KB

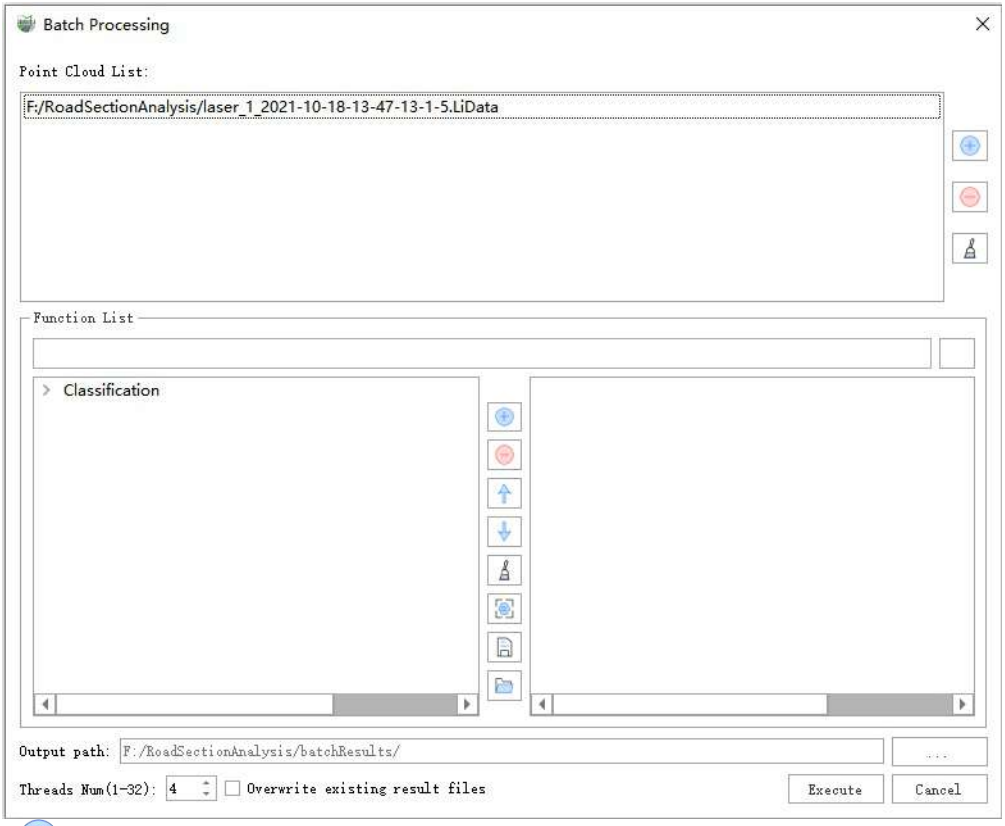
Batch Processing

Function Description

This function supports multithreaded batch processing for point cloud data, implementing multiple functionalities in a streamlined workflow. It supports LiData type data and offers two invocation methods: via dialogue box and command line. The dialogue box invocation also allows for saving the order of operations and parameter settings for future use with other data. The steps to use both invocation methods are detailed below.

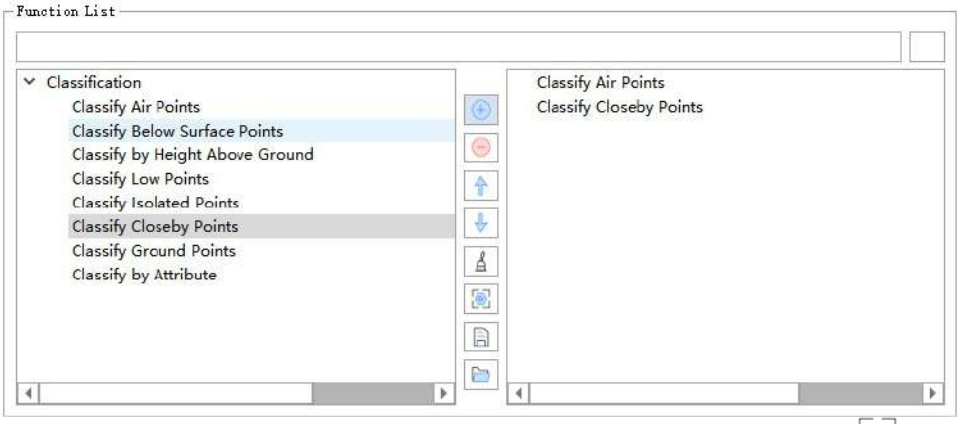
Steps for Dialogue Box Invocation

1.Click the "Batch Processing" button in the file tab to open the following interface. The file list displays all point cloud data in the system. The functionality section lists the features available for batch processing, and the right-side list presents the order of operations.



2.Click the button on the right side of the file list to select the point cloud data for batch processing. Click the button to clear all point cloud data. Select the files you want to remove from the file list, hold down the Ctrl key for multi-selection, and click the button to remove the selected data. Currently, LiData files are supported.

3.Double-click the feature on the left-side list (or select the feature and click the button), and a function parameter setting interface will pop up. After setting the parameters, they will be displayed in the right-side list.



4.(Optional) Double-click the function in the right-side list (or select the function and click the button) to modify the function parameter settings.

5.(Optional) Select a function in the right-side list and click the and buttons to adjust the execution order of the functions.

6.(Optional) Click the button to clear all selected functions.

7.(Optional) Click the button to save the order and parameter settings of the batch processing operations as a .LiProcessList file.

8.(Optional) Click the button to open the.LiProcessList file. The saved operation workflow will be loaded into the function list on the right.

9.You can set the number of threads for multithreaded batch processing in the thread count dialogue box (default is 4). When the thread count is set to 1, single-threaded batch processing is utilized.

10.After setting the output path, click the Execute button to start the batch processing operation in the order of the function list. All intermediate results during this process will be saved to the output path.

> Note: This function only applies to point cloud data, including LiData files. Functions that don't require parameters (e.g., PCV, point cloud normalization, etc.) will be directly added to the selected function list upon double click.

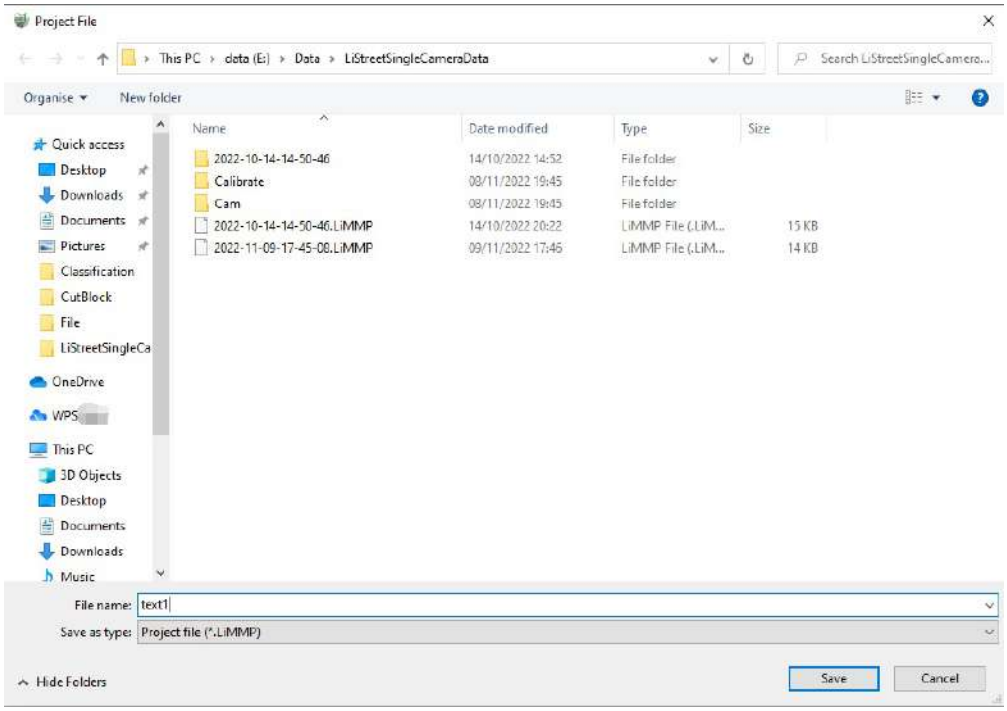
11.You can enter the function name in the search box for a quick search. Please pay attention to the software language version.

Save Project

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

Save As Project

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



Close Project

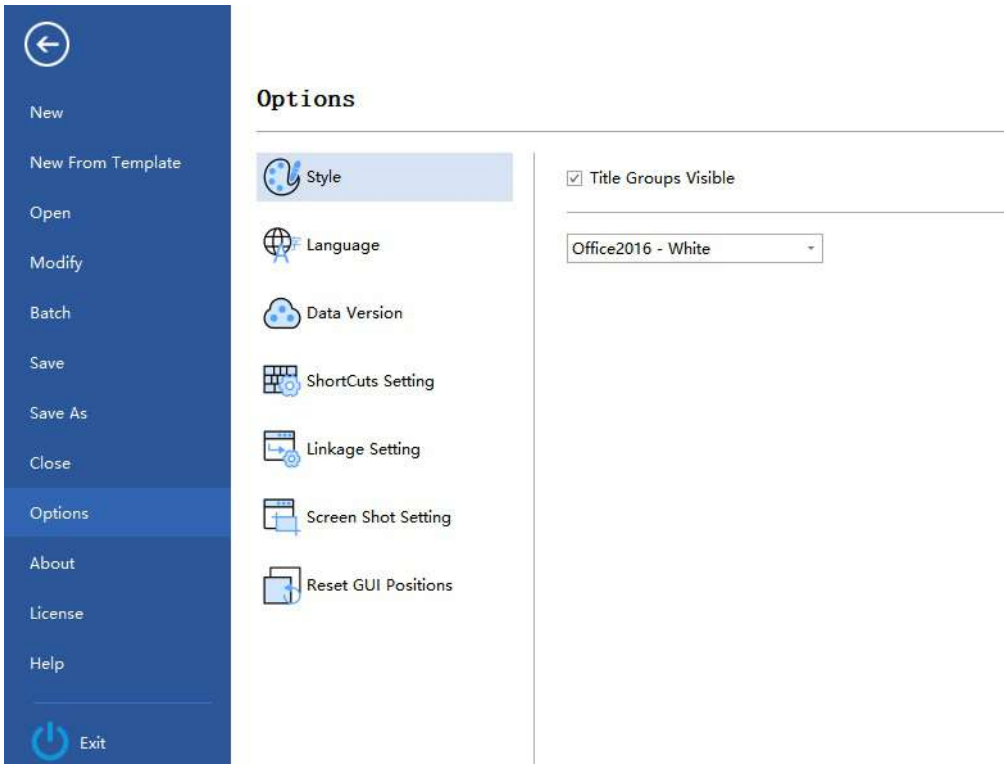
Function Description: Allows the user to close an opened project. The software will automatically save the project file being edited.

Options

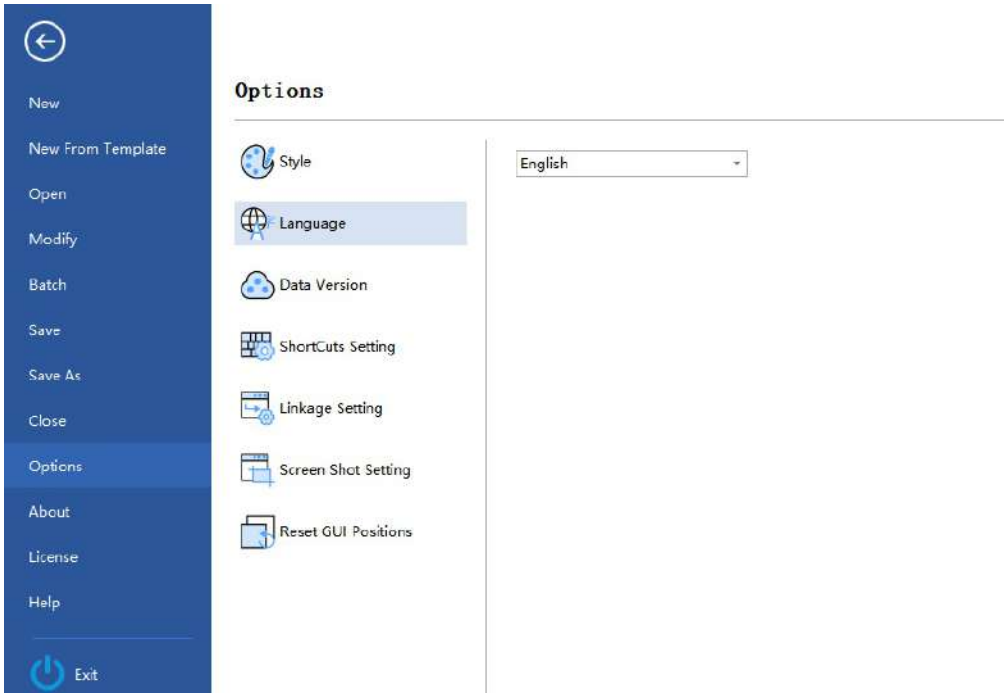
Function Description: By clicking on the 'Options', you can make some settings for the software.

Steps

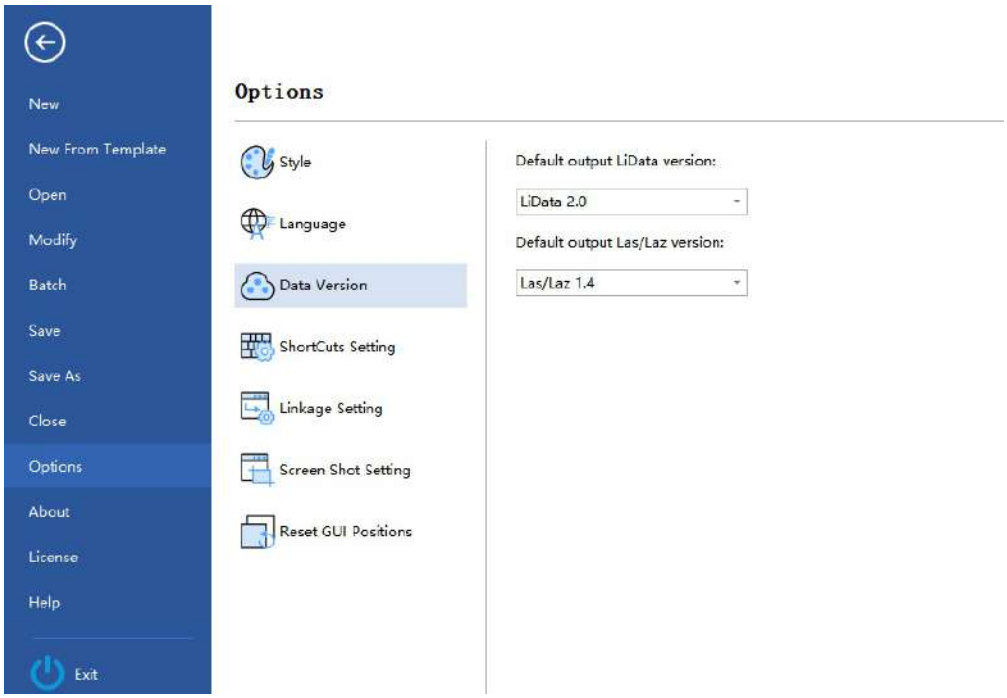
1.**Style**: You can set the display style of the software.



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



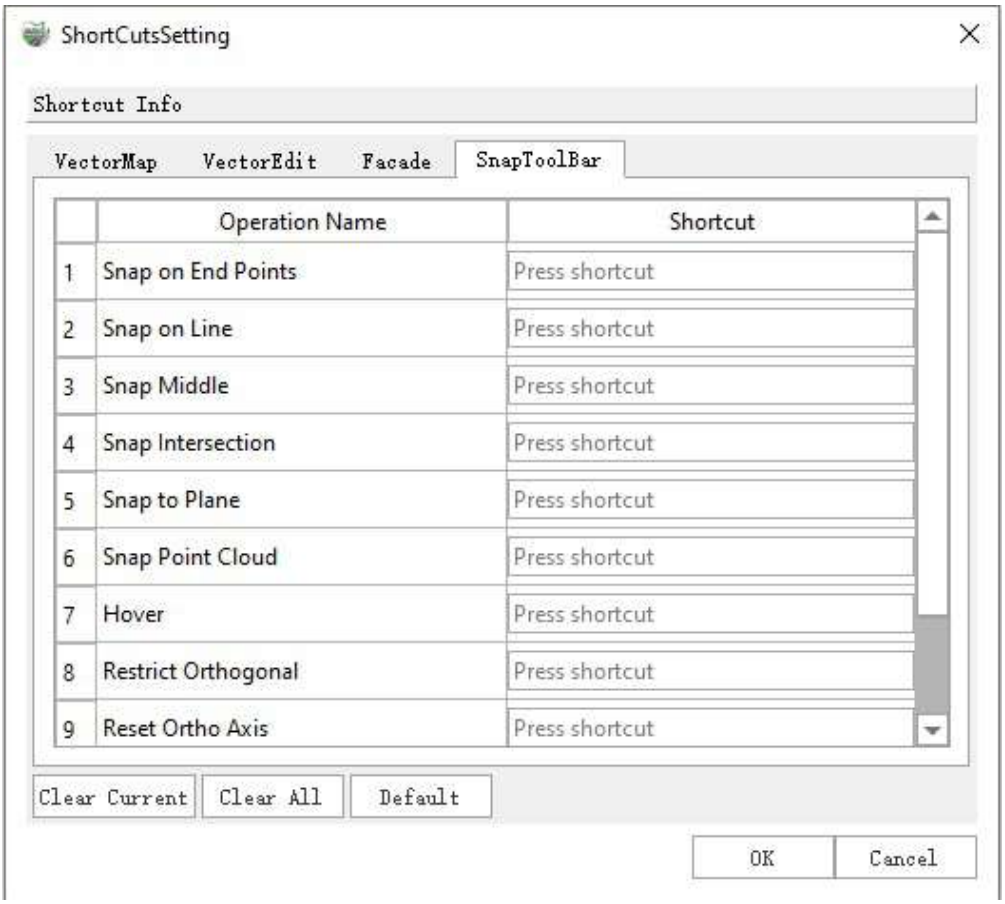
3.**Data Version**: Set the data version for input LiData format data and output as Las/Laz format, as well as the default point cloud storage format.



4.**Shortcut Settings**: You can set shortcuts for functions in the Vector Editing, Facade Measurement, and Capture modules.

Steps

（1）Click the Shortcut Configuration button , a shortcut configuration dialog box pops up.



(2) In the Shortcut Configuration dialog box, you can freely combine keyboard letters, numbers, Shift, Ctrl, Alt symbols, etc. Click OK to take effect.

(3) Click "Clear This Line" to delete the shortcut of the selected function.

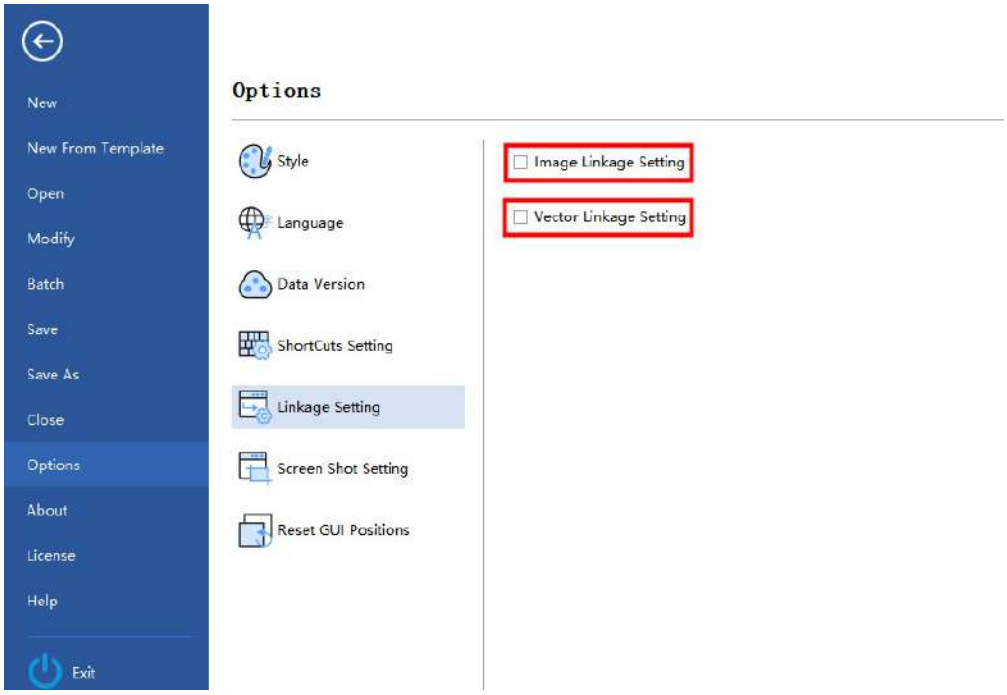
(4) Click "Clear All" to delete all configured shortcuts.

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

5.**Jump Settings**: The software jump settings are divided into vector jumps and the image jumps. Vector jumps control the 3D window jump after selecting a vector object; image jumps control the jump between photos and viewing angles in panorama or camera mapping projects.

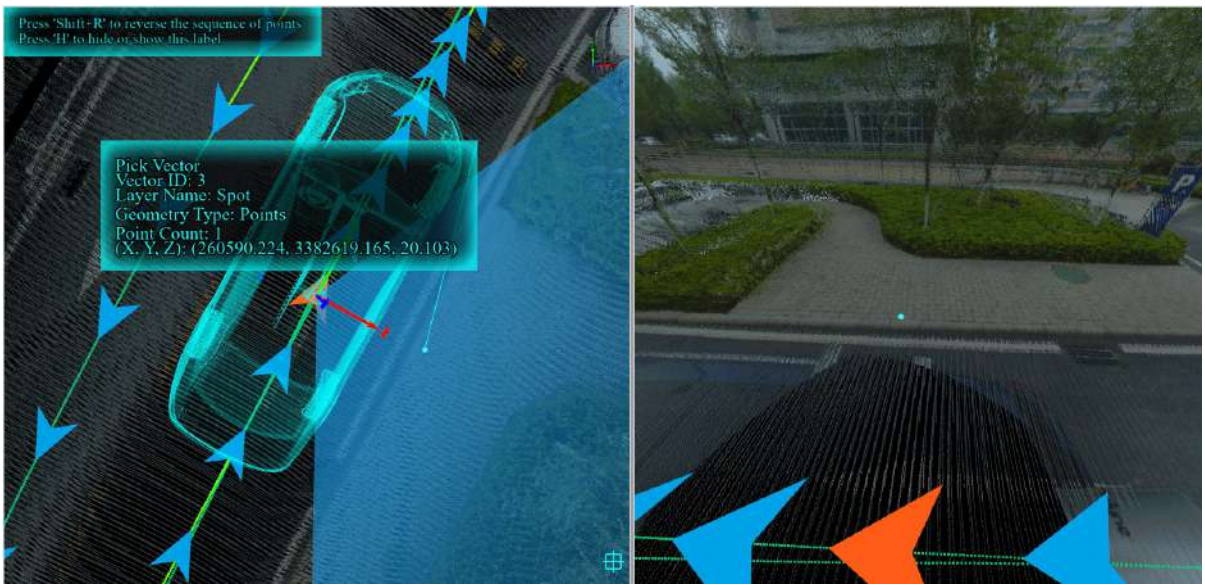
Steps

(1) The software jump settings page is as shown:



Jump Settings Page

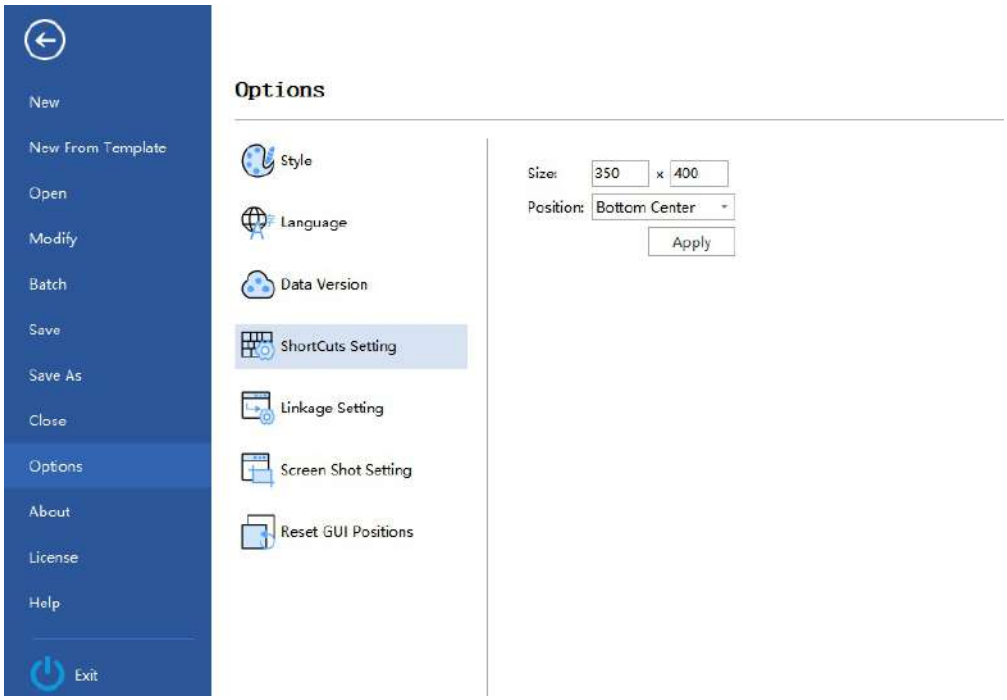
（2）After checking the image jump option, in the panorama and camera mapping projects, when selecting a vector, the image will jump to the nearest frame to the first point of the current vector, and the camera view will also face that vector. Similarly, when selecting a point cloud, the image will jump to the nearest frame to the selected point, and the camera view will also face that point. When drawing a vector, the image will jump to the nearest frame to the first point of the drawing, and the camera will also face the first point of the drawing. For example, selecting a vector in the panoramic window is as shown:



Panoramic Image Jump

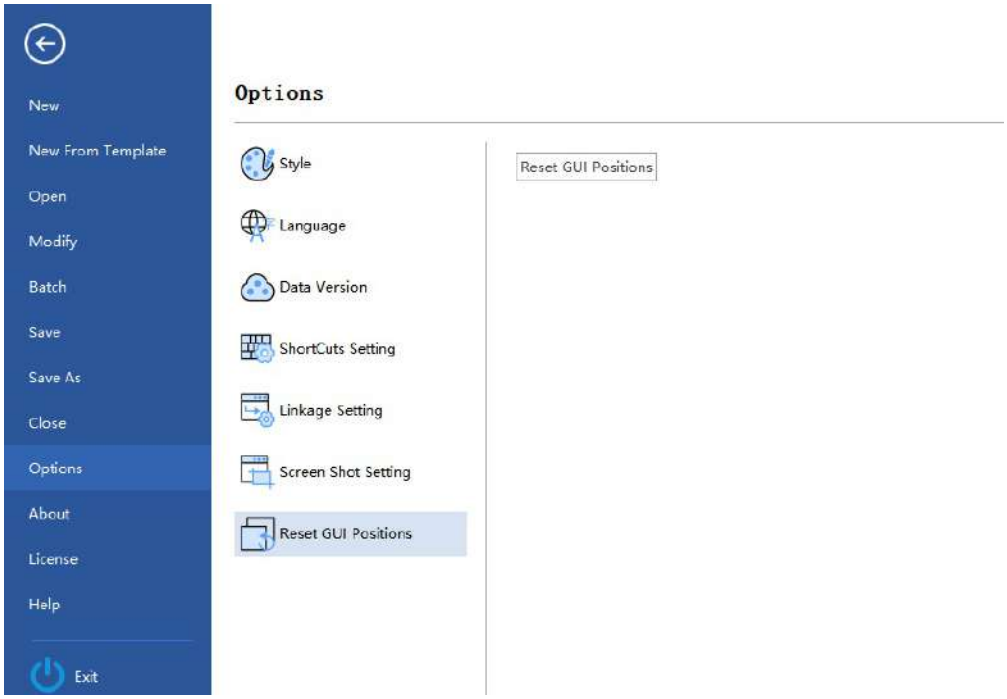
（3）After checking the vector jump setting, when you select a vector, the vector not only highlights, but also the vector will be displayed in the middle of the 3D window at a certain scale; if you deselect the vector jump setting, when you select a vector, the vector only highlights, and the 3D window will not jump.

6.**Screenshot Settings**: Set the size of the image when taking a screenshot.



- **Size**：The first item is the width and the second item is the height, both in pixels. After setting, click the 'Apply' button on the right to save the current settings and apply them to subsequent screenshot operations.
- **Position**：The locations for capturing images include: center position, bottom center, and top center.

7.**Restore Interface Position**: If a user accidentally drags the movable toolbar out of the interface and cannot restore it, click "Restore Interface Position" and then restart the software to reset the toolbar.



Default Keyboard Shortcuts

1.Common Platform Shortcuts

Function	Shortcut

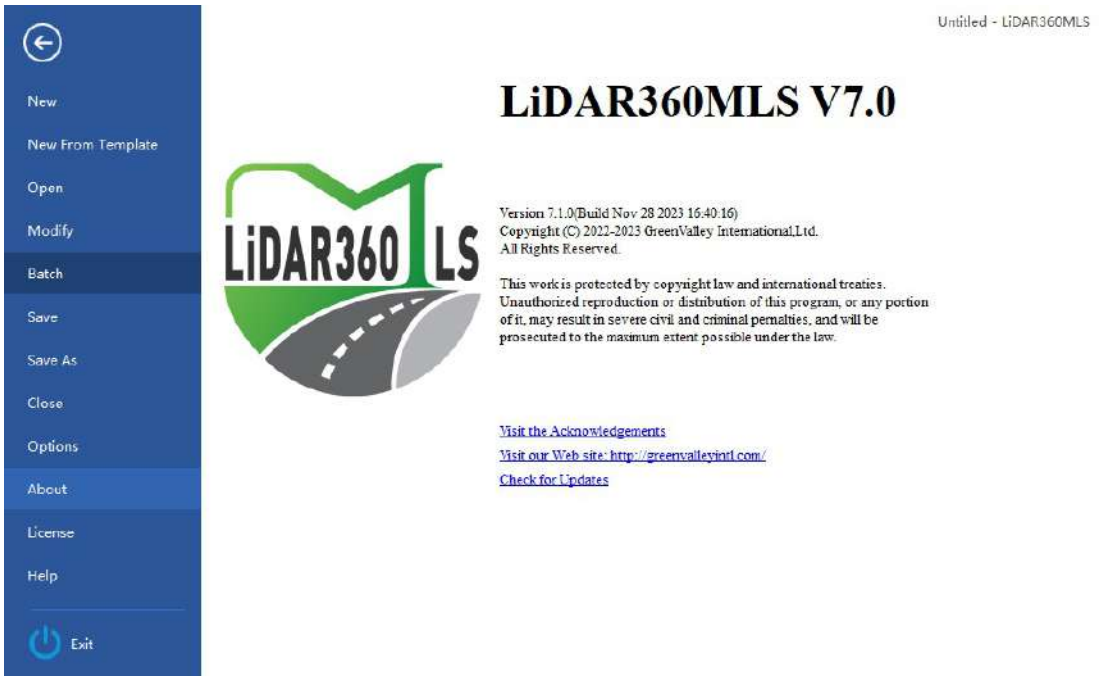
Exit LiDAR360 MLS software	Alt+F4
Save LiDAR360 MLS project	Ctrl+S
Switch between orthogonal/perspective projection	F3
Data upward translation	↑
Data downward translation	↓
Data leftward translation	←
Data rightward translation	→
Zoom in	+
Zoom out	-
Rotation	Left mouse button
Translation	Right mouse button or hold scroll wheel
Scaling	Scroll wheel
Global view	Spacebar

2.Common Function Shortcuts

Function	Shortcut	Description
Line, Polygon	s/S	Short press 's' to switch between drawing modes, from drawing straight lines to drawing arcs. Arcs are drawn using a three-point mode, where the first point is the last point before pressing 's', the second point is the endpoint of the arc, and the third point controls the arc's curvature.
Line, Polygon	b/B	Short press 'b' to backtrack the drawn nodes.
Add traffic signs, façade measurement move/rotate	q/Q;r/R	Rotate clockwise; Rotate counterclockwise

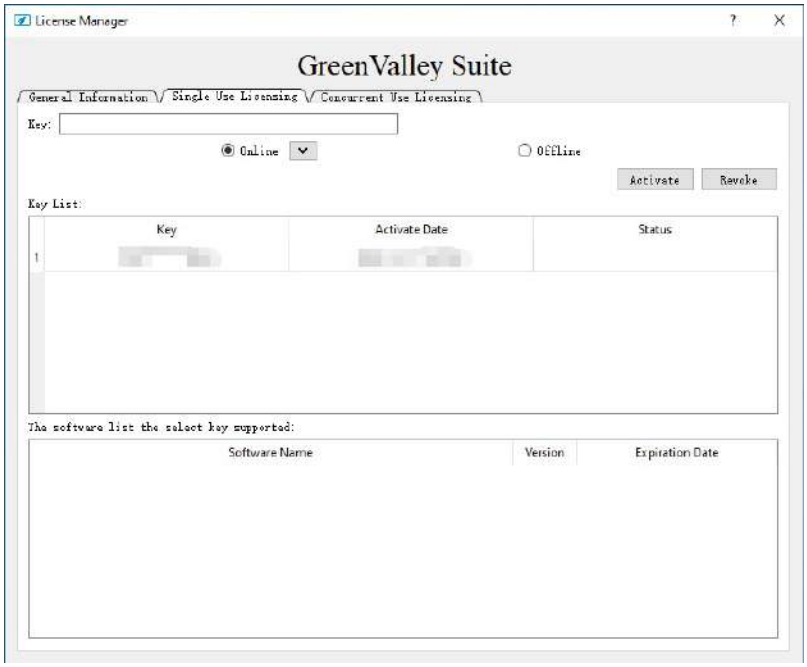
About

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



License

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



Help

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

Exit

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

Tools

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

Measurement

Select and Cut

Cross Selection

Roaming

Import

Export

Unroll

Measurement

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

- Single-point selection
- Multi-point selection
- Density measurement
- Length measurement
- Area measurement
- Angle measurement
- Gradient Measurement
- Height measurement
- Volume measurement
- Two Stage Volume
- Two Stage VolReport
- Mouse Hover

Single point selection

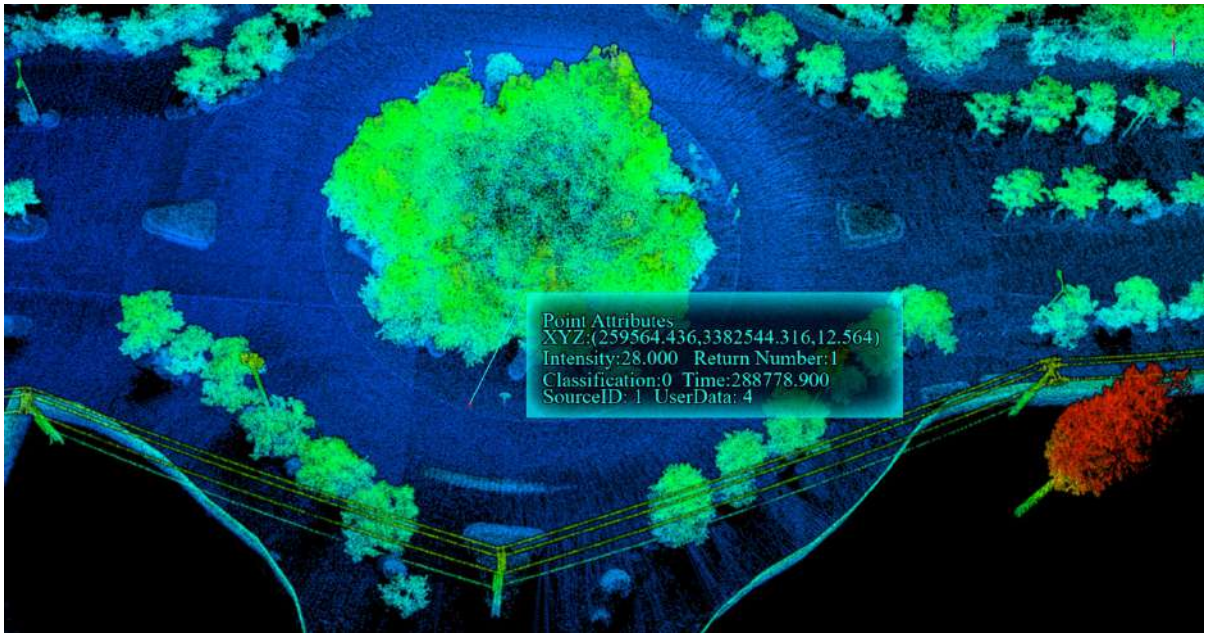
Function Description: The single point selection tool can interactively query the attribute information of a single point in the point cloud data, including the location, intensity, echo frequency, category, GPS time, etc. of the point.

Steps

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

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

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



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

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

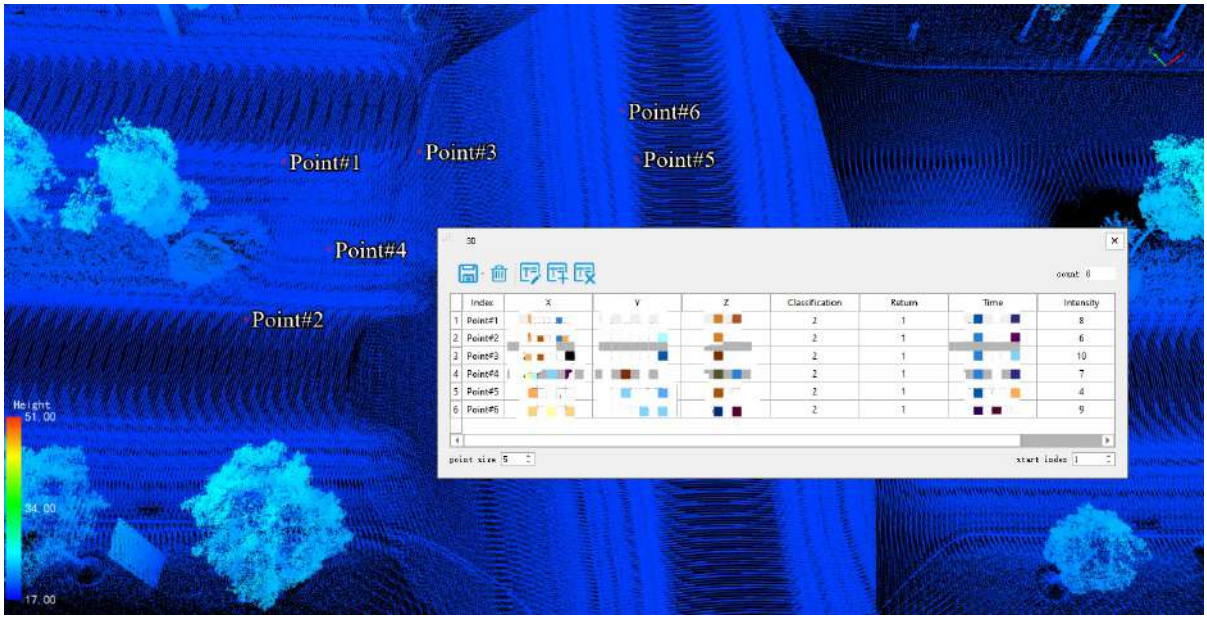
Multi-point selection

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

Steps

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

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



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

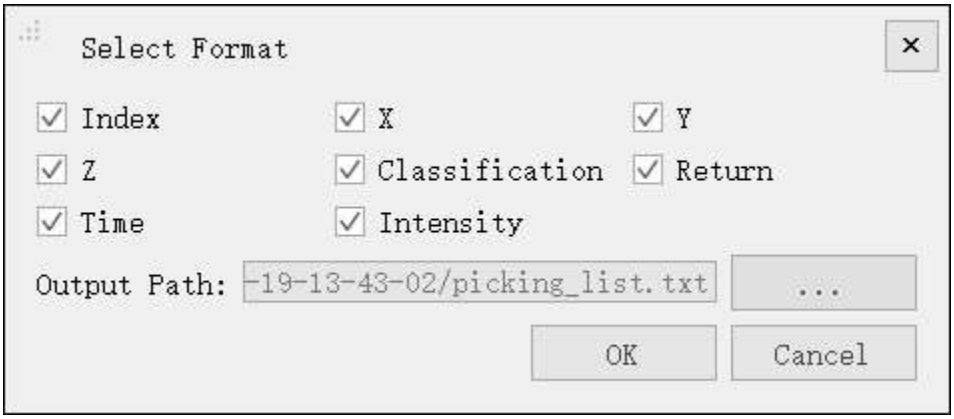
3.Click any row in the selection list and click the button deletes the point.

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

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

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

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

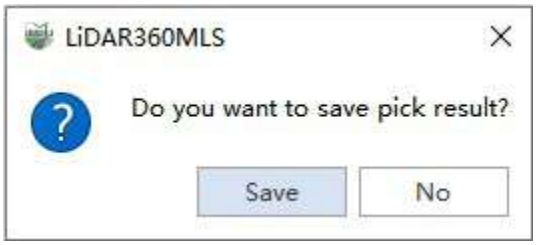


Parameter setting

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

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

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



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

Density measurement

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

Steps

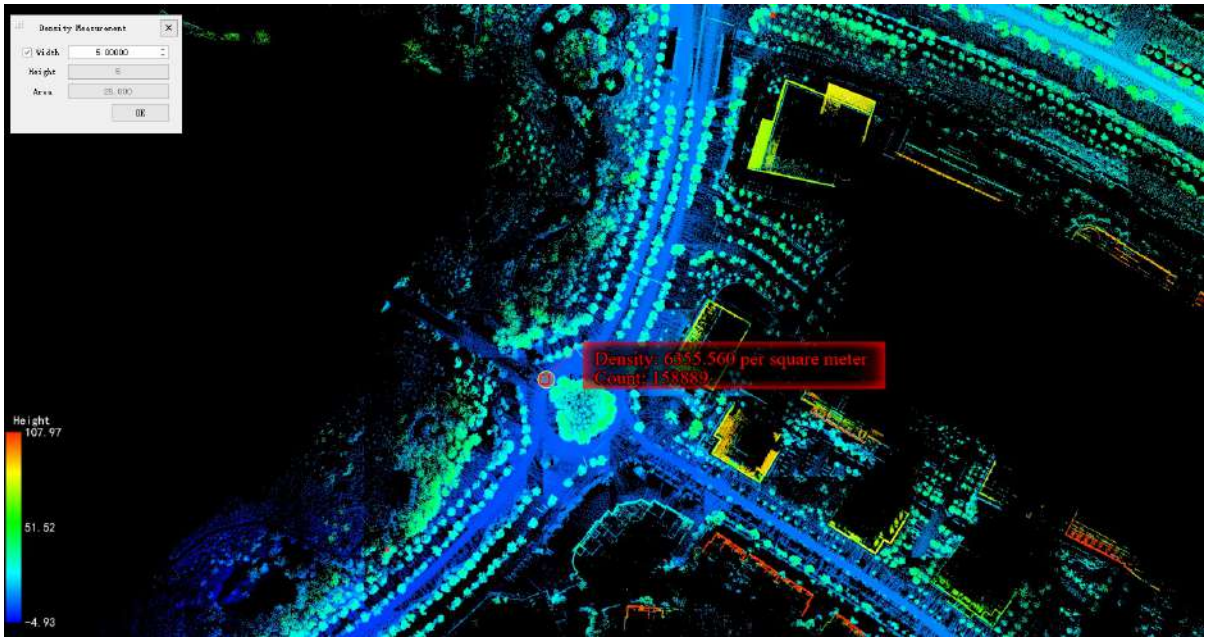
1.Click on the **Density** button in the Measurement toolset, open the point density measurement tool, and the 3D display window will automatically switch to orthogonal projection.

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

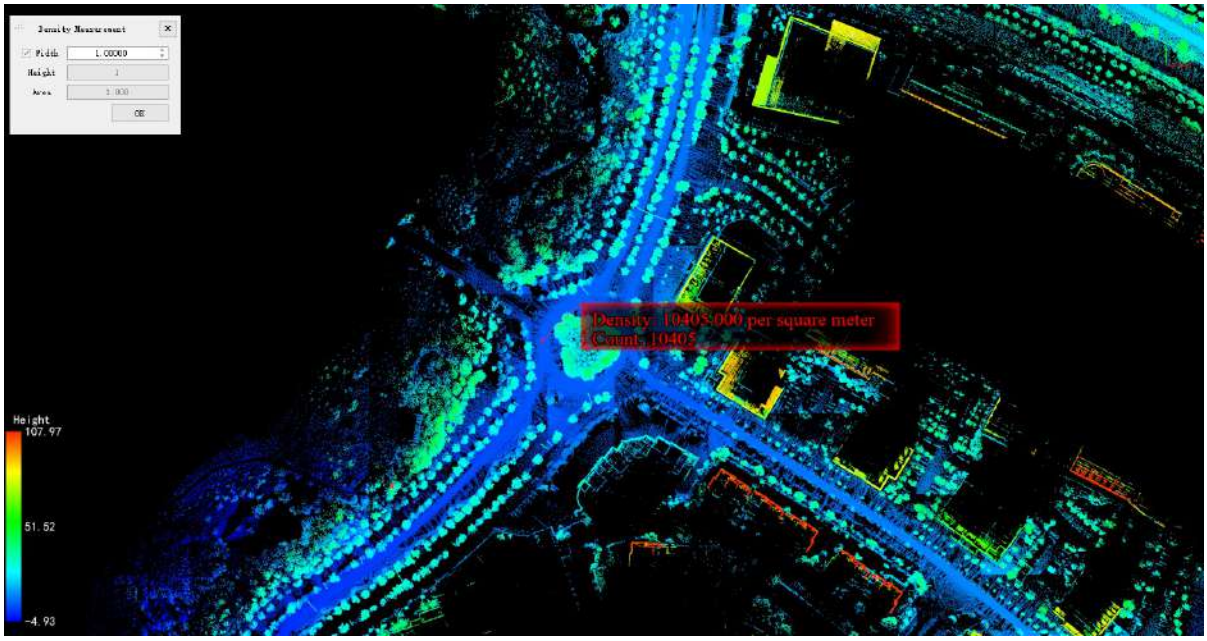
3.If "Width" is checked, you can enter the width value, the height value will be automatically set to the same as the width value, and the area is set to the product of width and height. Use the left mouse button to click to select the measurement area. If "Width" is not checked, the height value and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

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

Default width (width is 5):



Modify the width (width is 1):



Parameter setting

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

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

Length measurement

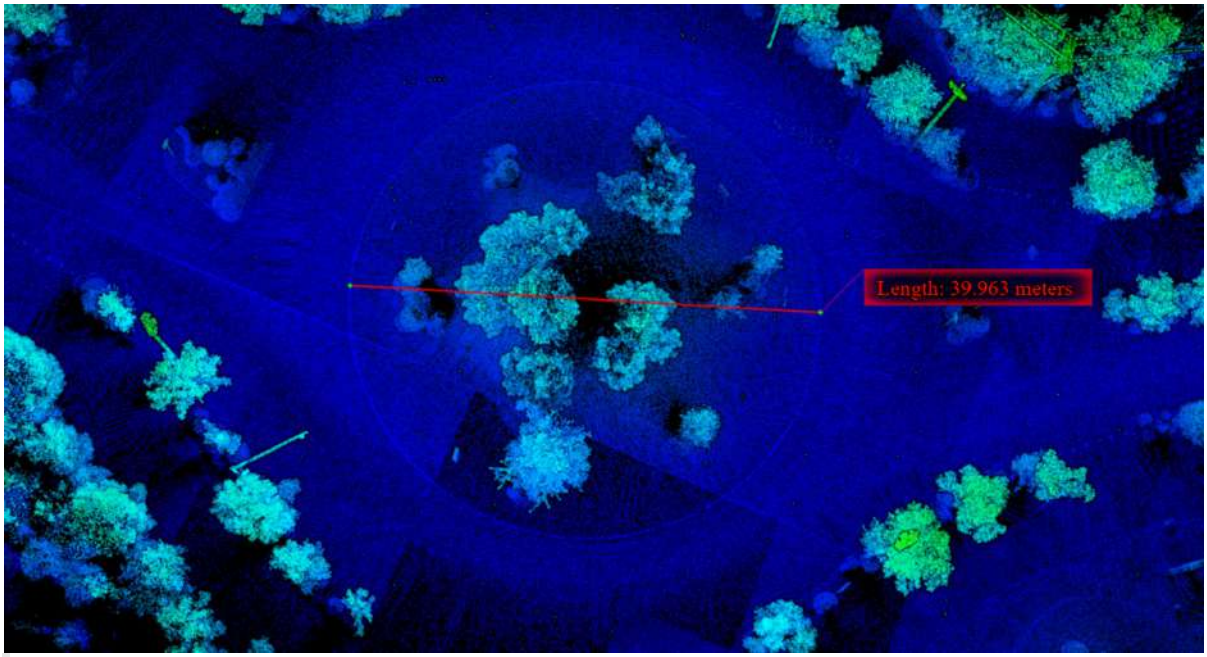
Function Description: The length measurement tool uses a mouse click to interactively query the distance information between multiple points. The measurement result represents the Euclidean distance of the point in the three-dimensional space.

Steps

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

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

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



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

Area measurement

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

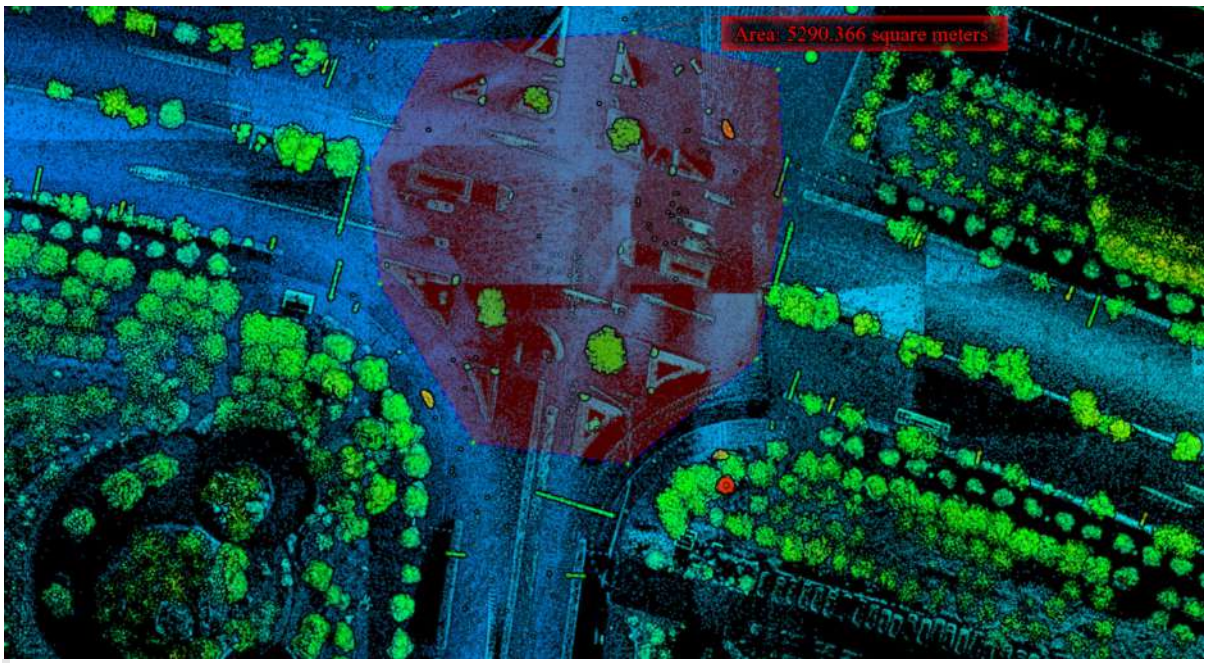
Steps

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

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

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

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



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

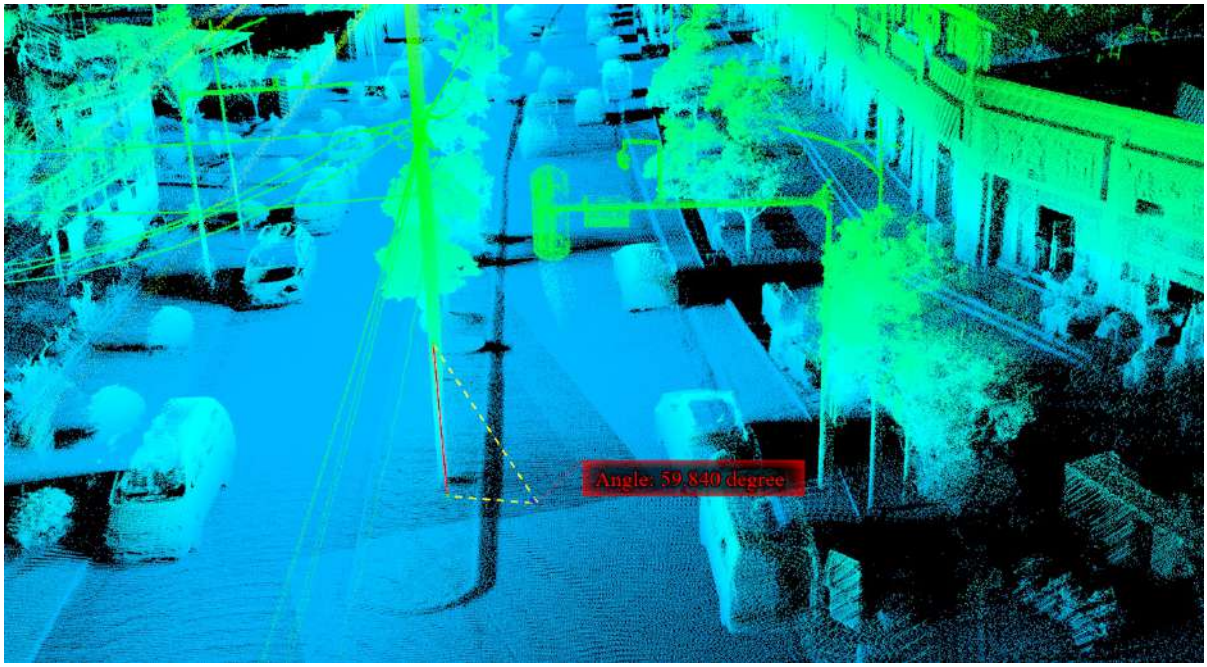
Angle Measurement

Function Description: The angle measurement tool allows you to interactively select measurement points using a mouse click and query the pitch angle between three points in a 3D view.

Steps

1.Click the "Angle" button in the "Measure" menu.

2.In the 3D view window, left-click the data points one by one with your mouse to select three points. The angle formed by these points is the measured angle (as shown in the figure).



3.You can right-click the mouse before completing the measurement to go back and deselect one point.

> Note: This feature is only available in the 3D view window.

Gradient measurement

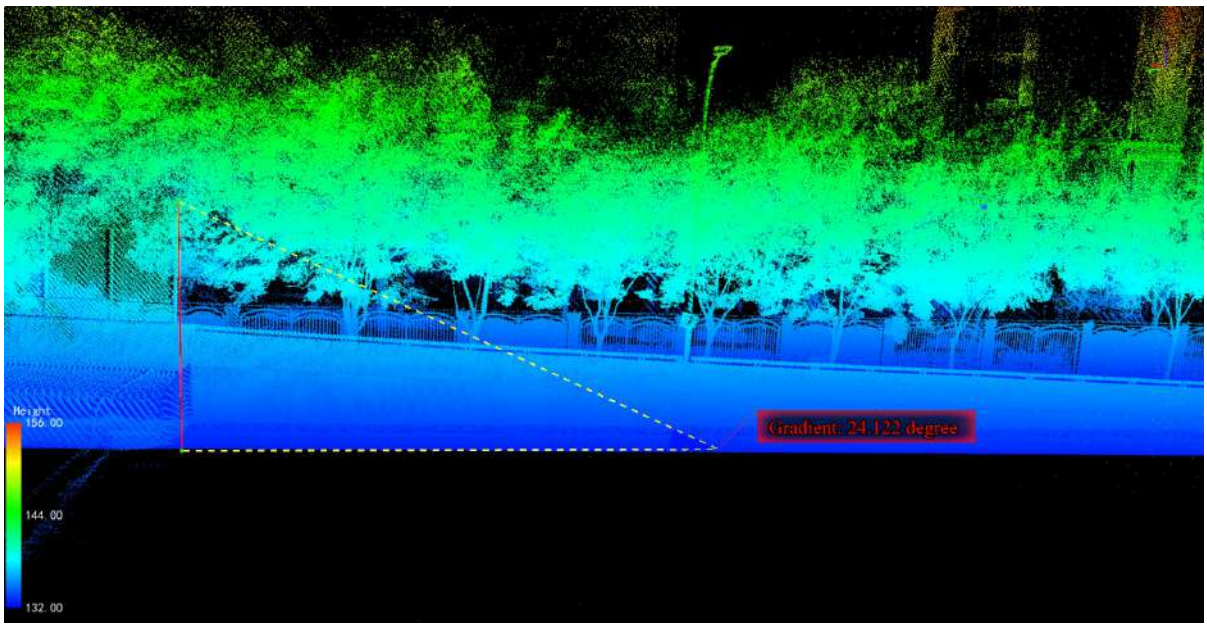
Function Description: The gradient measurement tool allows the user to measure the gradients between a set of selected points. In the 3D view, the pitch gradient between the two points is queried, that is, the gradient between the connecting line from starting point to the end point and the horizontal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

Steps

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

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

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



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

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

Height measurement

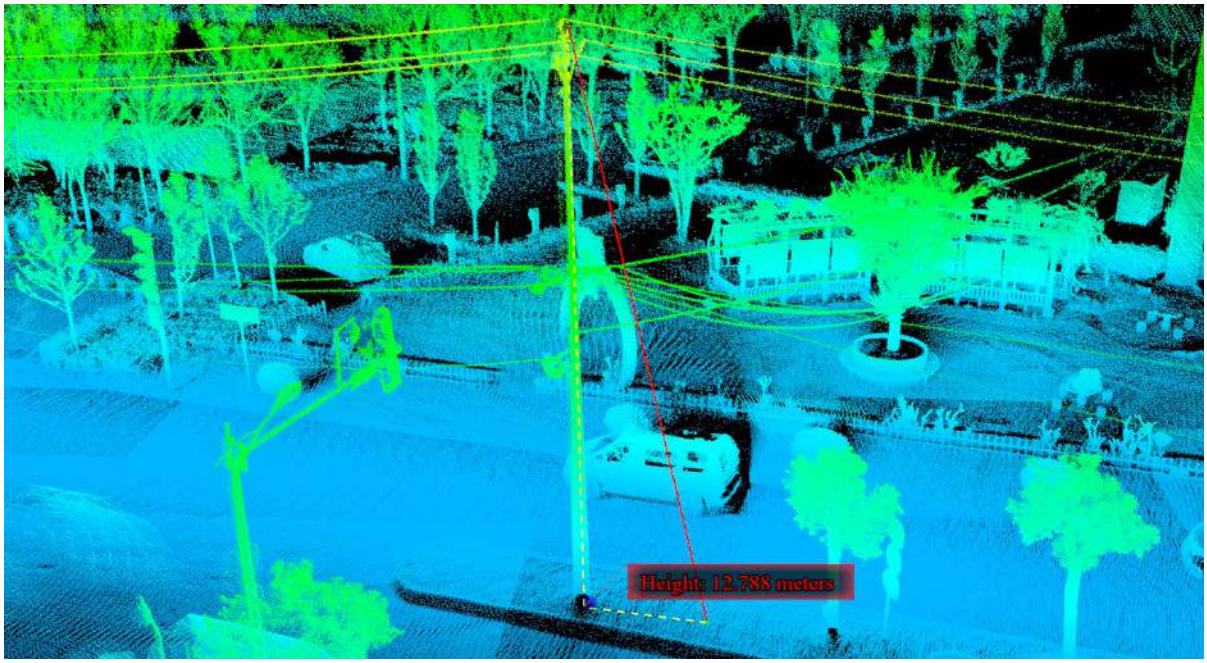
Function Description: The height measurement tool interactively selects the measurement point with a mouse click and queries the relative height between the two points.

Steps

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

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

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



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

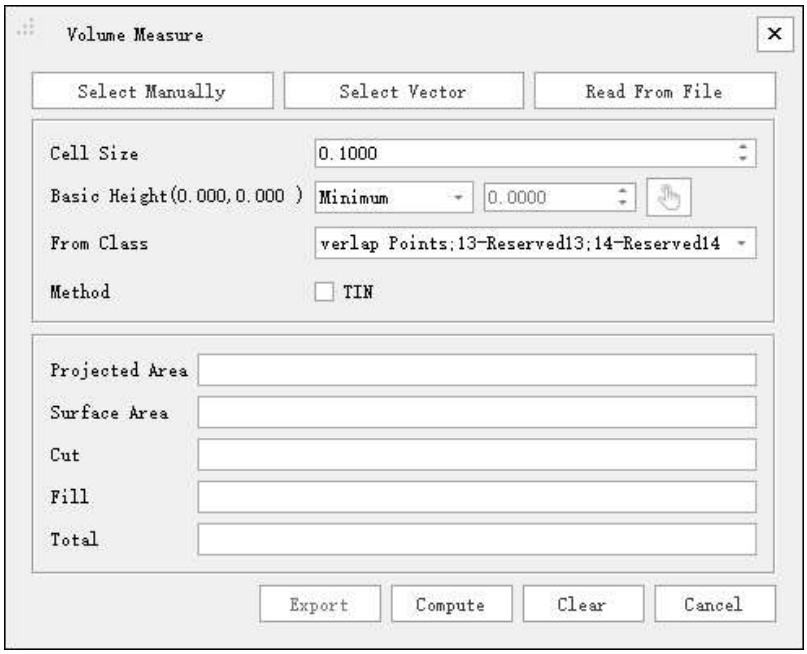
> Note: This function can only be used in the 3D view window.The fallback function can only be used before double-clicking the mouse.

Volume Measurement

Function Description: This tool is applicable to point cloud data which calculates filling, cutting and total amount relative to a reference height. It's commonly used in volume measurement of coal pile and hull.

Steps

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



Parameter setting

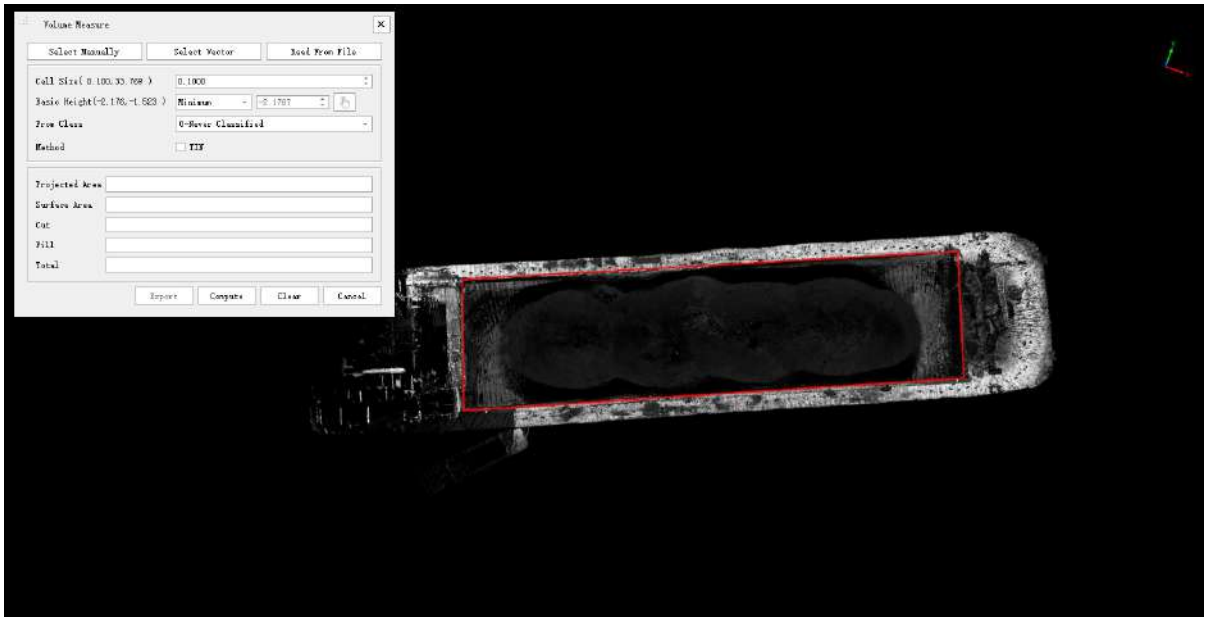
- **Select Manually**：Manually create the outer contour on the point cloud.
- **Select Vector**：Manually select an existing polygon as the outer contour.
- **Read From File**：Read geometric information of the outer contour from an external file. Supported formats include .dxf, .shp, and .txt.
- **Cell Size**：This parameter defines the minimum calculation unit size for volume measurement. A smaller value results in a more accurate calculation.
- **Basic Heigth**：This parameter defines the reference surface for volume measurement of cut and fill.
  - **Minimum**（**Default**）：Take the minimum Z value of the outer contour as the height value of the plane.
  - **Fitted Plane**：Fit the best plane based on the selected points.
  - **Custom**：Specify the height provided by the user as the reference surface height value for volume measurement.
- **From Class**：This parameter defines the point cloud point categories involved in the operation.

  > Effectively filter out noise.

- **Method**：This parameter defines whether to construct an irregular triangulated network (TIN) during volume measurement. Building an irregular TIN can make the measurement more accurate but may reduce the calculation speed. Users can choose whether to use this method based on their needs.

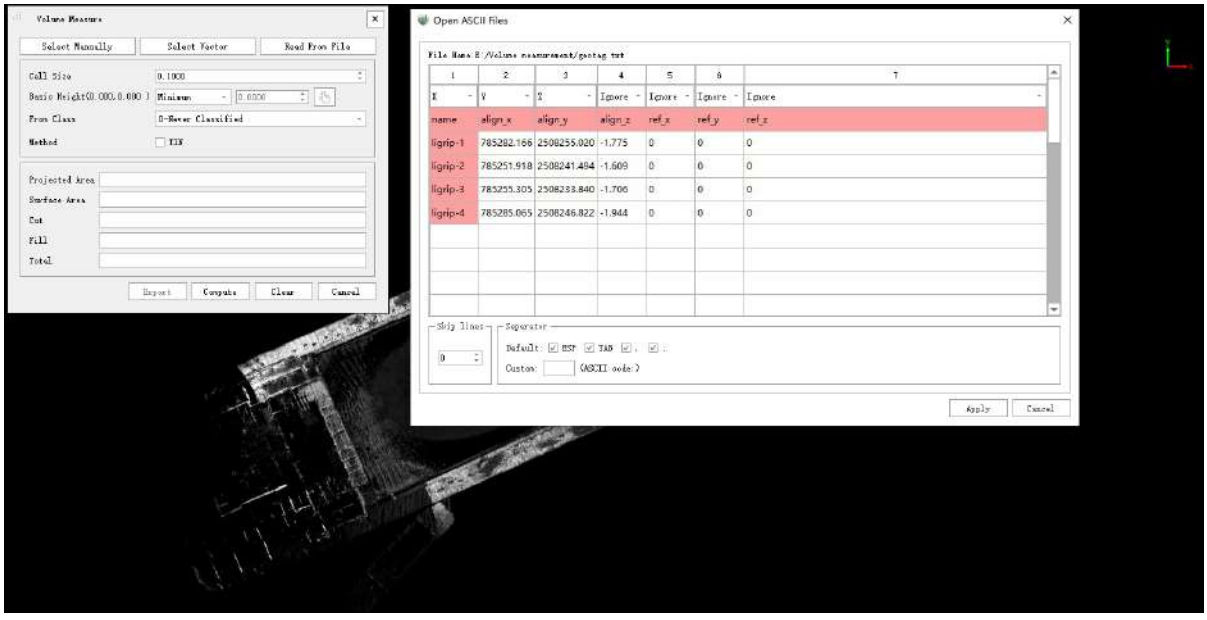
> Note: This function can only be used in the 3D view window.

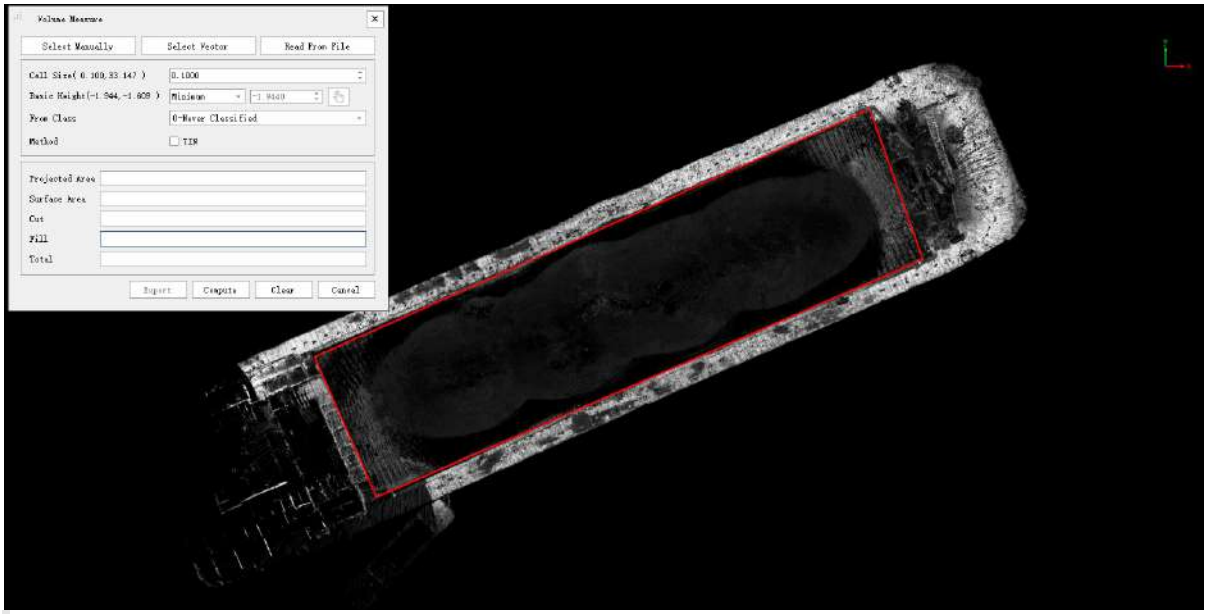
2.Select Manually：Click the **Select Manually** button, and the window is automatically adjusted to the top view mode. Then, in the window where the point cloud data is loaded, use the left mouse button to consecutively click on the region of interest (to generate a reference plane for volume measurement, select at least three points). Double-click the mouse to end the selection, and the selected area is outlined in red dashed lines in the scene."



3.Select Vector：Simply use the left mouse button to select an existing polygon.

4.Read From File：Click the 'Read From File' button, choose the geotag.txt file matched with the current project generated after the computation. The software automatically reads the align_x, align_y, and align_z columns from this file and automatically fits to generate the reference plane.





Note：

1、If GCP computation has been performed on the data, you should choose the geotag_optimization.txt file after GCP registration. At this point, a file settings list will appear. You need to select the X, Y, and Z columns corresponding to the outline range of the current volume measurement data, and skip the first row.

2、When using a device for field control point acquisition, it is only necessary to hit the target measurement object's footprint. For example, for a sand barge, hitting four intersection points is sufficient.

3、Before volume measurement, it is necessary to use the cleaning tool to remove excess point clouds and noise in order to improve calculation accuracy.

4、Supports importing external vector formats (shp, dxf) for volume measurement.

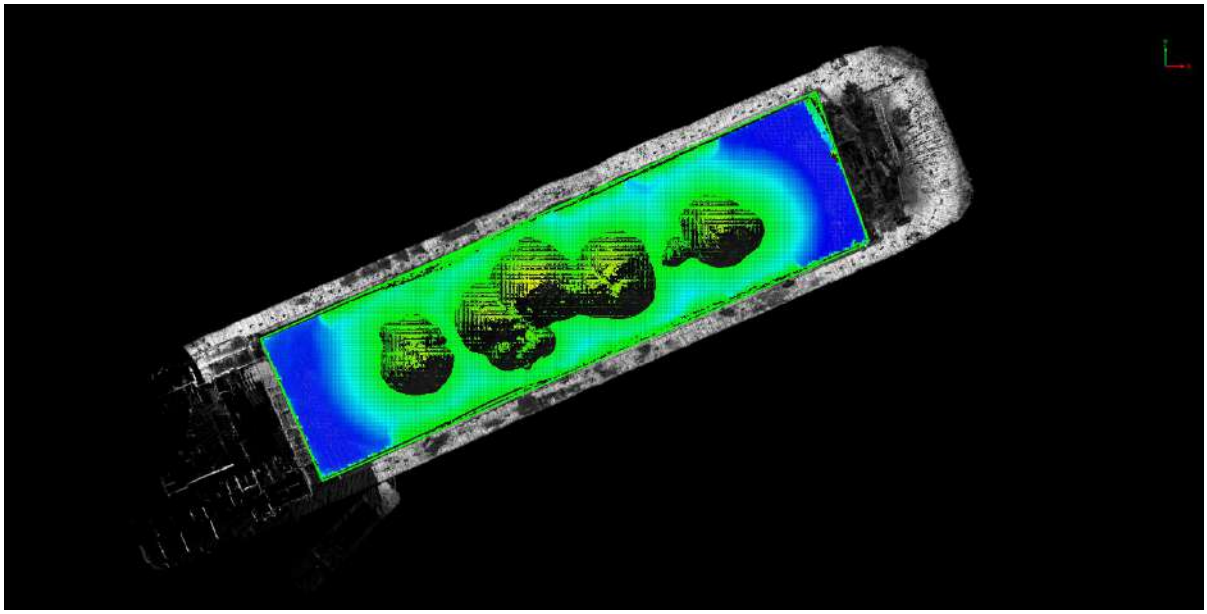
5.Set the cell size. For sand barge volume measurement, a cell size of 0.1 or 0.2 is generally suitable.

6.Set the reference surface for volume measurement. The calculation methods for the reference surface include minimum value, average value, and user input. When choosing a file for volume measurement, the average value is generally selected for the reference surface.

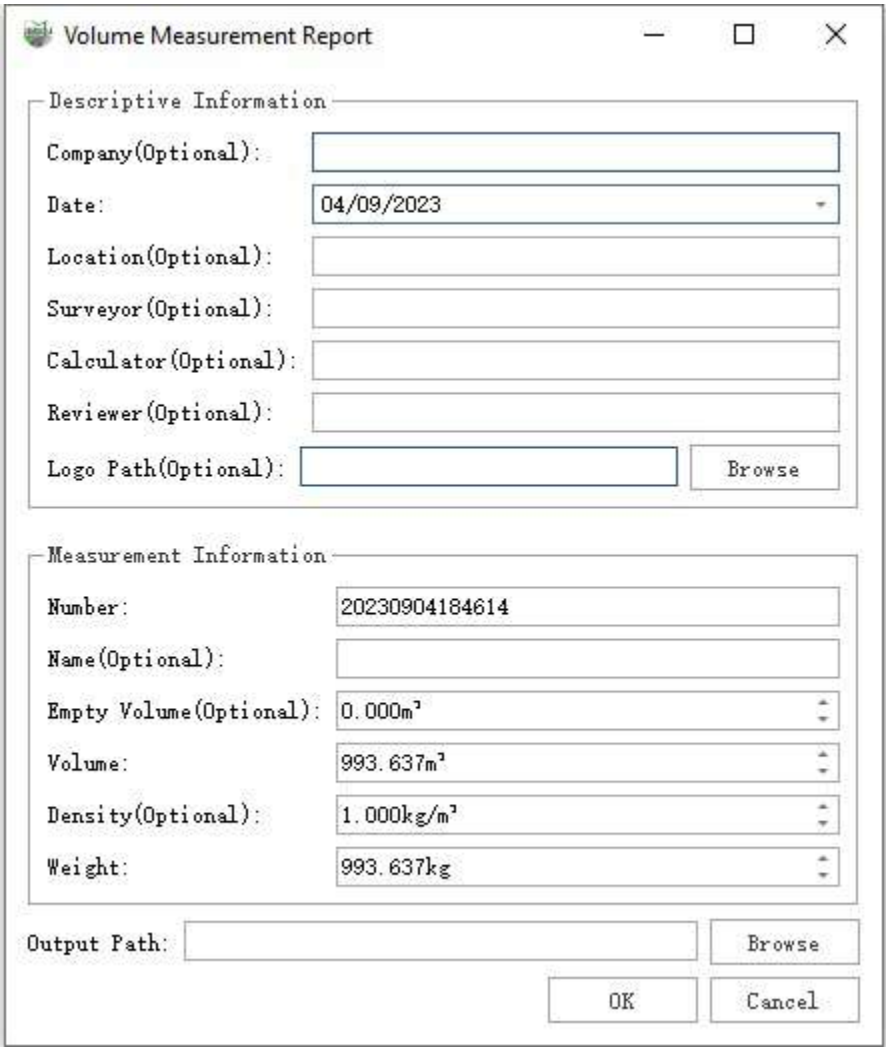
7.Select source categories to participate in the calculation based on your requirements.

8.The volume measurement methods include TIN and grid. Unchecking TIN selects the grid method.

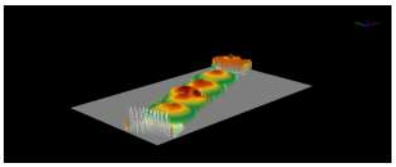
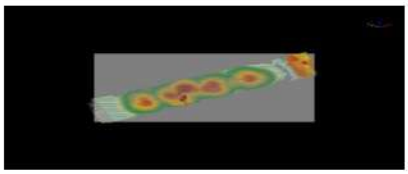
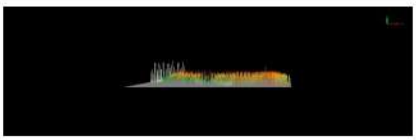
9.Click the **calculate** button to generate cut, fill, and cut-fill results. At the same time, the space occupied by the current measurement data is displayed in the scene, as shown in the figure.



10.Click export, and a information bar for the volume report will appear. At this point, you can enter some basic measurement information, such as measurement unit, measurement location, personnel, logo, empty volume (for sand barges, granaries, etc.), pile density, pile name, etc.



10.1 Then click browse to select the directory for the output report. Finally, click OK to output the final HTML report. The report includes the basic information you entered, volume values, mass, etc., and automatically generates oblique views, top views, and side views of the pile. If you want to output a PDF document, you can use the browser's print-to-PDF function. If you need to modify content, you can use PDF editing tools.

Volume Measurement Report							
Company		Time	2023-09-28	Location			
Number	Name	Size(L*W*H)	Area(m2)	Empty Volume(m3)	Volume(m3)	Density(kg/m3)	Weight(kg)
20230928163849		(35.727*20.479*1.117)	257.106	0	289.069	1	289.069
3D View							
Front Isometric View							
Top View							
Side View							
Surveyor		Calculator		Reviewer		GreenValley International	

> If there are trajectory lines in the report that you don't want to display, you need to manually turn it off before exporting the report.

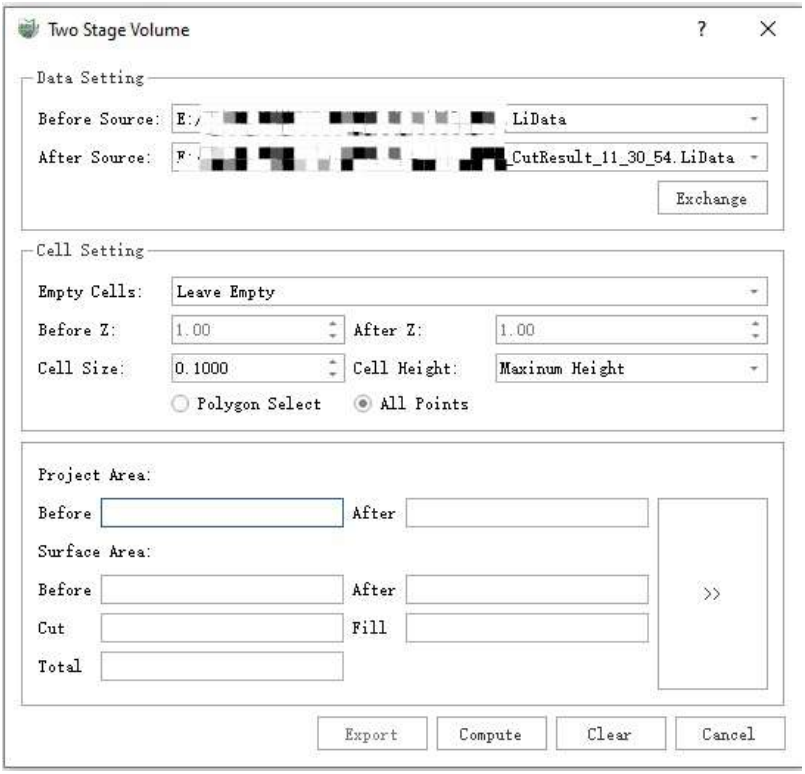
Two Stage Volume

Function Description：The two-phase volume measurement tool is primarily designed for measuring volumes using point cloud data from the same location but at different time periods. It calculates various information such as projected area, surface area, cut volume, fill volume, and total change between the two periods.

> You need to create at least two point cloud projects.

Steps

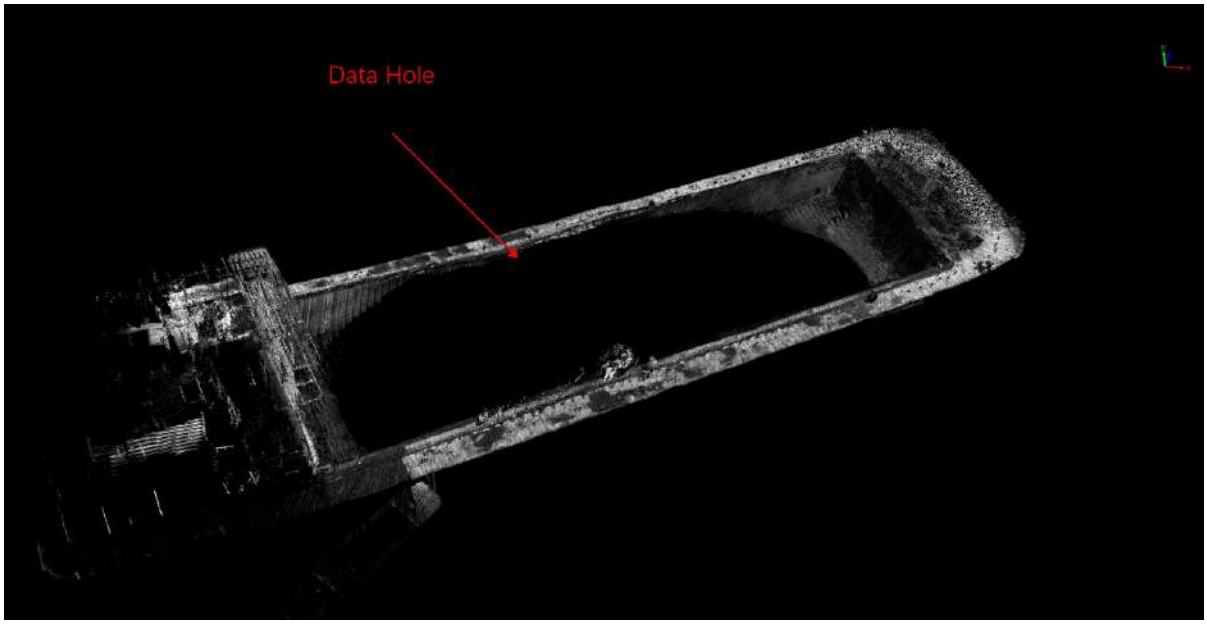
1.Click the **Measurement** menu, then select the **Two Stage Volume** utton, Open the Two-Phase Volume Measurement dialog box.



Parameter dialog box

Parameter setting

- **Data Setting**：Select the pre-phase and post-phase data
  - **Before Source**：The pre-phase data at the same location serves as the baseline data.
  - **After Source**：The post-phase data at the same location.
  - **Exchange**：Quickly swap pre-phase and post-phase data.
- **Cell Setting**：Use a uniform grid for volume measurement. Here, you can configure the grid cell settings.



Data example.

- **Empty Cells**: Data often contains voids, which can result in empty cells in the grid. Here, you can configure how to handle these empty cells.

  > Volume calculations are performed only within grid cells where both pre-phase and post-phase data are present. No calculations are performed in cells lacking data from either phase. Please configure empty cell handling accordingly based on the actual data.

  - **Leave Empty**: No action is taken for empty cells.
  - **Interpolation**: Perform triangular mesh interpolation on both pre-phase and post-phase data separately to fill in all data voids.
  - **Mininum Height**: Calculate the maximum and minimum heights of the data separately and write the empty cell locations into the minimum height.
  - **Maxinum Height**: Calculate the maximum and minimum heights of the data separately and write the empty cell locations into the maximum height.
  - **Average Height**: Calculate the maximum and minimum heights of the data separately and write the empty cell locations into the average height.
  - **Usr Defined**: After selecting this option, the pre-phase Z and post-phase Z buttons below become active, allowing you to set the Z-values for empty cell locations separately.
- **Before Z, After Z**: Combine with user-defined options for empty cells.
- **Cell Size**: Set the size of individual grid cells.

  > The default is 0.1, and it should be set based on the point density of the point cloud data. It is recommended to use the default value. If the point density is low, you can consider increasing it accordingly.
  >
  > A larger size leads to faster calculations and coarser results, while a smaller size results in slower calculations but relatively more accurate results. However, setting a size smaller than the point spacing can lead to loss of accuracy, so it's not recommended to set it too small.

- **Cell Height**: Set the basis for cell height calculation.
  - **Maxinum Height**: Use the highest Z-value from the point set that falls within the current cell as the cell height. (Default)

    - **Mininum Height**: Use the lowest Z-value from the point set that falls within the current cell as the cell height.
    - **Average Height**: Use the average Z-value from the point set that falls within the current cell as the cell height.
- **Polygon Select**: Define the calculation region by manually creating a polygon area.
- **All Points**: Define the calculation region by using the entire data area. (Default, suitable for scenarios with relatively small data volumes)
- **Project Area**:
    - **Befor, After**: Display the calculated projection areas for both pre-phase and post-phase separately.
- **Surface Area**:
    - **Befor, After**: Display the calculated surface areas for both pre-phase and post-phase separately.
- **Cut**: Display the calculated fill volume

> At the same cell location, if the height of the post-phase data in that cell is lower than the height of the pre-phase data in the same cell, it's considered as excavation or cut volume.

- **Fill**: Display the calculated fill volume.

> At the same cell location, if the height of the post-phase data in that cell is higher than the height of the pre-phase data in the same cell, it's considered as fill volume.

- **Compute**: After setting the parameters, click the calculate button with the mouse to start the calculation.
- **Clear**: Clear the result display box.
- **Cancel**: Exit the function.
- **Export**: The report export function is not available by default. It becomes available only after completing a calculation without making any parameter changes. For detailed instructions, please refer to the Two-Phase Volume Measurement Report Export.

2.Following the manual provided above, configure the parameters, and then click 'Calculate'.

Example of the results

> Excavation is displayed in blue, fill is displayed in red, and green represents no change between pre-phase and post-phase data.

Two Stage VolReport

Function Description：Export volume measurement reports for two time periods.

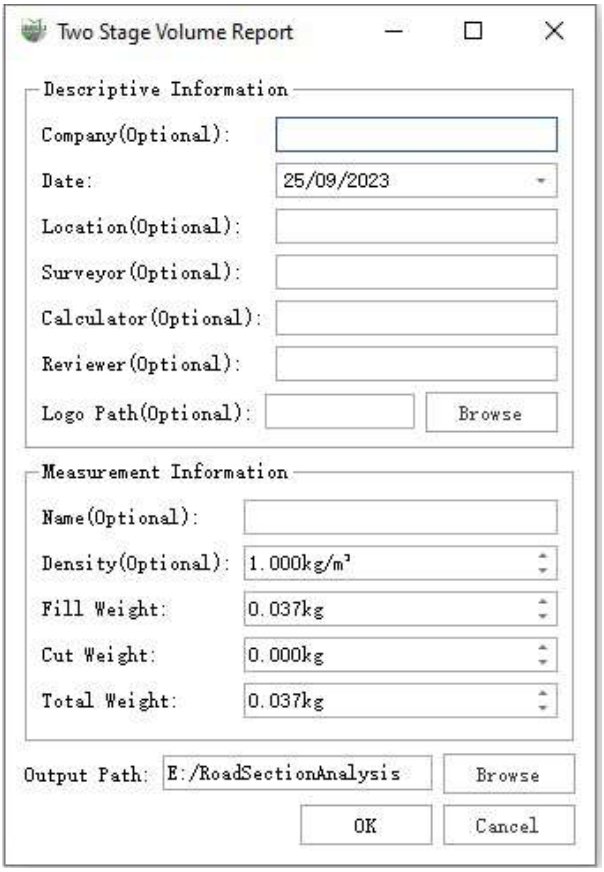
> Combine it with the Two Stage Volumefunction.

Steps

1.Click the **Measurement** menu, then select the **Two Stage Volume** button. Set the parameters and perform the calculation.

Result example.

2.Click the **Export** button to open the report information settings interface

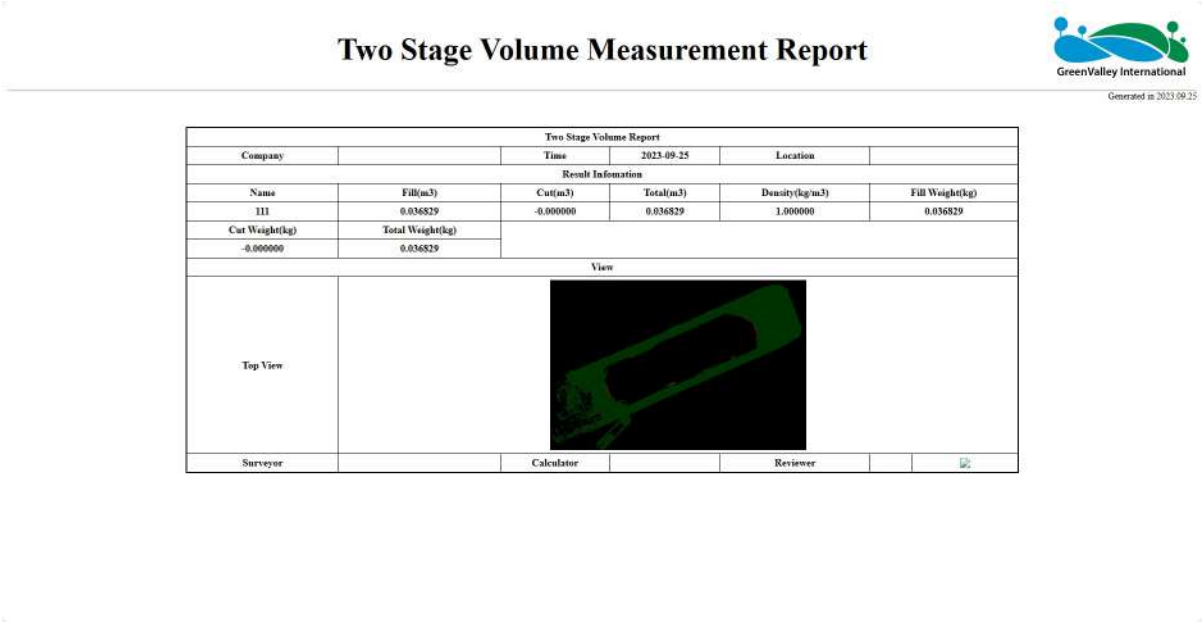


Dialog box

Parameter setting

- **Descriptive Information**: The text content information of the report.
- **Measurement Information**: You can set the density to calculate the mass of cut and fill, as well as the total change in mass.

Report example



Two Stage Volume Measurement Report

GreenValley International

Generated in 2023.09.25

Two Stage Volume Report					
Company		Time	2023-09-25	Location	
Result Infomation					
Name	Fill(m3)	Cut(m3)	Total(m3)	Density(kg/m3)	Fill Weight(kg)
111	0.036829	-0.000000	0.036829	1.000000	0.036829
Cut Weight(kg)	Total Weight(kg)				
-0.000000	0.036829				
View					
Top View					
Surveyor		Calculator		Reviewer	

Mouse Hover

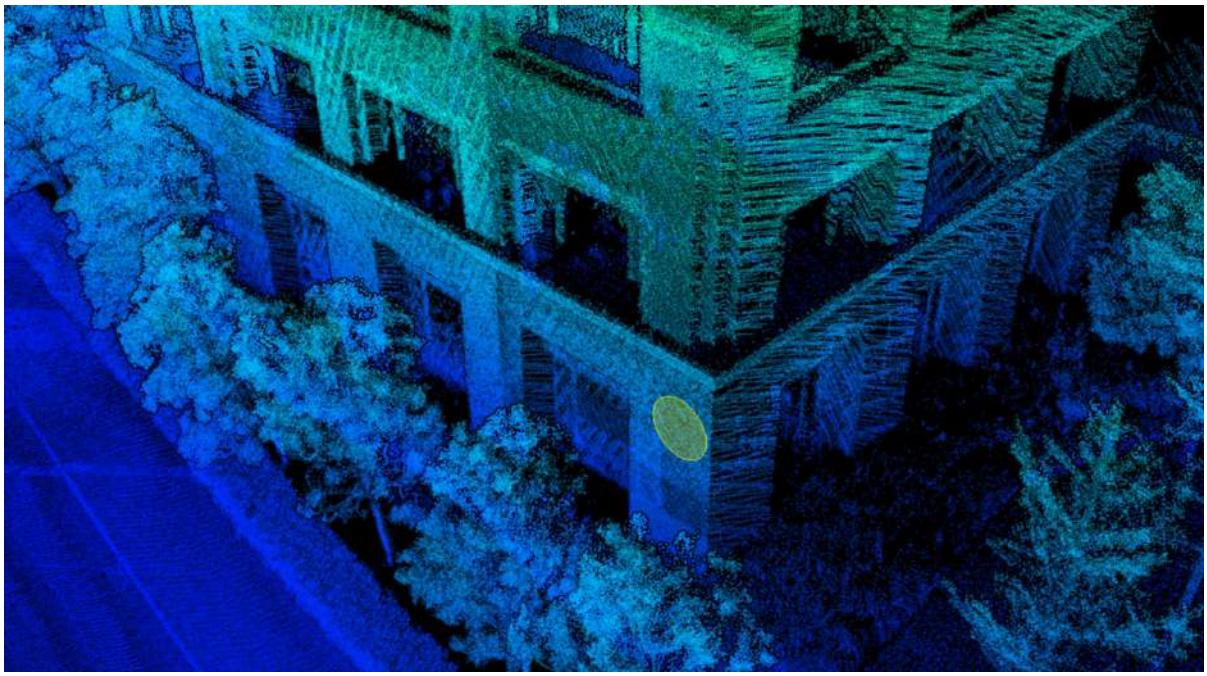
Function Description: The mouse hover query tool takes the point cloud within a certain pixel range for feature fitting at the position where the mouse is currently gliding, which is used to assist in measuring planes, ridges, corners and other local features.

Steps

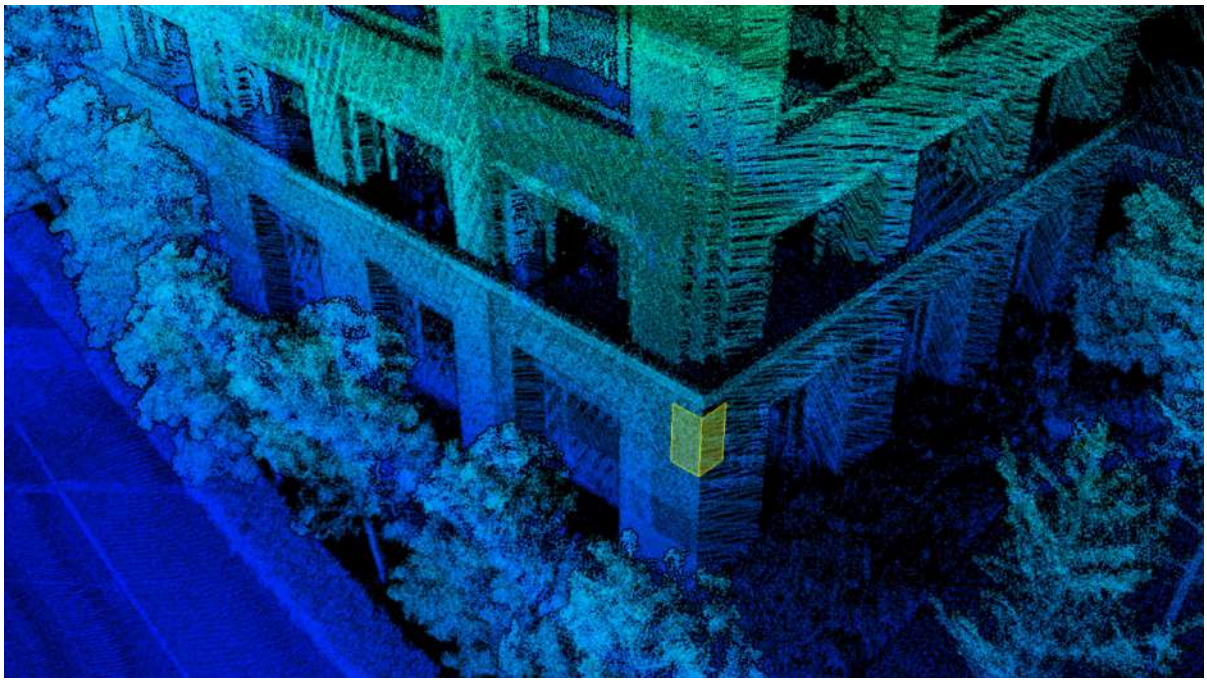
1.Click the **Use mouse hover query** check box in the **Mouse hover** to turn on the hover tool.

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

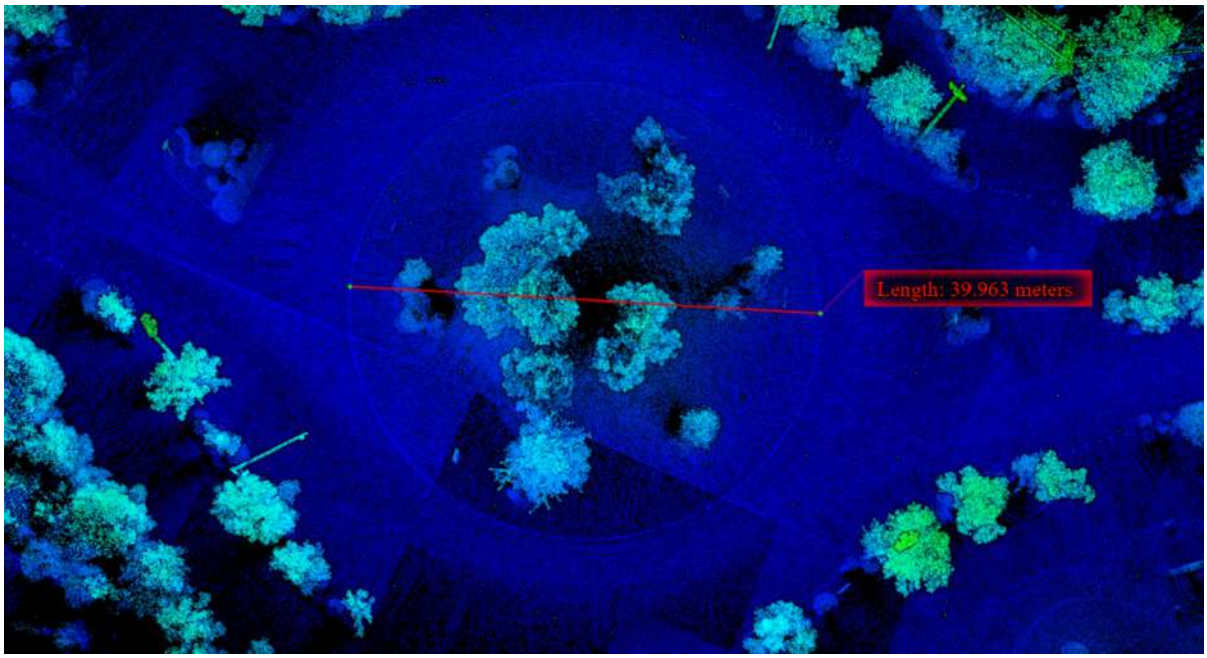
3.The effect of hovering the mouse to detect the plane is shown in the figure.



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



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



Select

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

- Polygon Selection
- Rectangle Selection
- Sphere Selection
- Circle Selection
- Line Above Selection
- Line Below Selection
- Plane
- Subtract
- Plane Above
- Plane Below
- Plane Inside
- Plane Distance Setting
- InCut
- OutCut
- Save and Cancel
- Clip By Polygon

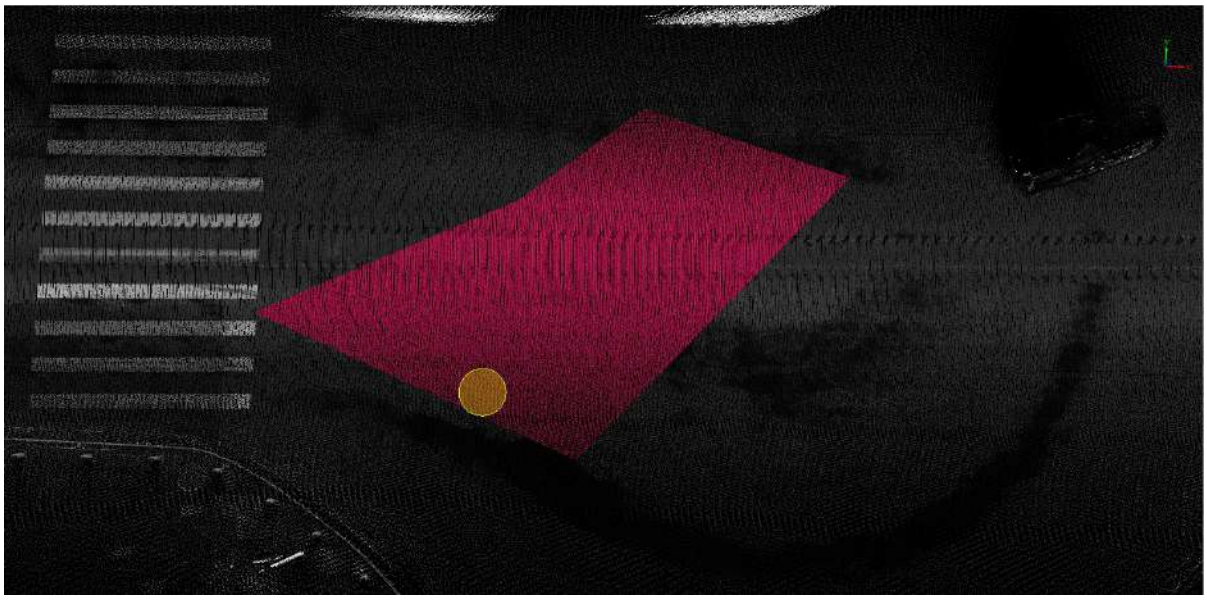
Polygon Selection

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

Steps

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

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



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

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

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

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

Rectangle Selection

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

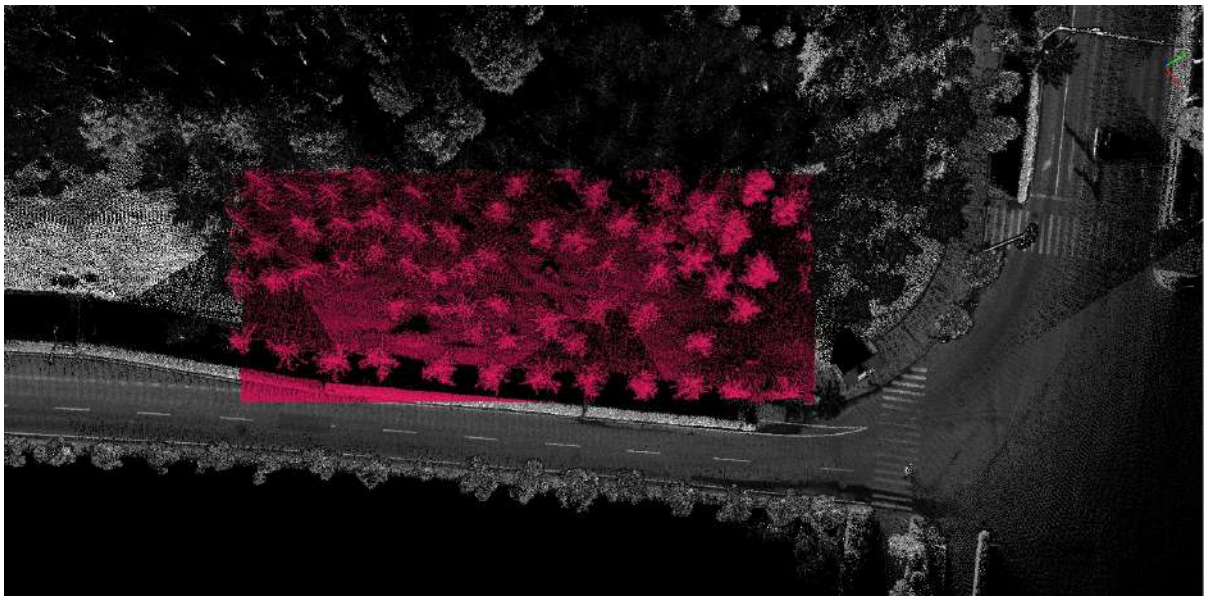
Steps

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

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

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

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



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

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

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

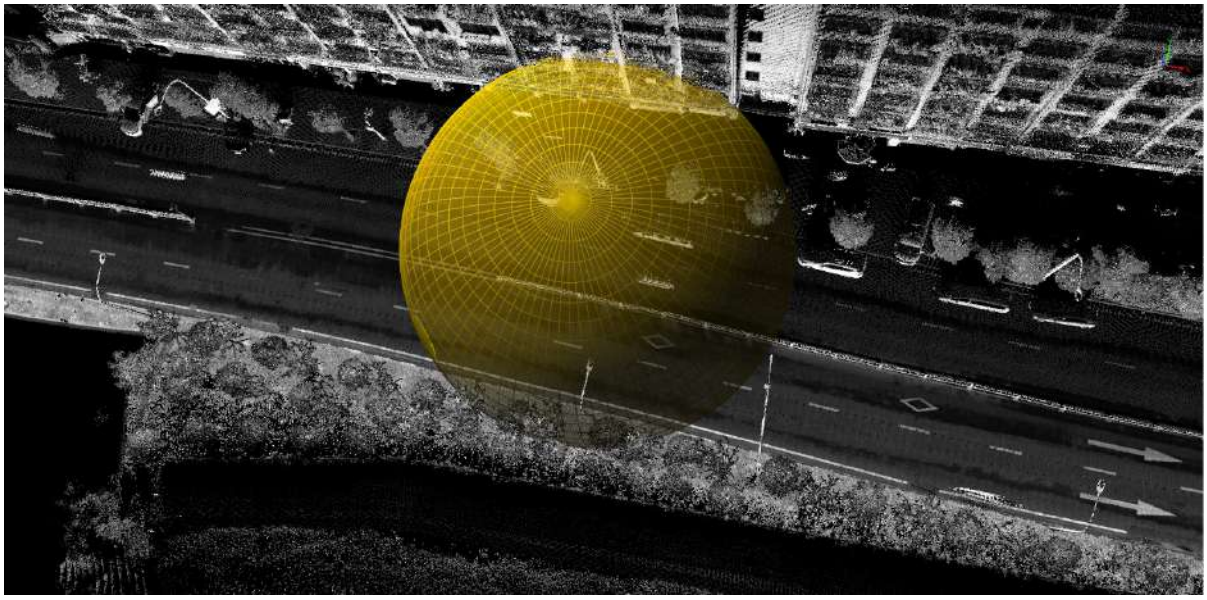
Sphere Selection

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

Steps

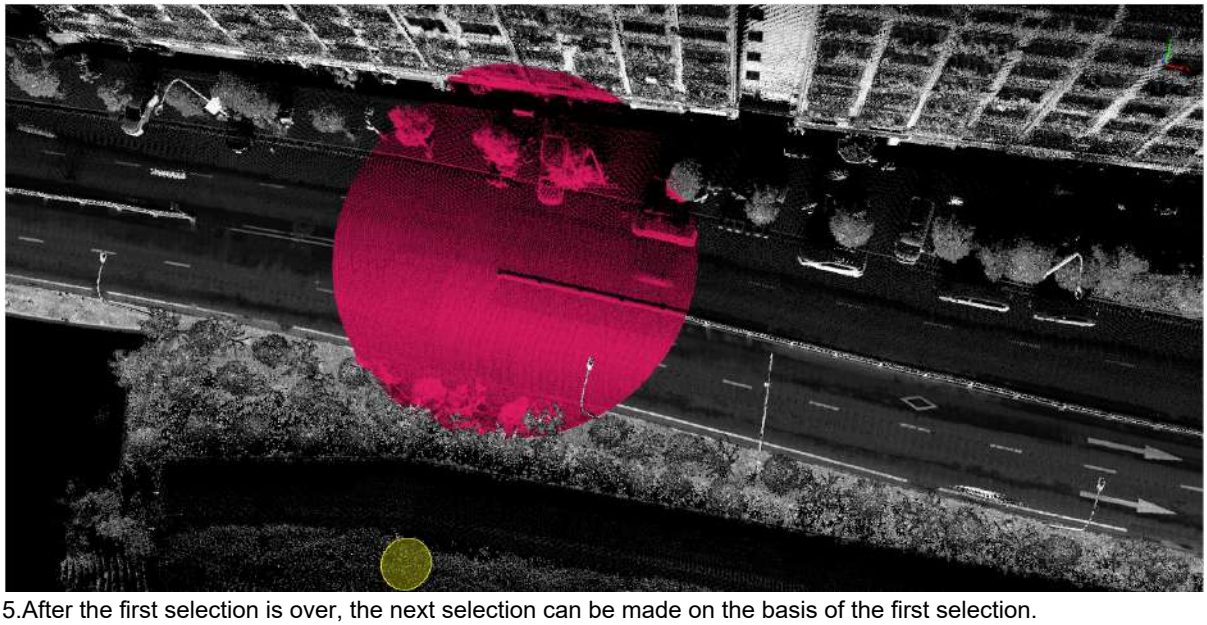
1.After clicking **Sphere Selection** button, the button is in the selected state and the function is activated.

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



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

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



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

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

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

Circle Selection

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

Steps

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

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

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

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

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

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

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

Line Above Selection

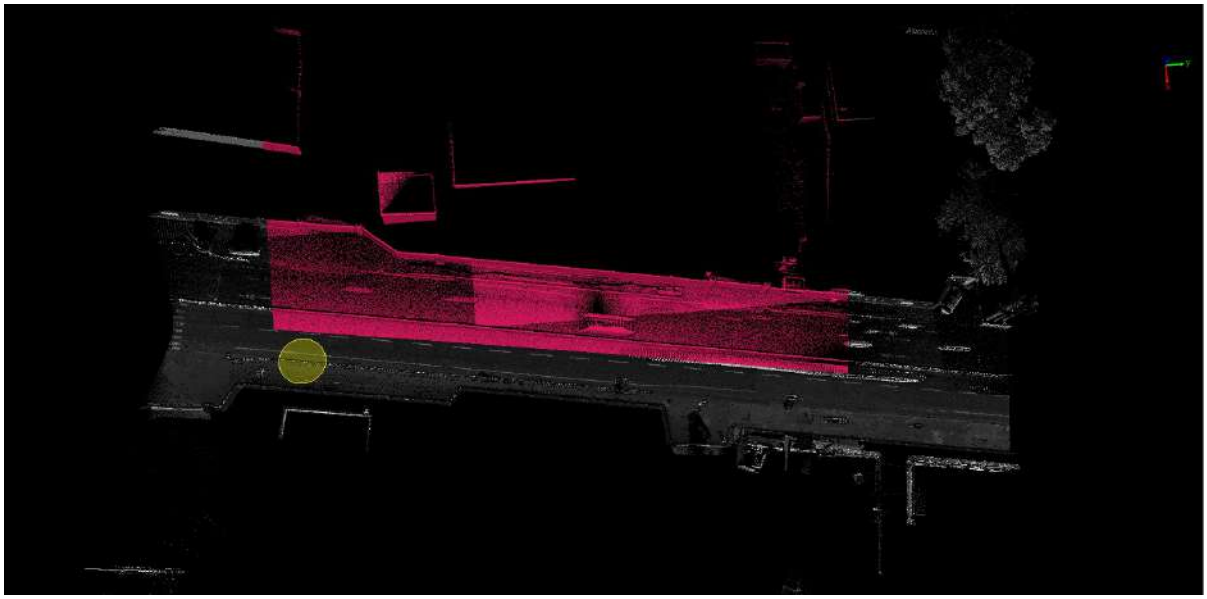
Function Description: This tool allows the user to select the point cloud according to the online method.

Steps

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

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

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



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

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

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

Line Below Selection

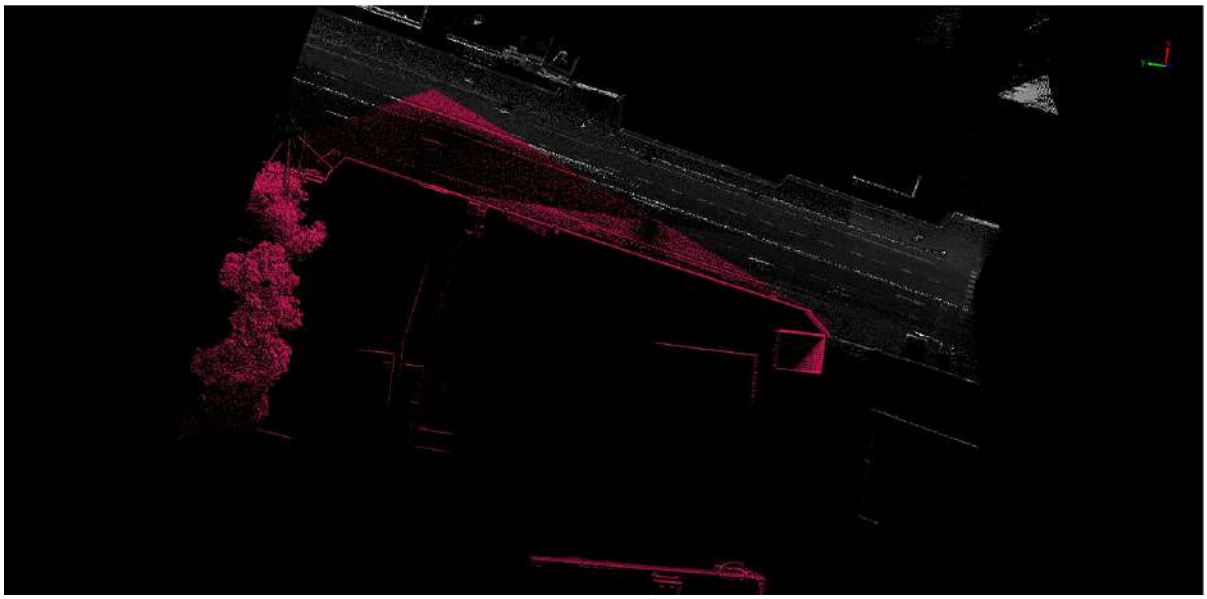
Function Description: This tool allows the user to select the point cloud according to the offline method.

Steps

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

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

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



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

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

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

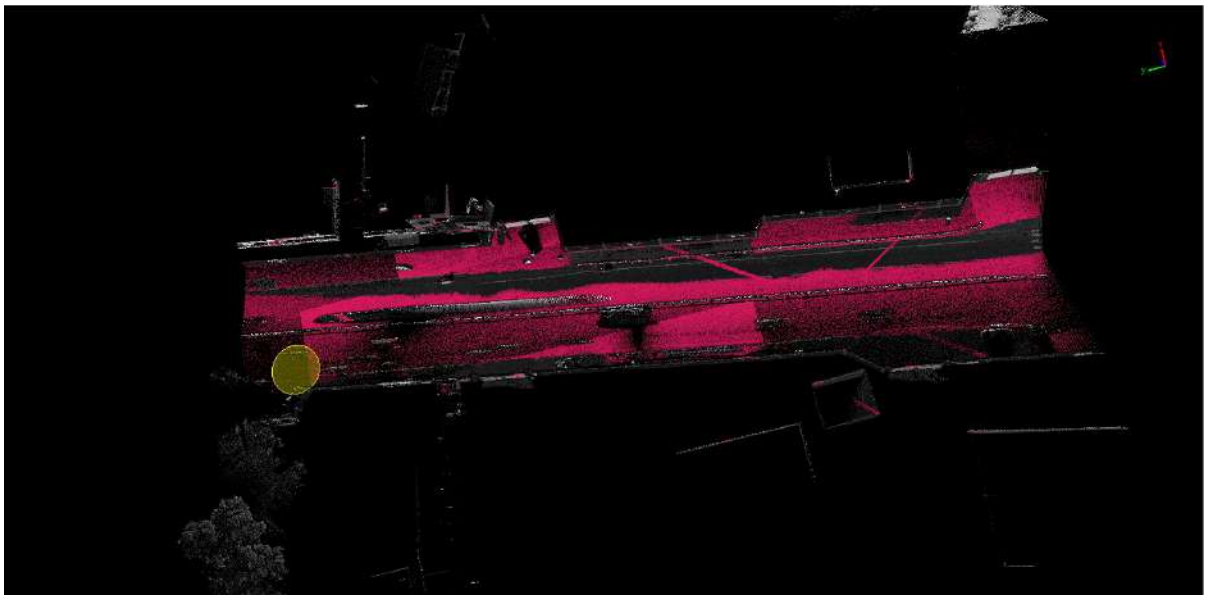
Plane

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

Steps

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

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



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

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

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

Subtract

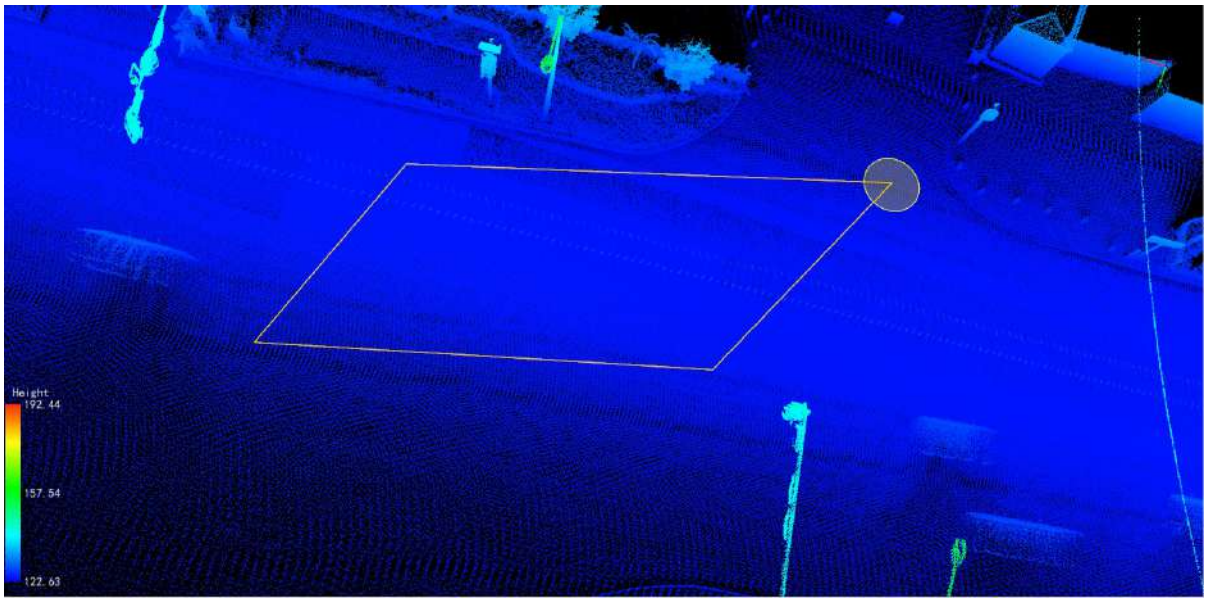
Function Description: This tool allows the user to deselect an existing selection and indicates the current selection state, and is used to control which the selected areas are added or subtracted. This function cooperates with Polygon Selection, Rectangle Selection, Sphere Selection, Circle Selection,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

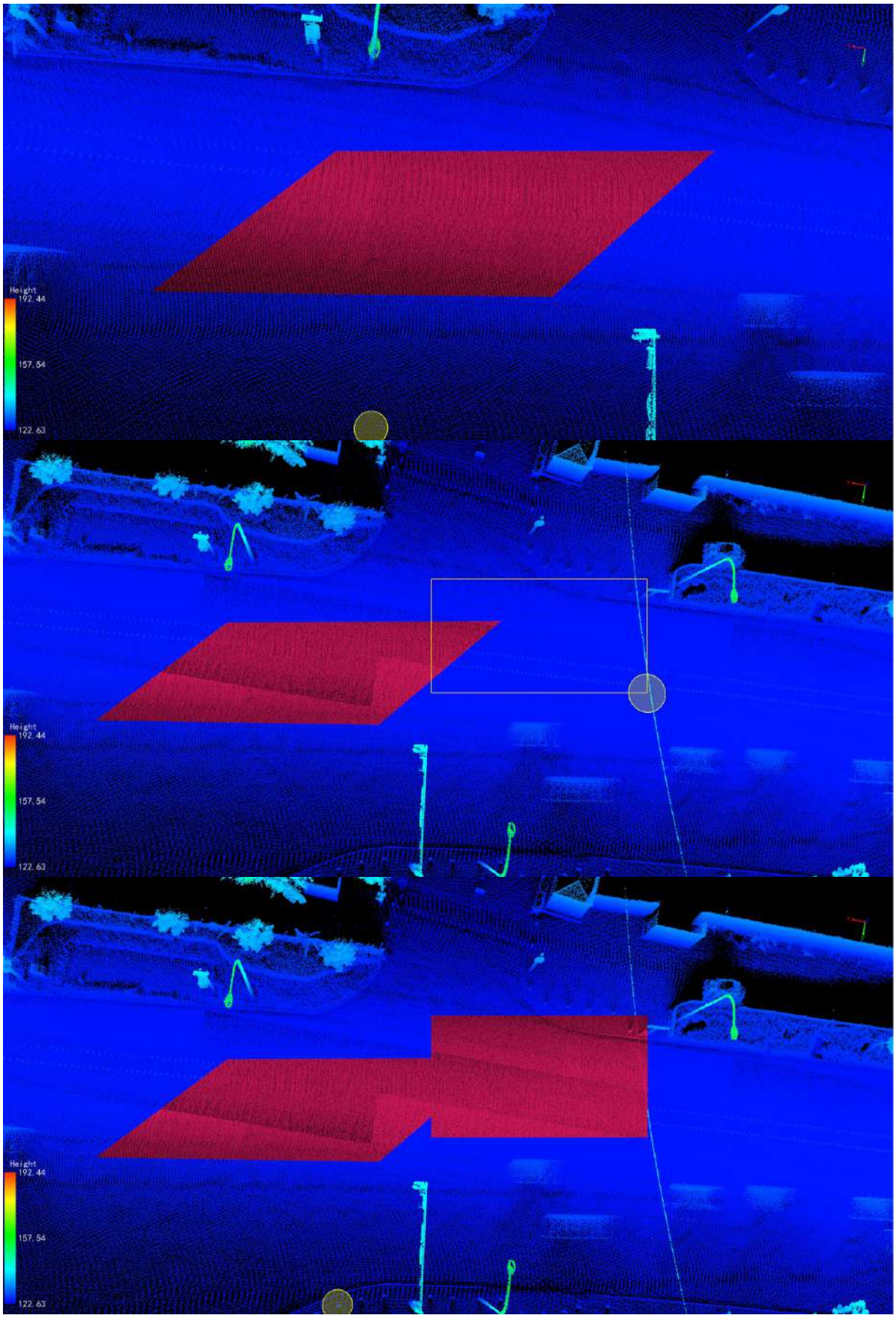
Steps

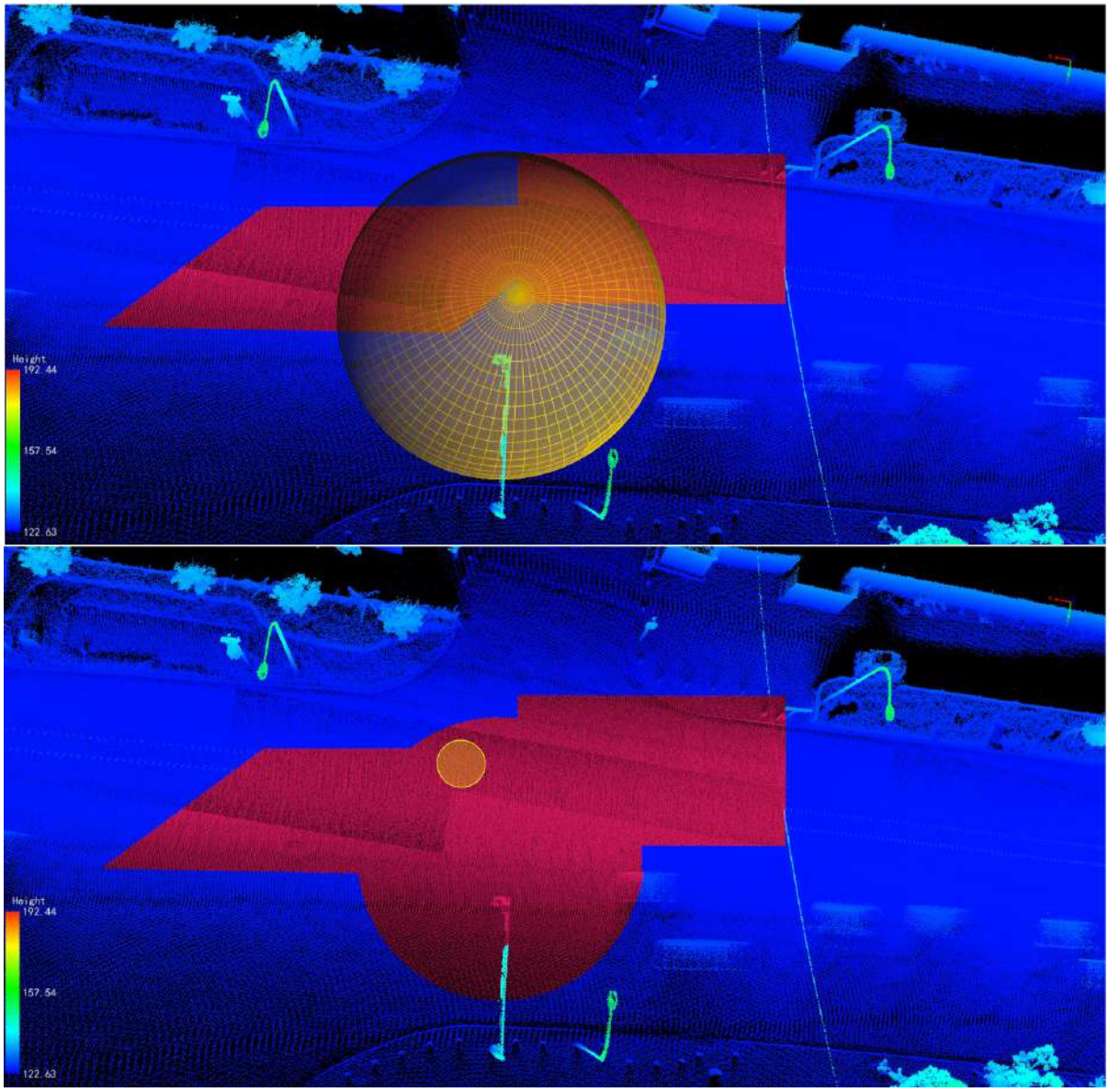
1.After clicking **Subtract** button, the button is in the selected state and the function is activated.

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

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

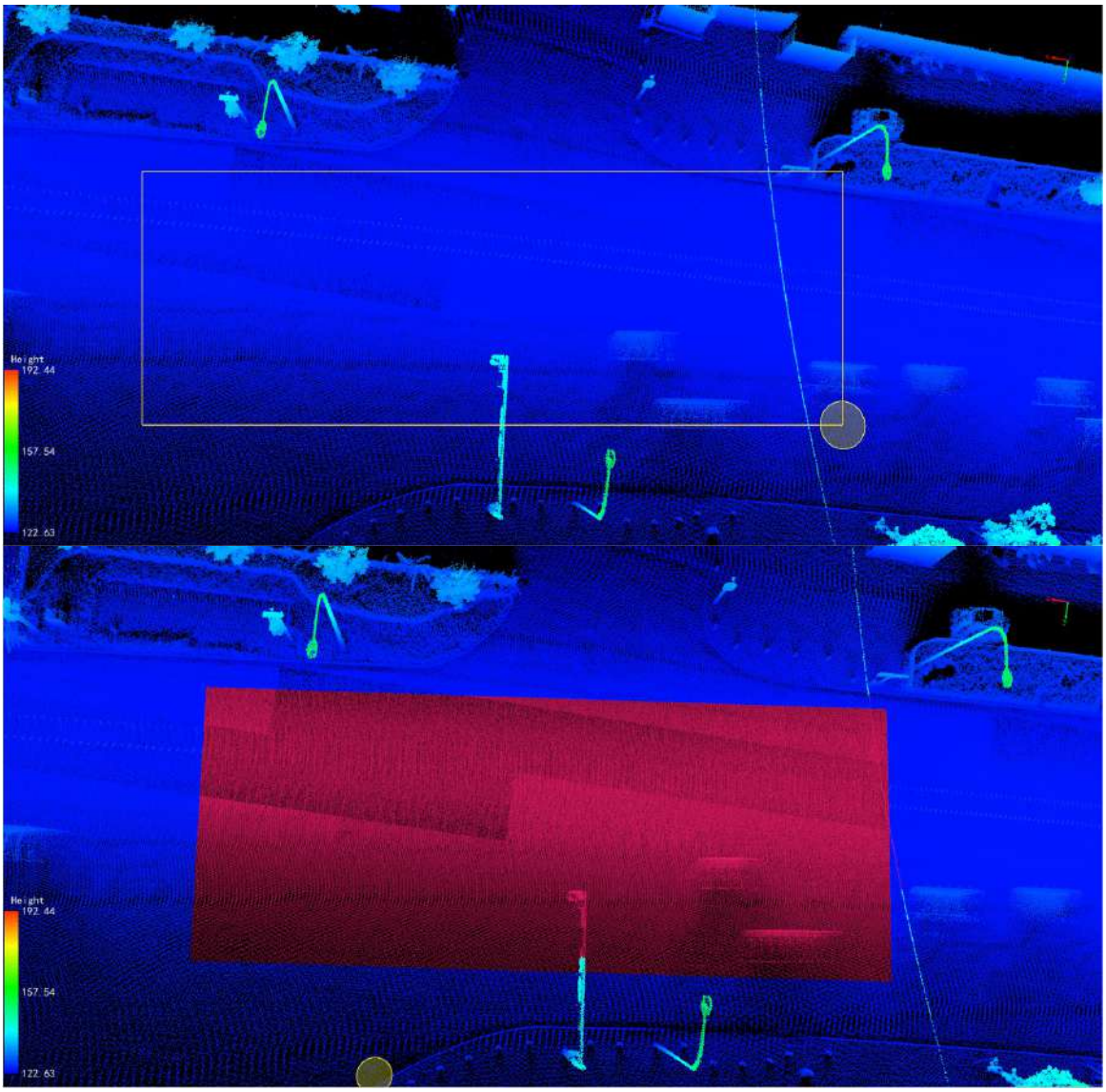



Height
192.44
157.54
122.63
Height
192.44
157.54
122.63
Height
192.44
157.54
122.63

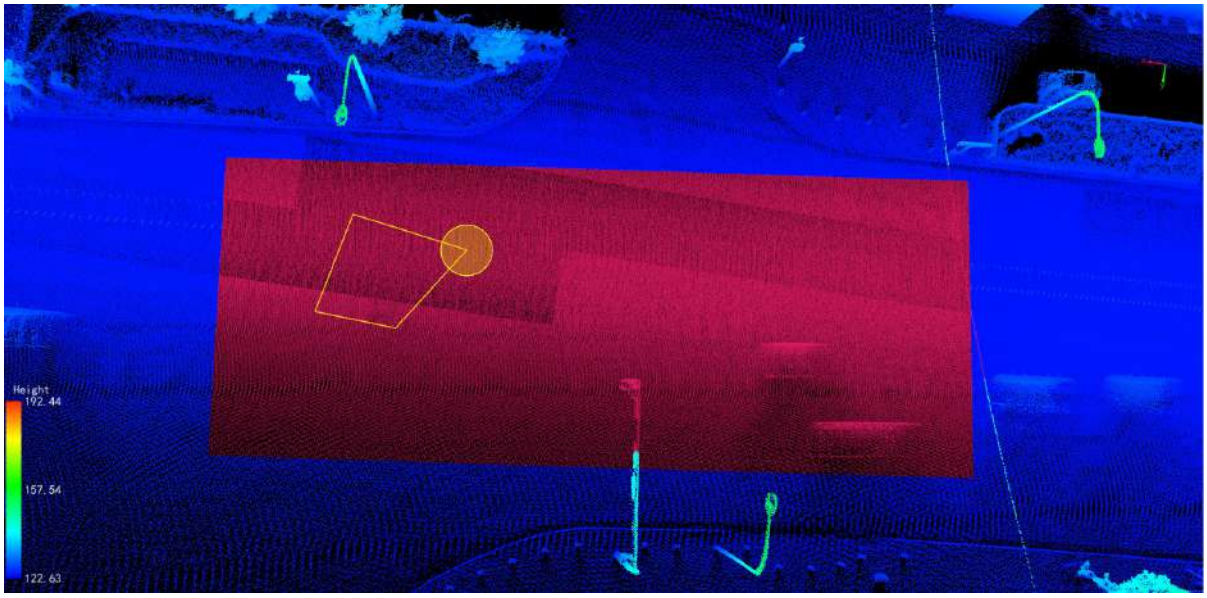


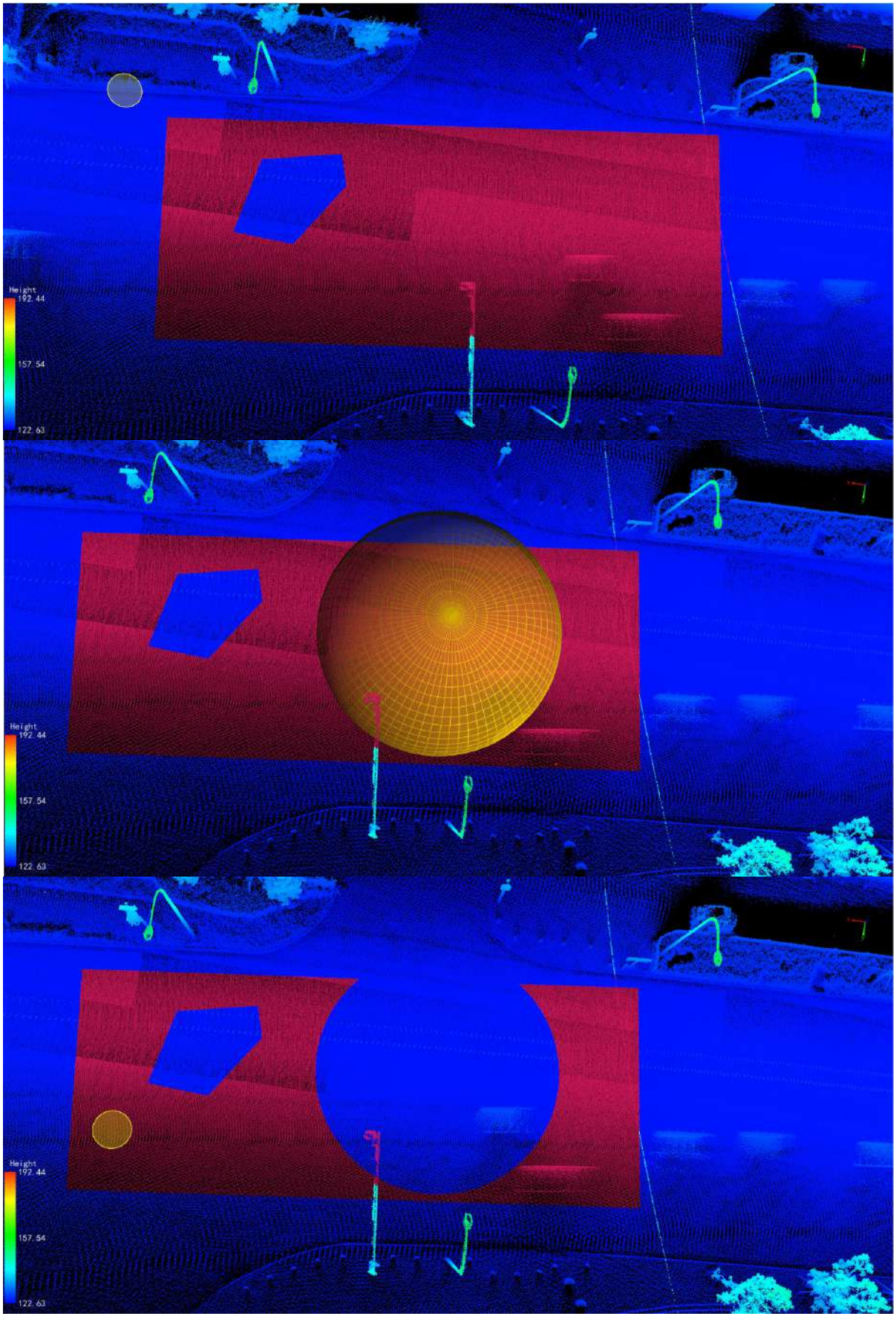
3.Press and select this function. When the button is in the selected state, the selection result is in the minus state, and the selected area is subtracted from the common part of the currently newly selected area as the final selection result.

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



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





Parameter setting

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

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

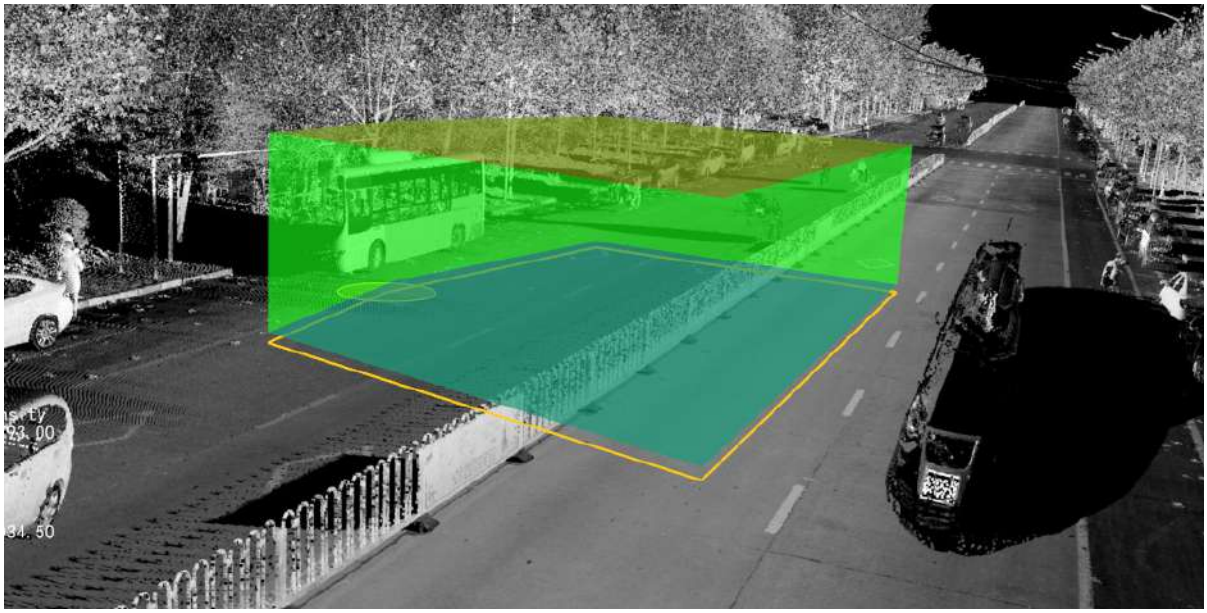
Plane Above

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

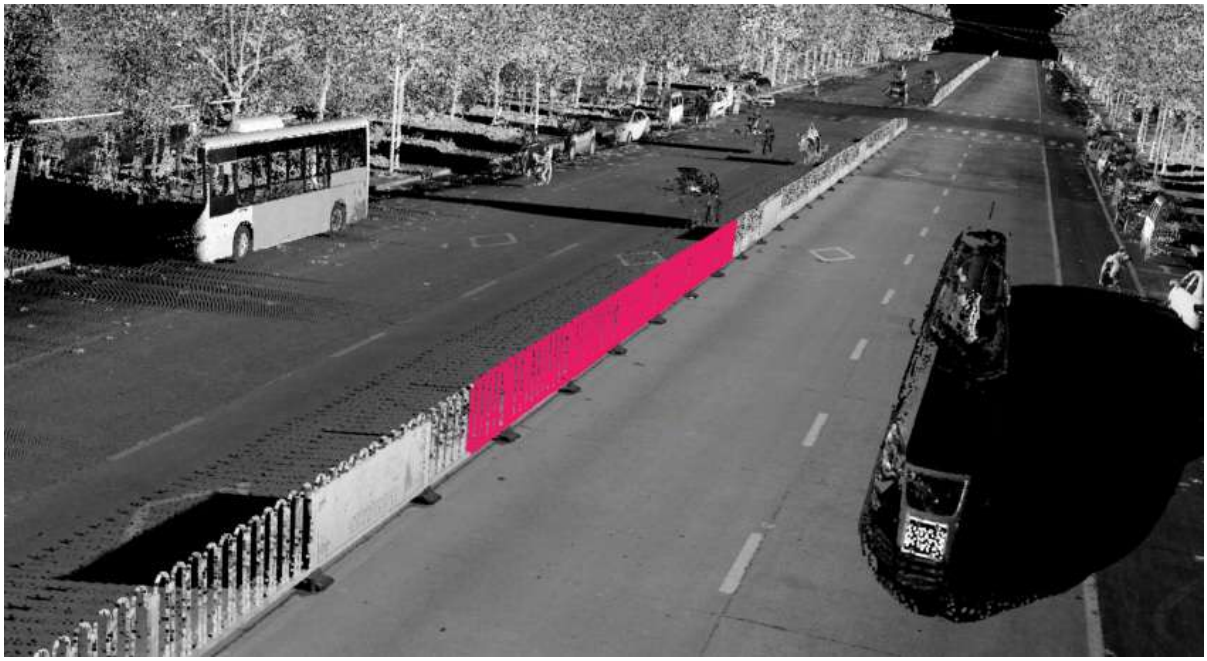
Steps

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

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



candidate region



Result map

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

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

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

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

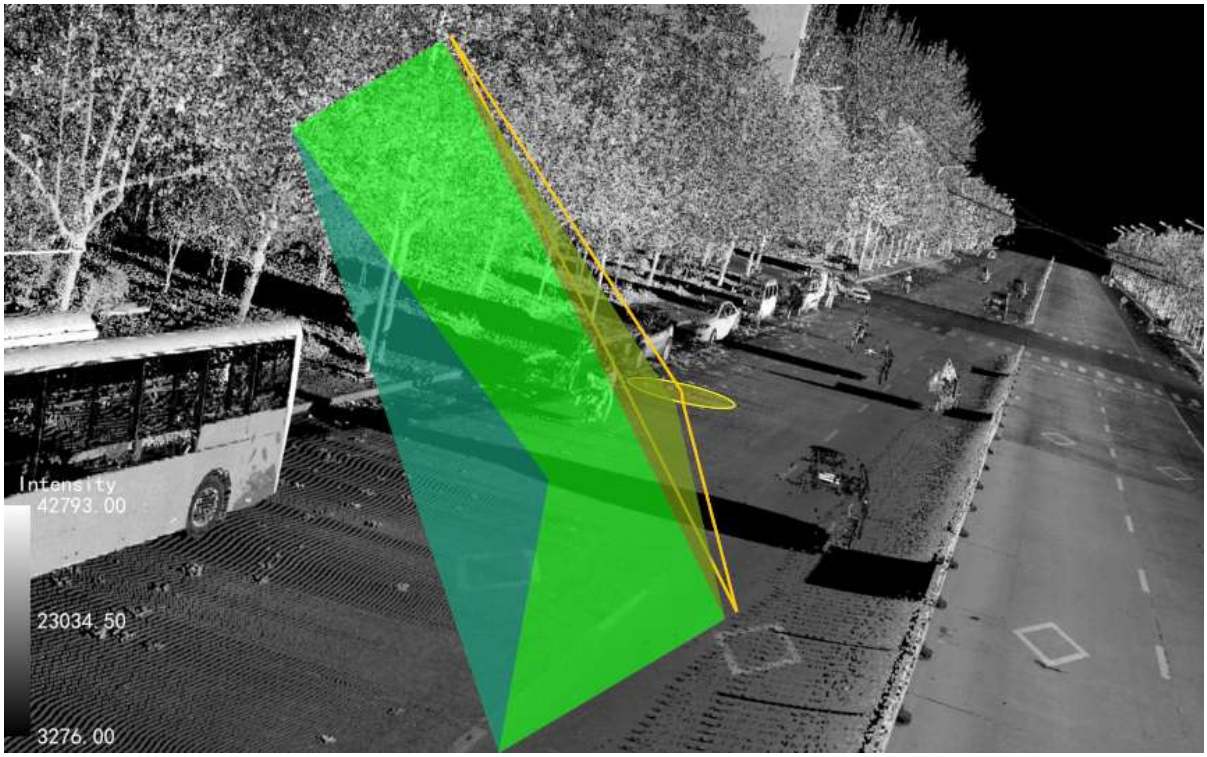
Plane Below

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

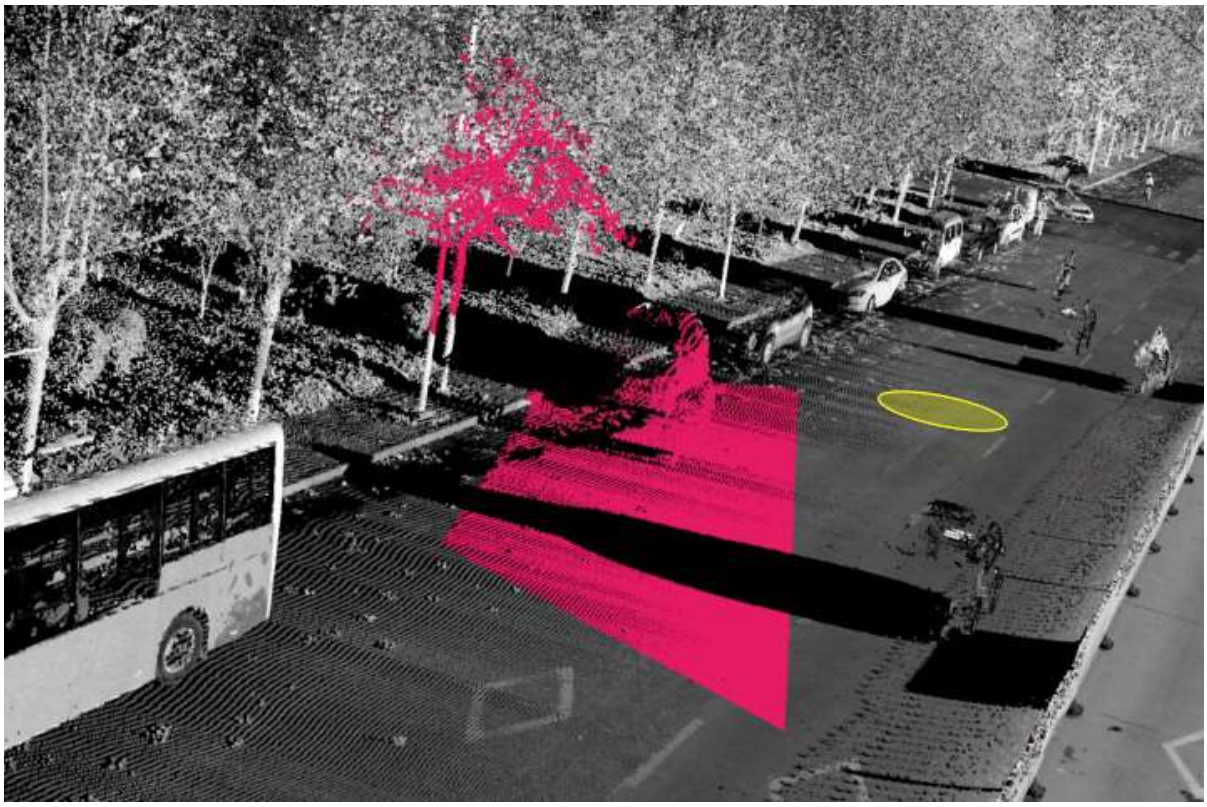
Steps

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

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

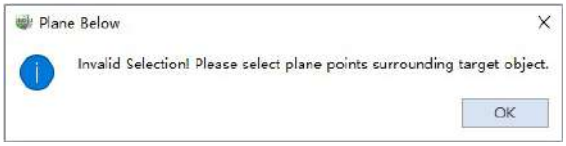


candidate region



Result map

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



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

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

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

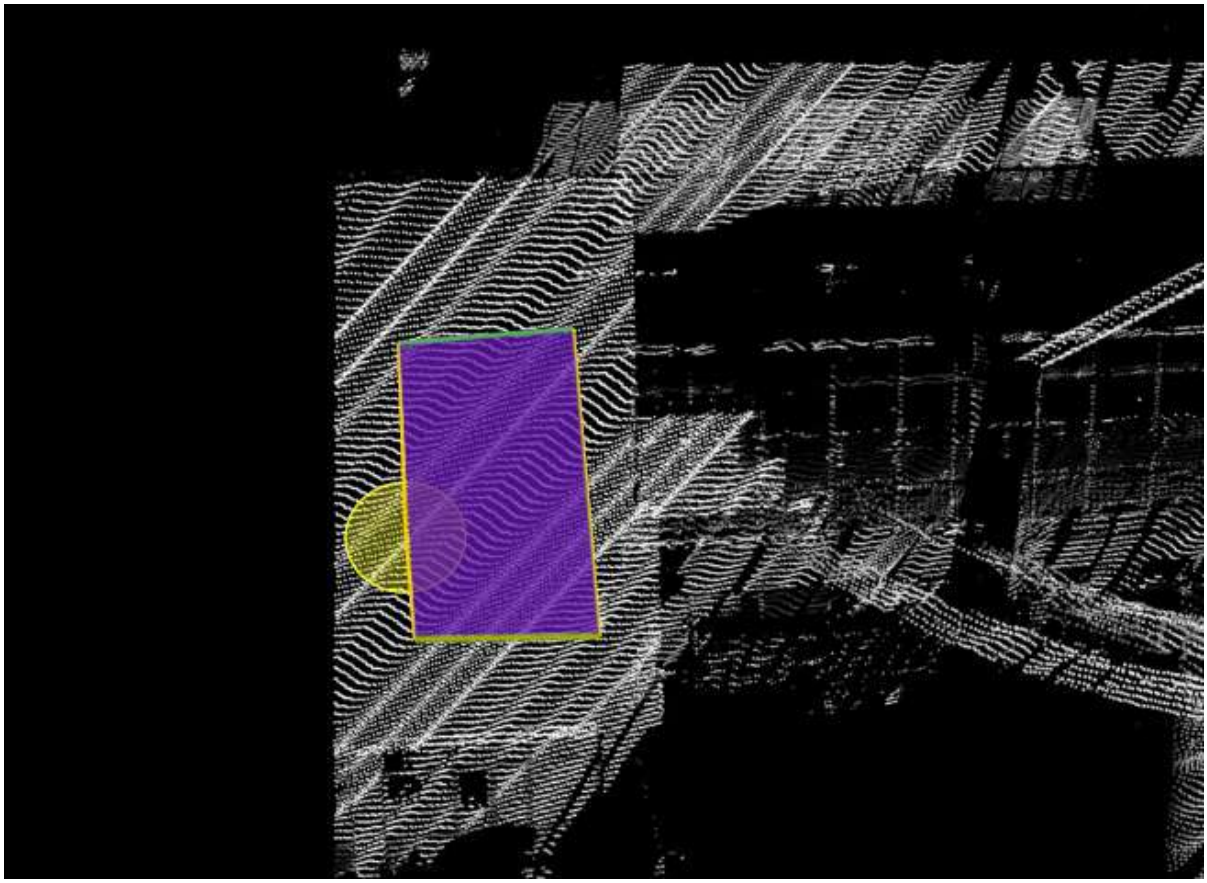
Plane Inside

Function Description：Draw a plane, and select the points inside the plane.

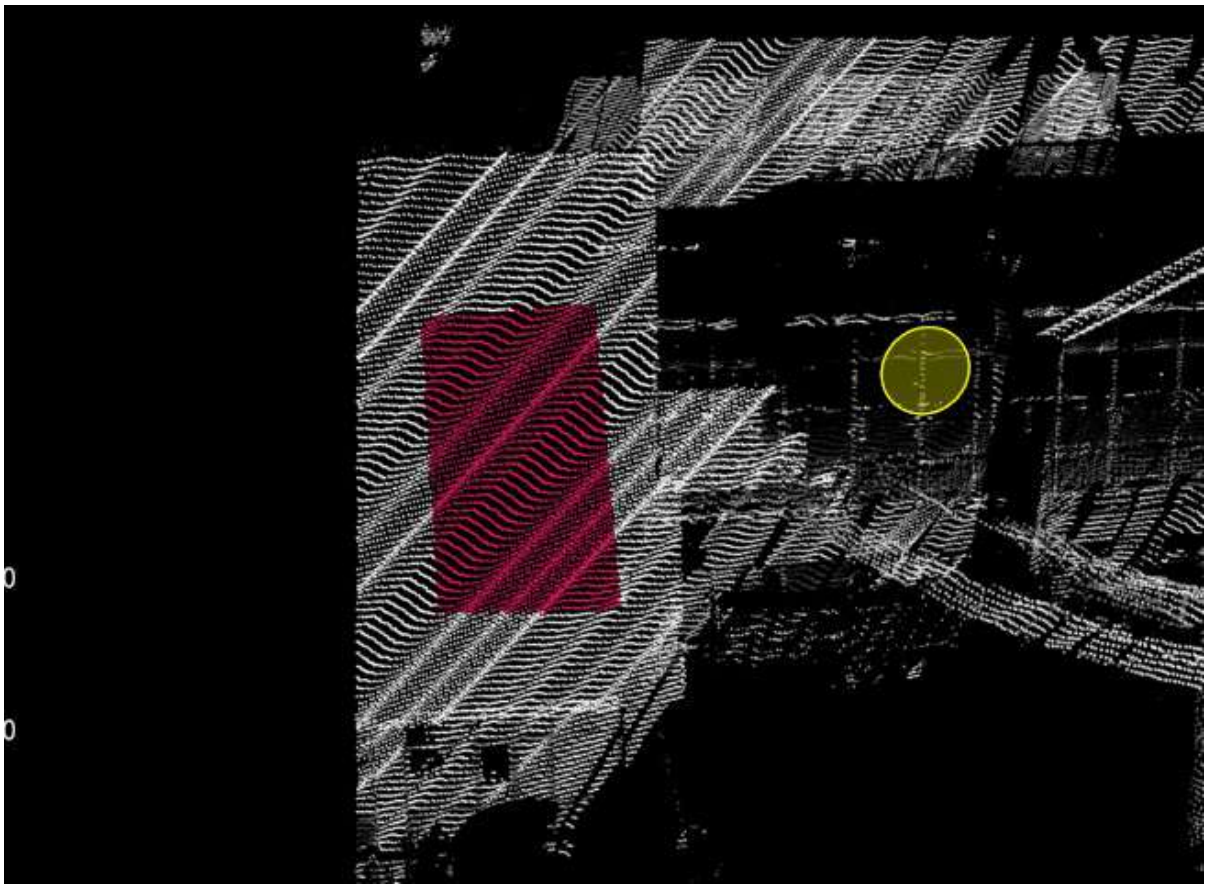
Steps

1.After clicking the **Plane Inside** button, it enters a selected state, activating the function.

2.Identify and highlight (in red) the point cloud within the selected region, as shown in the image.

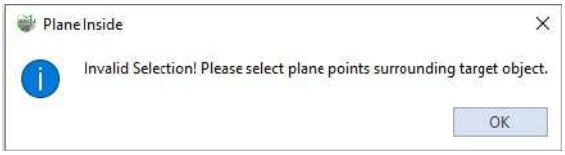


Candidate region



Result image

3.When a plane is drawn and there are no points on the plane, a popup dialog will appear to notify.



Notification

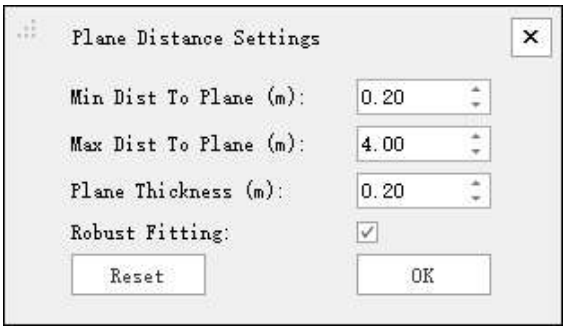
4.After the first selection is completed, you can perform another selection based on the initial selection.

5.Each selection will add or subtract selected regions based on the toggle selection state.

> Note: This feature is applicable only to point cloud data.

Plane Distance Setting

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



Description

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

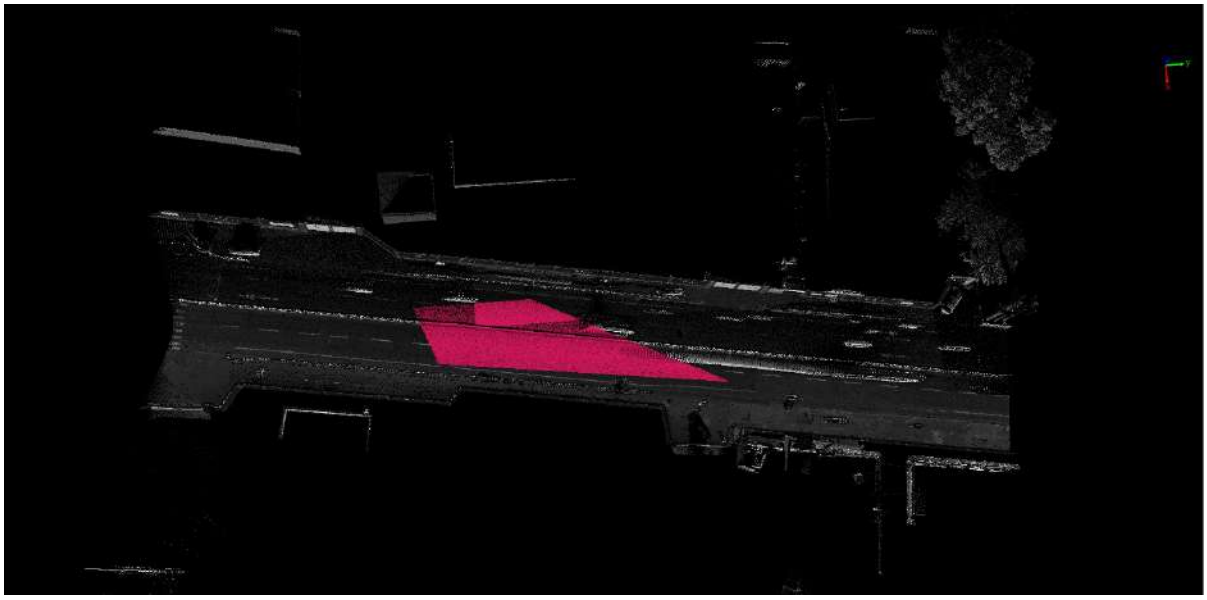
InCut

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

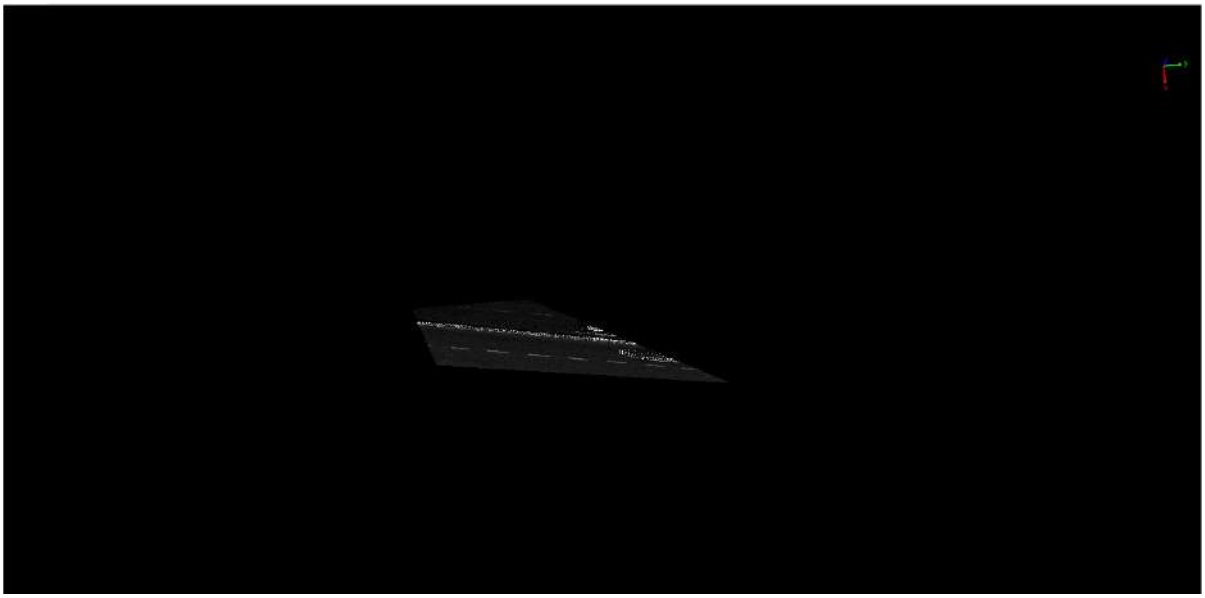
Steps

Click **InCut** button:

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



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



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

Parameter setting

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

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

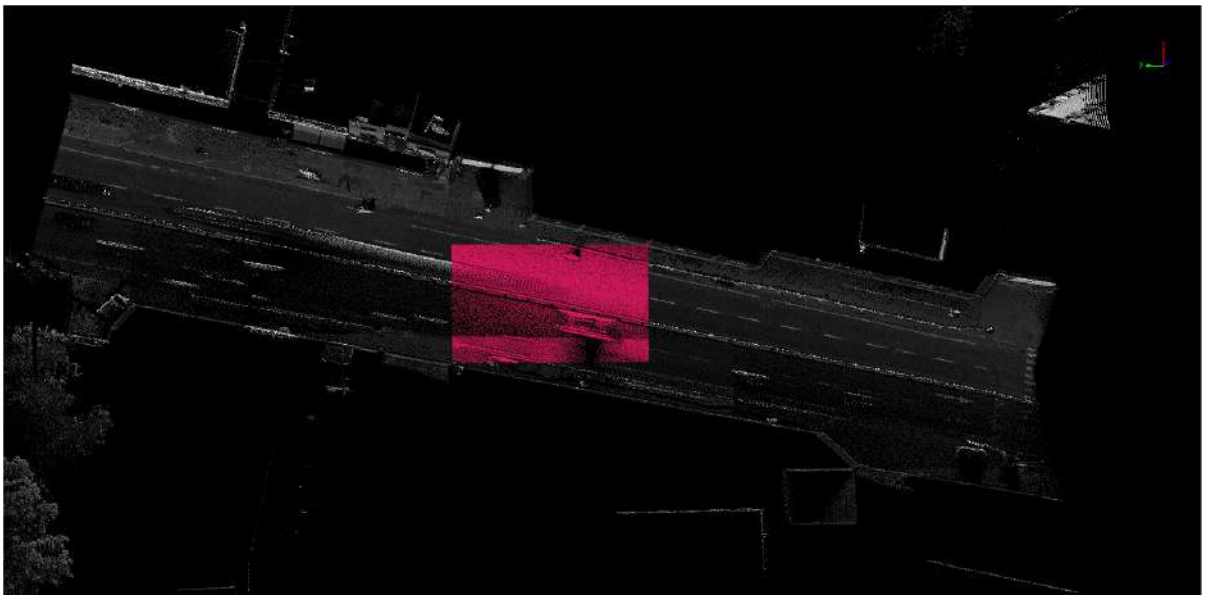
OutCut

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

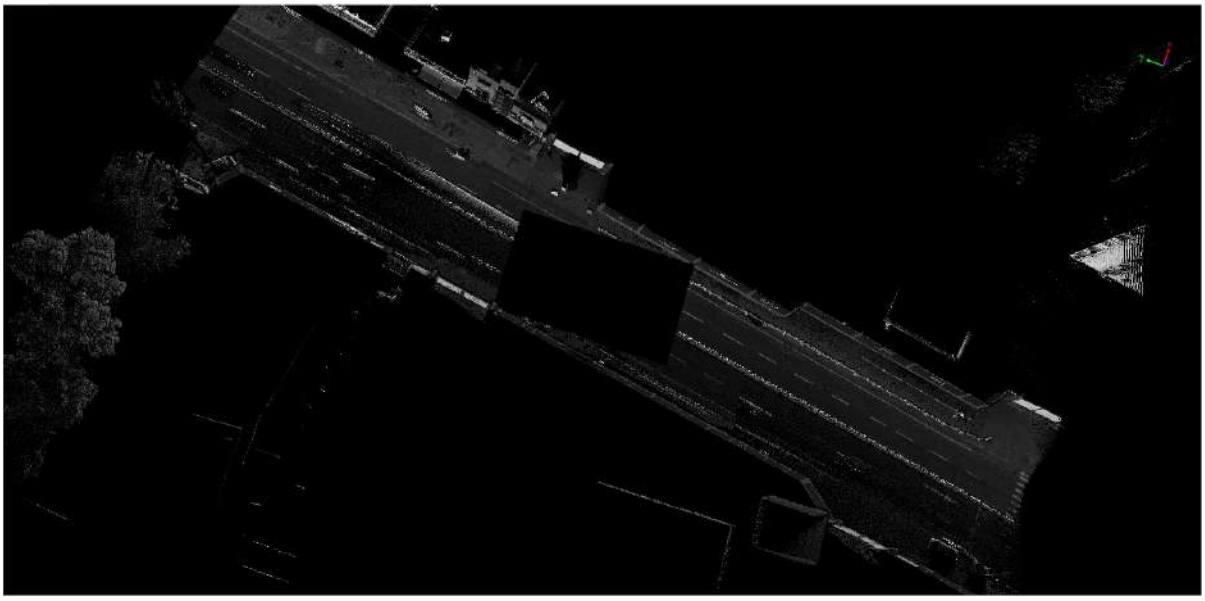
Steps

1.Click **OutCut** button:

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



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



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

Parameter setting

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

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

Save and Cancel

Save

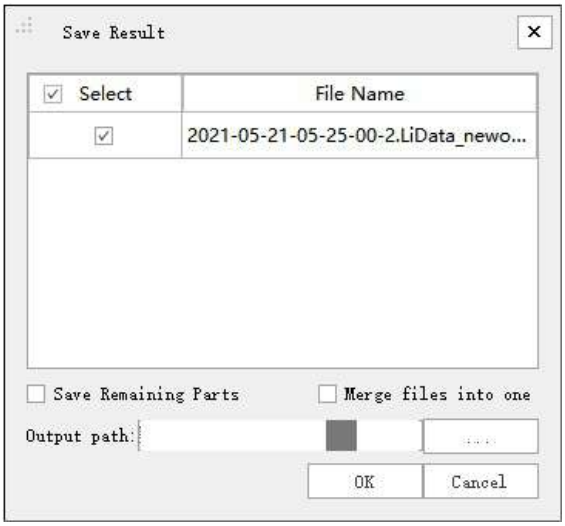
Function Description: This tool allows the user to save the resulting point cloud after cropping into a new point cloud file.

Steps

Click **Save** After the button:

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

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



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

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

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

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

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

Cancel

Click **Cancel** button, cancel the result of the last cropping, and the selection area corresponding to the crop will also be cancelled.

Parameter setting

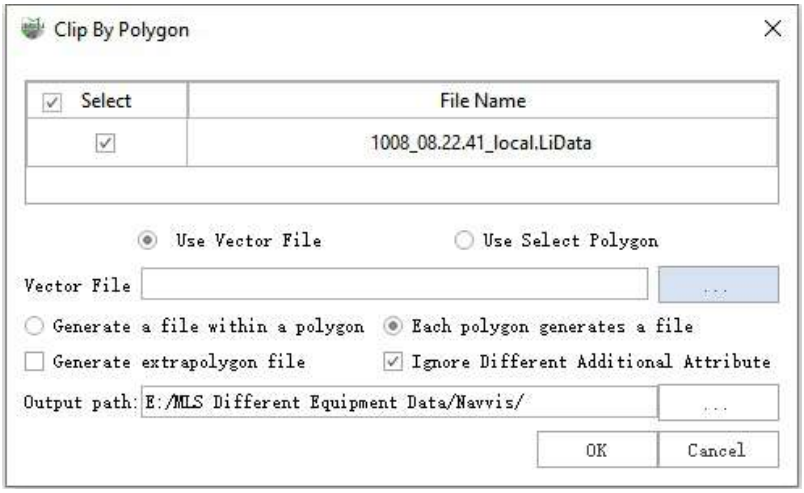
- Shortcut key: Ctrl + Z, cancel the result of the last crop, and the selection area corresponding to the crop will also be cancelled.

Clip By Polygon

Function Description：Clip point clouds using different formats of vector files and polygon vector objects.

Steps

1.Click **Clip By Polygon** button, then the pop-up box pops up:

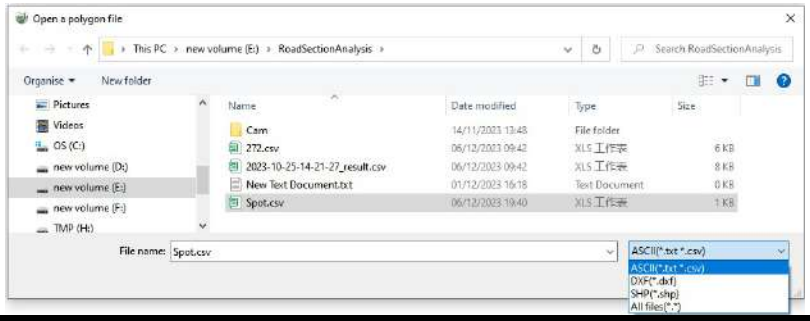


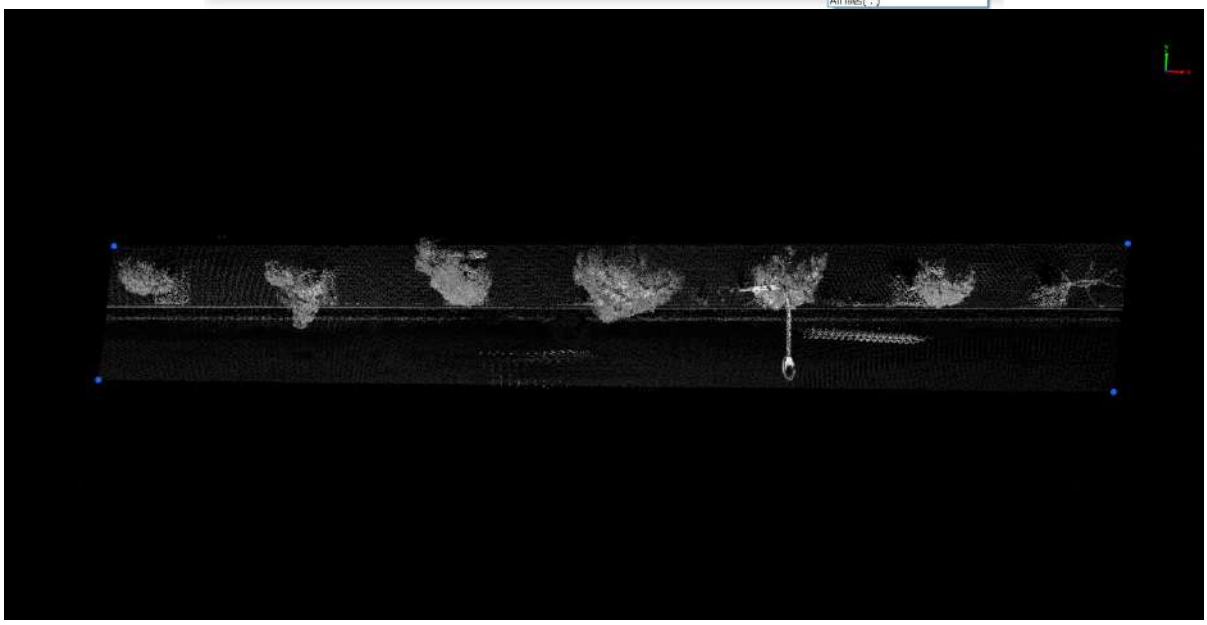
Description

- **Use Vector File**：This mode refers to selecting external files for point cloud data clipping.
- **Vector File**：This mode refers to selecting external files for point cloud data clipping.
  - **.txt、.csv formats**：It must include the planar coordinates of points and should be stored in sequence.
  - **.dxf、.shp formats**：The elements inside must be either polygon features or closed line features.
- **Use Select Polygon**：This mode refers to selecting existing polygon vector objects or closed lines in the project for clipping; multiple selections can be made by holding down Ctrl.
- **Generate a file within a polygon**：When selecting output as a single file, extract all polygon data from the vector file to clip the point cloud and integrate the results into a single file."
- **Each polygon generates a file**：When selecting output as multiple files, extract all polygon data from the vector file to clip the point cloud, generating a separate file for each segment of the point cloud."
- **Generate extrapolygon file**：When this option is selected, points outside the polygons will also be generated into a separate file.
- **Ignore Different Additional Attribute**：In the case of processing multiple input point clouds, if this option is selected, only identical additional attributes will be merged. Otherwise, merging will not be possible as long as there are any differing additional attributes.
- **Output path**：Output folder path, after the operation is executed, it generates newly clipped files."

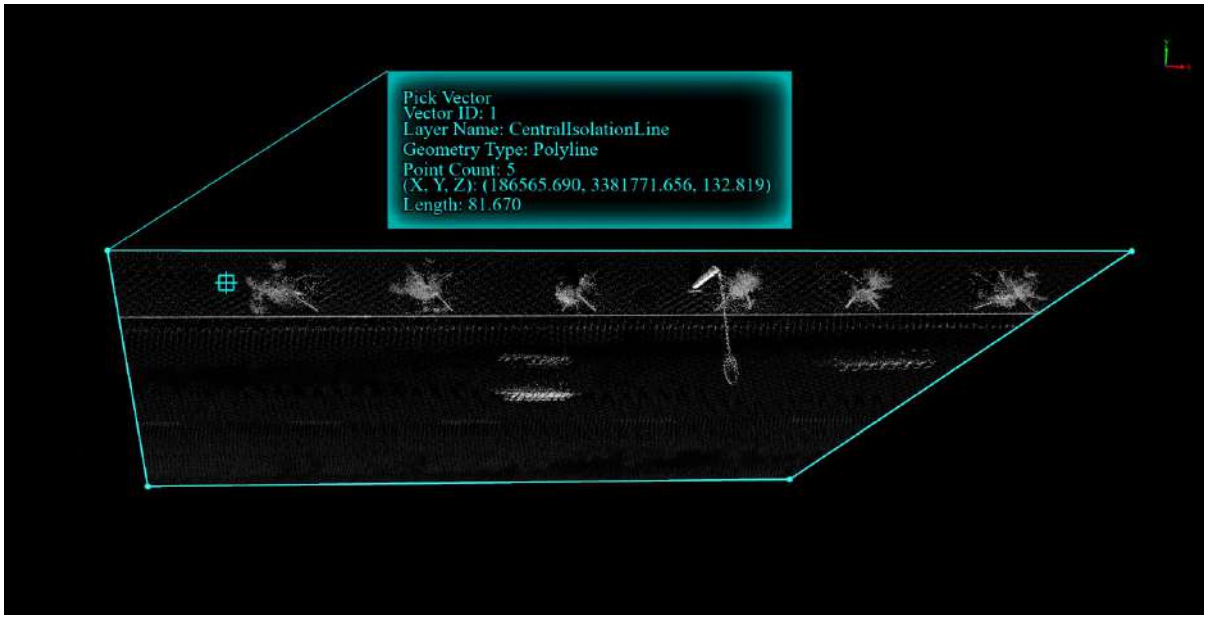
2.Clipping effects using different methods:

（1）Use Vector File：Supports point vector files in ***.txt** and ***.csv** formats, and closed boundary line vector files in *.dxf and* .shp formats. In point vector files, vector elements need to be connected and in the same order.

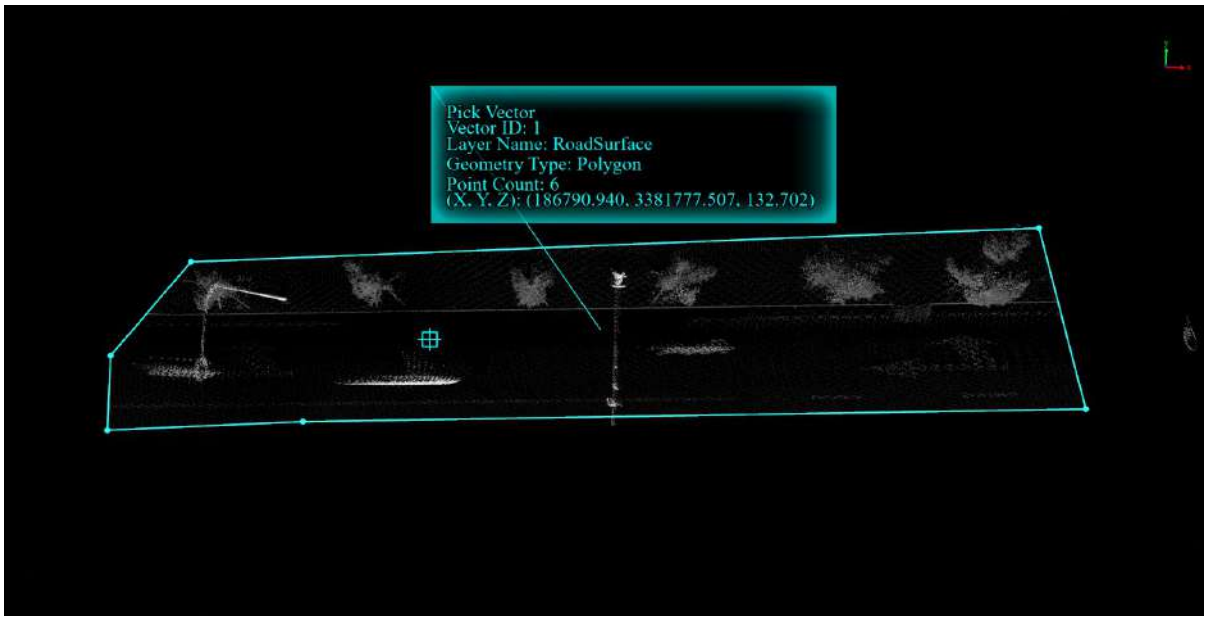




Point Vector File

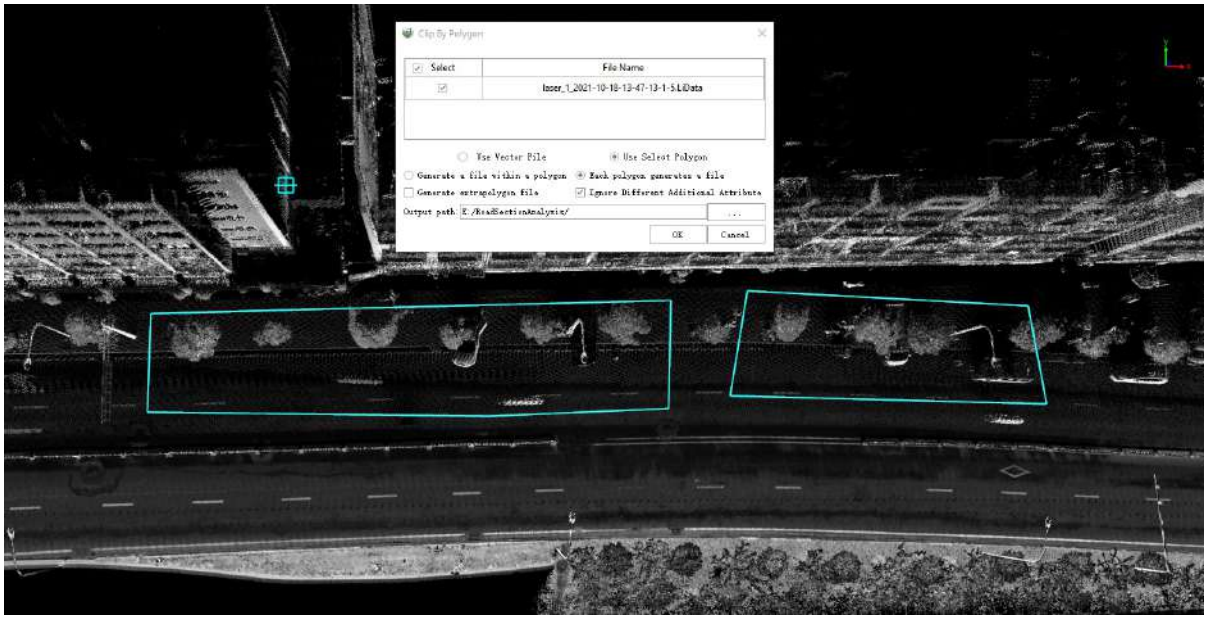


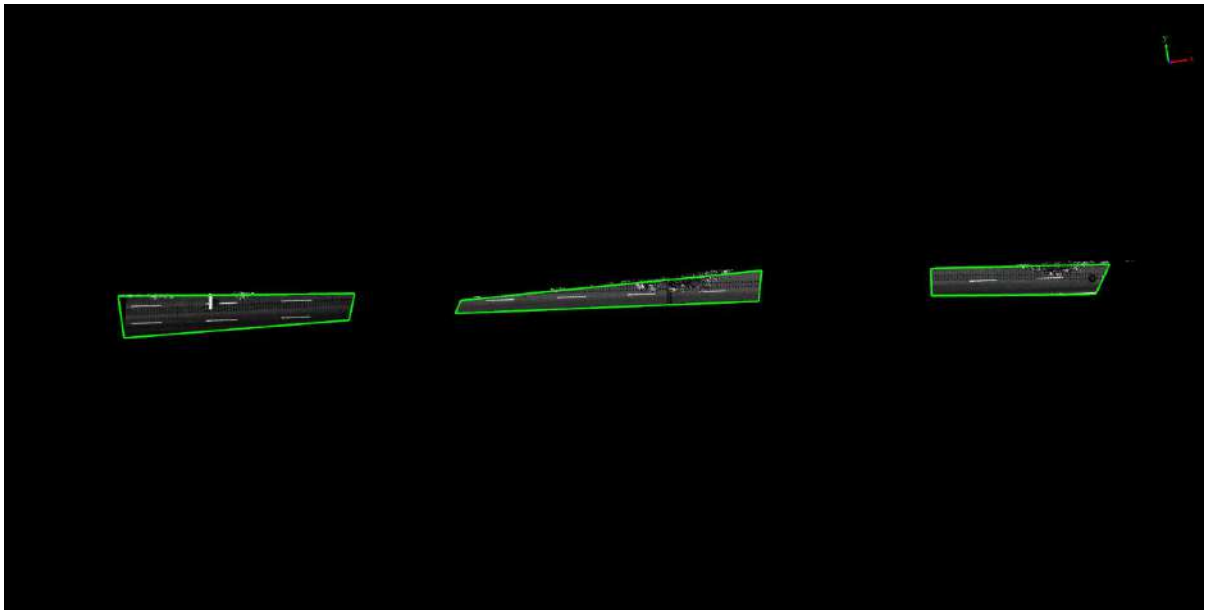
Line Vector File



Polygon Vector File

（2）Use Select Polygon：Holding down Ctrl allows you to select multiple polygons, and holding down Ctrl again allows you to deselect the selected vector objects.





Use Select Polygon

Cross Section

The functions included in the cross section are：

Cross Section

Horizon Section

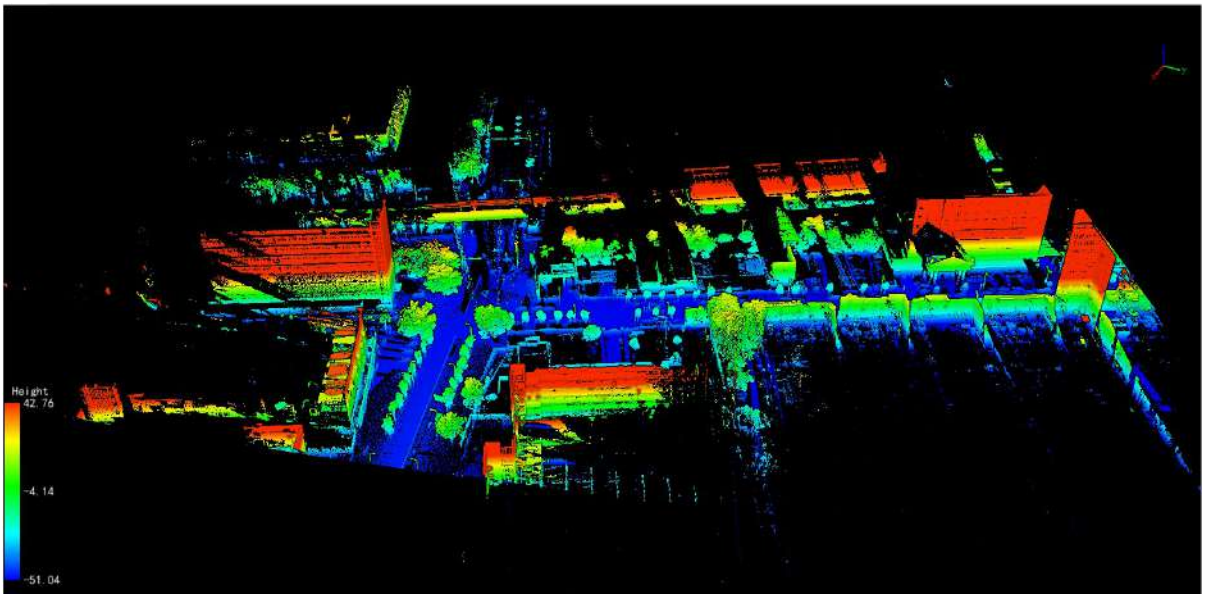
Cross Section

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

Steps

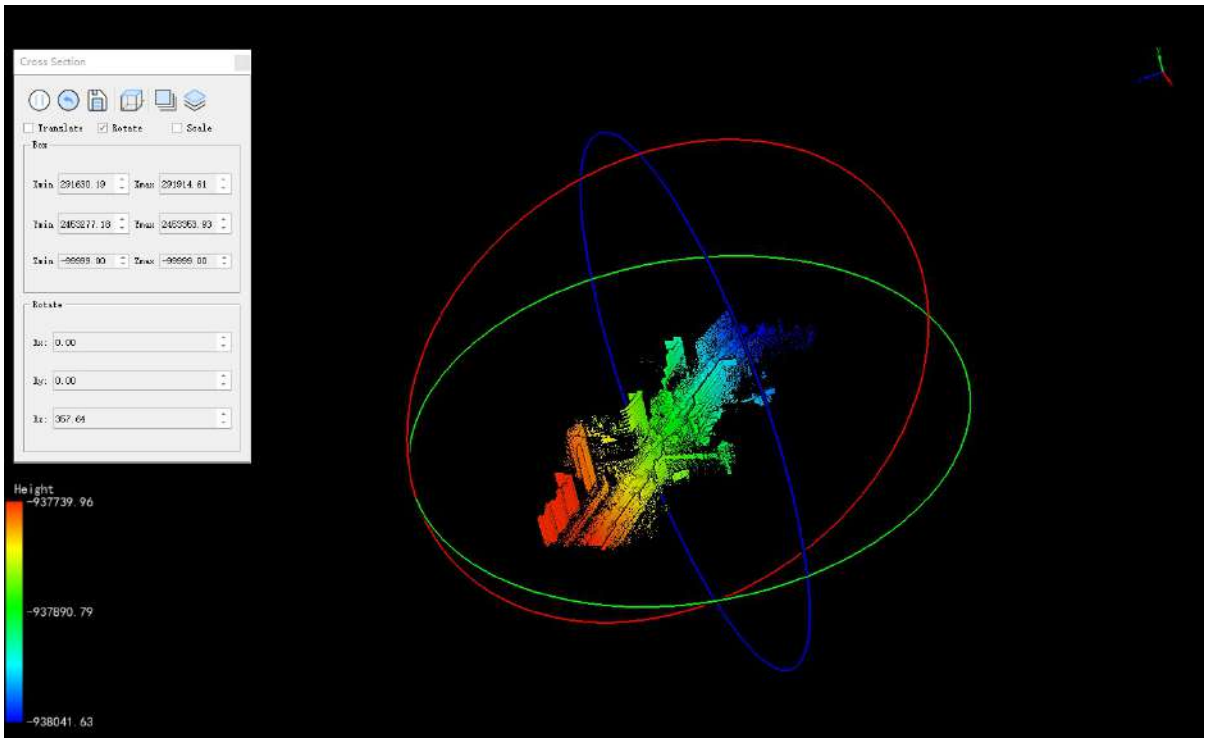
1.Open the LiDAR360 MLS software and load the corresponding project file.

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



The point cloud is displayed as elevation+EDL mode

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



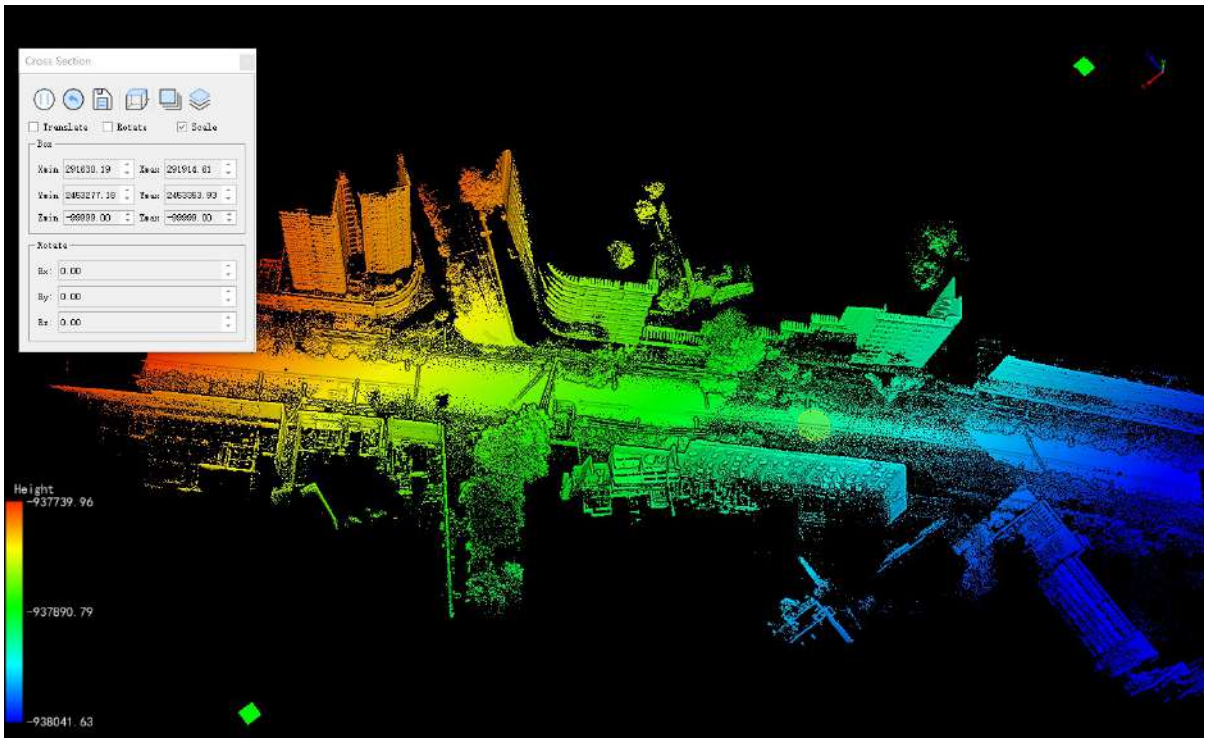
Adjust the enclosure

Parameter setting

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

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

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



Cut according to the z axis

6.Click button to save the cropped point cloud.

7.Click button to pause the cross-selection.

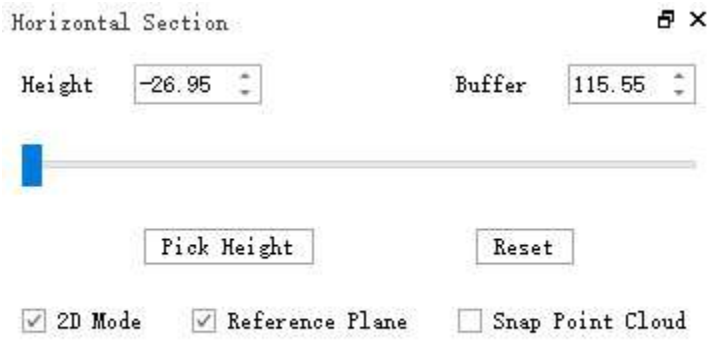
8.Click button sets the view mode selection and displays each default view of the currently activated view.

Horizon Section

Function Description：The Horizon Section tool is a basic tool that allows quick vertical sectioning of point clouds to assist other functions.

Steps

1.Open the LiDAR360 MLS software and click on the Horizon Section button to open the corresponding sidebar. By default, the 2D mode and Reference Plane options are enabled, while the Point Cloud Snap option is disabled.



Horizon Section Setting

Parameter Settings

- **Height (Translate)**: Controls the position of the horizontal section. By default, it is set to the lowest point of all the point clouds in the view.
- **Buffer (Rotate)**: Controls the thickness of the vertical section. By default, it is set to the difference between the highest and lowest points of all the point clouds in the view.

- **Select Height (Scale)**: When activated, allows selecting a point in the point cloud to set the height value to the elevation of the selected point.
- **Reset (Box)**: Resets the height and buffer parameters to their default values.

- **2D Mode (Rotate)**: Locks the view to a top-down perspective with a perspective projection, synchronized with the toolbar's 2D Mode option.
- **Reference Plane (Rotate)**: Enables or disables a virtual plane aligned with the z-axis at the same elevation as the Height parameter.
- **Point Cloud Snap (Rotate)**: Synchronizes with the toolbar's Point Cloud Snap option.

2.Manually enter the height or use the slider located below to adjust the section height. Optionally, input the point cloud buffer thickness. The point cloud horizontal section will be previewed in real-time.

3.Enabling the Reference Plane is necessary for collecting baseline measurements in the Elevation Measurement tool.

Roaming

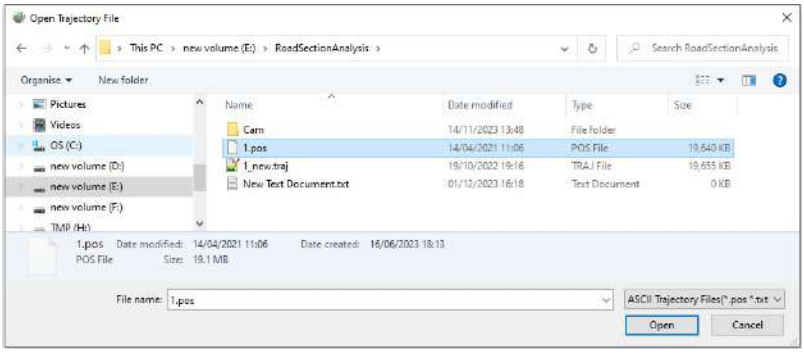
The Roaming toolset primarily allows the user to track roaming and camera roaming. Just switch the top menu bar to **tool**.

Trajectory Roaming

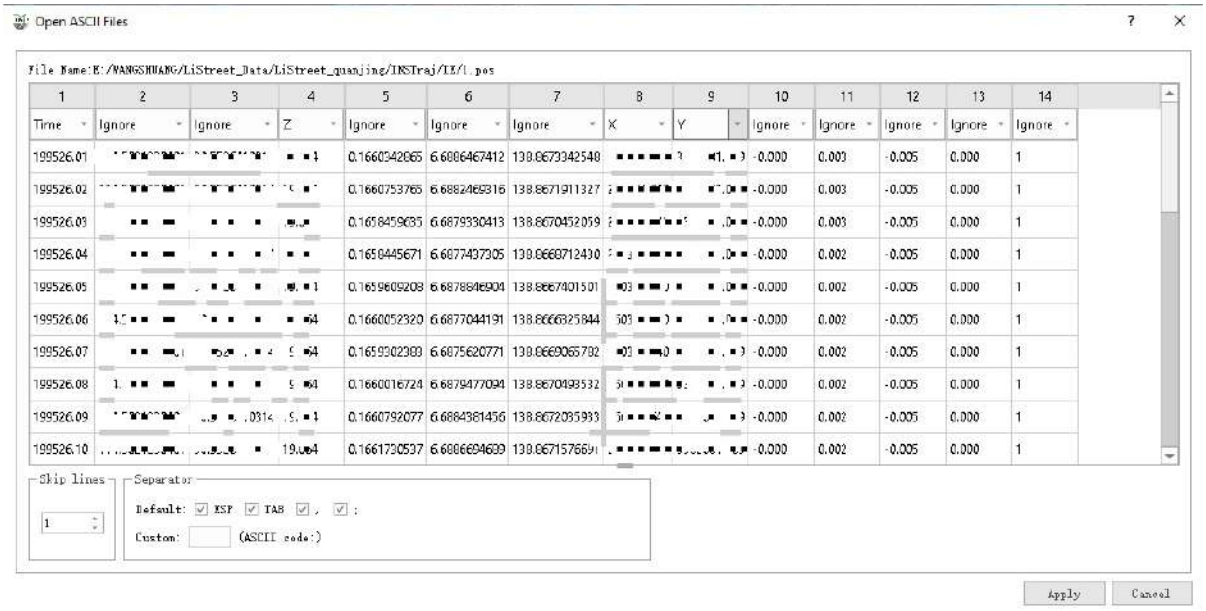
Function Description: Based on the roaming of the data trajectory, this tool allows the user to control the camera and move it according to the trajectory points, imitating the process of the vehicle collecting data in the real scene.

Steps

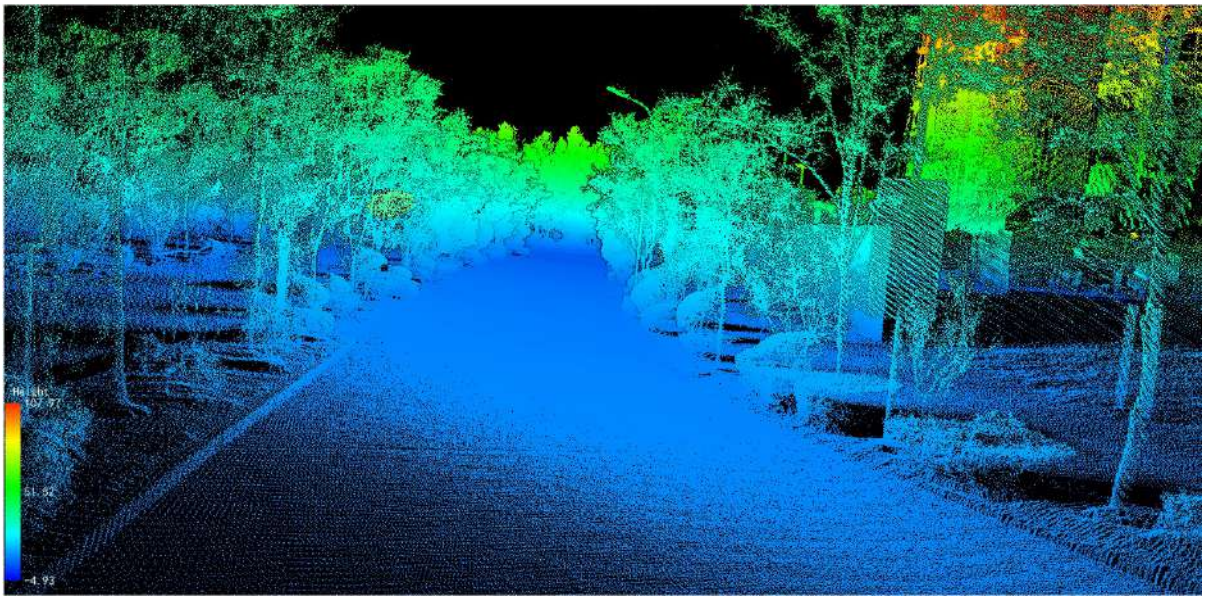
1.Click **Trajectory Roaming** button, the track selection interface pops up, which supports the import of track files in multiple formats, and generally selects the solved by default *.pos* format is enough.



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



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



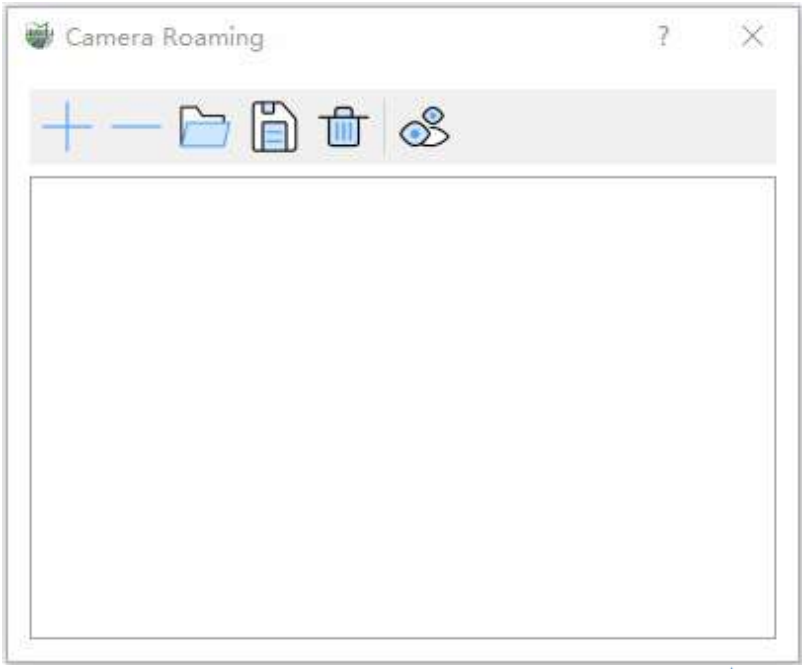
4.The user can click **Trajectory Roaming** button again to end the track roaming operation.

Camera Roaming

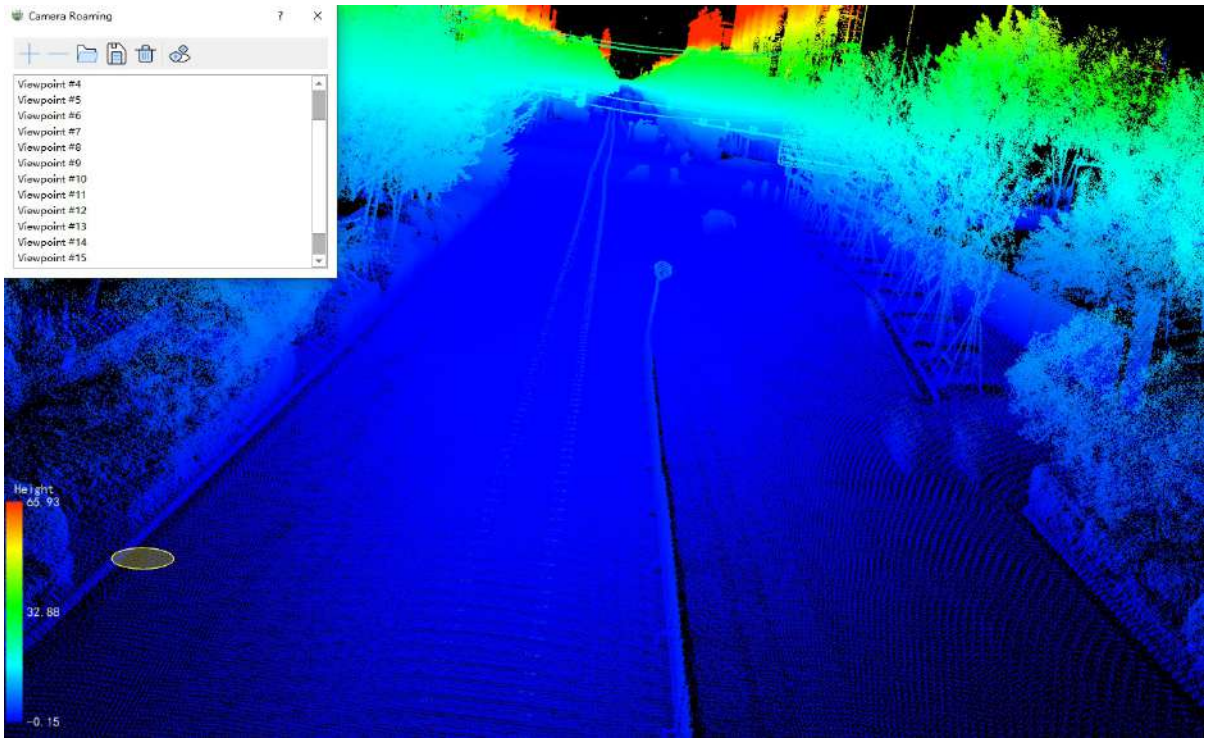
Function Description: Based on the roaming of user-defined viewpoints, This tool allows the user to control the camera can be and move it according to the viewpoints, as well as imitating the process people observing data in real scenes is realized, which enhances the immersion and presentation of the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

Steps

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

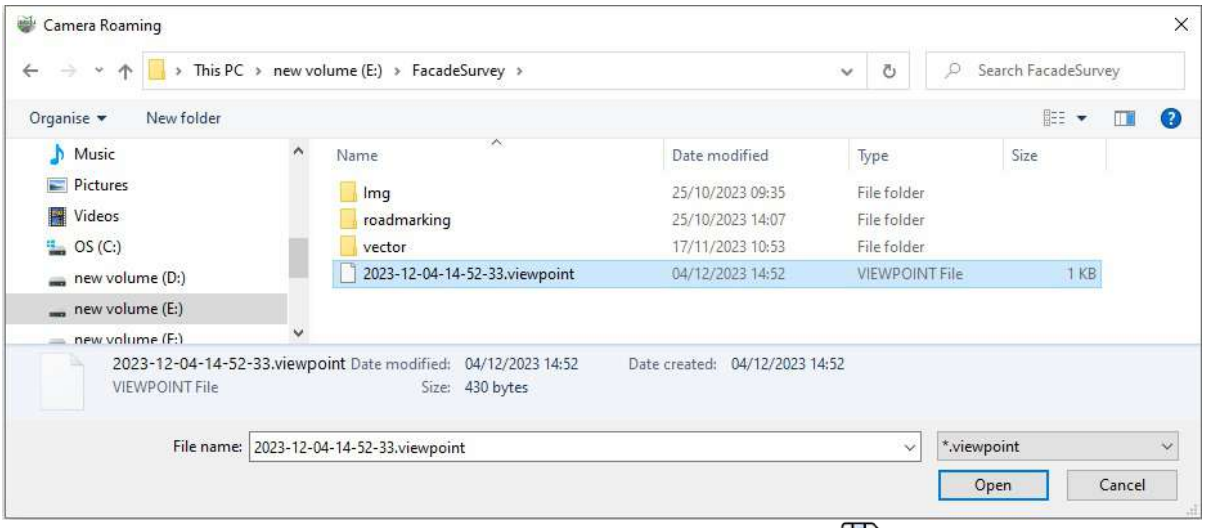


2.By adjusting the appropriate viewing angle, the user clicks **Add Current Viewpoint** button, add the current viewpoint position to the roaming path, as the keyframe of the roaming, the new viewpoint will appear in the viewpoint list box.

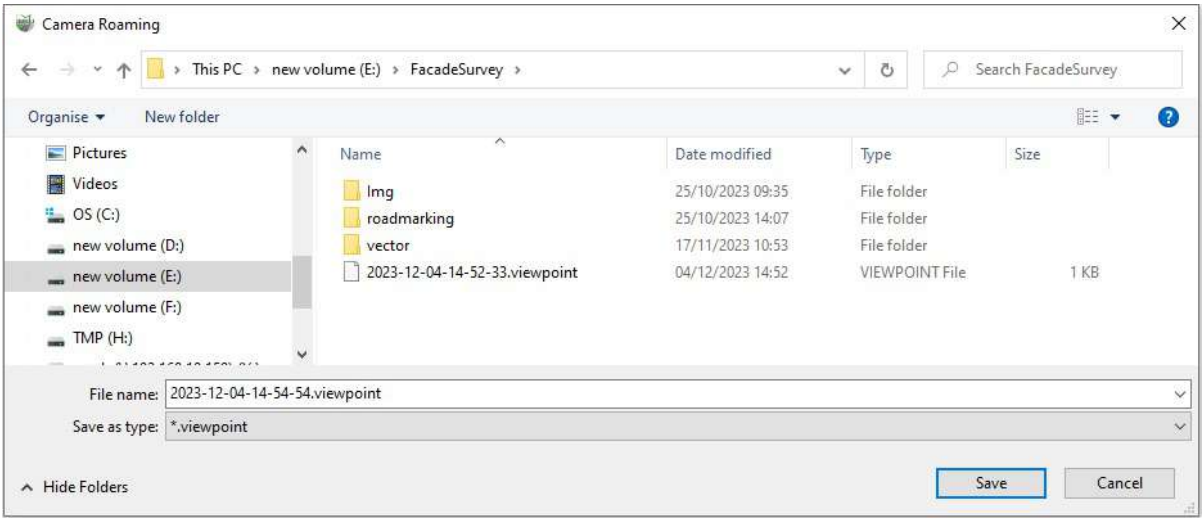


3.Users can also click **Remove Selected Viewpoint (optional)** —— button to delete unwanted or problematic viewpoints.

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

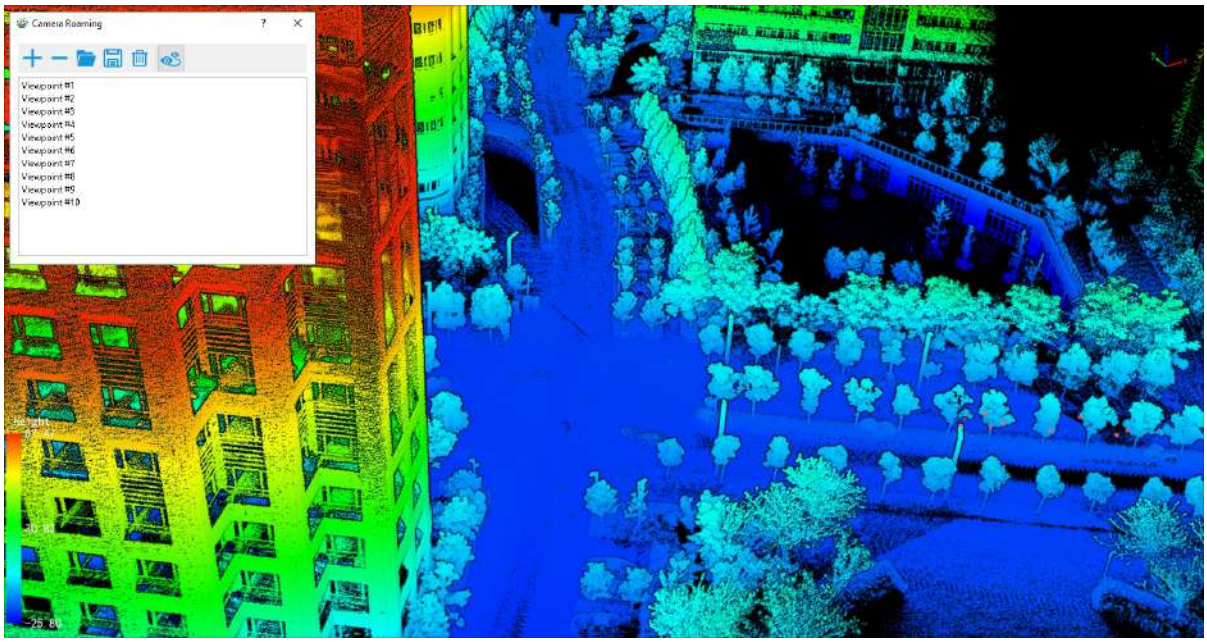


5.After adjusting the viewpoint, the user can click **Save Viewpoints (optional)** button, to save all viewpoints in the viewpoints list box as keyframe information in a viewpoints list file.



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

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



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

Import Vector

Importing vectors includes importing external vector files such as shp, dxf, etc. It also involves importing vector points/lines/polygons from TXT or CSV files or importing vectors from Gpkg files. Below are the details for each.

- Import Shape
- Import TXT
- Import Gpkg

Import Vector

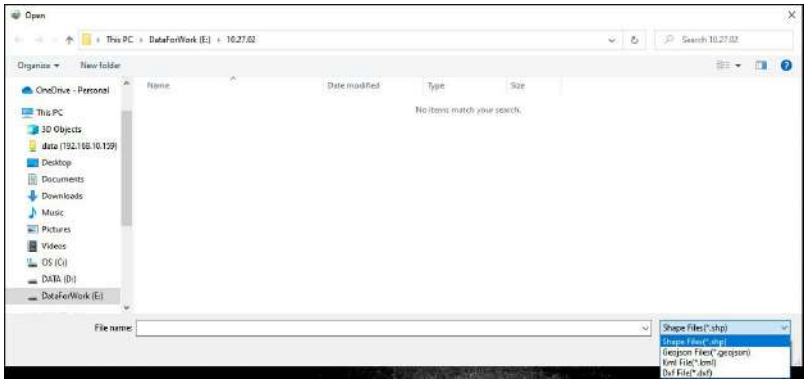
Function Description：The import vector feature allows you to import external files such as SHP, DXF, KML, etc., into the project file.

Steps

1.Click the "Import Vector" button, a dialog box will pop up. Choose the desired vectors and formats based on your needs.

Description

- **Add**: After clicking the button, a dialog box for selecting files will appear, supporting the import of external vectors in four formats: SHP, DXF, GeoJSON, and KML.



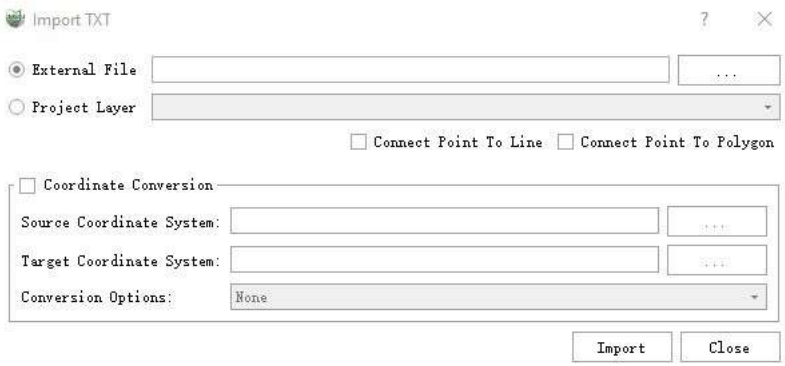
- **Delete**: Delete the shp file that has been added.
- **Clear**: Clear all added shp files and clear the table.
- **Coordinate Conversion (optional)**: If the coordinate transformation option is selected, coordinate transformation can be directly applied during the import of external vector files.

Import TXT

Function Description：This function allows you to import external CSV/TXT files with recorded coordinates as vector layers into your project.

Steps

1.Click the **Import TXT** button to bring up the parameter setting dialog box.



Parameter description

- **External File**：After selecting an external CSV/TXT file, import by configuring the column where XYZ is located.
- **Project Layer**：Choose a table from the project file, and import by configuring the column where XYZ is located.
- **Connect Point to Line**：When this option is checked, the imported points will be connected into lines in sequence based on the designated ID column.

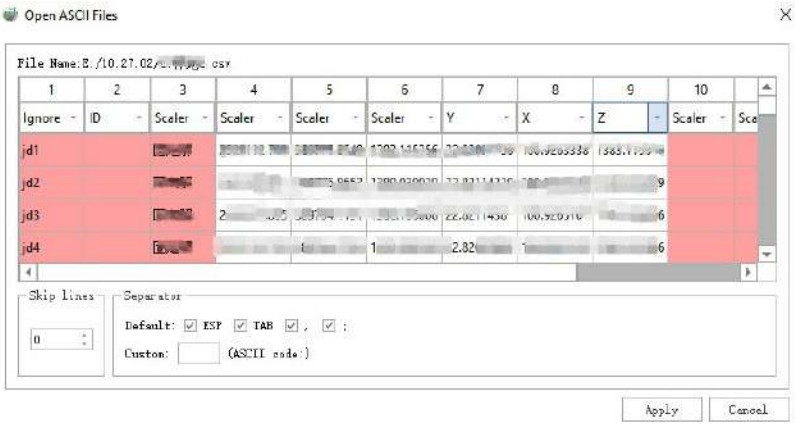
> Note: Points with the same ID, only when there are two or more, will be connected into lines; points that do not meet this requirement will be filtered out by default.

- **Connect Point to Polygon**：When this option is selected, the imported points will be connected into polygons in sequence based on the designated ID column.

> Note: Points with the same ID will be connected into polygons only when there are three or more points with the same ID. Points that do not meet this requirement will be filtered out by default.

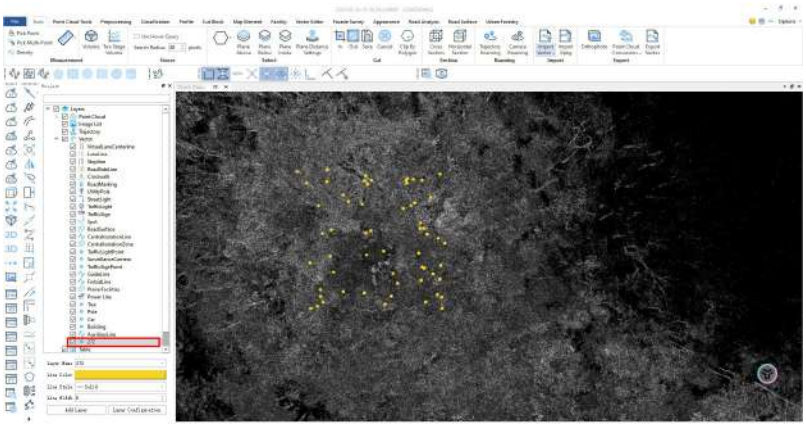
- **Coprdinate Conversion(Optional)**：If the coordinate transformation option is selected, you can choose the current file's coordinate system and select the target coordinate system. This will convert the newly added vector in SHP format to the target coordinate system and integrate it into the software.

2.Select an external CSV/TXT file, the "Open ASCII File" dialog box will pop up, the default will automatically be the first line of the table as the table header, which must be specified in the two columns of X, Y, Z, if not specified, the elevation of the imported data are all 0. You can specify the form of large coordinates of the XYZ or latitude/longitude as the height of the XYZ. XYZ, only when the need for point to line / surface, you must specify the ID where the identification column, the rest of the column can be ignored as Ignore Ignore or the default will be set to the attribute Scale, only when set to Scale, after the import, the contents of the other columns will be displayed in the attribute table. The number of skipped rows and separator can be set by yourself according to the content of the file.

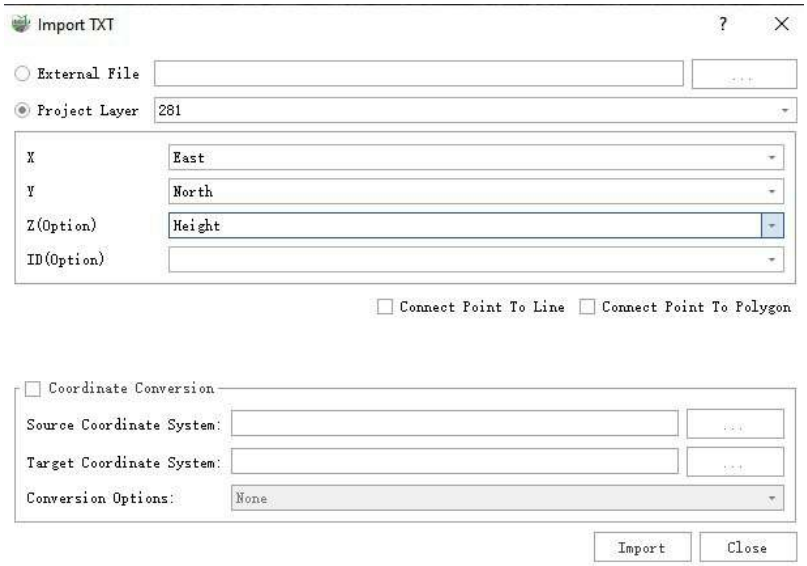


> Note: If you specify the latitude and longitude height as XYZ, you must do the coordinate conversion, as shown in the example, the source coordinate system select WGS84 (EPSG:4326), the target coordinate system select the coordinate system of the current project CGCS2000 / 3-degree Gauss-Kruger CM 102E (EPSG:4543), after the setting is correct, the imported points can be superimposed with the point cloud to display the point cloud. The imported points can be overlaid with the point cloud only when the settings are correct.

The imported data is shown in the figure below, without geometric conversion will be imported according to the point layer, the layer name using the file name



3.Import from the existing table of the project: Select the current project option, select a table that has been imported from the project in the drop-down list, the software will automatically parse the table header, and under the XYZ option below, select the corresponding columns of XYZ correspondingly. Other settings with step 2

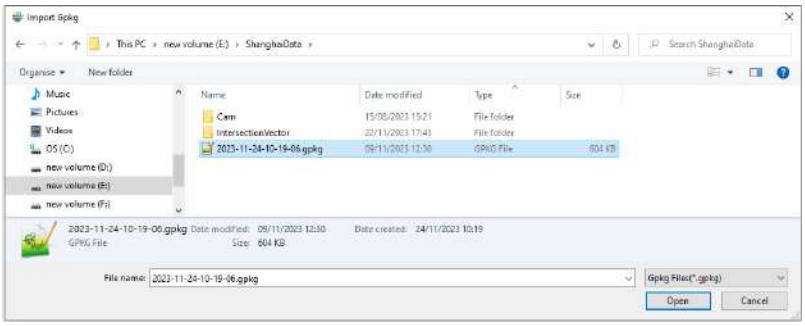

Import TXT
External File
Project Layer
281
X
East
Y
North
Z(Option)
Height
ID(Option)
Connect Point To Line
Connect Point To Polygon
Coordinate Conversion
Source Coordinate System:
Target Coordinate System:
Conversion Options:
None
Import
Close

Import Gpkg

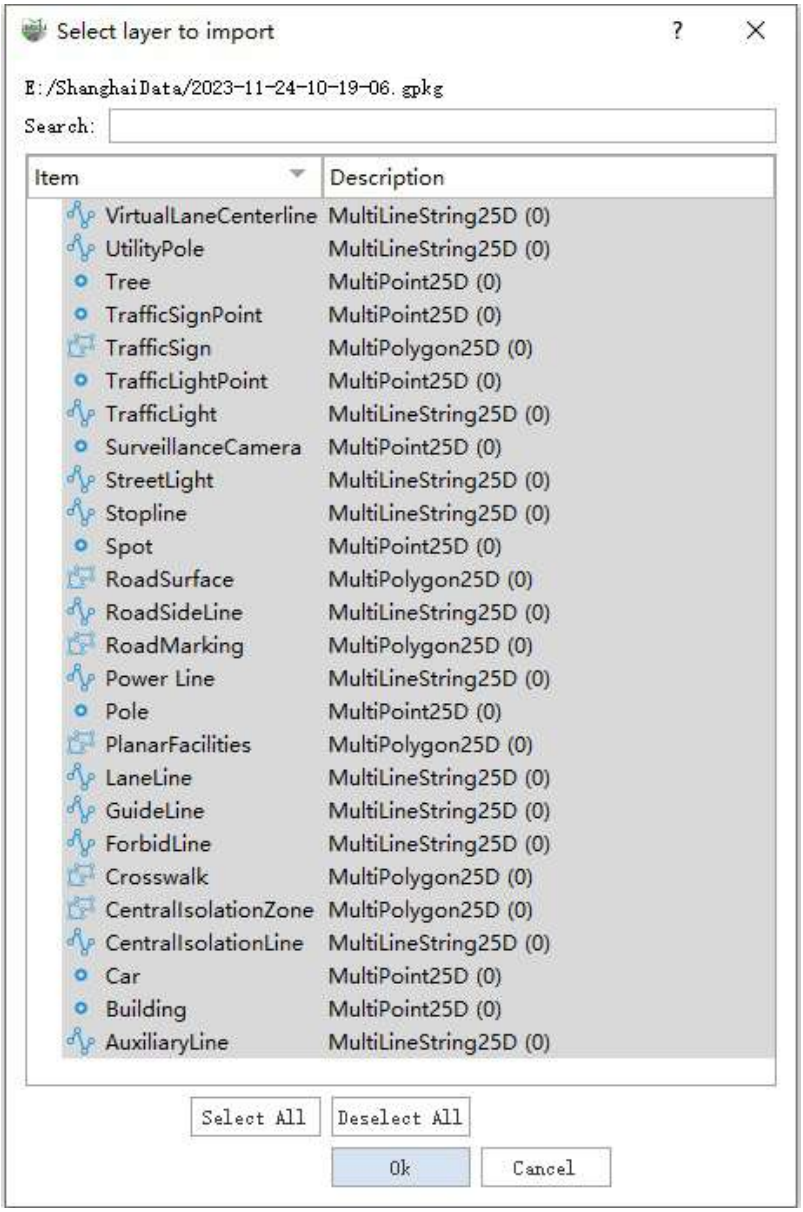
Function Description：Import an external *.gpkg file into the project.

Steps

1.Click the **Import GPKG** button, a file selection dialog will pop up. Choose the *.gpkg file you want to import and click 'OK'.



2.In the pop-up dialog for selecting layers to import, you can view and select the details of the content to be imported. The specific parameter explanations are as follows:



- **Search**: You can filter the content to be imported by keywords, such as layer name keywords or layer type keywords.
- **Item**: Import the names of layers in the database.
- **Description**: Import the type of each layer in the database, along with the number of vector objects contained in the current layer.
- **Select All**: You can quickly select all layers.
- **Deselect All**: You can quickly deselect all layers.

> Note: Left-click on a layer with the mouse's left button to select it. While holding down the Ctrl key, you can make multiple selections of layers.

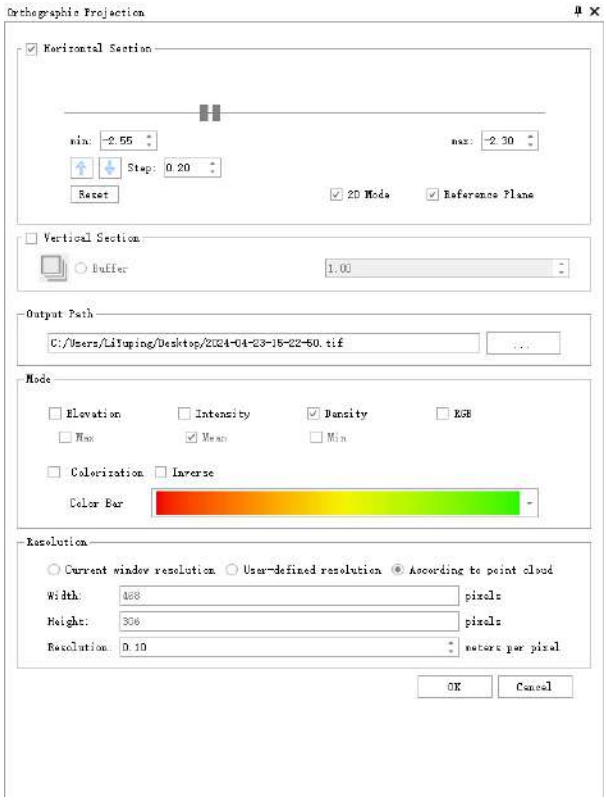
Export tool

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

- Orthophoto
- Point Cloud Conversion
- Export Vector

Export Orthophoto

1.Click on the **Orthophoto** button to open the Orthophoto Export Settings window. As shown below, you can export the elevation, intensity, density, and RGB images of the point cloud in conjunction with the Horizontal Section Tool and Vertical Section Tool. The exported image supports tif and png formats, and the results can be opened and measured in other GIS software.



Orthophoto Export Settings Window

Horizontal Section Tool

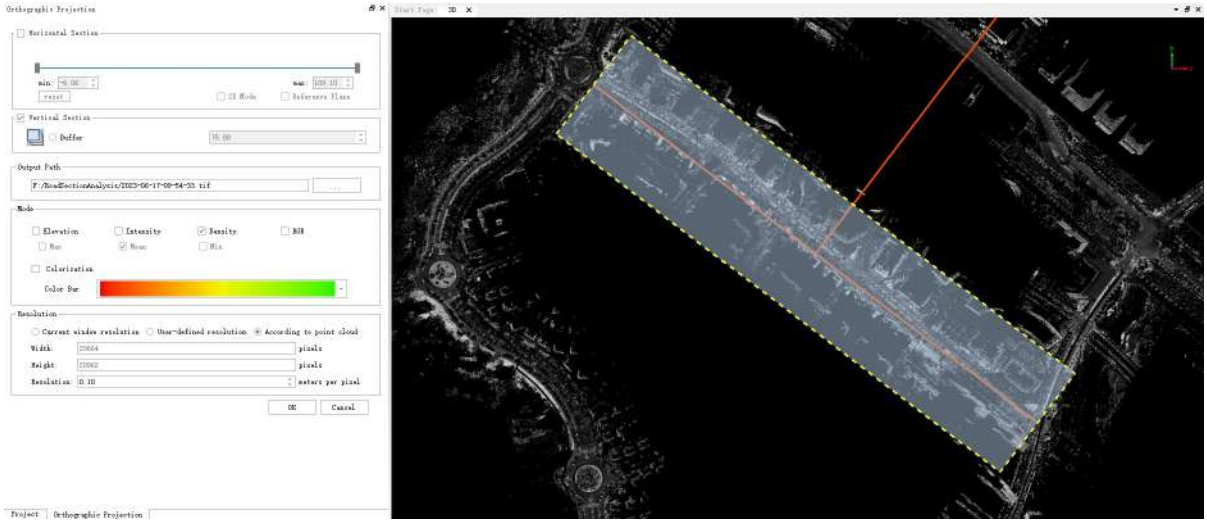
1.Check the Horizontal Section to set the minimum and maximum values of the horizontal section through a dual slider. The 3D window will display the results in real time. You can reset the current settings by clicking on the Reset button. Check the 2D mode to switch between 2D and 3D modes in the 3D window. Checking Reference Plane can turn on or off the reference plane in the current 3D window. It looks like this:

Horizontal Section

Vertical Section Tool

1.Check Vertical Plane and click on the button. The 3D scene will automatically switch to the top view.

2.Use the mouse to interactively draw a rectangle. The first two points determine the axis direction, and the third point determines the buffer size. Double-click to the end and slice the section, as shown below.



Vertical section area selection

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

Vertical Section Slice Result

4.To fix the section buffer, you can click on the Buffer radio button and adjust the buffer value.

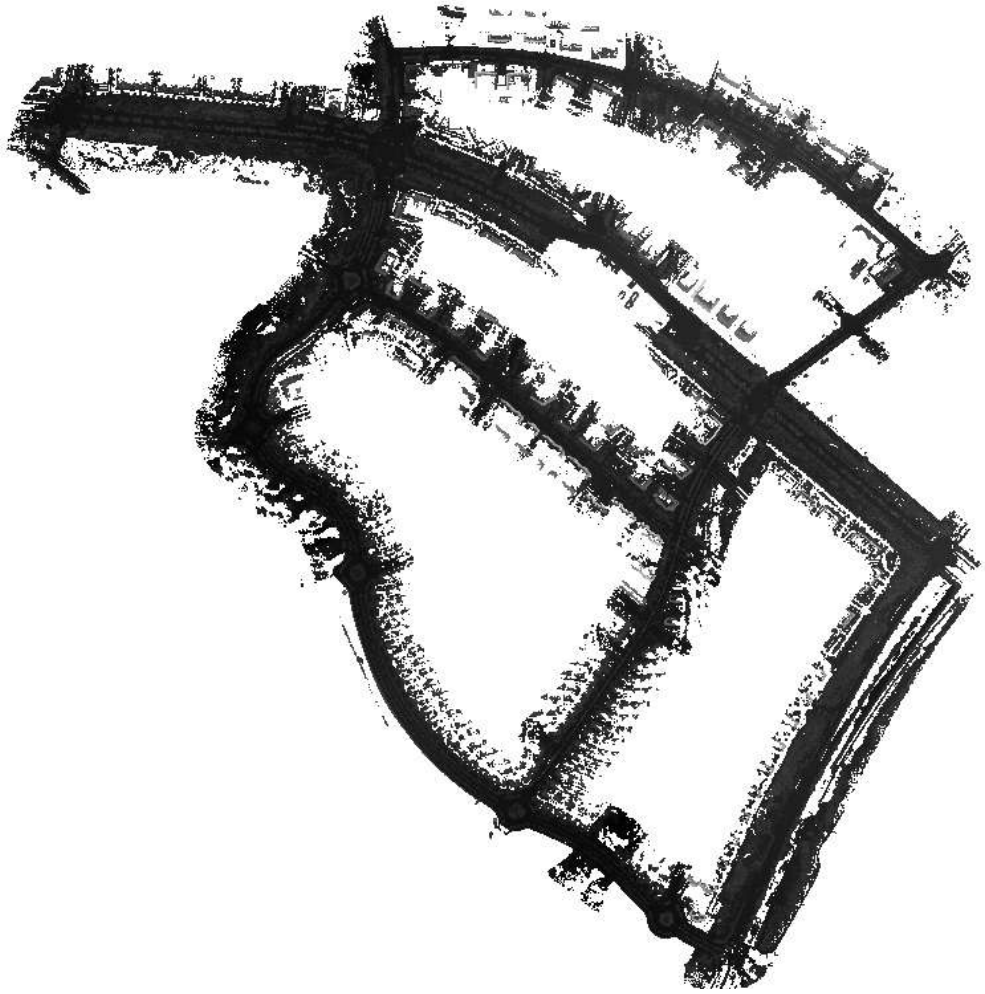
Orthophoto Export

Export by Elevation

1.Check Elevation in the Orthophoto Export Settings Window-Mode. The point cloud in the 3D window will be displayed according to the elevation. The Point Cloud Color Mode Setting window on the right side of the software will pop up. You can select the current display color bar and other settings through the Point Cloud Color Mode Setting window, as shown below:

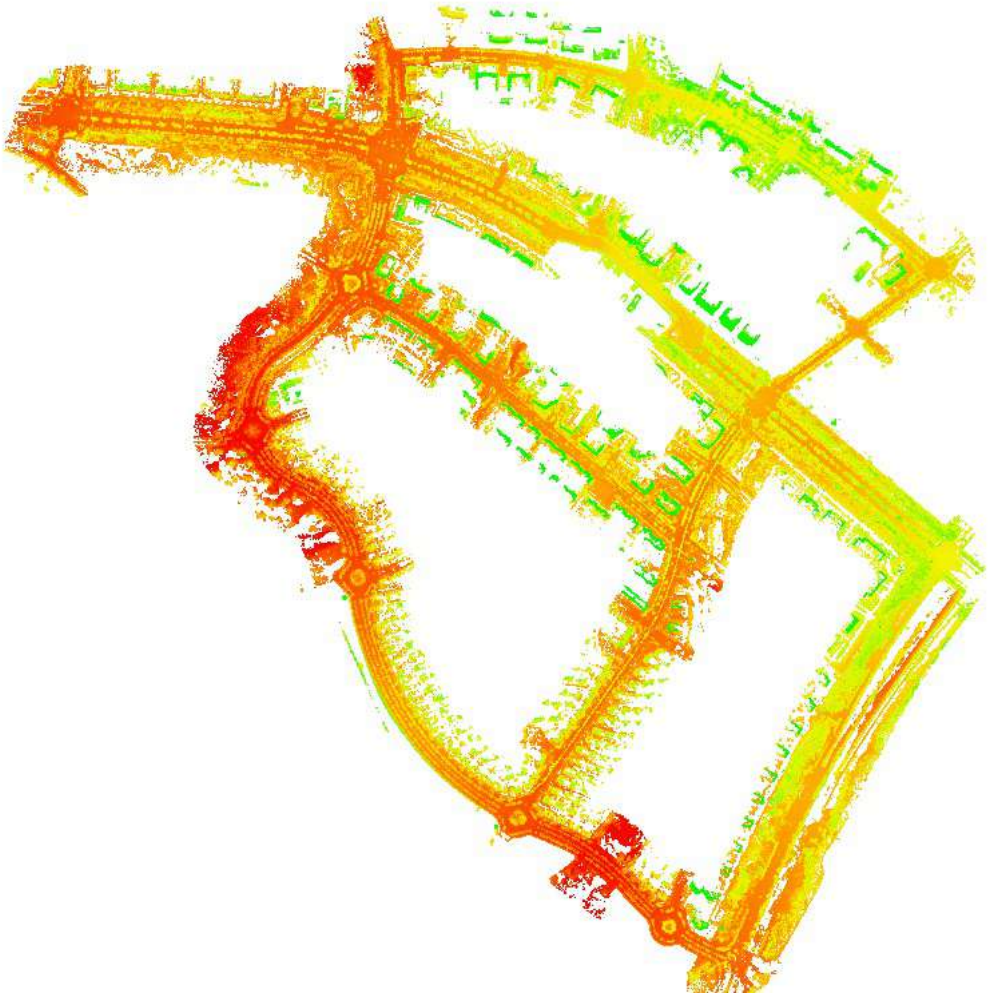
Display by Elevation

2.After setting the point cloud display, you can choose whether the exported image is colorized. If not checked, a binary image will be output. As shown below:



Export Binary Image by Elevation

3.If Colorized is checked, a color image will be output. As shown below:



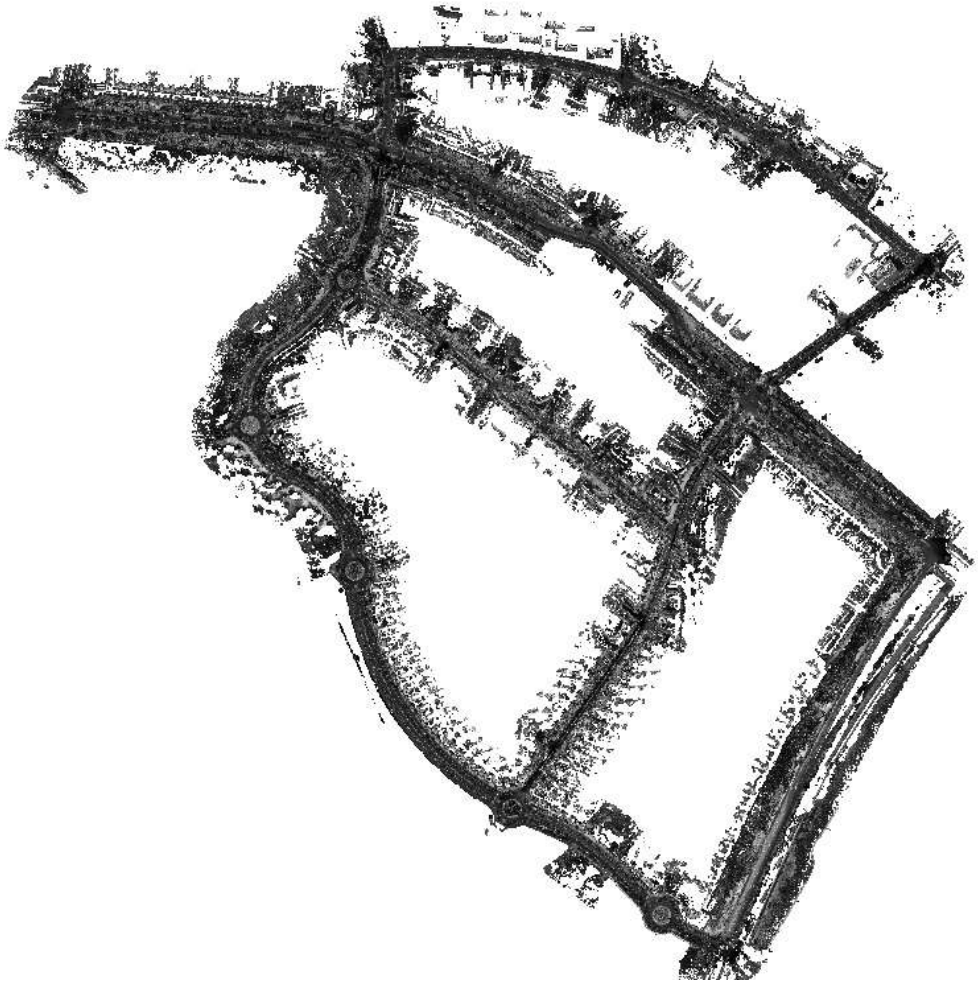
Export Color Image by Elevation

Export by Intensity

1.Check Intensity in the Orthophoto Export Settings Window-Mode. The point cloud in the 3D window will be displayed according to the intensity. The Point Cloud Color Mode Setting window on the right side of the software will pop up. You can select the current display color bar and other settings through the Point Cloud Color Mode Setting window, as shown below:

Display by Intensity

2.After setting the point cloud display, you can choose whether the exported image is colorized. If not checked, a binary image will be output. As shown below:



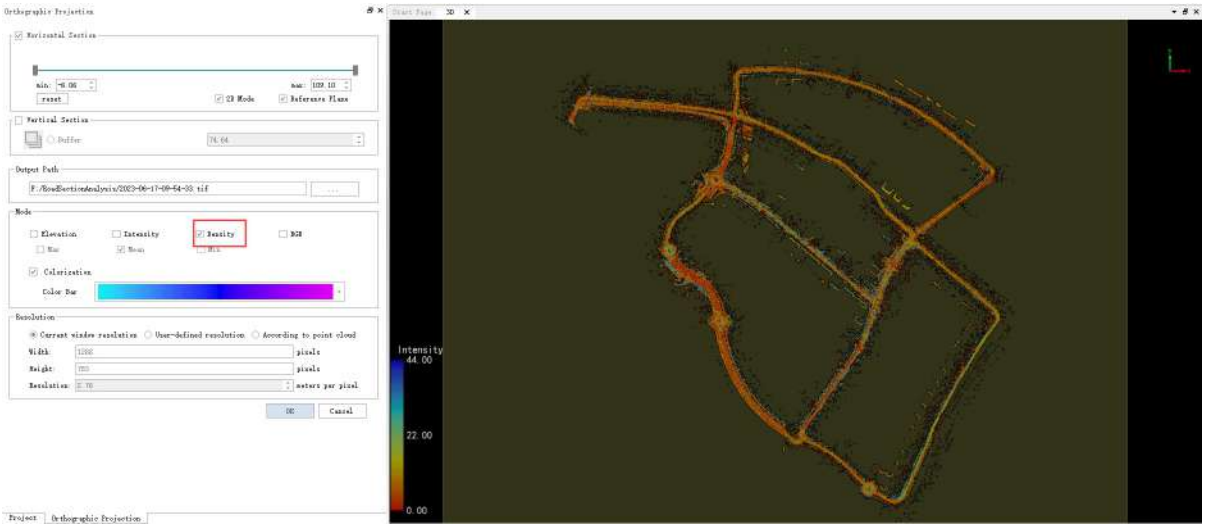
Export Binary Image by Intensity

3.If Colorized is checked, a color image will be output. As shown below:

Export Color Image by Intensity

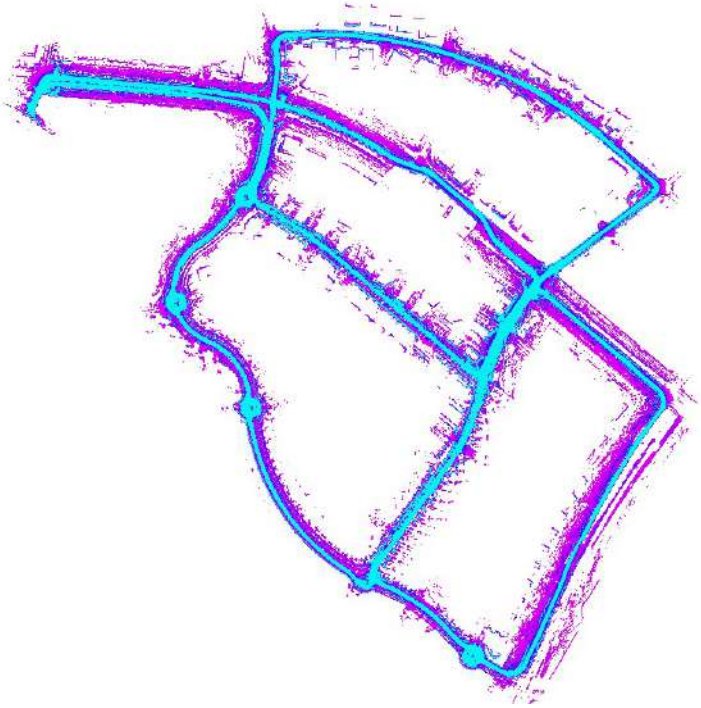
Export by Point Cloud Density

1.Check Density in the Orthophoto Export Settings Window-Mode. Since there is no option to display according to density in the point cloud display mode, there will be no change in the 3D window. At the same time, the color bar for exporting images according to point cloud density will be set in the Orthophoto Export Settings Window. If Colorized is not checked, a binary image will be output. As shown below:



Export Binary Image by Density

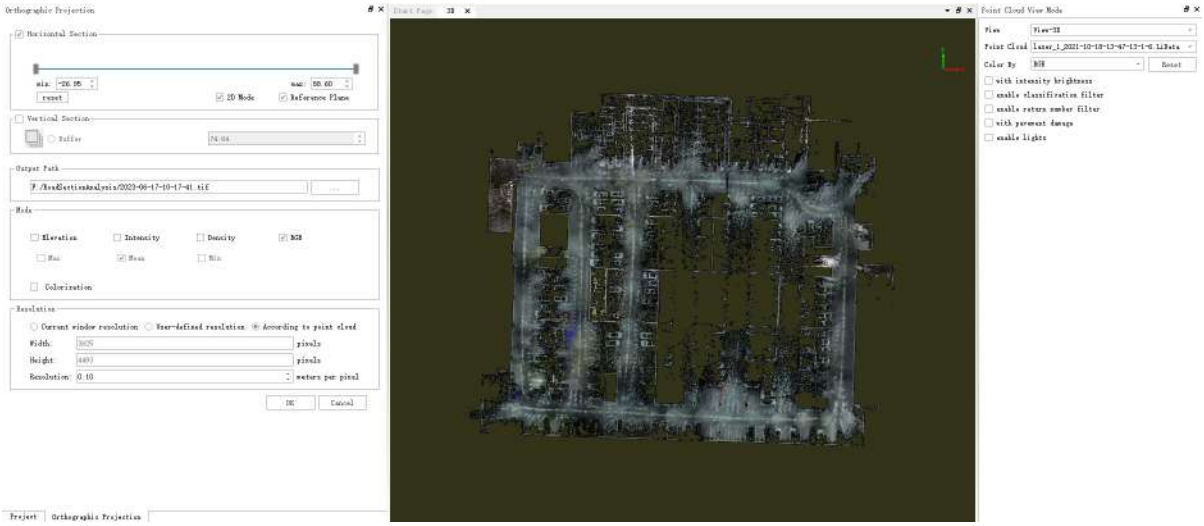
2.If Colorized is checked, a colorized image will be output. As shown below:



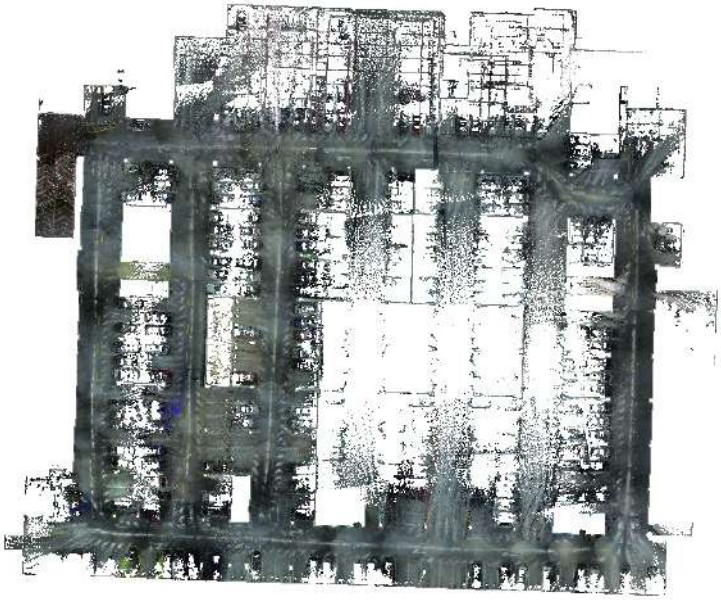
Export Color Image by Density

Export by Point Cloud RGB

1.Check RGB in the Orthophoto Export Settings Window-Mode. If the point cloud contains RGB information, the point cloud in the 3D window will be displayed according to RGB. If the point cloud does not have RGB information, the software output window will give a warning. After checking RGB, the colorization setting will be invalid, and only the color image containing point cloud RGB information can be exported. As shown below:



Display by RGB



Export Color Image by RGB

> Note: In each mode of export, there are options to check the maximum value, minimum value, and average value. This setting is effective for the mode settings exported according to point cloud elevation and intensity.
>
> - **Maximum value**: When calculating the pixel value of each pixel in the image, it is calculated according to the maximum value of all point cloud elevations or intensities mapped to the current pixel.
> - **Minimum value**: When calculating the pixel value of each pixel in the image, it is calculated according to the minimum value of all point cloud elevations or intensities mapped to the current pixel.
> - **Average value**: When calculating the pixel value of each pixel in the image, it is calculated according to the average value of all point cloud elevations or intensities mapped to the current pixel.

Resolution Settings

There are currently three options for setting the resolution of the exported image:

- **Current Window Resolution**: The aspect ratio and resolution of the output image will be exactly the same as the 3D window. And it does not support setting.
- **User-defined Resolution**: The aspect ratio of the output image will remain the same as the 3D window, and the resolution of the image supports user setting through the world coordinate accuracy mapped by each pixel.
- **According to Point Cloud**: The aspect ratio of the output will be calculated through the bounding box of all current point clouds. The aspect ratio of the image is consistent with the calculated bounding box, and the resolution of the image supports user settings through the world coordinate accuracy mapped by each pixel.

Point Cloud Format Conversion

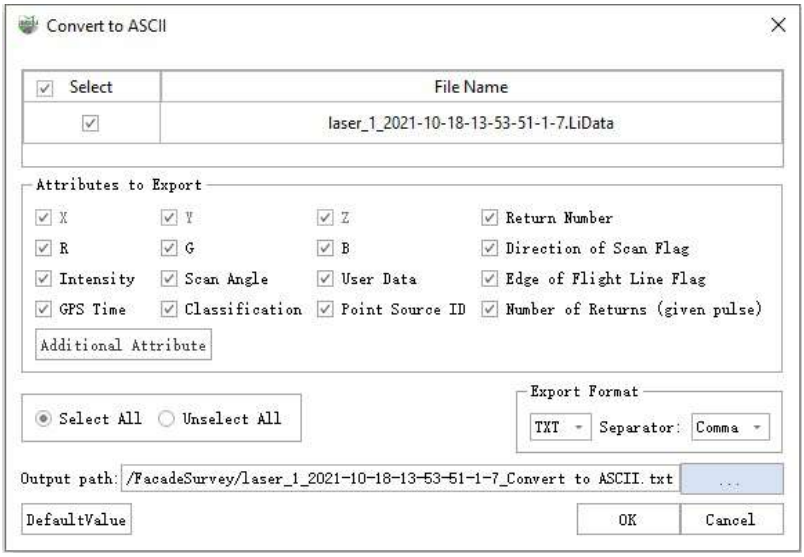
Feature Description: The point cloud conversion supports the export of various point cloud data, including converting the point cloud to ASCII, Las, Laz, E57, Ply, and LiData point cloud formats. Detailed introductions will follow.

Convert to ASCII

Feature Description: The "Convert to ASCII" tool can convert LiData point clouds to ASCII format, which is a text format that can be easily viewed in a text editor.

1.Click Convert to ASCII button

2.Usage:Click Point Cloud Conversion -> Convert to ASCII



Parameter Settings

- **Input Data**: Choose the LiData file(s) you need to export. The input file can be a single-point cloud data file or multiple data files.
- **Exported Attributes**: Supports basic attribute types and additional attributes.
- **Export Format**: The software supports txt and CSV suffix formats, and the separators support commas, spaces, and TABs;
- **Output Path**: The output folder path. The new file will be generated after the function is executed.

Convert to Las

Feature Description: The "Convert to Las" tool can convert LiData point clouds to Las format, that is, the standard Lidar point cloud data format.

1.Click Convert to Las button

2.Usage：Click Point Cloud Conversion -> Convert to Las

> The conversion to Laz format follows the same procedure.

Parameter Settings

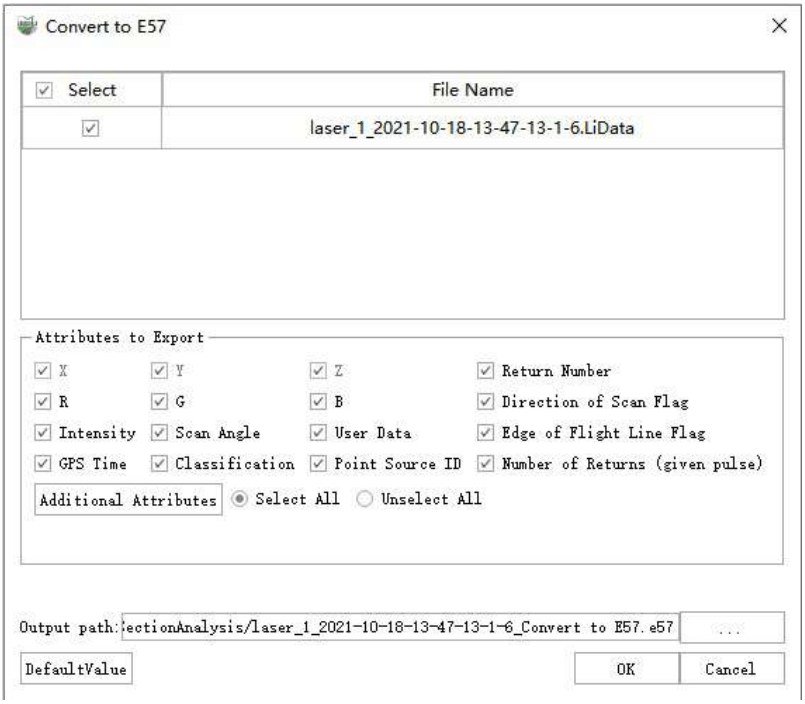
- **Input Data**: Choose the LiData file(s) you need to export. The input file can be a single-point cloud data file or multiple data files.
- **Exported Attributes**: Supports all Las attribute types and additional attributes.
- **Color Range**: There are three RGB ranges to choose from.
  - **None**: Output according to the original RGB range of the selected file.
  - **0~255 (8-bit)**: Map the RGB range to 0255 for output. If the color range of the user-selected file is 01 or 065535, it can be mapped to 0255 for output.
  - **0~65535 (16-bit)**: Map the RGB range to 065535 for output. If the color range of the user-selected file is 01 or 0255, it can be mapped to 065535 for output.
- **Intensity Range**: There are three intensity ranges to choose from.
  - **None**: Output according to the original intensity range of the selected file.
  - **0~255 (8-bit)**: Map the intensity range to 0255 for output. If the intensity range of the user-selected file is 01 or 065535, it can be mapped to 0255 for output.
  - **0~65535 (16-bit)**: Map the intensity range to 065535 for output. If the intensity range of the user-selected file is 01 or 0255, it can be mapped to 065535 for output.
- **Source Unit**: The unit of the LiData file to be exported. The point cloud unit supported by the software is meters, so this parameter is set to meters by default and cannot be modified.
- **Target Unit**: The unit for exporting to Las data. You can choose meters, decimeters, centimeters, millimeters, feet, and inches.
- **Las Version (default value 1.4)**: The version to export as LAS, supports 1.2~1.4, referencing LiData format. When converting from a higher version to a lower one, attribute fields that exceed the range will be zeroed out.
- **Output Path**: The output folder path. The new file will be generated after the function is executed.

Convert to E57

Feature Description: The "Convert to E57" tool can convert LiData point clouds to E57 format.

1.Click Convert to E57 button

2.Usage:Click Point Cloud Conversion -> Convert to E57



Parameter Settings

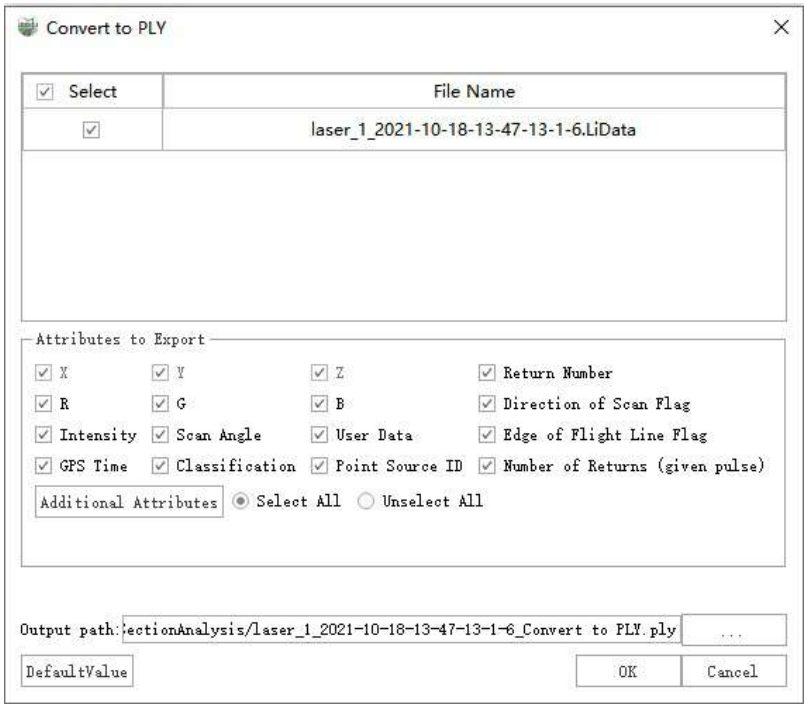
- **Input Data**: Choose the LiData file(s) you need to export. The input file can be a single-point cloud data file or multiple data files.
- **Exported Attributes**: Supports basic attribute types and additional attributes.
- **Output Path**: The output folder path. The new file will be generated after the function is executed.

Convert to Ply

Feature Description: The "Convert to Ply" tool can convert LiData point clouds to Ply format.

1.Click Convert to Ply button

2.Usage: Click Point Cloud Conversion -> Convert to Ply



Parameter Settings

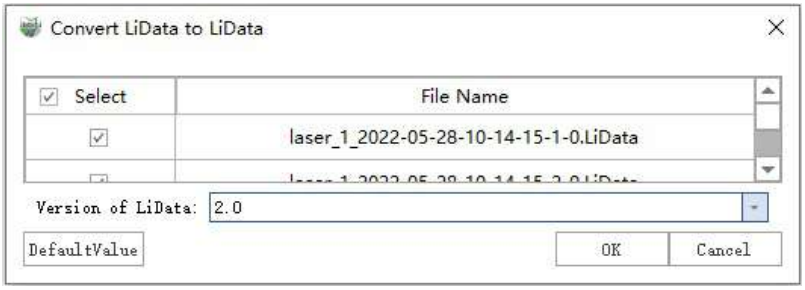
- **Input Data**: Choose the LiData file(s) you need to export. The input file can be a single-point cloud data file or multiple data files.
- **Exported Attributes**: Supports basic attribute types and additional attributes.
- **Output Path**: The output folder path. The new file will be generated after the function is executed.

Version Conversion

Feature Description: Conversion between point cloud versions.

1.Click Convert LiData to LiData button

2.Usage:Click Point Cloud Conversion -> Convert LiData to LiData



Parameter Settings

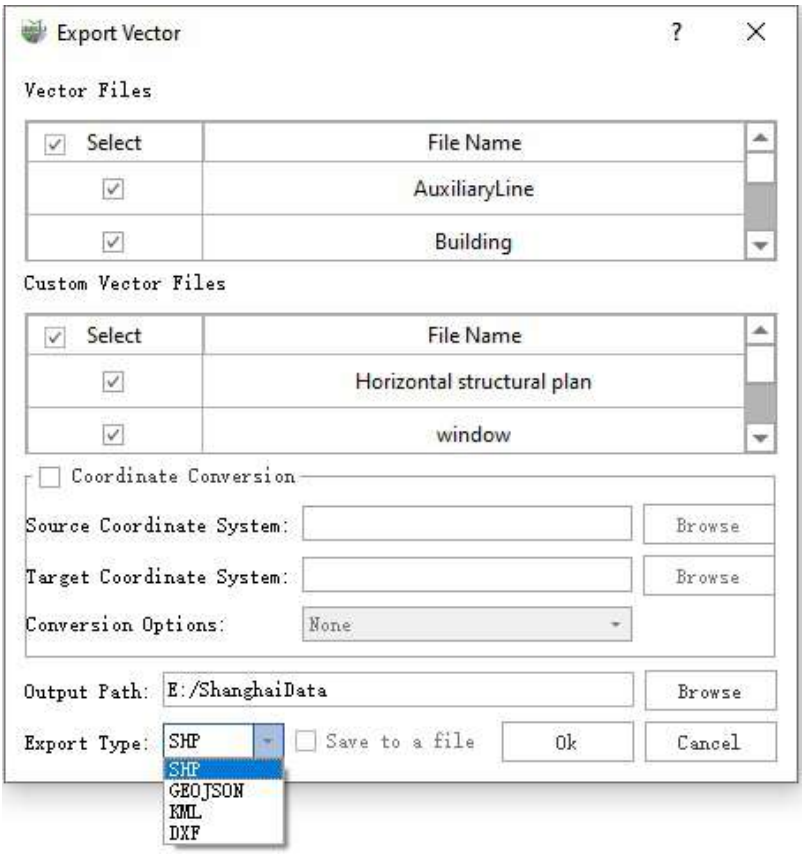
- **Input Data**: Choose the LiData file(s) you need to export. The input file can be a single-point cloud data file or multiple data files.
- **LiData Version**: Supported versions.

Export Vector

Function Description：The Export Vector dialog box allows you to select the layers you want to export, choose the output directory, and select the vector file format for export. Follow the steps below for detailed instructions.

Steps

1.Click on the **Export Vector** button .



Instructions

- **Vector Files**: Initialize the table to display all the built-in layers in the software, and by default, all layers are selected for export. Users can manually check/uncheck the layers as needed.
- **Custom Vector Files**：Display the layers customarily added to the project, and by default, all layers are selected for export. Users can manually check/uncheck the layers as needed.
- **Output Path**: The output path is initially set to the same directory as the point cloud data. You can click the Browse button to change the output path.
- **Export Type**: Currently, four-vector file formats are supported for export: Shapefile, GeoJSON, KML, and DXF.
- **Save as a Single File**: This option is specific to DXF documents and allows you to save all vectors in a single file.

Unroll

The point cloud unrolling feature allows you to specify a move thickness based on selected or drawn lines, curves, cylinders, etc. It unfolds the point cloud onto a single plane and enables you to save it.

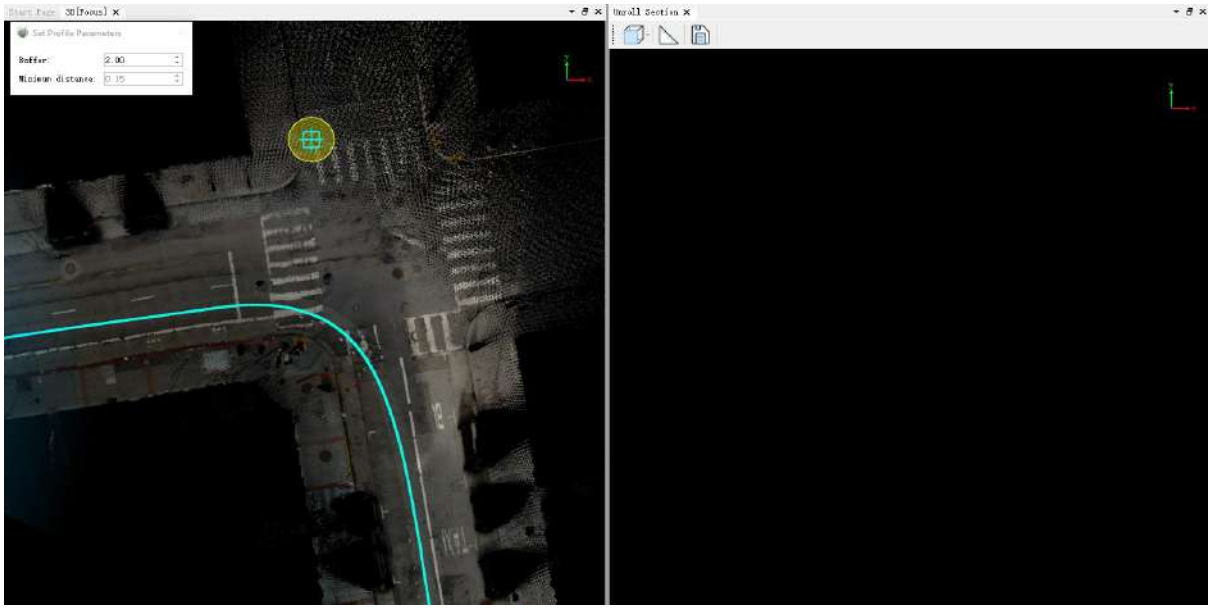
- Unroll Polygonal Section
- Unroll Curved Section
- Unroll Cylindrical Section

Unroll Polygonal Section

Function Description：The polygon section unfolding tool utilizes multiple connected polygon sections to slice the point cloud vertically and unfold it onto the same plane.

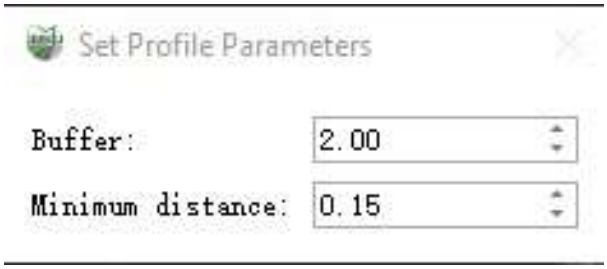
Steps

1.Open the LiDAR360MLS software and click on the Unroll Polygonal Section button. The 3D window will default to 2D mode, and a section unfolding window will pop up. Additionally, the polygon section unfolding button will lock other functionalities, and only manually lifting this button will exit this function and allow access to other functionalities. See the illustration below:



Unroll Polygonal Section

The parameter settings window will be displayed in the top left corner of the 3D window. As shown in the image:

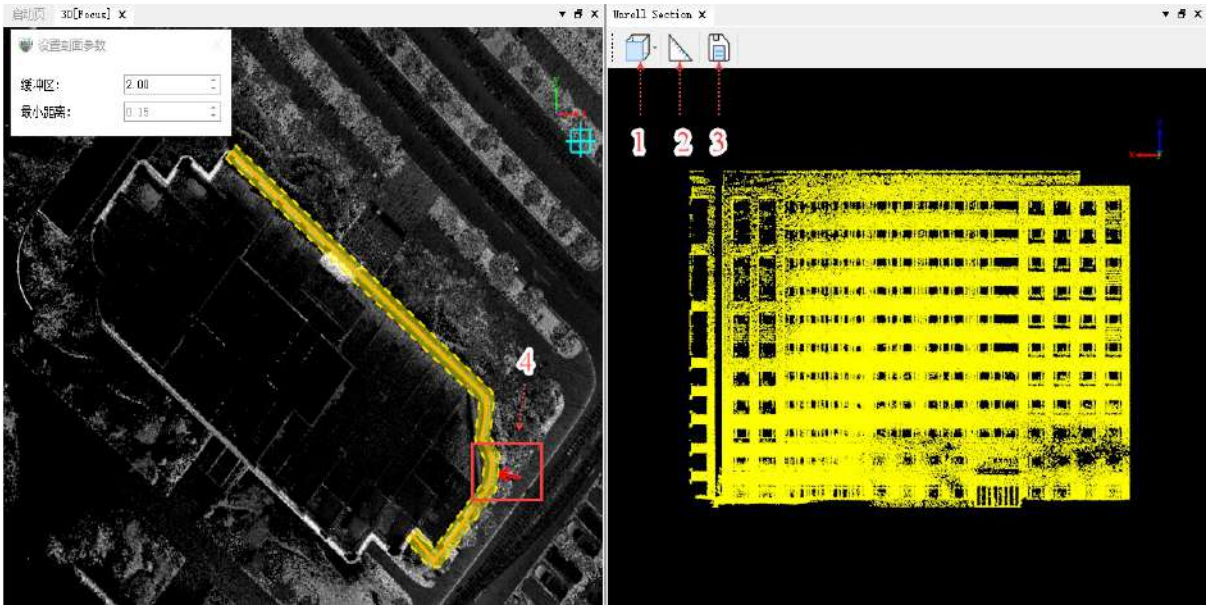


Parameter Settings Page

Parameter Setting

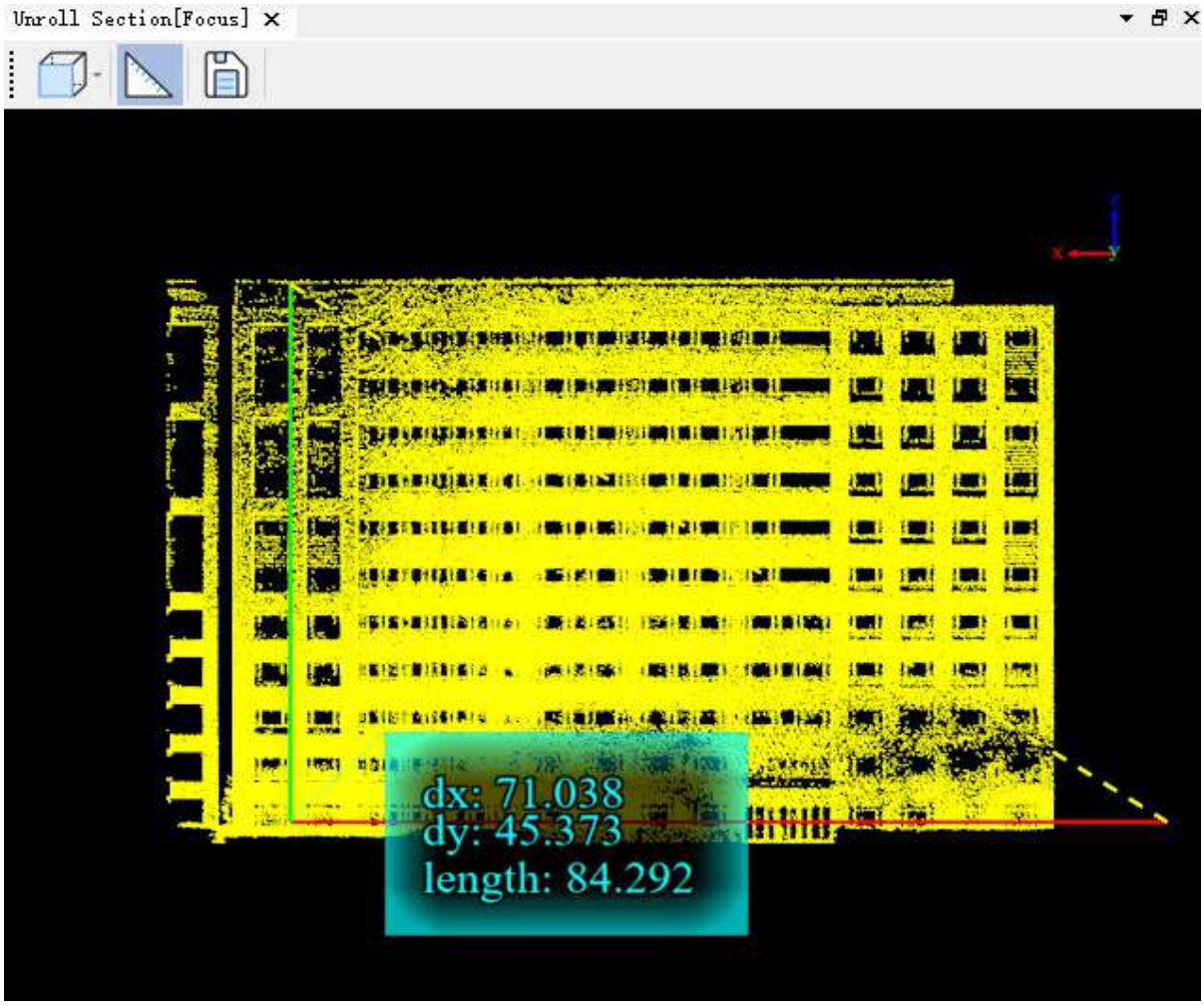
- **Buffer**：Control the width of the section.
- **Minimum Distance**：The surface unfolding operates similarly to the unfolding method of a differential curve. This parameter adjusts the granularity of the differentiation. It is disabled when unfolding polygonal or cylindrical sections and can only be adjusted when unfolding surfaces.

2.After completing the parameter settings, draw the section line that needs to be unfolded in the main window using the line drawing method. Once the drawing is completed, double-click the left mouse button to perform the section unfolding operation. After the progress bar completes, the result of the section unfolding will be displayed in the section window, as shown in the image.



The result of unfolding the polygon section.

The arrow at position 4 in the above image indicates the direction of observation for the unfolded section result. Button 1, located at position 1, allows setting the front or back view. Button 2, located at position 2, enables measurement operations on the unfolded result, as shown in the image.

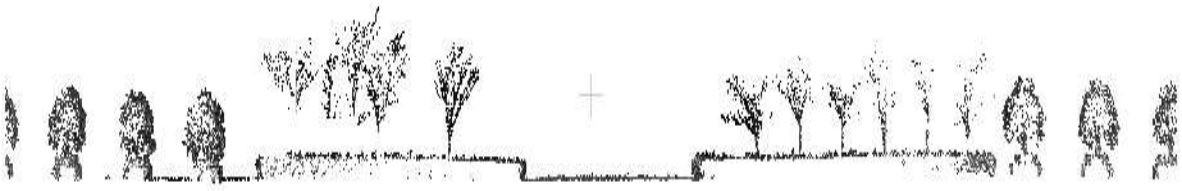


Measuring the unfolded polygon section result.

Button 3 in the unfolded cylinder section result image allows saving the section unfolding result. You can choose to save the format as LiData format point cloud for further processing in LiDAR360MLS or save it as Las format point cloud for further processing in third-party software. The saved point cloud will retain the attributes of the original point cloud.

Select the format for saving the result.

The saved LiData point cloud can be opened in LiDAR360MLS.



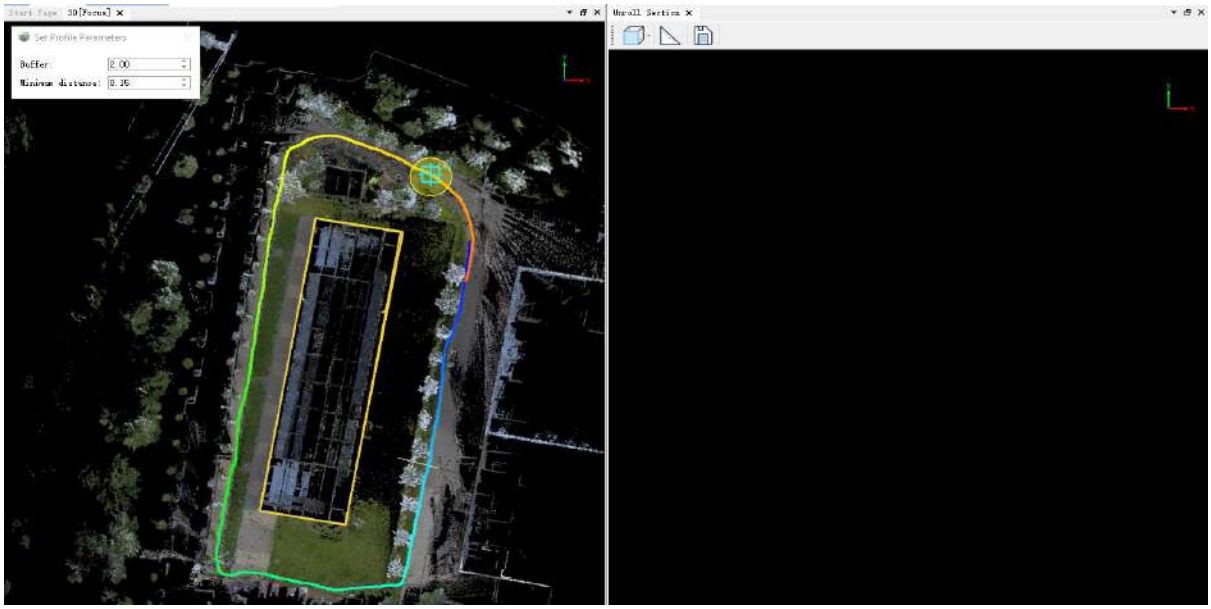
The saved Las point cloud can be opened in third-party software.

Unroll Curved Section

Function Description：The Unroll Curved Section tool uses surfaces to intercept a point cloud in the vertical direction and expand it to the same plane.

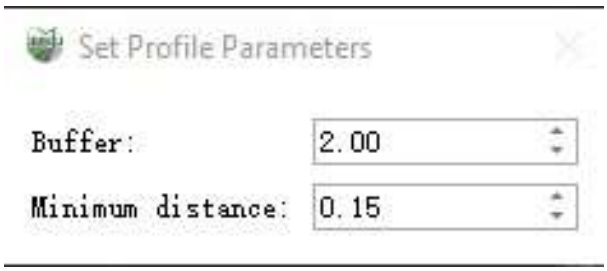
Steps

1.Click the Unroll Curved Section button. The 3D window will default to 2D mode, and a section unfolding window will pop up. Additionally, the "Unroll Surface Section" button will lock other functionalities, and only manually lifting this button will exit this function and allow access to other functionalities. See the illustration below:



Unroll Curved Section

The parameter settings window will be displayed in the top left corner of the 3D window. See the illustration below:

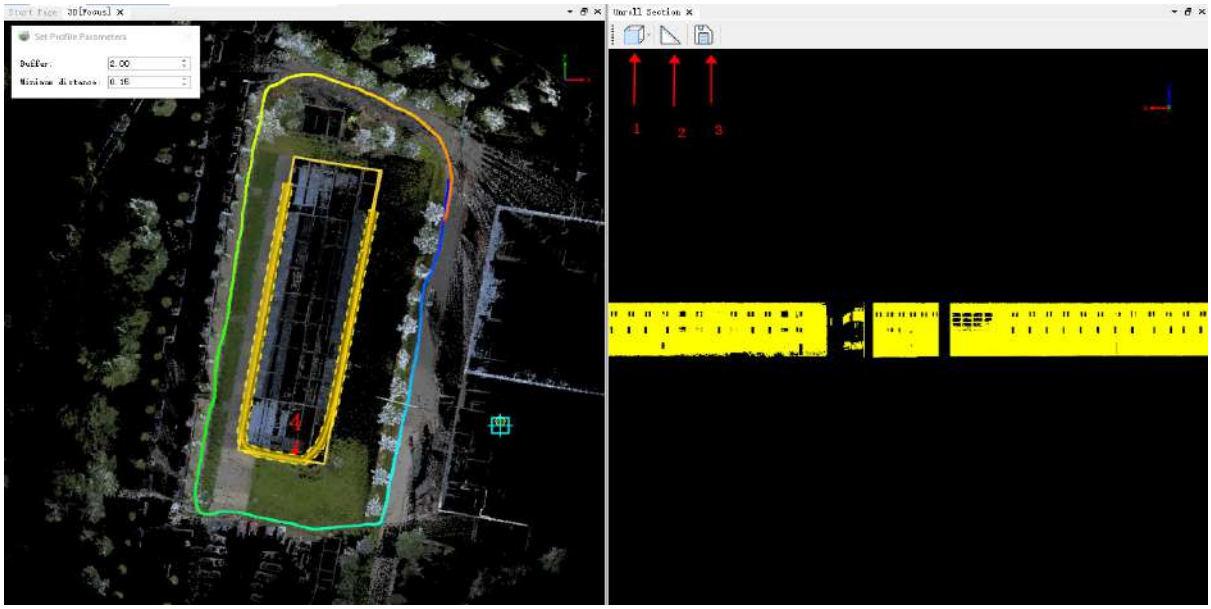


Parameter Setting Page

Parameter Settings

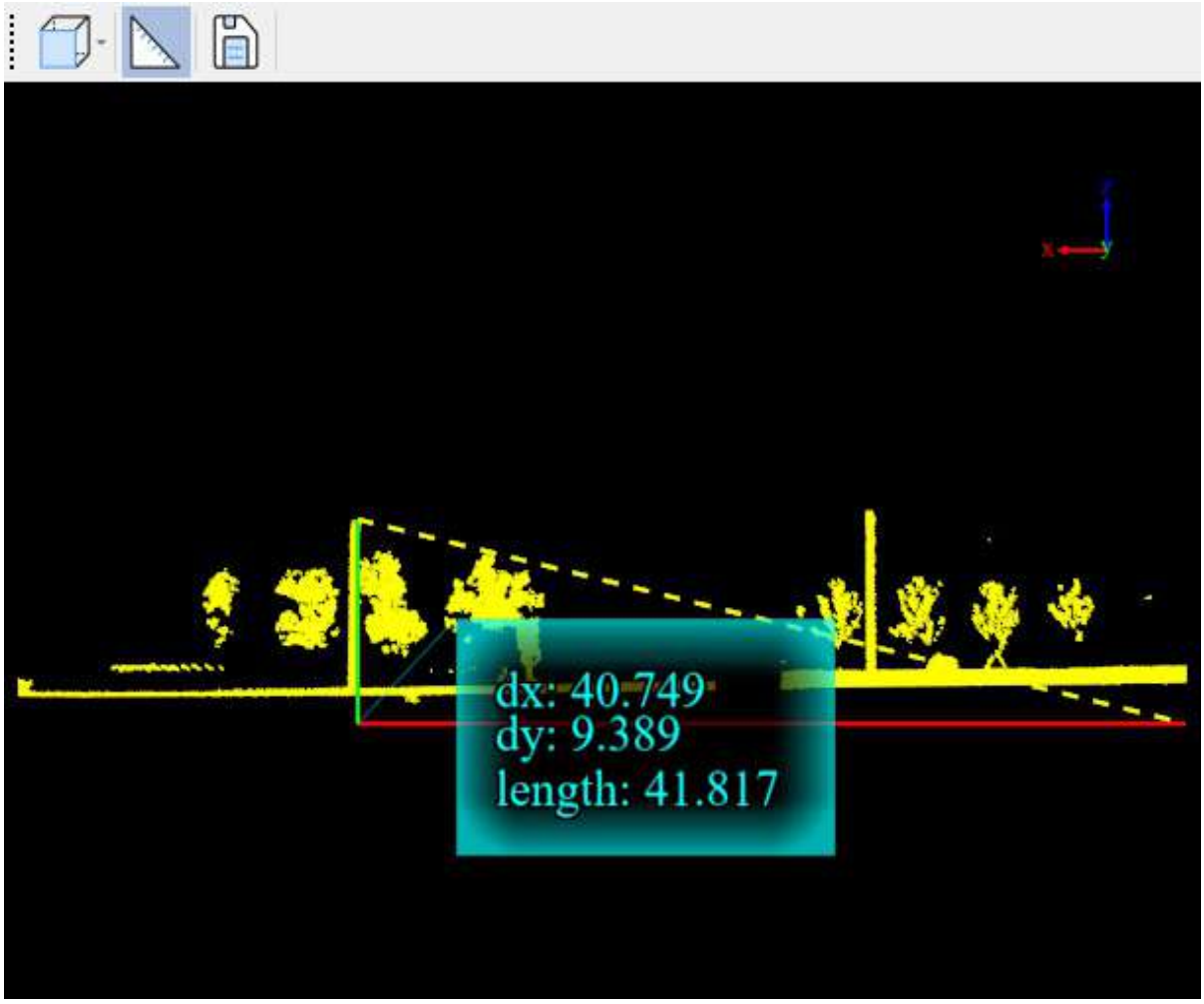
- **Buffer**:Control the width of the section.
- **Minimum Distance**：The surface unfolding operates similarly to the differential curve unfolding method. This parameter sets the granularity of differentiation. It is disabled when unfolding polygonal or cylindrical sections and can only be adjusted when unfolding surfaces.

2.After completing the parameter settings, draw the surface that needs to be unfolded in the main window using the B-spline drawing method. Once the drawing is completed, double-click the left mouse button to perform the section unfolding operation. After the progress bar completes, the result of the section unfolding will be displayed in the section window, as shown in the illustration below:：



The result of unroll curved section.

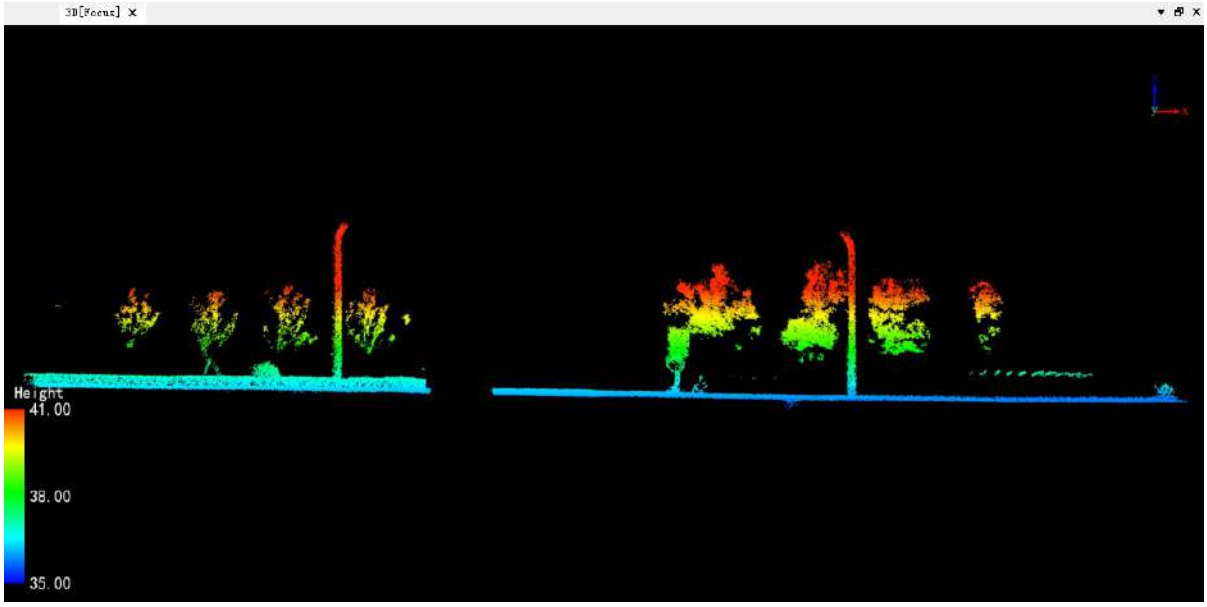
The arrow at position 4 in the above image indicates the direction of observation for the unfolded section result. Button 1 allows setting the front or back view. Button 2 enables measurement operations on the unfolded result, as shown in the image.



Measuring the unroll curved section result.

Button 3 in the unfolded surface section result image allows saving the unfolded section result. You can choose to save the format as LiData format point cloud for further processing in LiDAR360MLS, or save it as Las format point cloud for further processing in third-party software. The saved point cloud will retain the attributes of the original point cloud.

Select the format for saving the result.



The saved LiData point cloud can be opened in LiDAR360MLS.

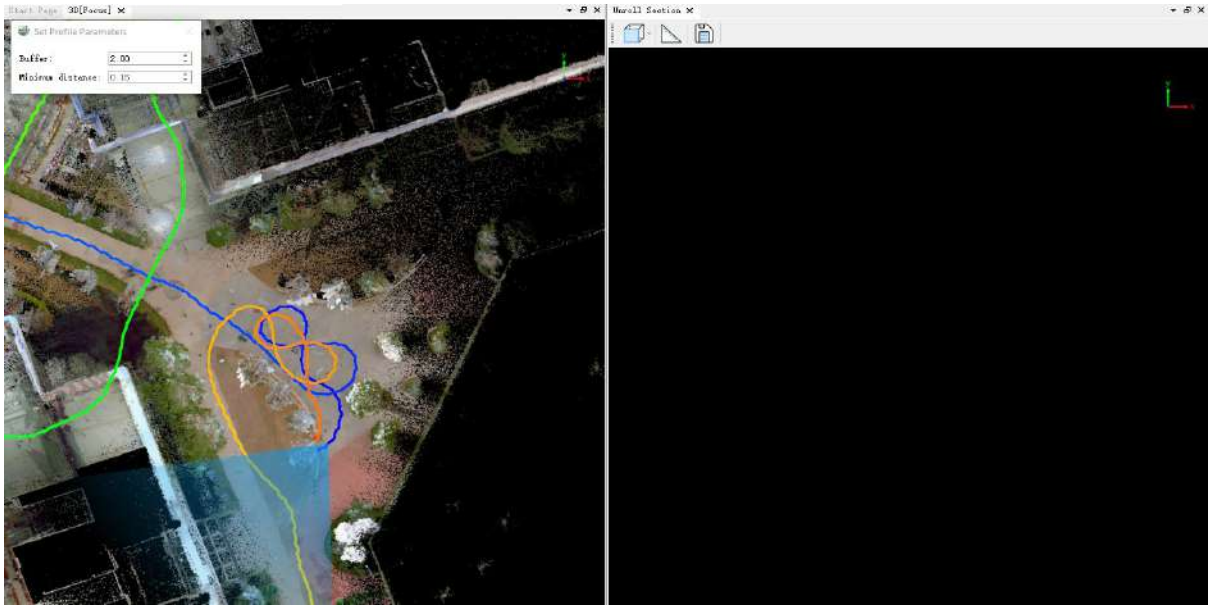
The saved Las point cloud can be opened in third-party software.

Unroll Cylindrical Section

Function Description：The cylindrical section unfolding tool utilizes a cylinder to intercept the point cloud in the vertical direction and unfold it onto the same plane.

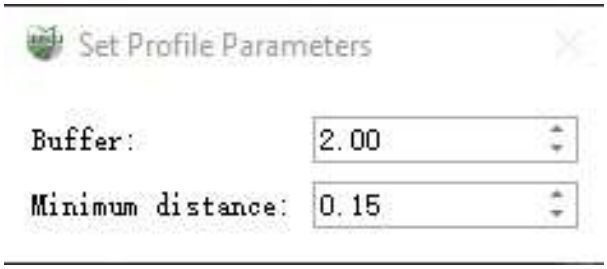
Steps

1.Open the LiDAR360MLS software and click on the **Unroll Cylindrical Section** button. The 3D window will default to 2D mode, and a section unfolding window will pop up. Additionally, the cylinder section unfolding button will lock other functionalities, and only manually lifting this button will exit this function and allow access to other functionalities. See the illustration below:



The cylinder section unfolding is activated.

The parameter settings window will be displayed in the top left corner of the 3D window. As shown in the image:

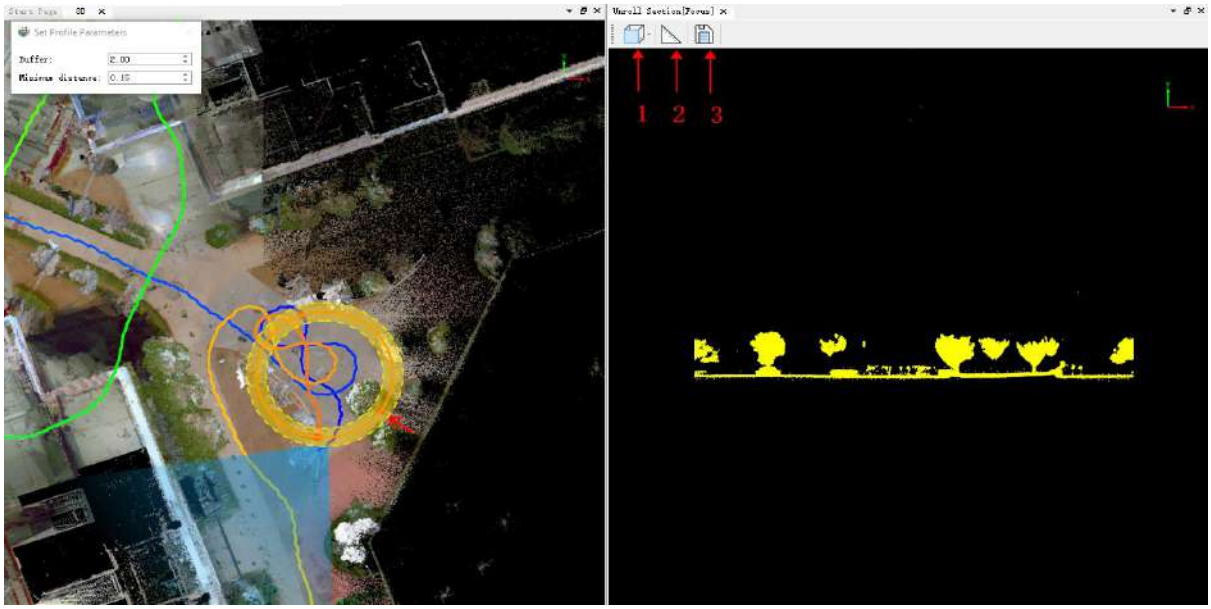


Parameter Settings Page

Parameter Setting

- **Buffer**：Control the width of the section.
- **Minimum Distance**：The surface unfolding operates similarly to the unfolding method of a differential curve. This parameter adjusts the granularity of the differentiation. It is disabled when unfolding polygonal or cylindrical sections and can only be adjusted when unfolding surfaces.

2.After completing the parameter settings, draw the cylinder that needs to be unfolded in the main window using the three-point circle drawing method. Once the drawing is completed, double-click the left mouse button to perform the section unfolding operation. After the progress bar completes, the result of the section unfolding will be displayed in the section window, as shown in the image.



The result of unfolding the cylinder section.

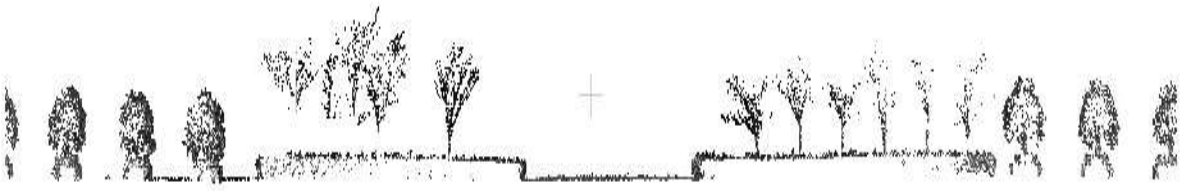
The arrow at position 4 in the above image indicates the direction of observation for the unfolded section result. Button 1, located at position 1, allows setting the front or back view. Button 2, located at position 2, enables measurement operations on the unfolded result, as shown in the image.

Measuring the unfolded cylinder section result.

Button 3 in the unfolded cylinder section result image allows saving the section unfolding result. You can choose to save the format as LiData format point cloud for further processing in LiDAR360MLS or save it as Las format point cloud for further processing in third-party software. The saved point cloud will retain the attributes of the original point cloud.

Select the format for saving the result.

The saved LiData point cloud can be opened in LiDAR360MLS.



The saved Las point cloud can be opened in third-party software.

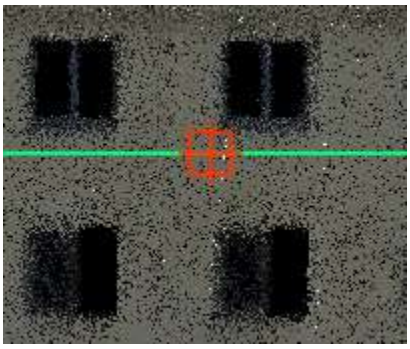
Catch

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

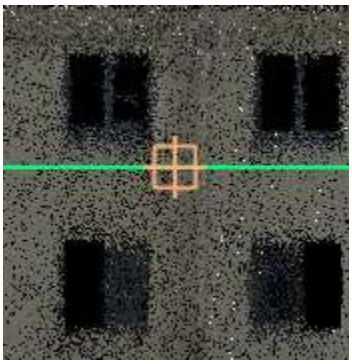
Steps

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

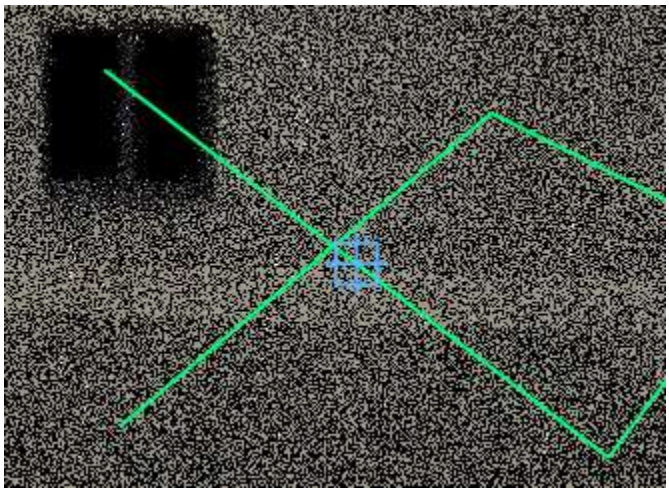
1.Snap on End Points: Capture each node of the vector data, and the state of the cross wire captured by the node is red.



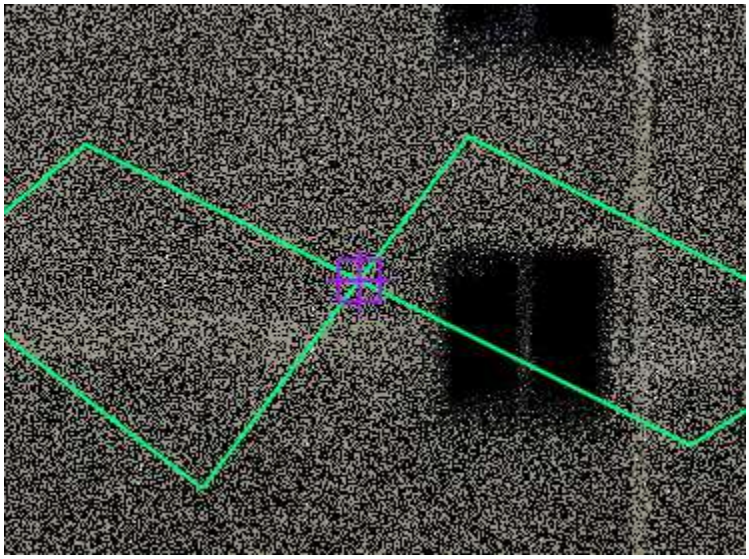
2.Snap on line: Capture the edges of vector data, such as any position on the edges of lines and polygons, the state of the line captured to the cross wire is orange.



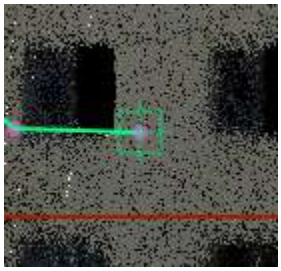
3.Snap middle: Capture the midpoint position of the vector line, and the state of the cross wire captured by the midpoint is blue.



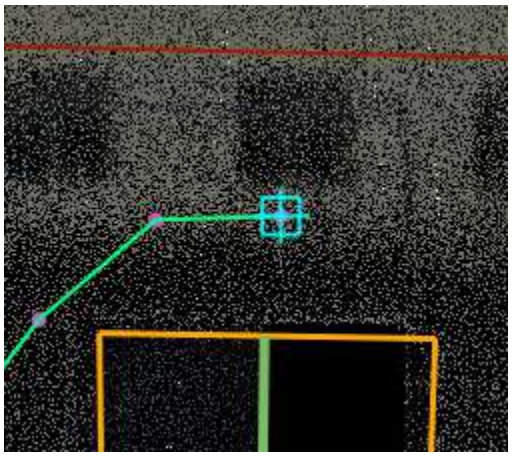
4.Snap intersection: Capture the intersection of the cross line, and the state of the cross wire captured at the intersection is purple.



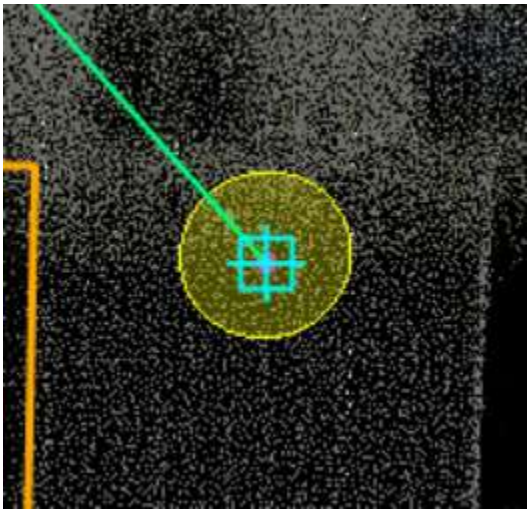
5.Snap Plane: Capture any point on the reference plane, and the state of the cross wire captured by the plane is green.

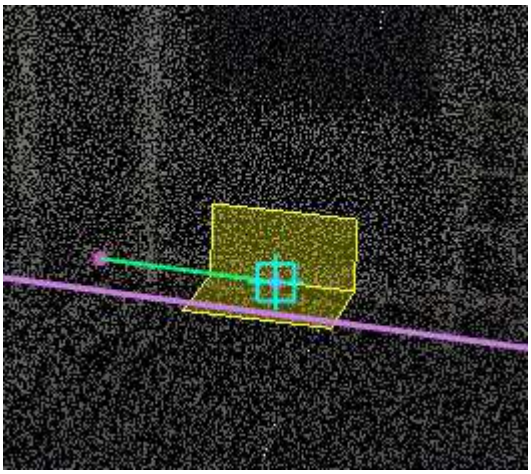


6.Snap Point Cloud: Capture the real point cloud coordinate points on the point cloud, and the state of the cross wire captured by the point cloud is light blue.



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





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

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

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

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

Point Cloud Tools

The point cloud tool includes basic point cloud processing functions.

- Point Cloud
- Extract

Point Cloud

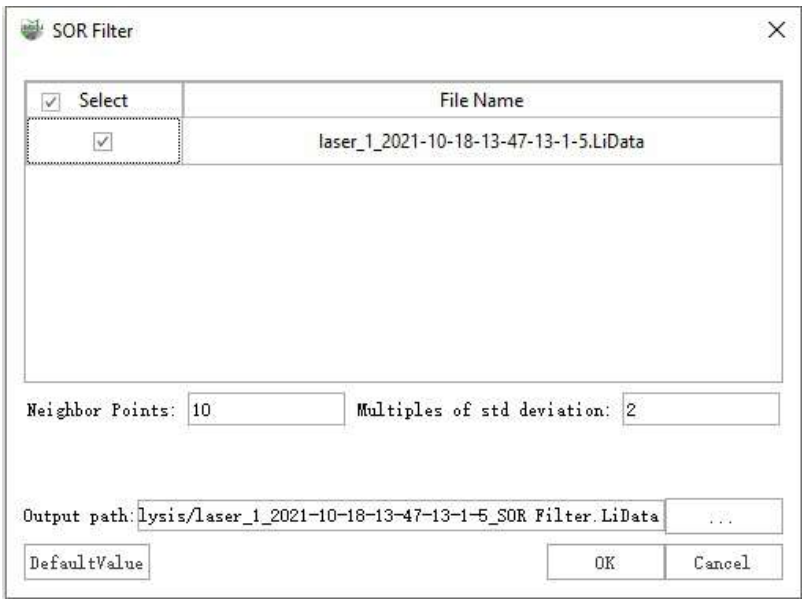
- SOR Filter
- Noise Filter
- SubSample
- Merge Data
- Normalization By Ground
- DeNormalization
- Compute Normals
- Transform GPS Time
- Brightness Contrast
- Define Projection
  - Geoid Model
- Projection
- Transform
  - CreatingConversionRelationships

SOR Filter

Function Description：Common noises include high level gross errors and low level gross errors. High level gross error is usually caused by the returns of high-flying objects (such as birds or aircraft) during the process of data collection; low-level gross error are returns with extremely low attitudes caused by the multipath effect of a laser pulse. The Outlier Removal tool aims to remove these errors as much as possible and therefore improve the data quality.
The algorithm will first search for each point's neighboring points within a user-defined area and calculate the average distance from the point to its neighboring points. Then, the mean and standard deviation of these average distances for all points are calculated. If the average distance of a point to its neighbors is larger than maximum distance (maximum distance = mean + n * standard deviation, where n is a userdefined multiple number), it will be considered as an outlier and be removed from the original point cloud.

Steps

1.Click the **SOR Filter** button to bring up the **SOR Filter pop-up box**.



Description:

- **Input Data**：The input file can be a single point cloud data file or multiple data files. File Format:*.LiData.
- **Neighbor Points (default value is "10")**: The number of points required in the neighborhood to calculate the average distance of each point. If there are not enough points found, the algorithm will not be executed.
- **Multiples of std deviation (default value is "5")**: The factor multiplied by the standard deviation to calculate the maximum distance.
- **Output path**: Path of the output file. After the function being executed, a new file will be generated. When more than one files are entered, the path needs to be set to a folder.

> Note: The algorithm of this function can be performed repeatedly to improve the denoising results. The outlier removal results is limited if the noises are to dense.

2.Comparison images before and after noise reduction.

Before denoising

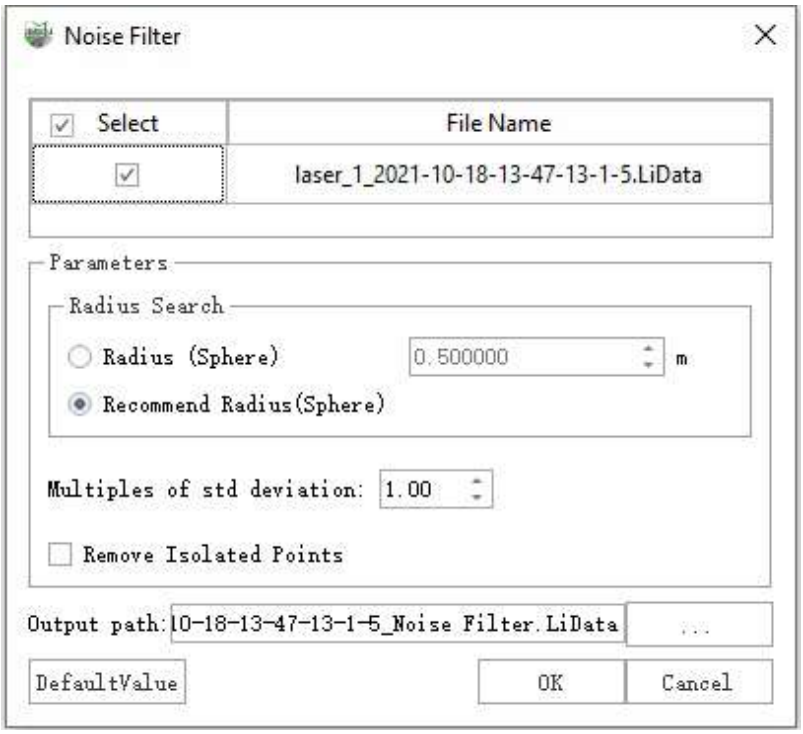
After denoising

Noise Filter

Function Description：Remove the outliers from the point cloud data. Different from the remove outliers function, this function has a better processing performance (filters more outliers) on objects with a flat surface (such as walls, interior tunnel walls, and the ground).

Steps

1.Click the **Noise Filter** button to bring up the **Noise Filter pop-up box**.



Description:

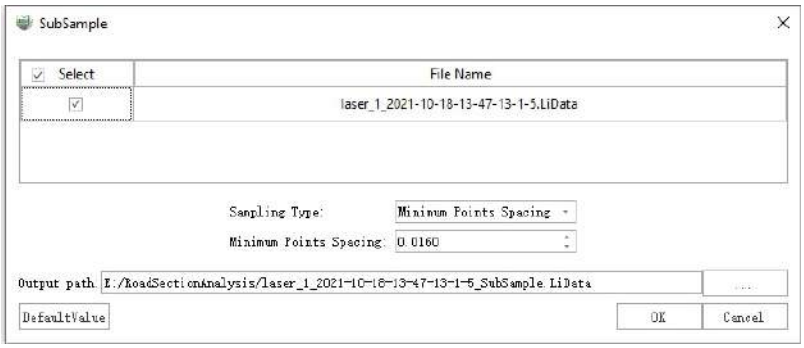
- **Input Data**：The input file can be a single point cloud data file or multiple data files. File Format:*.LiData.
- **Radius（m）(Default value is “0.5m”)**: Set the radius of the fitting plane. This function can be used when users know the approximate density of the point cloud data.
- **Recommend Radius**：Automatically calculate the appropriate searching radius according to the input point cloud data.
- **Multiples of std deviation (Default value is “1.0”)**: Using the relative error (sigma) as a parameter for outliers removal. The algorithm will automatically calculate the standard deviation (stddev) of a point P's surrounding fitting plane. If the distance, d, from this point to that plane is less than sigma * stddev, this point, P, will be kept. The reduction of this relative error results in removing more points. Conversely, more points will be retained. The change of this parameter will not influence the efficiency.
- **Remove Isolated Points**：The point will be treated as an isolated point when there is less than 4 points within the distance of the searching radius (cannot create a fitting plane with less than 4 points).
- **Output path**：The point cloud data will be save at this path after the outliers are removed.

SubSample

Function Description：Subsampling point clouds, namely, reducing the number of point clouds, LiDAR360 offers three methods for resampling: minimum point spacing, sampling rate, and octree.

Steps

1.Click the **SubSample** button to bring up the **SubSample pop-up box**.



Description:

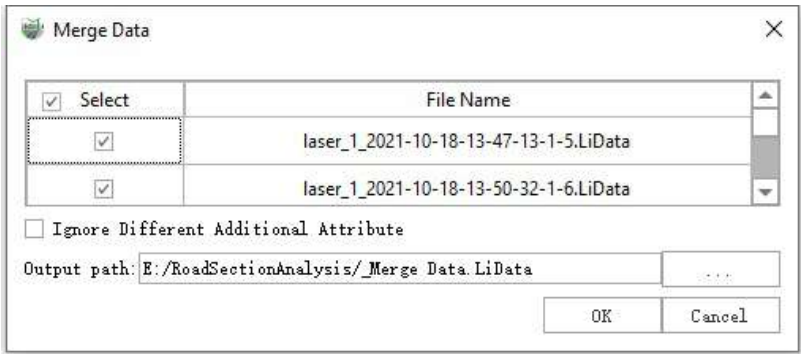
- **Input Data**：The input file can be a single point cloud data file or multiple data files. File Format: *.LiData.
- **Sampling Type**：This parameter defines the resampling method.
  - **Minimum Points Spacing (default, default value is "0.0000")**: Users need to set a minimum point spacing between two points so that the minimum three-dimensional distance between any two points in the sampled point cloud will not be less than this value. The larger the value is set, the fewer points will be kept.
  - **Sampling Rate (default value is "99.99%")**: Users need to set the percentage of reserved points. In this mode, LiDAR360 will randomly retain the specified number of points. Reserved Points = Total Number of Points * Sample Rate. The value of this parameter ranges from 0 to 100%. The smaller the value is set, the fewer points will be kept.
  - **Octree (default value is "21")**: This method allows users to select an "octree" subdivision level and build 3D voxels for the input point cloud. Only the point closest to each voxel center will be retained. The "octree" subdivision level ranges from 1 to 21. The smaller the value is set, the fewer points will be kept.
- **Output path**：Path of the output file. New file will be generated after the function being executed.

Merge Data

Function Description：Merge multiple point cloud files into a single point cloud file.

Steps

1.Click the **Merge Data** button to bring up the **Merge Data pop-up box**.



Description:

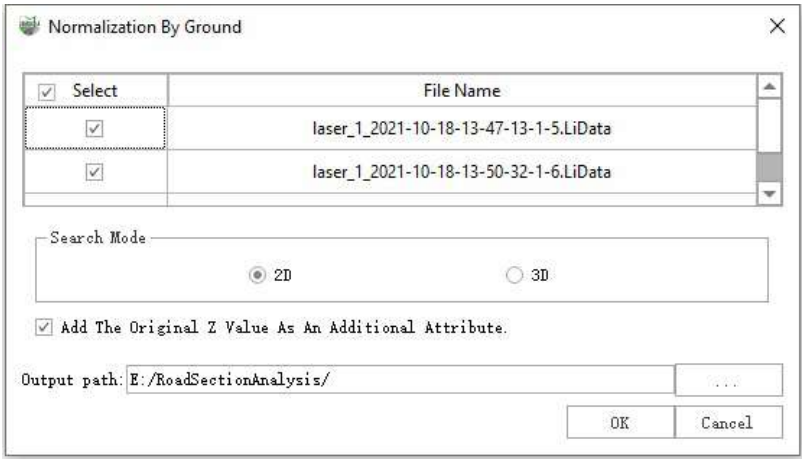
- **Input Data**：The input file can be a single point cloud data file or multiple data files. File Format: *.LiData.
- **Ignore Different Additional Attributes**: When the input data sets are more than one, if this option is checked, then only the same additional attributes will be merged. If there are any different additional attributes and this option is unchecked, the mergence cannot be done.
- **Output path**: Path of the output folder. After the algorithm being executed, a new file will be generated.

Normalization By Ground

Function Description：The normalization tool can remove the effects of topographic relief on the elevation value of point cloud data. This function requires that the input data has already been classified into ground points and non-ground points. The normalization process is performed by subtracting the terrain elevation (represented by the elevation of the closet ground point to each point) from each point's Z value.

Steps

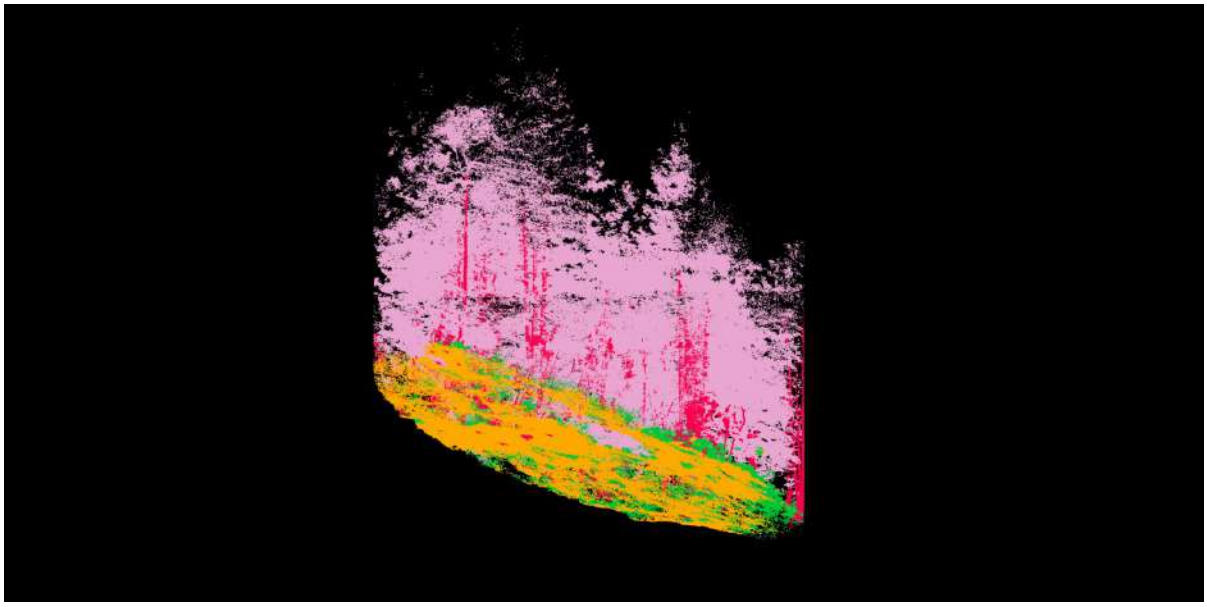
1.Click the **Normalization By Ground** button to bring up the **Normalization By Ground pop-up box**.



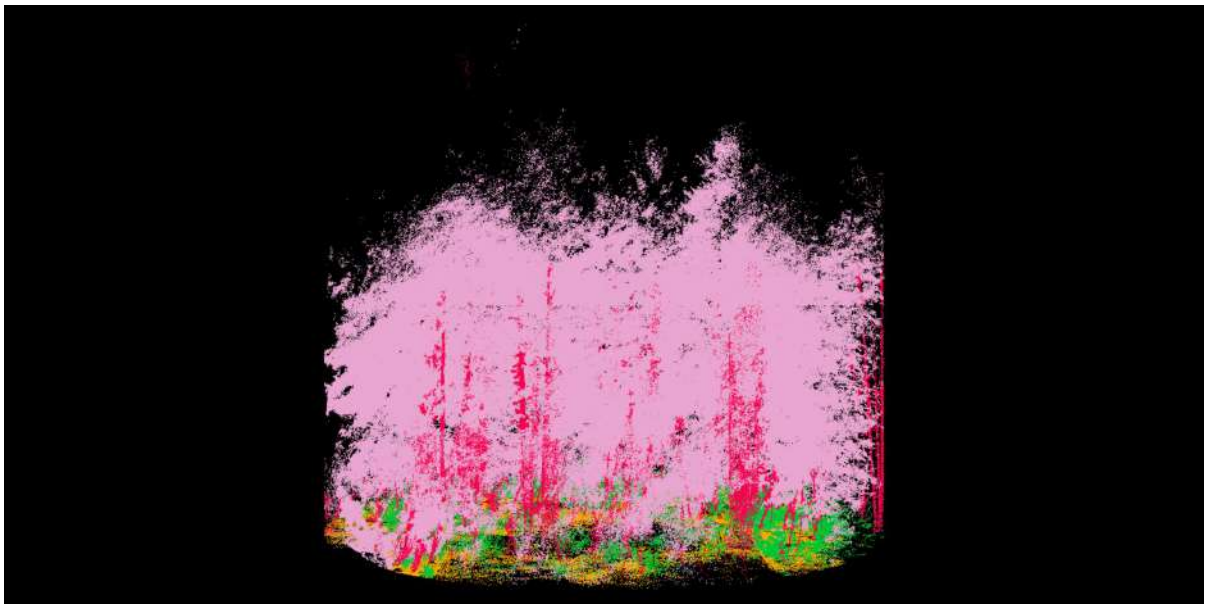
Description:

- **Input Data**：The input file can be a single point cloud data file or multiple data files. File Format:*.LiData.
- **Search Mode**: 2D method is suitable for most terrains, while 3D method is suitable for cliff terrain and can solve the problem of incorrect vegetation height after normalization. However, steep slopes or other terrains are recommended to use 2D search.
- **Add Z Value to Additional Attributes**: Add the Z value of the current point cloud to the additional attribute table. If this option is not checked, then the normalized point cloud cannot be denormalized.
- **Output path**: Path of the output file. After the function being executed, the normalized new file will be generated. When entering more than one file, the path needs to be set as a folder.

2.Comparison of the post-processing results.



Before Normalization By Ground



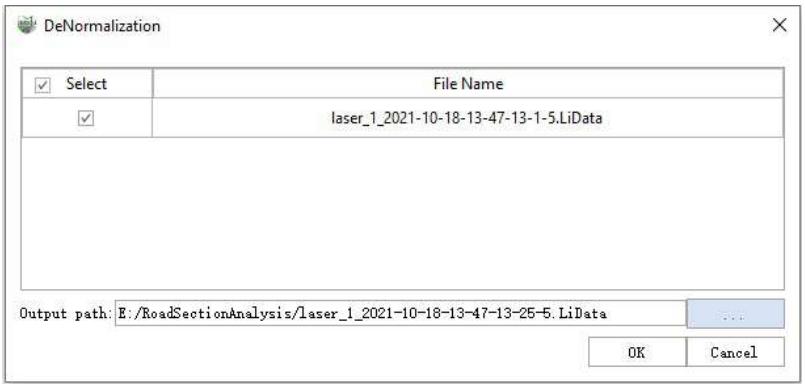
After Normalization By Ground

DeNormalization

Function Description：Denormalization tool can reset the Z value of the normalized data. Users need to check the option "Add Original Z Value to Additional Attributes" when performing Normalize by Ground Points. After denormalization, the Z value will be reset as the original Z value stored in the additional attributes.

Steps

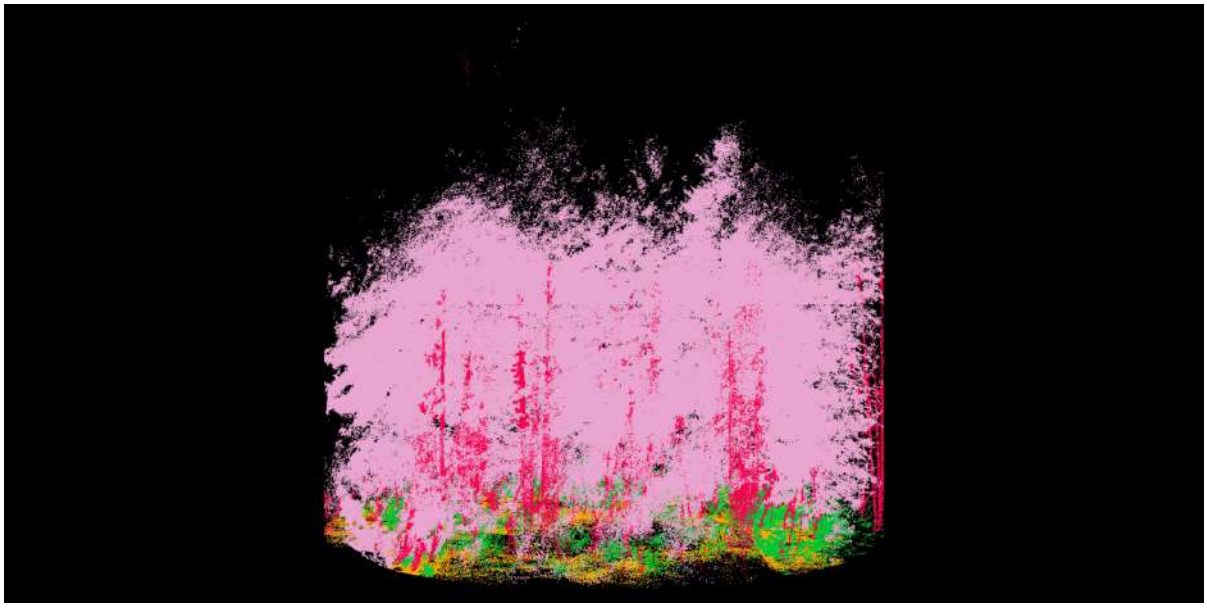
1.Click the **DeNormalization** button to bring up the **DeNormalization pop-up box**.



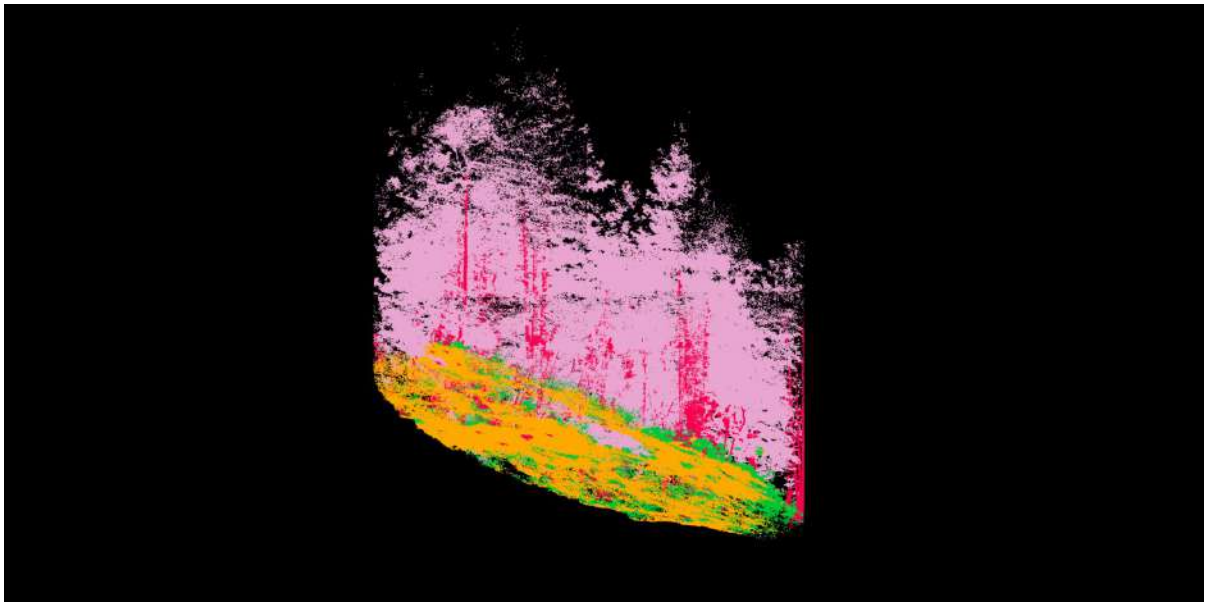
Description:

- **Input Data**：The input file can be a single point cloud data file or multiple data files. File Format:*.LiData.
- **Output path**: Path of the output file. After the function being executed, a new file will be generated. When more than one files are entered, the path needs to be set to a folder.

2.Comparison of the post-processing results.



Before DeNormalization



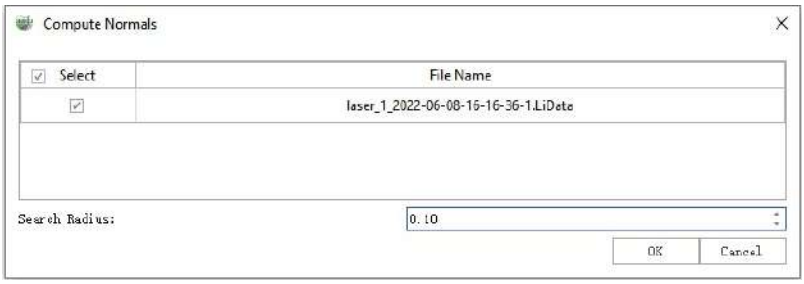
After DeNormalization

Compute Normals

Function Description：Calculate normals for point clouds and write them into the additional attributes of LiData.

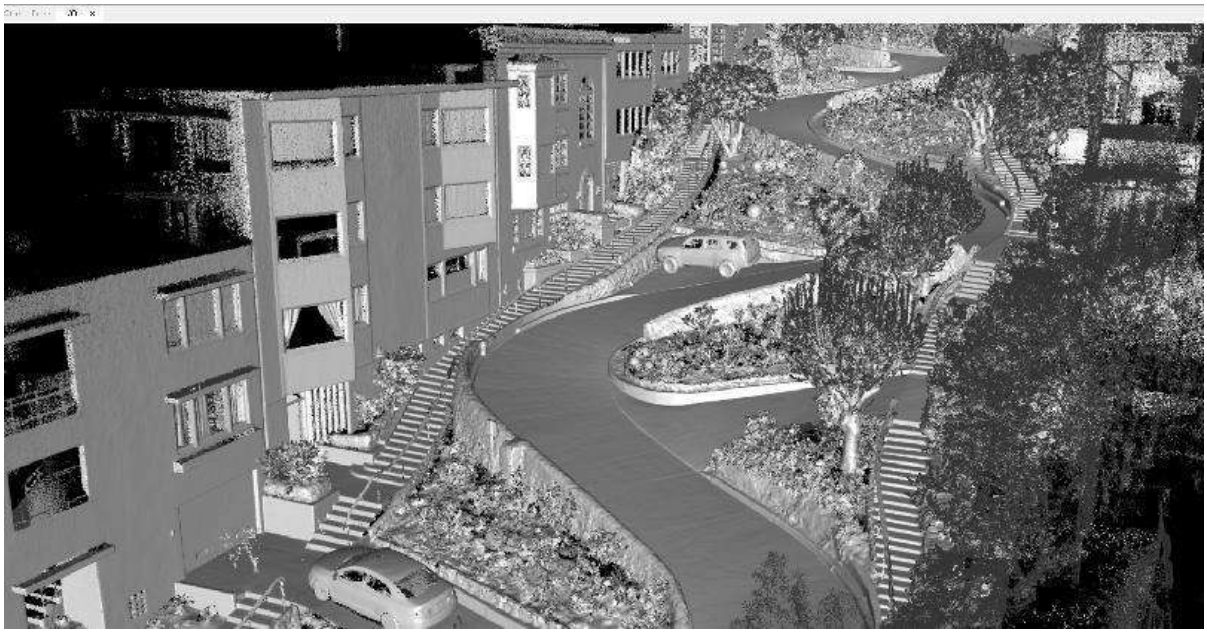
Steps

After launching the software, click on the **Compute Normals** button to open the parameter settings window:



- **Search Radius**: The neighborhood search radius. The algorithm obtains the neighboring points by performing a neighborhood search for each point, and then calculates the normal direction based on these points. Both too large and too small values will lead to incorrect calculation of the normals, and too large values will also lead to reduced efficiency of the algorithm. This value can be set to 5 times the average point spacing to get the best results.

2.The effect after calculating the normal is shown below:

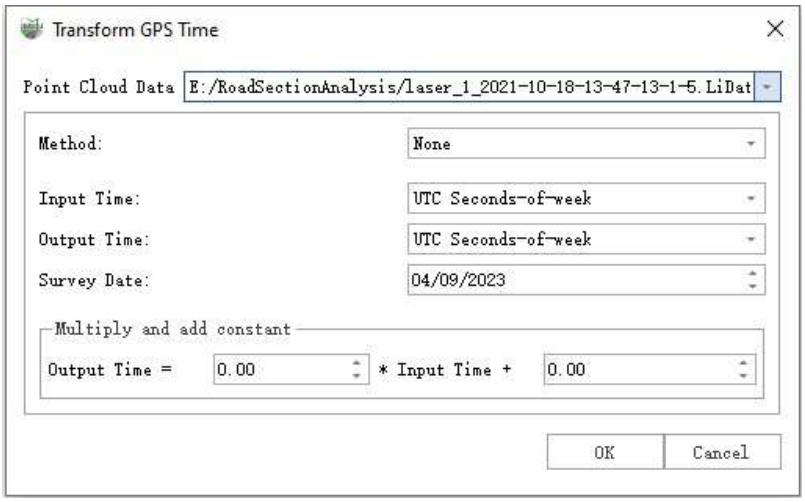


Transform GPS Time

Function Description: This function converts time recorded in LiData data to time in different time references, including GPS seconds of day, GPS seconds of week, UTC seconds of day, UTC seconds of week, GPS coordinate time, GPS standard time, and UNIX time.

Steps

1.Click the **Transform GPS Time** button to bring up the **Transform GPS Time pop-up box**.



Description:

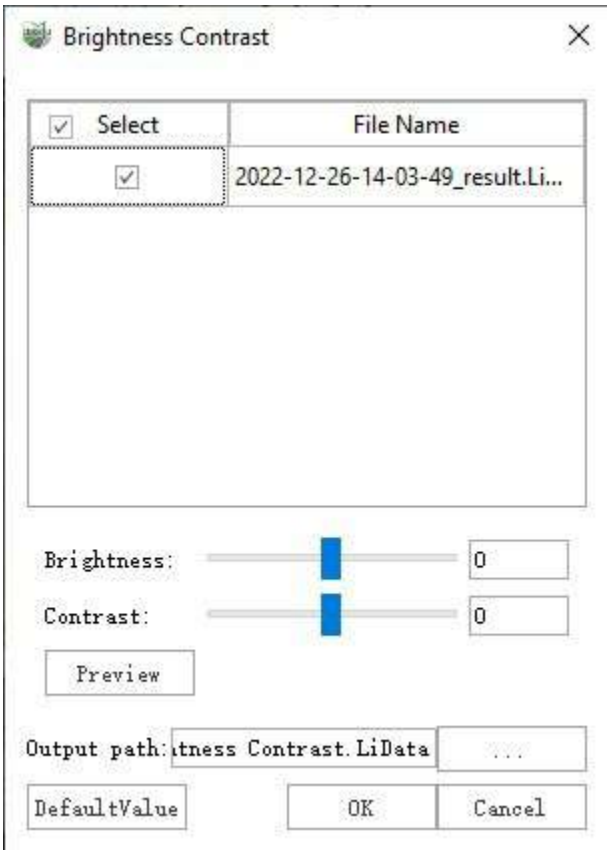
- **Input Data**: Enter a LiData point cloud data file.
- **Method**: You can choose from various methods for time conversion on LiData point cloud files, including:
  - **None**: Do not perform time reference conversion.
  - **Source and target system**: Select the time reference of the source LiData file and the time reference of the target LiData file. Also, choose the acquisition date for the conversion.
  - **Multiply and add constant**: Perform a linear transformation on time using the formula: Output Time = a * Input Time + b.
- **Input Time**: Select the time reference of the source LiData file.
- **Output Time**: Select the time reference of the target LiData file.
- **Survey Date**: Choose the date of point cloud data collection.

Brightness Contrast

Function Description：For point clouds containing color information, adjust the brightness and contrast of the point cloud and then save it.

Steps

1.Click the **Brightness Contrast** button, adjust the two parameters until the preview effect meets expectations, then click **OK** to apply the adjustments.



Parameter Description:

- **Input Data**: The input file can be either a single point cloud data file or a point cloud dataset. The data to be processed must be opened in the software. Only point clouds containing color information will be displayed in the list.
- **Brightness**: Adjust by sliding or directly inputting brightness values. The range is from -100 to +100.
- **Contrast**: Adjust by sliding or directly inputting contrast values. The range is from -50 to +50.
- **Preview**: After adjusting the parameters, click to preview the effect of the current parameters in the window. The preview only affects the selected point cloud data.
- **Output Path**: The path for saving the resulting point cloud file.
- **Default Values**: Restore the initial brightness and contrast values.

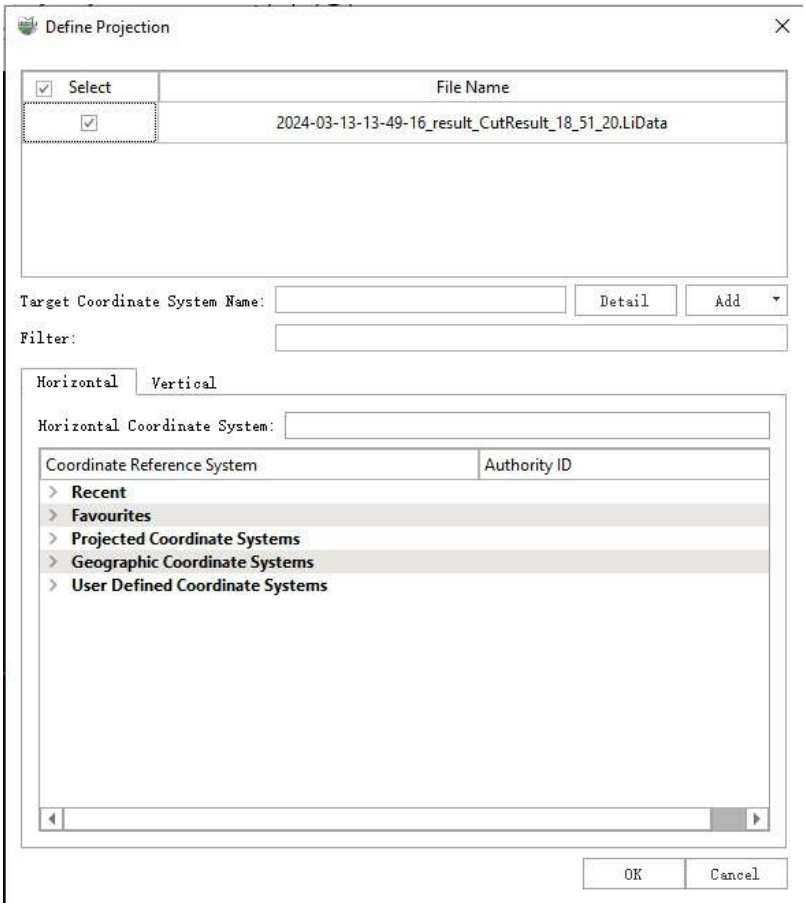
> Note: This functionality only applies to point clouds containing color information.

Define Projection

Function Description：Define a coordinate reference system for point cloud data. The coordinate reference system generally includes geographic coordinates (usually latitude and longitude coordinates), geocentric coordinates and projected coordinates (plane coordinates converted by projection). This feature only supports defining geographic and projected coordinates.

Steps

1.Click the **Define Projection** button to bring up the **Define Projection pop-up box**.



Description:

- **Select Input file**: The input file can be a single point cloud data file or multiple point cloud data files. File format: *.LiData.
- **Target Coordinate System Name**：Displays the name of the currently set coordinate system and supports custom setting of the coordinate system name.
- **Detail**：Displays detailed information about the currently set coordinate system.
- **Add Coordinate System(Add)**: Users can add external coordinate system settings or customize coordinate system settings. LiDAR360MLS software provides three ways to add external coordinate system:
  - **Add Geographic Coordinate System**
  - **Add Projected Coordinate System**

    - **Add Vertical Coordinate System**
- **Filter**: Users need to enter a customized coordinate system. By entering the coordinate system keywords, the corresponding coordinate system can be filtered from the **Coordinate reference systems of the world** table (for example: to set the point cloud coordinate system to WGS 84 / UTM Zone 49N, users can enter UTM 49N in the filter for fast screening, or enter its EPSG number 32649 for quick search.) Users can also import external coordinate system by clicking the **Add Coordinate System** button.

- **Set Coordinate System**: Select the coordinate system to be set, either from the default coordinate system or a custom coordinate system. Firstly, you can select the horizontal coordinate system in the **Horizontal** tab, and then the vertical coordinate system will be ellipsoid height by default. If you need to define a vertical coordinate system, you can switch to the **Vertical** tab to select a vertical coordinate system, and the two together form a combined coordinate system. The coordinate system selection interface contains the following nodes:

    - **Recent**: Records the recently used coordinate system.
    - **Favourites**: Coordinate system for adding collections.
    - **Projected Coordinate Systems**: All default projected coordinate systems are included.
    - **Geographic Coordinate Systems**: All default geographic coordinate systems are included.
    - **User Defined Coordinate Systems**: All custom coordinate systems are included.
    - **Vertical Coordinate Systems**: All default vertical coordinate systems are included.
    - **User Defined Vertical Coordinate Systems**: All custom vertical coordinate systems are included.

    > Right-click on any coordinate system, select the **Favourites** function from the pop-up menu, and the selected coordinate system will be added to bookmarks. Right-click on a bookmarked coordinate system, select the **Remove from Favourites** function, and the coordinate system will be removed from bookmarks.
    >
    > Right-click on any custom coordinate system, select the **Delete** function, to delete the custom coordinate system. If the deleted custom vertical coordinate system is being used in a custom composite coordinate system, it will prompt whether to delete the composite coordinate system simultaneously. Choosing "Yes" will delete all these coordinate systems, while choosing "No" will not proceed with the deletion operation.
    >
    > Right-click on any coordinate system, select the **Copy and Modify** function to modify the coordinate system parameters and save it as a custom coordinate system. The parameters of the currently selected coordinate system will be filled into the corresponding modification interface. For specific settings, refer to the "Adding Coordinate Systems" section on this page.

- **Set Geoid Model**: On the **Vertical** tab page, the user can set the geoid model of the vertical coordinate system. If the user need to use this option, the user need to provide the vertical reference grid data. For the method, please refer to the next section **Vertical Coordinate System**. At present, this setting has the following three situations.

    - When this option is not set, if the vertical datum grid data is provided, the appropriate geoid model will be automatically searched and used during coordinate transformation. If not provided, the geoid model will not be used during coordinate transformation (except for the EGM2008 model included in the software installation package).
    - If the selected vertical coordinate system can use the geoid model supported by default, it can be selected from the drop-down list, and the model will be specified when performing coordinate transformation. For the geoid models currently supported by default in LiDAR360MLS software, please refer to the Geoid Model page.
    - If the geoid model to be used is not included in the package provided by PROJ, the user can place the

model file to be used under the geoid folder in the software installation directory, and then add the geoid model by clicking the **Add Geoid** button to add the file (support tiff and gtx formats), the model will be added to the drop-down list, after selecting the added model, the custom geoid model will be used in the coordinate transformation process.

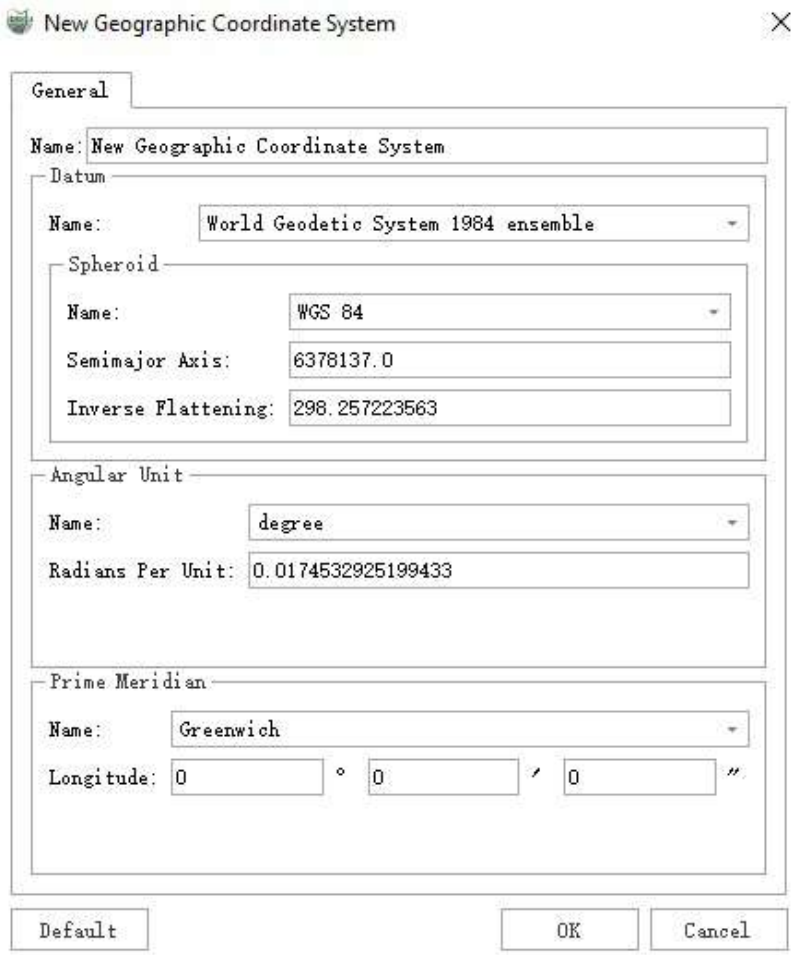
> If the coordinate range of the selected geoid model does not match the coordinate range after point cloud coordinate transformation, the model will not take effect.

- **Selected Coordinate System**: The information of the coordinate system selected by the user will be displayed here.
- After clicking **OK**, the WKT information representing the projection will be written to LiData.

Vertical Coordinate System: Users can customize the vertical coordinate system. Vertical coordinate system transformations and grid projections require vertical datum grid data for valid transformations. EGM2008 geoid model elevation conversion is provided by default in the software. If you need to support other regional geoid models, you need to download the corresponding grid data files. The grid files are provided by the official PROJ library, which can be downloaded here proj-data-1.8, or on the official website https ://proj.org/download.html to download the latest version of proj-data. After downloading, unzip the file to the geoid folder in the software installation directory, the default is C:\Program Files\GreenValley Suite\LiDAR360MLS\7.2.0.0\geoid, "7.2.0.0" is the version number, and there will be differences between different versions.

> Note: The custom geoid model must be placed in the geoid folder to take effect.

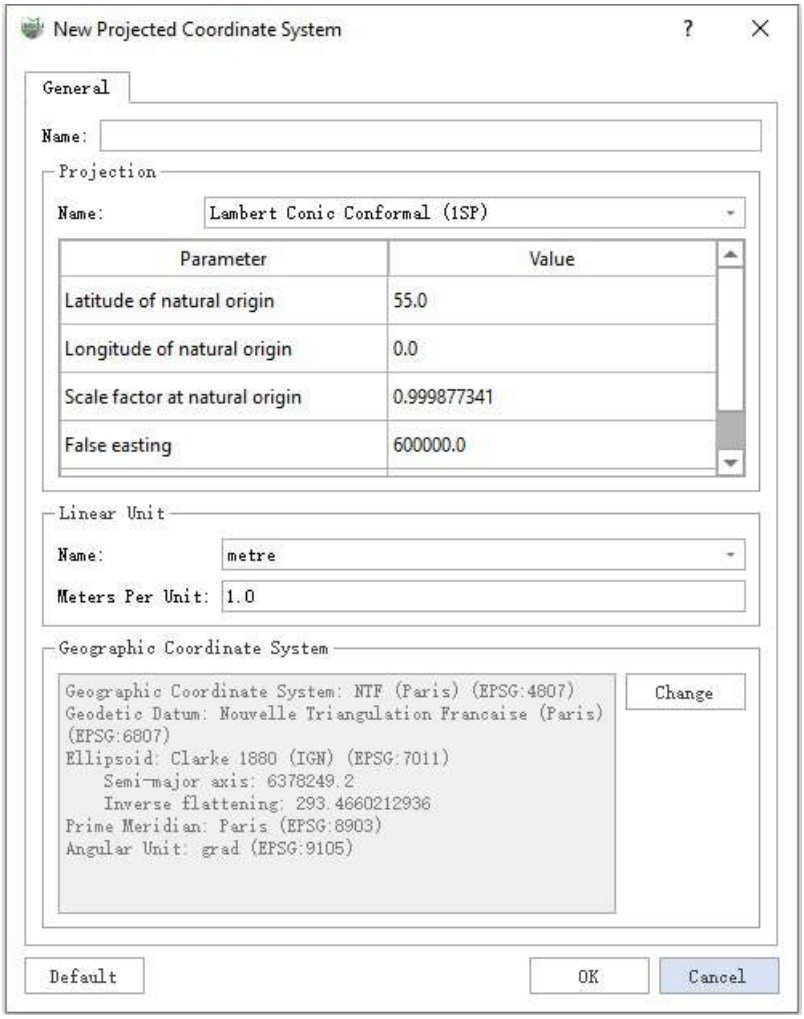
Add Geographic Coordinate System



- **Name**：Users can customize the name of a geographic coordinate system.

- **Datum**:
  - **Name**: Users can customize a datum name or select a known datum from the drop-down list.
  - **Spheroid**: If a user selects the datum from the drop-down list, the ellipsoid parameter does not require to be entered.
    - **Name**: The name of the spheroid.
    - **Semimajor Axis**: Spheroid's major semiaxes.
    - **Inverse Flattening**: Inverse flattening of the spheroid.
- **Angular Unit**:
  - **Name**: Users can customize the angle unit name, or select the angle unit name from the drop-down list.
  - **Radians Per Unit**: Define the unit of arc in the geographic coordinate system.
- **Prime Meridian**:
  - **Name**: Users can customize the name of the central meridian, or select the name of the central meridian from the drop-down list.
  - **Longitude**: Enter the central meridian longitude.

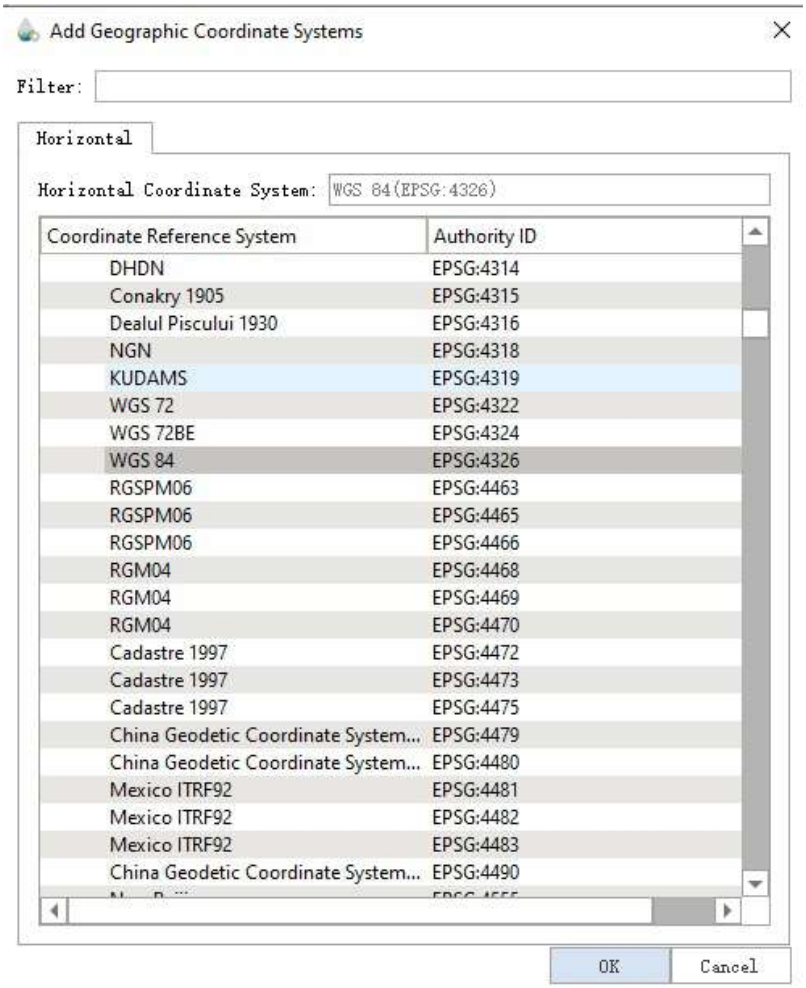
Add Projected Coordinate System



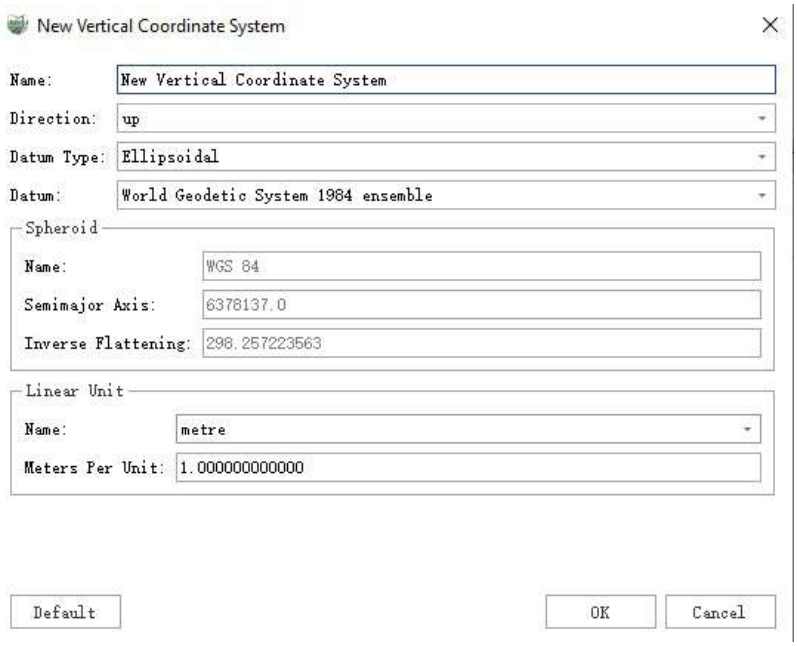
- **Name**: Users can customize the name of a projection coordinate system.
- **Projection**:
  - **Name**: Users can customize a projection name or select a known projection from the drop-down list. It should be noted that user-defined projections can be modified from a projection template selected from the drop-down list.
- **Linear Unit**:

- **Name**：Users can customize the linear unit name or select the linear unit name from the drop-down list.
- **Meters per Unit**: Input meters per unit.

- **Geographic Coordinate System**: User can click Modify to select a geographic coordinate system.
  - **Change**: When the user clicks the Change button, the following dialog box will pop up to modify the geographic coordinate system. Users can filter out the desired coordinate system through the Filter operation. Supports selection of default geographic coordinate system and custom geographic coordinate system.



Add Vertical Coordinate System



- **Name**: Users can customize the name of a vertical coordinate system.
- **Direction**: Sets whether the direction of the axis is up or down.
- **Datum Type**: Sets the datum type of the vertical coordinate system. Ellipsoidal is an ellipsoidal datum and Gravity-related is a gravity field datum.
- **Datum**: This option allows you to set the ellipsoid datum when the datum type is set to Ellipsoidal. When selected, the ellipsoid parameters will be automatically populated in the ellipsoid parameter list below.
- **Vertical Datum**: This option allows you to set the vertical datum when the datum type is set to Gravity-related.
- **Spheroid**: If a user selects the datum from the drop-down list, the ellipsoid parameter does not require to be entered.
  - **Name**: The name of the spheroid.
  - **Semimajor Axis**: Spheroid's major semiaxes.
  - **Inverse Flattening**: Inverse flattening of the spheroid.
- **Linear Unit**:
  - **Name**: Users can customize the linear unit name or select the linear unit name from the drop-down list.
  - **Meters per Unit**: Input meters per unit.
- **Default**: Restore default parameters.

Geoid Model

When performing coordinate transformation in the projection library, vertical coordinate system transformation and grid projection need to provide vertical datum grid data for effective transformation. EGM2008 geoid model elevation conversion is provided by default in the software. If you need to support other regional geoid models, you need to download the corresponding grid data files. The grid files are provided by the official PROJ library, which can be downloaded here proj-data-1.8, or on the official website https ://proj.org/download.html to download the latest version of proj-data. After downloading, unzip the file to the geoid folder in the software installation directory, the default is C:\Program Files\GreenValley Suite\LiDAR360MLS\7.2.0.0\geoid, "7.2.0.0" is the version number, and there will be differences between different versions.

Supported Geoid Model

Geoid Model	Vertical coordinate system
GEOID99	NAVD88 height - EPSG:5703
	NAVD88 height (ft) - EPSG:8228
	NAVD88 height (ftUS) - EPSG:6360
GEOID03	NAVD88 height - EPSG:5703
	NAVD88 height (ft) - EPSG:8228
	NAVD88 height (ftUS) - EPSG:6360
GEOID06	NAVD88 height - EPSG:5703
	NAVD88 height (ft) - EPSG:8228
	NAVD88 height (ftUS) - EPSG:6360
GEOID09	NAVD88 height - EPSG:5703
	NAVD88 height (ft) - EPSG:8228
	NAVD88 height (ftUS) - EPSG:6360
GEOID12A	NAVD88 height - EPSG:5703
	NAVD88 height (ft) - EPSG:8228
	NAVD88 height (ftUS) - EPSG:6360
	NMVD03 height - EPSG:6640
	PRVD02 height - EPSG:6641
	VIVD09 height - EPSG:6642
	ASVD02 height - EPSG:6643
	GUVD04 height - EPSG:6644
	NAVD88 height - EPSG:5703
	NAVD88 height (ft) - EPSG:8228
	NAVD88 height (ftUS) - EPSG:6360
	NMVD03 height - EPSG:6640

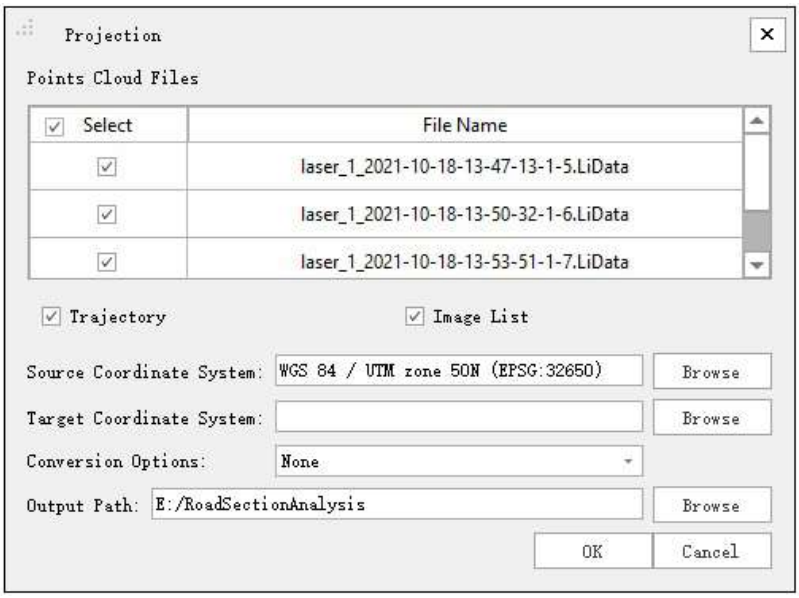
GEOID12B	PRVD02 height - EPSG:6641
	VIVD09 height - EPSG:6642
	ASVD02 height - EPSG:6643
	GUVD04 height - EPSG:6644
GEOID18	NAVD88 height - EPSG:5703
	NAVD88 height (ft) - EPSG:8228
	NAVD88 height (ftUS) - EPSG:6360
	PRVD02 height - EPSG:6641
	VIVD09 height - EPSG:6642
GGM10	NAVD88 height - EPSG:5703
	NAVD88 height (ft) - EPSG:8228
	NAVD88 height (ftUS) - EPSG:6360
OSGM15	Belfast height - EPSG:5732
	Douglas height - EPSG:5750
	Lerwick height - EPSG:5742
	Malin Head height - EPSG:5731
	ODN (Offshore) height - EPSG:7707
	ODN height - EPSG:5701
	ODN Orkney height - EPSG:5740
	St. Marys height - EPSG:5749
	Stornoway height - EPSG:5746
EGM96	EGM96 height - EPSG:5773
EGM2008	EGM2008 height - EPSG:3855
	MSL height - EPSG:5714
GSIGEO2001	JGD2011 (vertical) height - EPSG:6695

Projection

Function Description：This tool allows the user to project the coordinate system of the point cloud in the software.

Steps

1.Click the **Projection** button to bring up the **Projection pop-up box**.



Description:

- **Input Data**：The input file can be a single point cloud data file or multiple data files. File Format:*.LiData.
- **Trajectory**：Whether the trajectory is converted or not.
- **Image List**：Whether the image is converted or not.
- **Source Coordinate System**: The software automatically reads the coordinate system of the point cloud and vector files and displays them in the output box. Users can also click the browse button to manually select.
- **Target Coordinate System**: Click the Browse button on the right to select the projection coordinate system to be exported.
- **Conversion Options**: Select the method of projection conversion.
- **Output Path**: Select the output path after the point cloud and vector are converted, and the software automatically defaults to the path where the current point cloud is located.

Transform

Function Description：LiDAR360 MLS software supports multiple coordinate transformation methods. Users can select a transformation method to perform coordinate conversion according to their needs.

Steps

1.Click the **Transform** button to bring up the **Transform pop-up box**.

Description:

- **Input Data**： The input file can be a single point cloud data file or multiple data files. File Format: *.LiData.
- **Select or customize transformation method**: available for reference, Create Transformation Relationship.
- **Trajectory**：Whether the trajectory is converted or not.
- **Image List**：Whether the image is converted or not.
- **Output Path**：Path of the output folder. After the algorithm being executed, new files after coordinate conversion will be generated.

Create Transformation Relationship

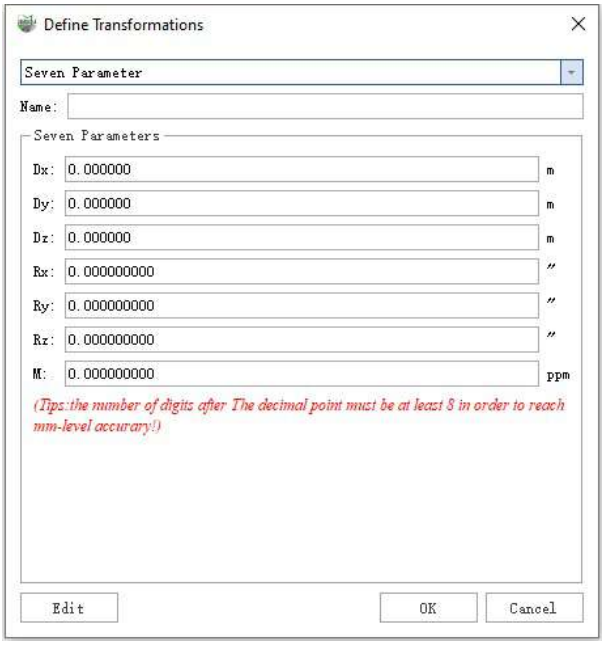
Function Description：The user can define and edit a variety of transformation relationships that can be saved to a history file for the user to reuse. These transformation relationships can be defined during "transformation relationship calculations" as well as when importing data.

> 1ppm = 0.001‰ is defined as one part per million (1 PPM)

Supported conversion relationship:

- Four parameters:
  - Translation parameter Dx (meter, default value: 0).
  - Translation parameter Dy (meter, default value: 0).
  - Rotation parameter Theta (second, default value: 0).
  - Zoom ratio t (default value: 1.0).
  - Elevation fitting: You can check whether to carry out elevation fitting. If elevation fitting is checked, you need to select predefined elevation fitting parameters from the drop-down list.

    > Note: When converting with Dx -0, Dy -0, Theta = 0, t = 1.0, the original paint cloud will be obtained.



The four-parameter formula can be written as follows:

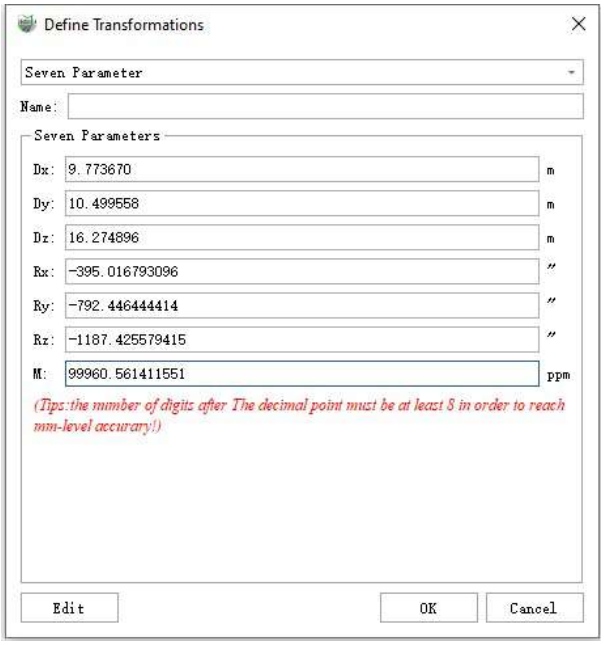
$$X_2 = X_0 + X_1 m COS\,\alpha - Y_1 m \sin\alpha$$
$$Y_2 = Y_0 + Y_1 m COS\,\alpha + X_1 m \sin\alpha$$

It can be solved according to the following formula:

$$\begin{bmatrix} X_2 \\ Y_2 \end{bmatrix} = \begin{bmatrix} 1 & 0 & X_1 & -Y_1 \\ 0 & 1 & Y_1 & X_1 \end{bmatrix} \begin{bmatrix} X_0 \\ Y_0 \\ C \\ D \end{bmatrix} \qquad m = \sqrt{C^2 + D^2} \qquad \alpha = \arctan\left(\frac{D}{C}\right)$$

- Seven parameters: The seven parameters in the Bursa model.
  - Translation parameter Dx (meter, default value: 0).
  - Translation parameter Dy (meter, default value: 0).
  - Translation parameter Dz (meter, default value: 0).
  - Rotation parameter Rx (meter, default value: 0).
  - Rotation parameter Ry (meter, default value: 0).
  - Rotation parameter Rz (meter, default value: 0).
  - Zoom ratio m (ppm).

    > Note: The scale m in the seven parameters is different from the scale t in the four parameters. The two can be transformed into each other. The conversion relationship is m = (t - 1.0) *1e6. Assuming t is 0.9999988, then m = (0.9999988-1.0)* 1e6 = -1.2.



The formula of Bursa's seven parameters can be written as follows:

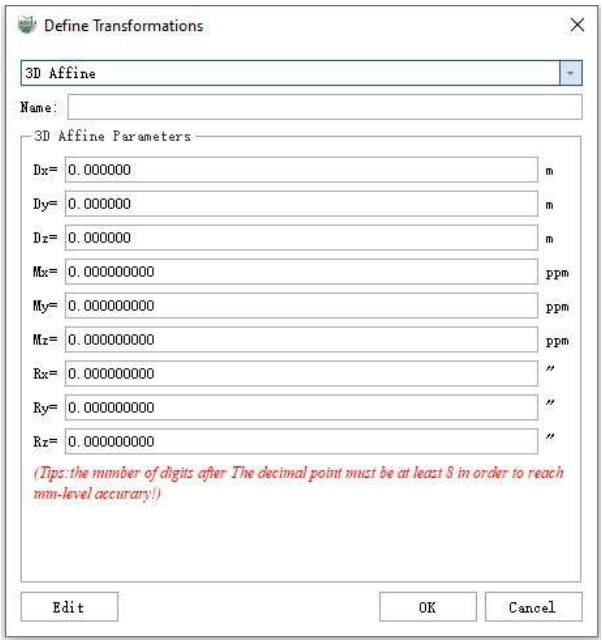
$$\begin{bmatrix} X_B \\ Y_B \\ Z_B \end{bmatrix} = \begin{bmatrix} X_A \\ Y_A \\ Z_A \end{bmatrix} + \begin{bmatrix} 1 & 0 & 0 & X_A & 0 & -Z_A & Y_A \\ 0 & 1 & 0 & Y_A & Z_A & 0 & -X_A \\ 0 & 0 & 1 & Z_A & -Y_A & X_A & 0 \end{bmatrix} \begin{bmatrix} dx \\ dy \\ dz \\ m \\ rx \\ ry \\ rz \end{bmatrix}$$

The seven-parameter model can be converted into 3D affine transformation parameters, and the conversion relationship is:

> Dx = Dx;
>
> Dy = Dy;

> Dz = Dz;
>
> Rx(″) = Rx(″);
>
> Rx(″) = Rx(″);
>
> Rx(″) = Rx(″);
>
> Mx(ppm) = m(ppm);
>
> My(ppm) = m(ppm);
>
> Mz(ppm) = m(ppm);

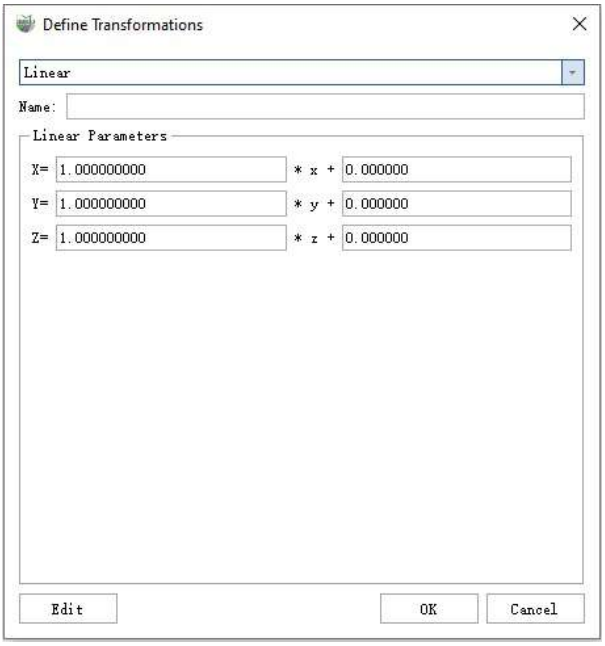
- 3D affine transformation:



- **Dx, Dy, Dz (m, default value is "0")**: Values (panning values) added to X, Y, Z.
- **Mx, My, Mz (ppm, default value is "0")**: Scale factors applied to X, Y, and Z. It has the same definition as the scale factor m in the seven parameters.
- **Rx, Ry, Rz (second, default value is "0")**: The angle of rotation about the X, Y, and Z axes in degrees.

  > Tips: the number of digits after The decimal point must be at least 8 in oder to reach mm-level accuracy.

- **Four parameters (displayed as two-dimensional transformation parameters)**: Two-dimensional transformation parameters, also known as XYMultiply.



Two-dimensional transformation parameters can be expressed in the following formula:

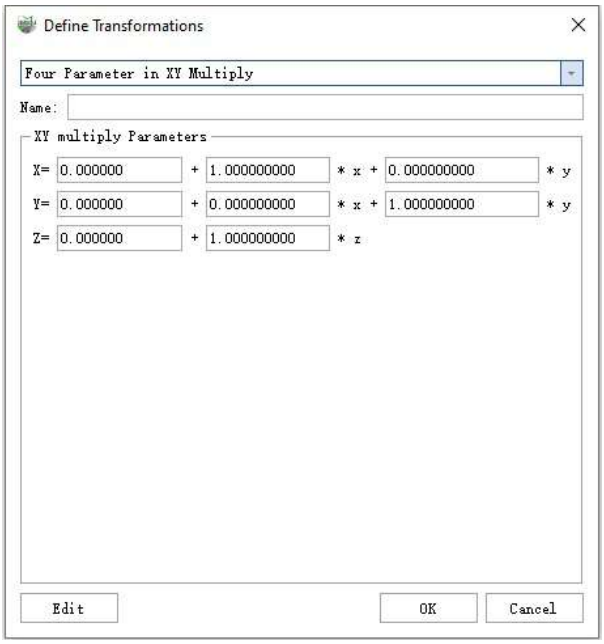
$$\begin{cases} X = P_x + a * S_x + b * S_y \\ Y = P_y + c * S_x + d * S_y \\ Z = P_z + e * S_z \end{cases}$$

Among them, Px, Py, Pz, a, b, c, d, e are the transformation parameters, Sx, Sy, Sz are the original coordinates, and X, Y, Z are the transformed coordinates. This is often used as a 2D Helmert transformation. The parameters can be converted from four parameters. Assuming that the four parameters are: Dx, Dy, theta, t, the conversion method is:

> Px = Dx;
>
> Px = Dy;
>
> Pz = 0.0;
>
> Define k = t;
>
> Define tRad = theta / 3600.0 / 180 * PI;
>
> Then:
>
> a = k * cos(tRad);
>
> b = -k * sin(tRad);
>
> c = k * sin(tRad);
>
> d = k * cos(tRad);
>
> e = 1.0;

- **Linear transformation**: The linear transformation is used to pan and zoom a point cloud. A panning

parameter and a scaling parameter need to be set to the X, Y, Z coordinates, respectively.

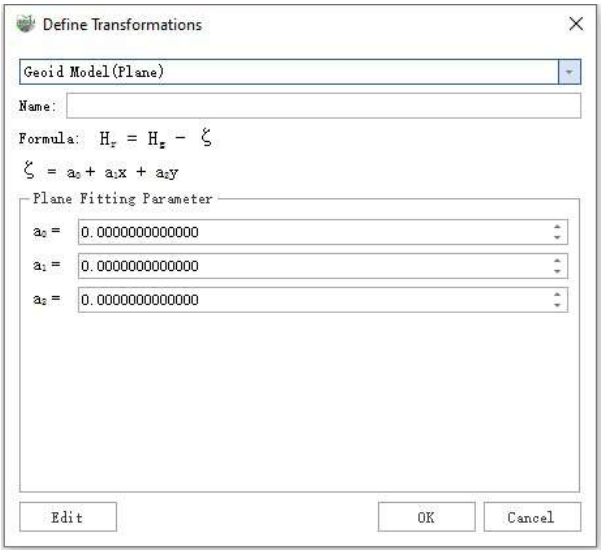


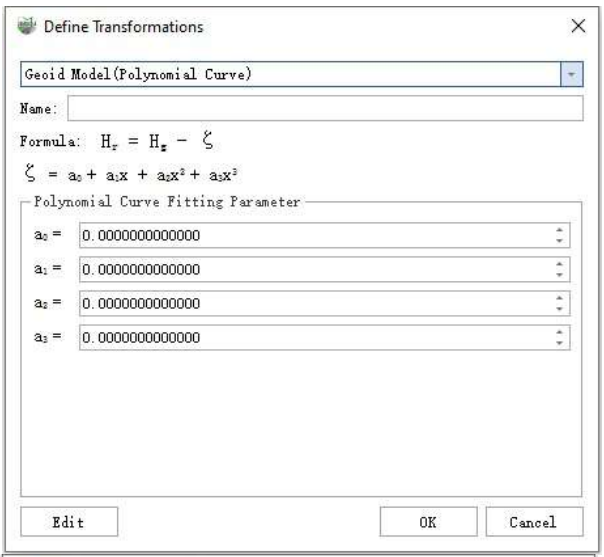
Linear transformation is calculated using the following formula:

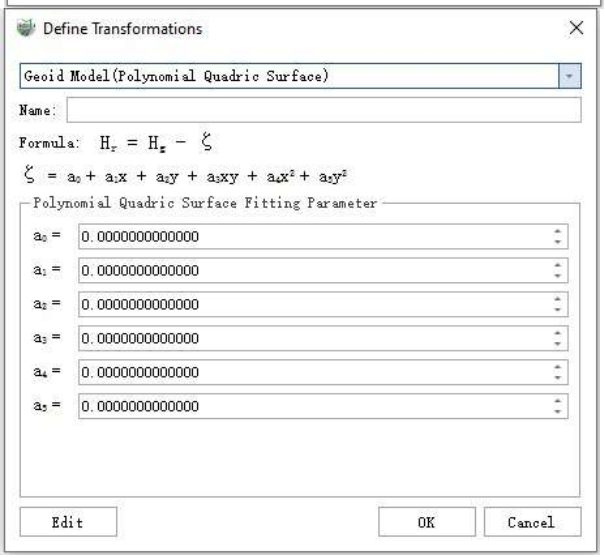
$$\begin{cases} X = S_x * x + P_x \\ Y = S_y * y + P_y \\ Z = S_z * z + P_z \end{cases}$$

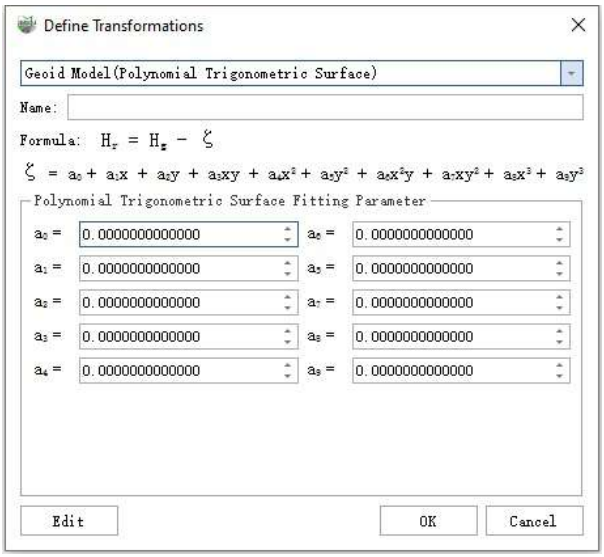
Among them: Sx, Sy. Sz are the scaling factors for the x, y, and z coordinates. Px, Py, and Pz are the panning parameters for the x, y, and z coordinates. x, y, and z are the original coordinates, and X, Y, and Z are coordinates obtained after the linear transformation.

- **Elevation fitting parameter**: It is used to calculate elevation anomalies when xy is known, and then correct Z.
  - Plane fitting. The plane fitting formula is $\zeta = a0 + a1x + a2y$; It contains three parameters a0-a2.
  - Linear interpolation. The linear interpolation formula is $\zeta = a0 + a1x + a2x^2 + a3x^3$; It contains four parameters a0-a3.
  - Conicoid Fitting. $\zeta = a0 + a1x + a2y + a3xy + a4x^2 + a5y^2$; It contains six parameters a0-a5.
  - The formula for the Fitting of the Cubic Bézier Curve and Surface is $\zeta = a0 + a1x + a2y + a3xy + a4x^2 + a5y^2 + a6x^2y + a7xy^2 + a8x^3 + a9y^3$; It contains ten parameters a0-a9.


Define Transformations
Geoid Model(Plane)
Name:
Formula: $H_r = H_z - \zeta$
$\zeta = a_0 + a_1x + a_2y$
Plane Fitting Parameter
a_0 = 0.0000000000000
a_1 = 0.0000000000000
a_2 = 0.0000000000000
Edit
OK
Cancel


Define Transformations
Geoid Model(Polynomial Curve)
Name:
Formula: $H_r = H_z - \zeta$
$\zeta = a_0 + a_1x + a_2x^2 + a_3x^3$
Polynomial Curve Fitting Parameter
a_0 = 0.0000000000000
a_1 = 0.0000000000000
a_2 = 0.0000000000000
a_3 = 0.0000000000000
Edit
OK
Cancel


Define Transformations
Geoid Model(Polynomial Quadric Surface)
Name:
Formula: $H_r = H_z - \zeta$
$\zeta = a_0 + a_1x + a_2y + a_3xy + a_4x^2 + a_5y^2$
Polynomial Quadric Surface Fitting Parameter
a_0 = 0.0000000000000
a_1 = 0.0000000000000
a_2 = 0.0000000000000
a_3 = 0.0000000000000
a_4 = 0.0000000000000
a_5 = 0.0000000000000
Edit
OK
Cancel


Define Transformations
Geoid Model(Polynomial Trigonometric Surface)
Name:
Formula: $H_r = H_z - \zeta$
$\zeta = a_0 + a_1x + a_2y + a_3xy + a_4x^2 + a_5y^2 + a_6x^2y + a_7xy^2 + a_8x^3 + a_9y^3$
Polynomial Trigonometric Surface Fitting Parameter
a_0 = 0.00000000000000
a_1 = 0.00000000000000
a_2 = 0.00000000000000
a_3 = 0.00000000000000
a_4 = 0.00000000000000
a_5 = 0.00000000000000
a_6 = 0.00000000000000
a_7 = 0.00000000000000
a_8 = 0.00000000000000
a_9 = 0.00000000000000
Edit
OK
Cancel

Extract

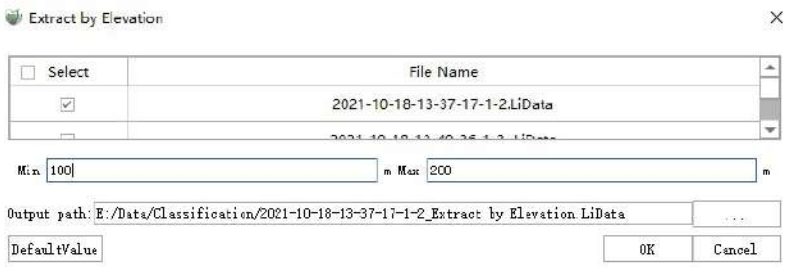
- Extract by Elevation
- Extract by Intensity
- Extract by Return
- Extract by Time
- Extract by Class
- Extract By Additional Attributes

Extract by Elevation

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

Steps

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



Parameter description:

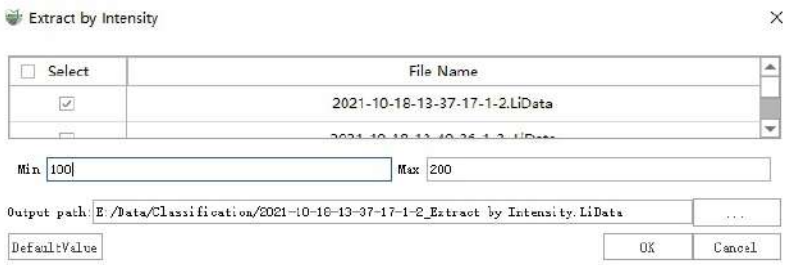
- **Input Data**: Input one or more point cloud data files. File Format: *.LiData.
- **Min (meters)**: The default is “100”. Users need to enter the minimum elevation value of the point cloud data to be extracted.
- **Max (meters)**: The default is “200”. Users need to enter the maximum elevation value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

Extract by Intensity

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

Steps

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



Parameter description:

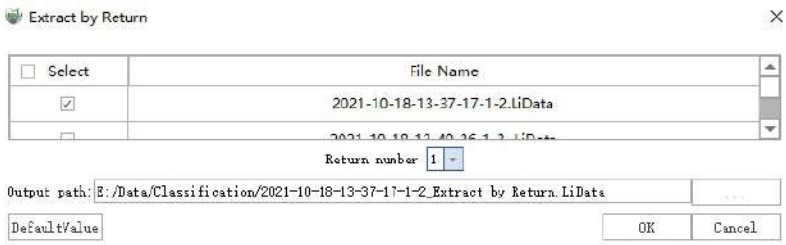
- **Input Data**: Input one or more point cloud data files. File Format: *.LiData.
- **Min (meters)**: The default is “100”. Users need to enter the minimum intensity value of the point cloud data to be extracted.
- **Max (meters)**: The default is “200”. Users need to enter the maximum intensity value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

Extract by Return

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

Steps

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



Parameter description:

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

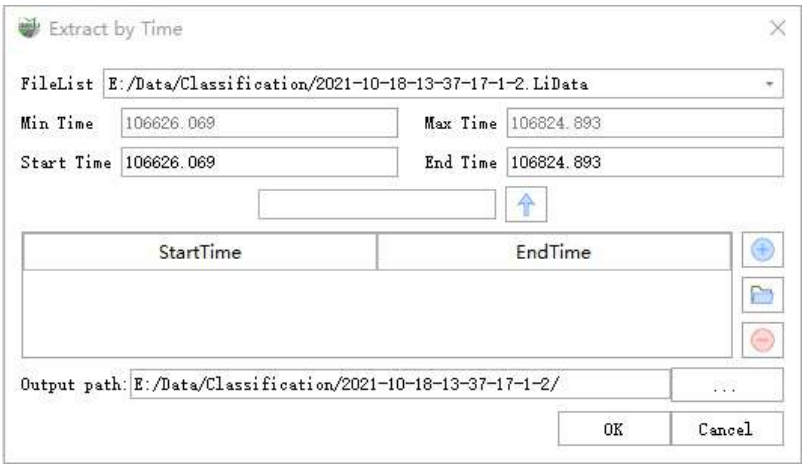
  > Note: IIf the data does not have the return number selected by the user, the extraction will not be executed.

Extract by Time

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

Steps

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



Parameter description:

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

Extract by Class

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

Steps

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

Parameter description:

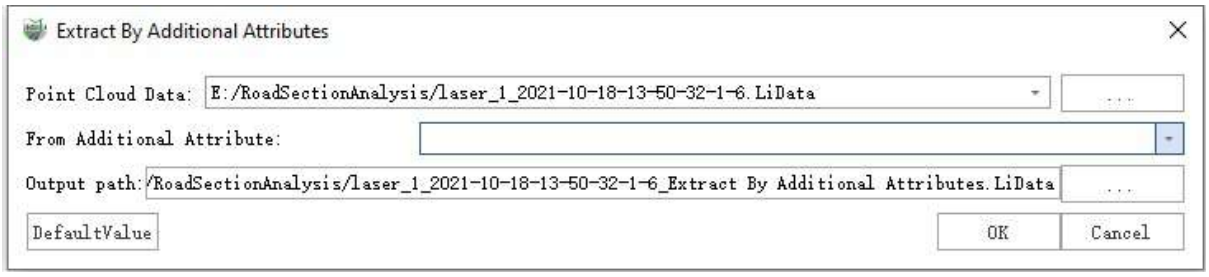
- **Input Data**: Input one or more point cloud data files. File Format: *.LiData.
- **From Class**: Users need to select the class to be extracted from the drop-down list. The unavailable status in the drop-down list represent the corresponding category that does not exist in the file.
- **Output path**: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

Extract By Additional Attributes

Feature Description：The Extract by Additional Attributes tool can extract point clouds based on the user-selected names, components, and extents of additional attributes. Additional properties support one or more components. Currently only the processing of a single point cloud is supported.

Steps

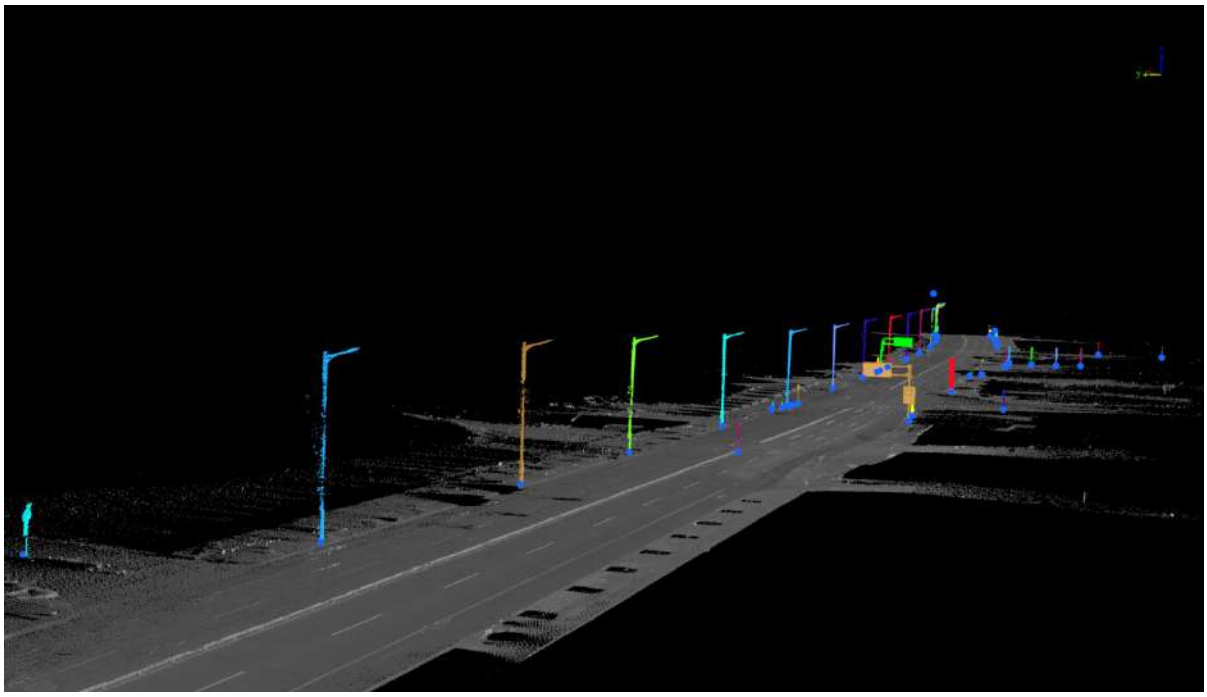
1.Click the **Extract By Additional Attributes** button to bring up the **Extract By Additional Attributes pop-up box**.



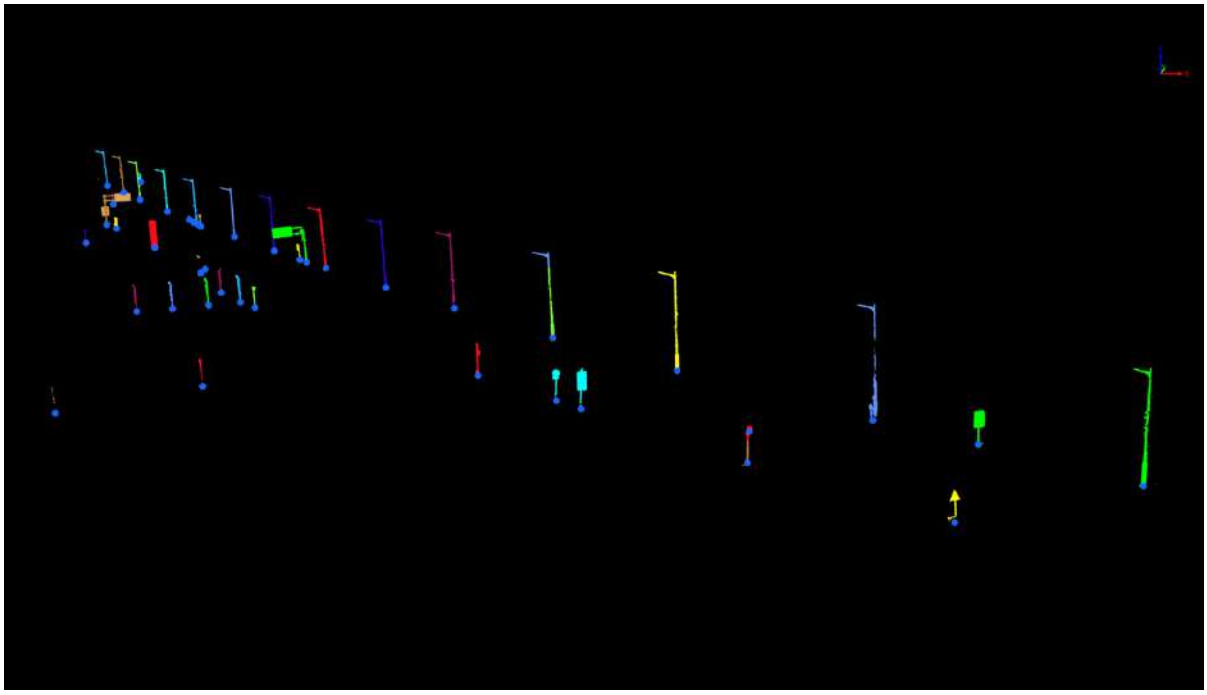
Description:

- **Point cloud Data**: Select the point cloud data file, currently only one point cloud data is supported. File format: *.LiData.
- **From Additional Attribute**: User needs to select the attached property name for filtering from the checkbox. This function cannot be run if the point cloud data does not contain additional attributes.
- **Output Path**: The path of the extracted output file after the function is executed.

2.The extracted additional attribute 'poleID' is displayed as follows:



Before extraction



After extraction

Classification

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

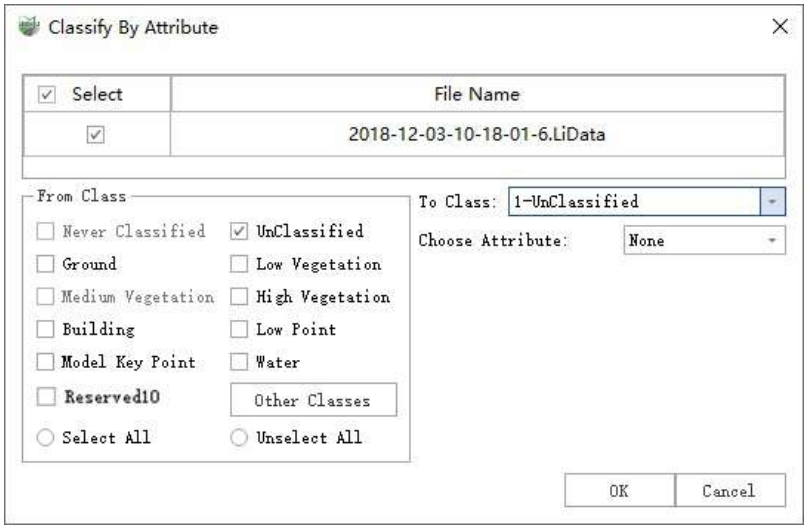
- Classify by Attribute
- Classify Ground Points
- Classify Air Points
- Classify by Coverage
- Classify by Cluster Size
- Classify Below Surface Points
- Classify by Height Above Ground
- Classify Low Points
- Classify Isolated Points
- Classify Closeby Points
- Classify by Polygons
- Classify by Polylines
- Cylinder Detection
- Classify by Region Grow
- Classify by Deeplearning
- Classify by Manual Editing
- Custom Deep Learning Classification

Classify by Attribute

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

Steps

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



Parameter description:

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

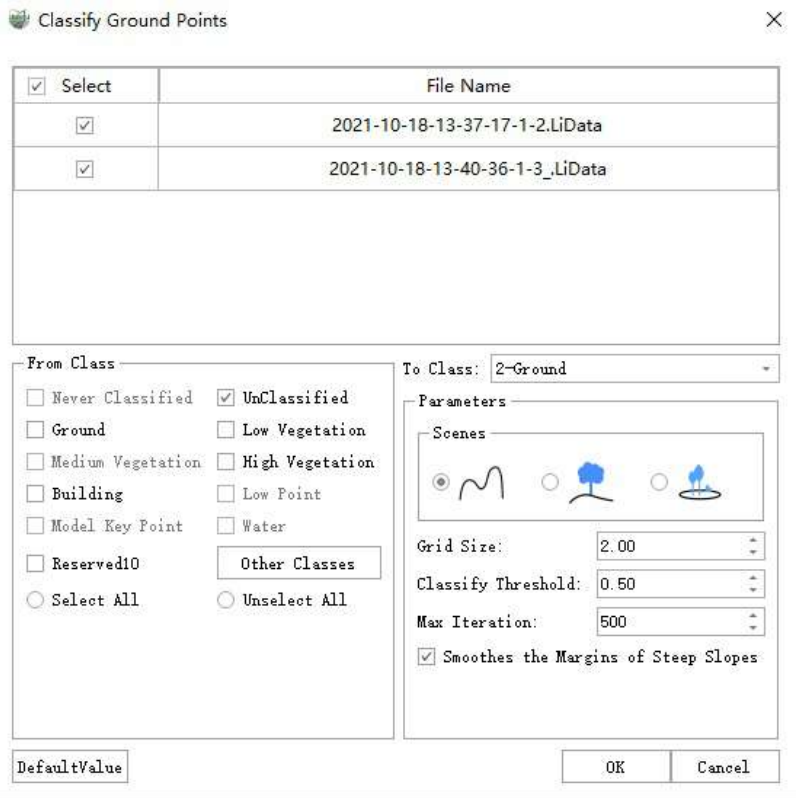
Classify Ground Points

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

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

Steps

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



Parameter description:

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

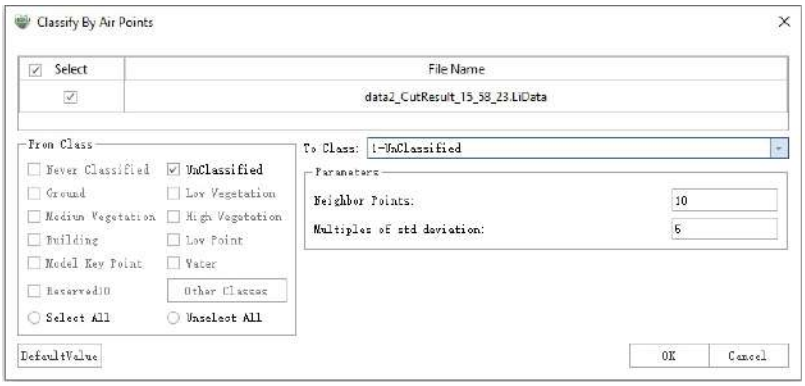
- **Scenes (Optional)**: For different terrain scenes, you can choose steep terrain, gentle terrain, and flat terrain.Check different terrain features to correspond to different default parameters
- **Grid Size (meters)**: The default is “1.0”, the resolution of the fabric node.1.0 is suitable for most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

Classify Air Points

Function Description: This function is to classify point clouds that are significantly higher than the surrounding points as airborne noise points. The algorithm idea is the same as the denoising principle in the data management module.

Steps

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



Parameter description:

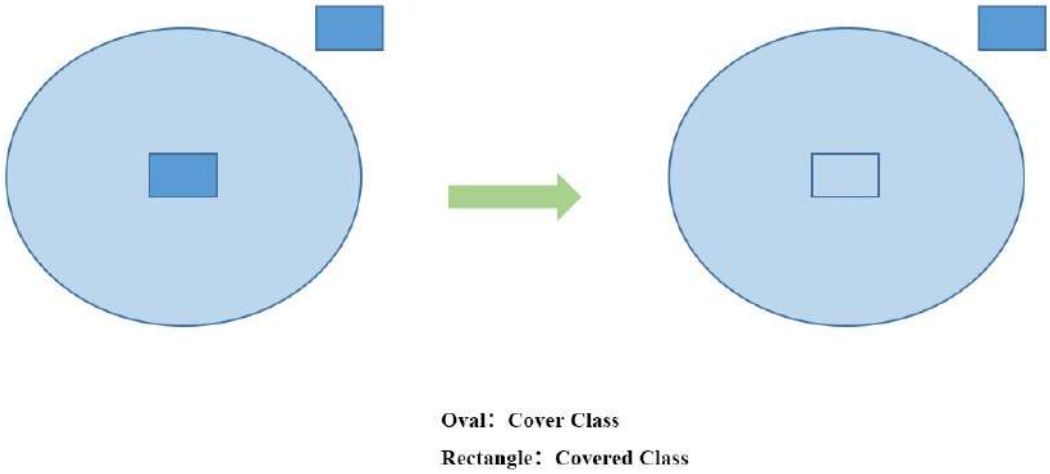
- **Input Data**: The input file can be a single point cloud data or a point cloud dataset.
- **From Class**: Source class(es).
- **To Class**: Target class.
- **Neighbor Points**: The default is “10”, the number of neighbors that will be used to determine whether a point is a noise in the sky.Calculate the distance between each point to the nearest point and calculate the standard deviation of the nearest distances.
- **Multiples of std deviation**: The default is “5”, if the deviation of points beyond the minimum allowable threshold,they are considered as noise in the sky. The larger the threshold,the less the noise will be divided into.
- **DefaultValue**: Click this button to set all parameters as default.

Classify by Cover

Function Description：Transform the category of the covered class into the covering class using geometric distribution of the covering and covered classes.

Principle

As shown in the diagram below, when the points of the covered class is located inside the cover class, category A of the covered class is converted to category B of the cover class.

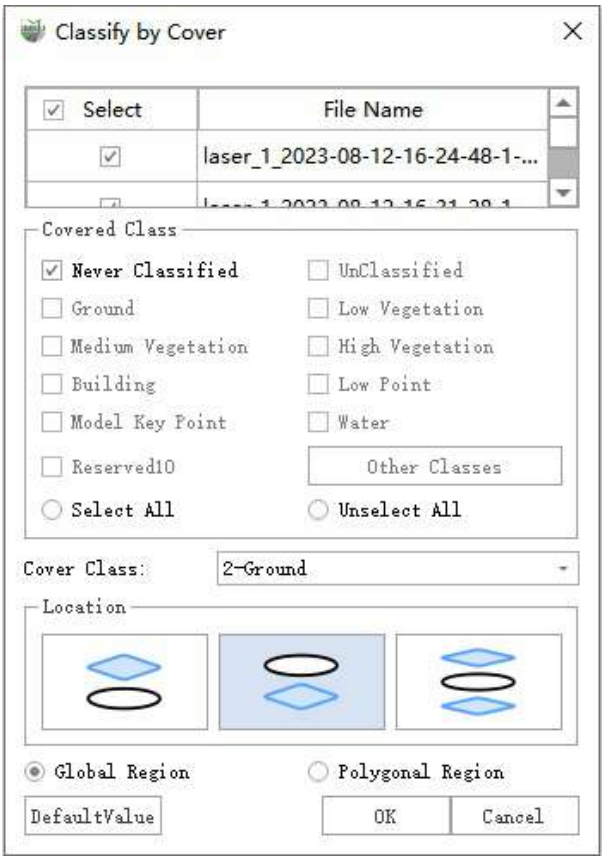


Principle Diagram

Steps

1.Click the **Classify by Cover** button in the Classification panel.

2.The **Classify by Cover** dialog will appear.

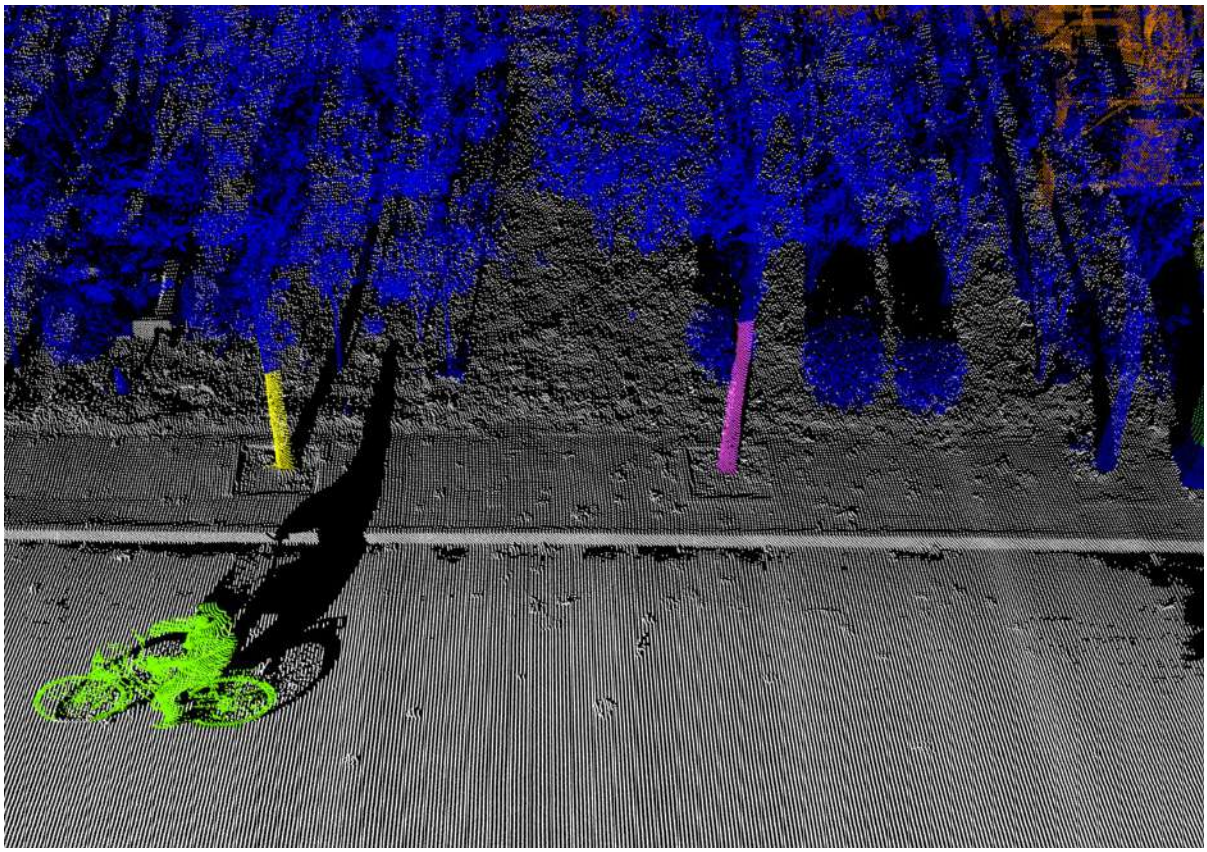


Classify by Cover Dialog

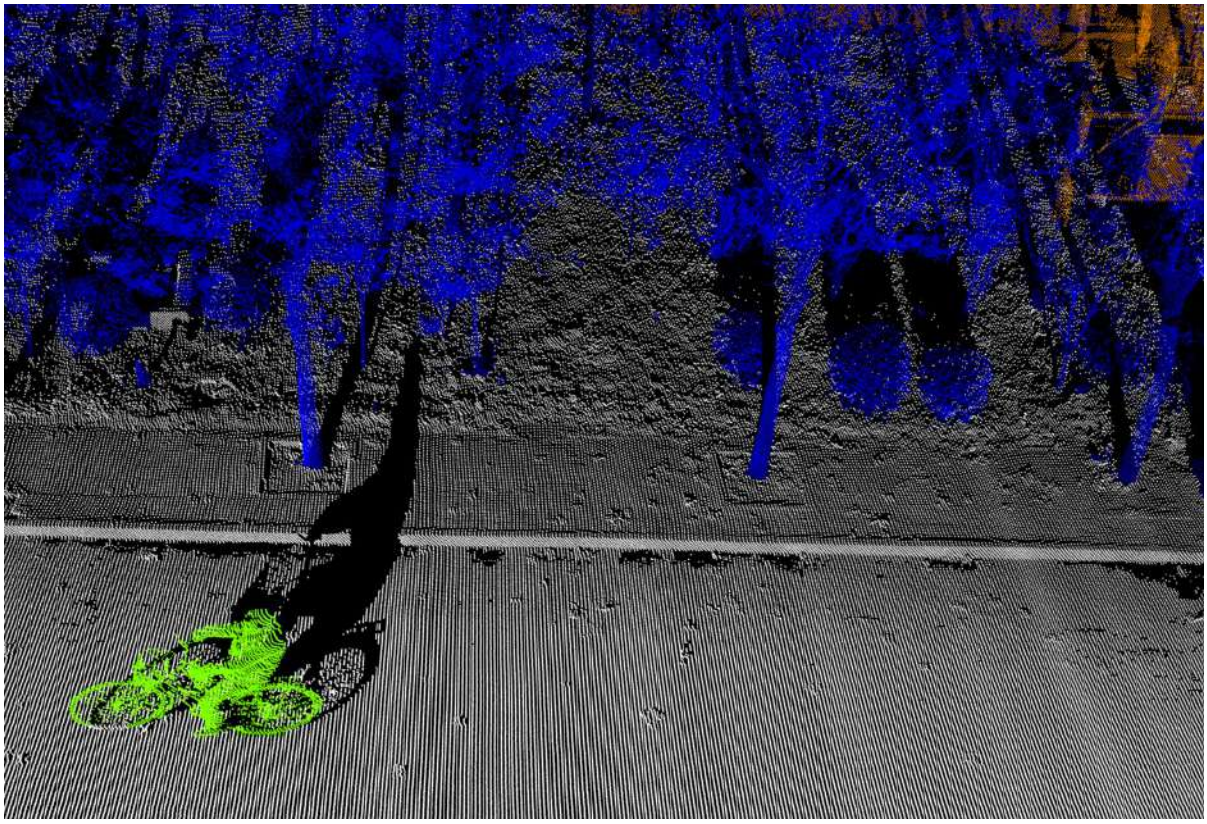
Parameters:

- **Select**: Select the point cloud files for classification.
- **Covered Class**: The class of the point sets to be converted.
- **Cover Class**: The point sets for cover calculation.
- **Location**: Indicate the relative positions of the covered class and the covering class.
  - **Above**: The covered class is located above the covering class.
  - **Below**: The covered class is located below the covering class.
  - **A11**: The covered class and the covering class overlap on the XY plane.
- **Global**: Default selected. Perform covering calculation between the covering class and the covered class in the entire point cloud file.
- **Select by Polygon**: Default not selected. Construct a polygonal region by sequentially clicking the left mouse button. Perform cover calculations for the point sets located within the region.

3.Result Display



Before processing, the cover class is shown in blue



After processing

Classify by Cluster Size

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

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

Parameter dialog box

Parameter dialog box

Parameter description:

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

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

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

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

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

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

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

  > Multiple polygons can be framed at once.

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

Steps

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

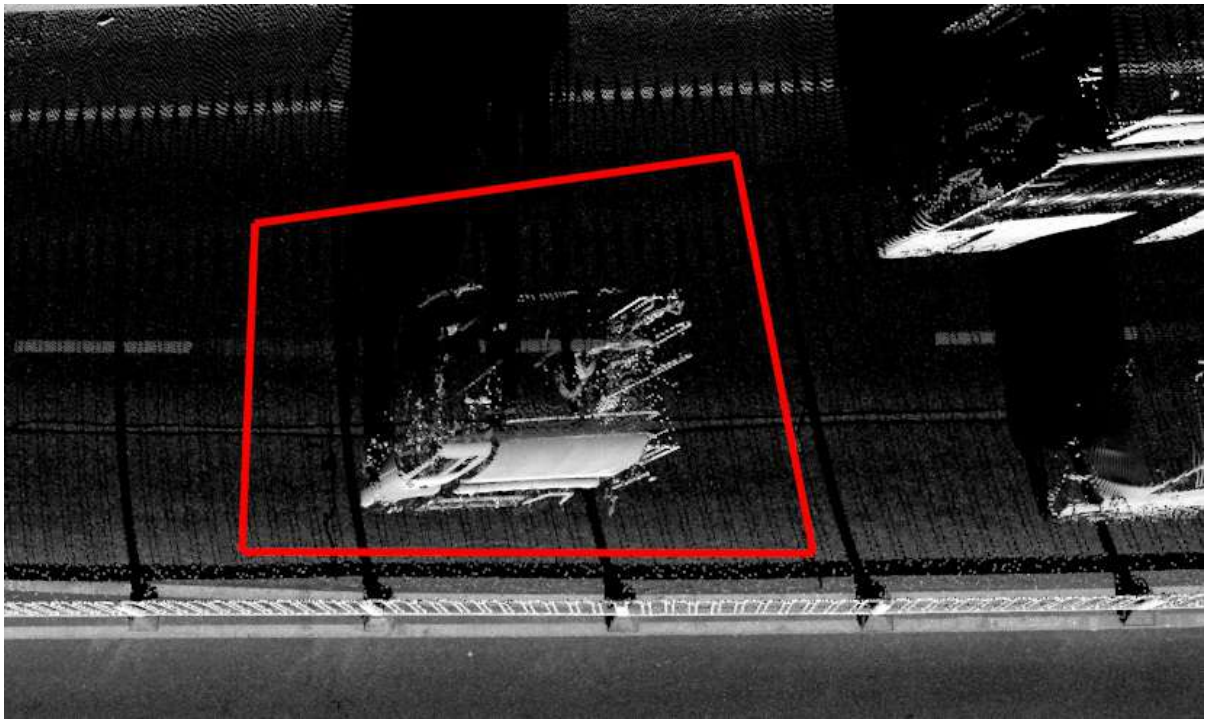
2.Pop up the **Parameter Settings dialog box**.

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

4.Select the operating mode and perform the calculation.

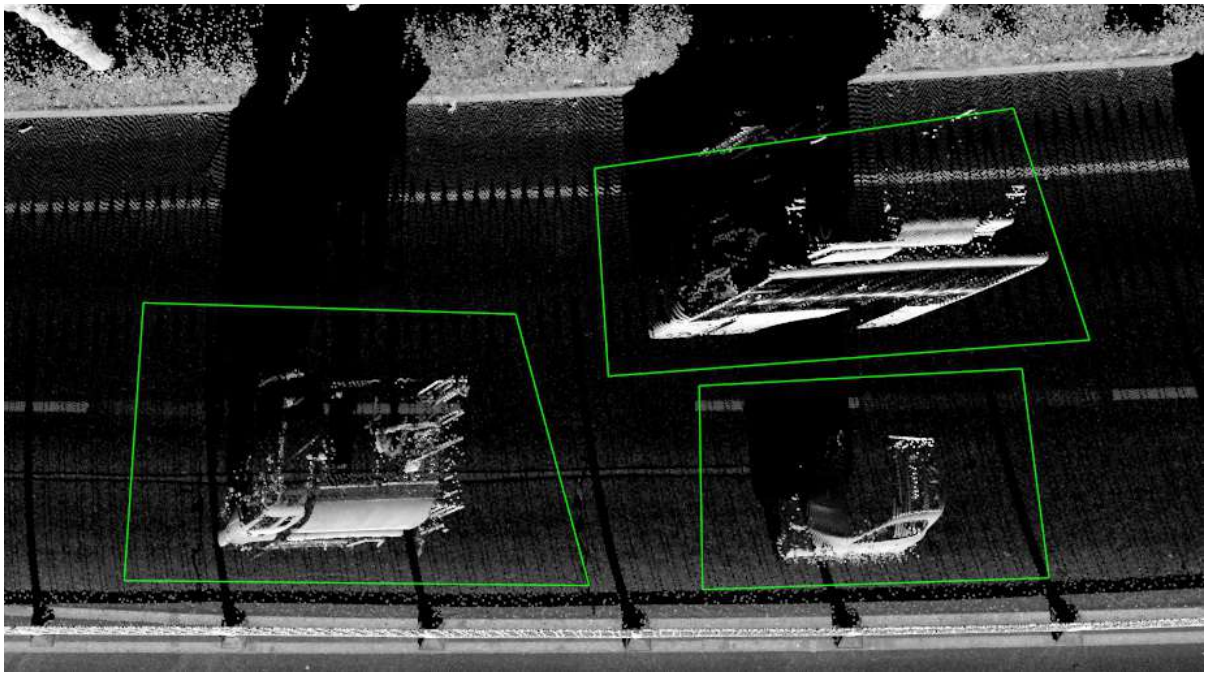
Select by Polygon

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



Select by Polygon

2.Repeat step 1 until the desired range.



Select by Polygon

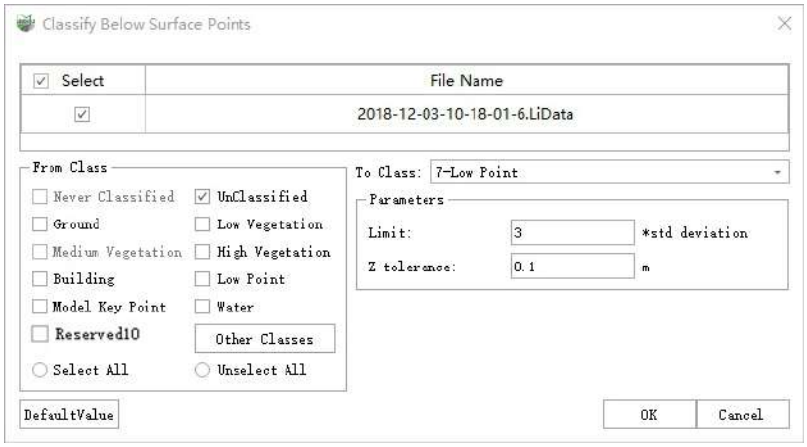
Classify Below Surface Points

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

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

Steps

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



Parameter description:

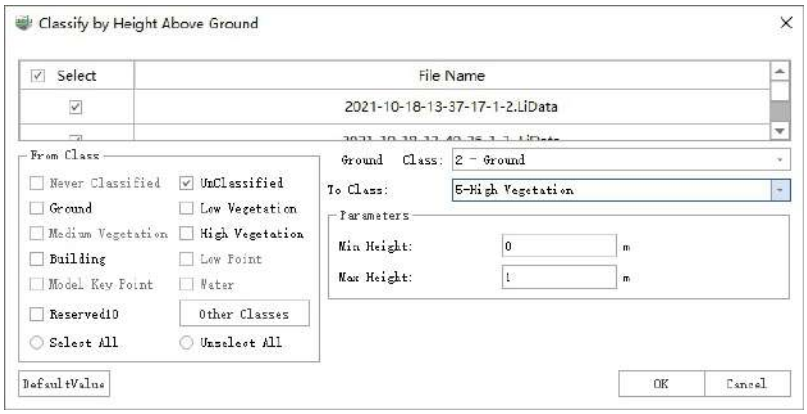
- **Input Data**: The input file can be a single point cloud data or a point cloud dataset.
- **From Class**: Source class(es).
- **To Class**: Target class.
- **Limit**: The default is “3".The multiple of the mean squared error of the neighboring point fitting plane of the unclassified points. The larger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

Classify by Height Above Ground

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

Steps

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



Parameter description:

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

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

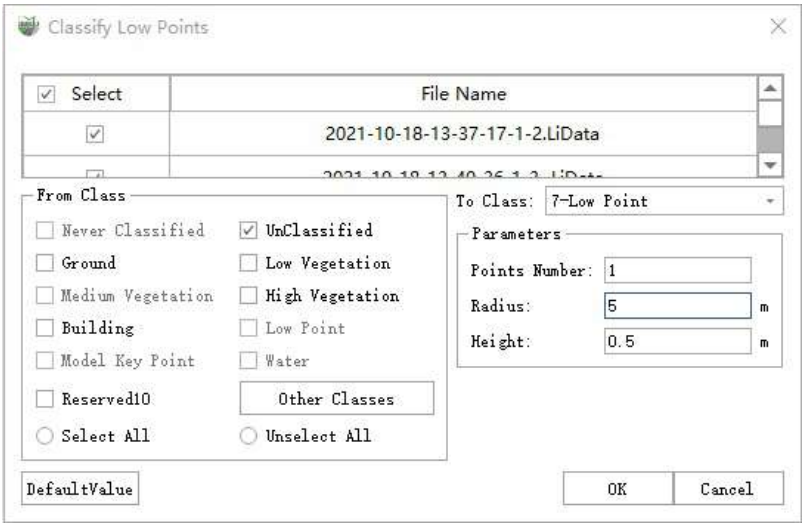
Classify Low Points

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

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

Steps

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



Parameter description:

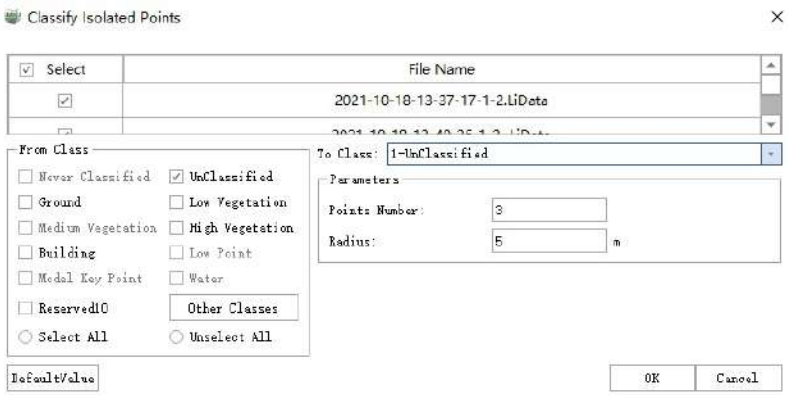
- **Input Data**: The input file can be a single point cloud data or a point cloud dataset.
- **From Class**: Source class(es).
- **To Class**: Target class.
- **Points Number**: The default is “1”. When the number of points is set to 1, the single low point is classified. If it is greater than 1, cluster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

Classify Isolated Points

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

Steps

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



Parameter description:

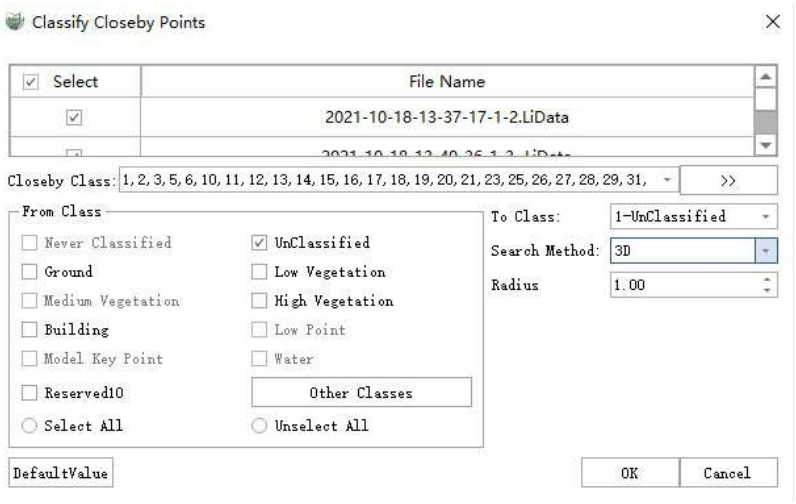
- **Input Data**: The input file can be a single point cloud data or a point cloud dataset.
- **From Class**: Source class(es).
- **To Class**: Target class.
- **Points Number**: The default is “3". If the number of points in the neighboring radius are less than or equal to the value, the point is considered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

Classify Closeby Points

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

Steps

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



Parameter description:

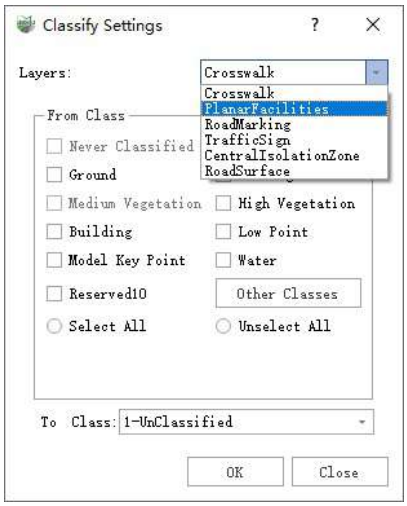
- **Input Data**: The input data could be one point cloud file or a set of point cloud files.
- **Closeby Class**: The points of this class which is close to the source points will be classified.
- **From Class**: The points close to this class will be classified.
- **To Class**: Target class of the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

Classify by Polygons

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

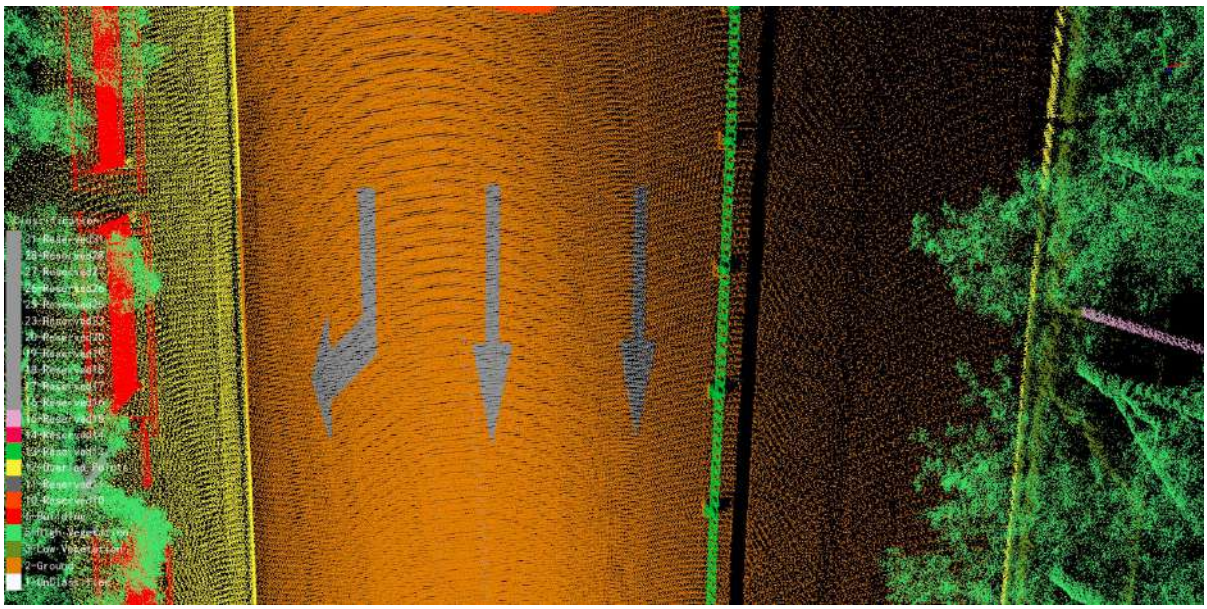
Steps

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

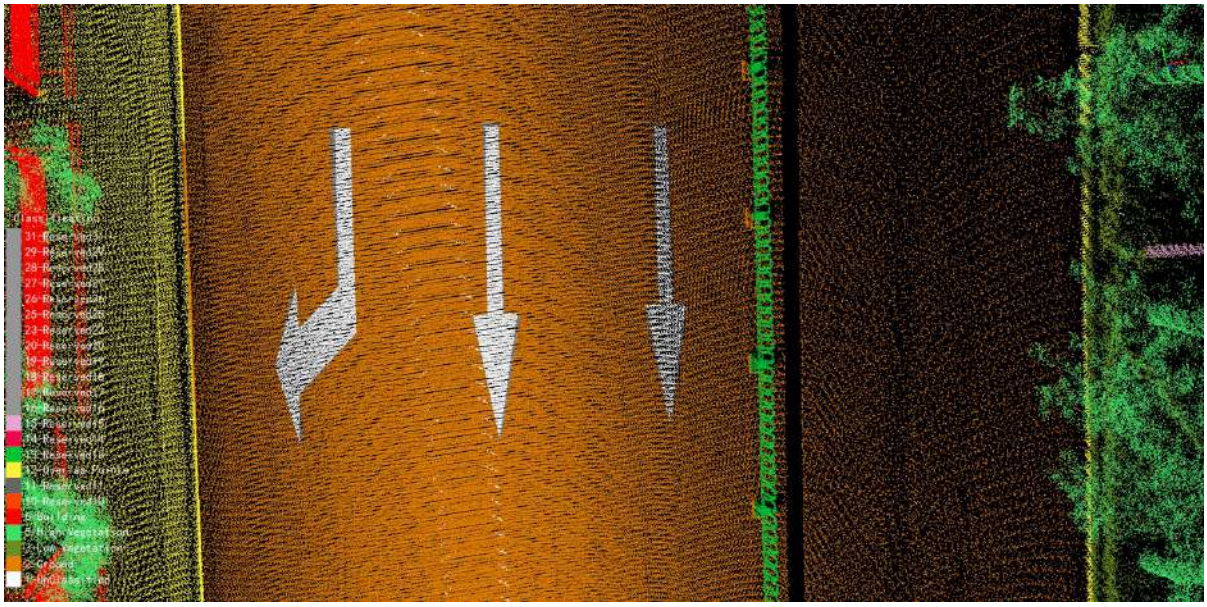


Parameter description:

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



Before Classify by Polygons



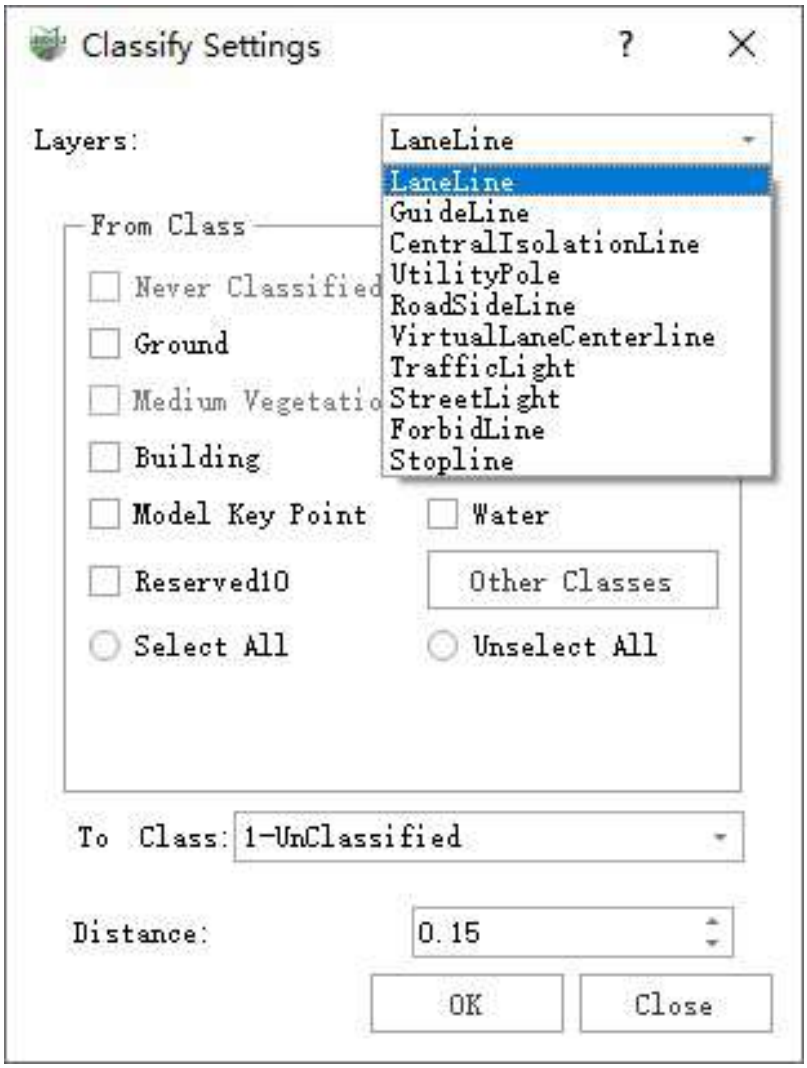
Effect diagram after Classify by Polygons

Classify by Polylines

Function Description: Classify point clouds based on vector lines.

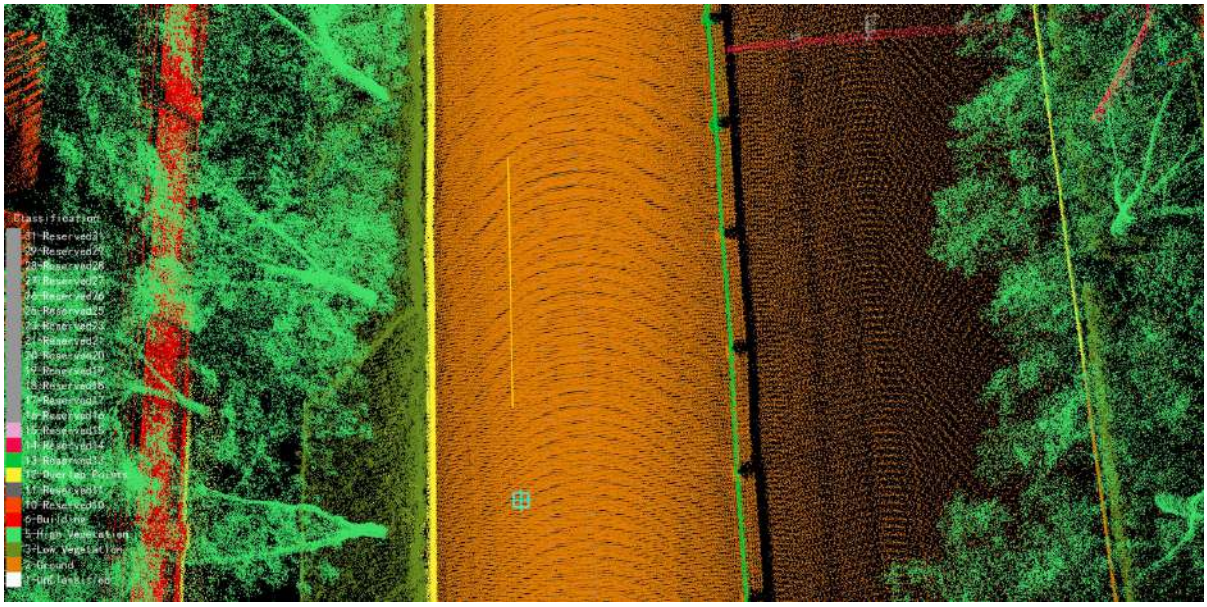
Steps

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

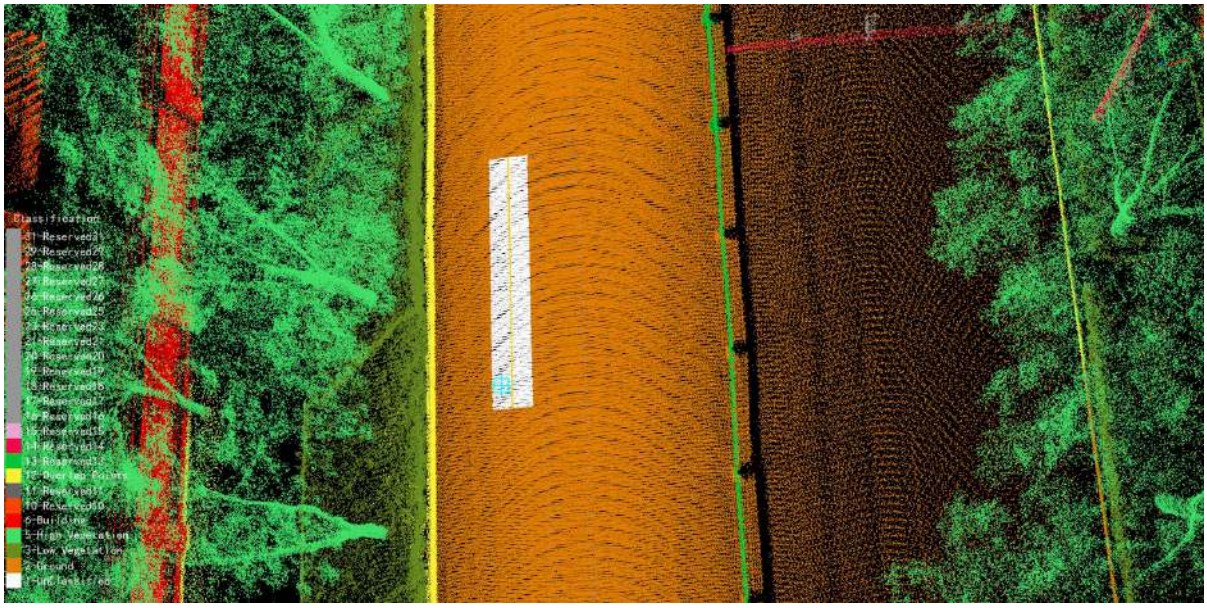


Parameter description:

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



Before Classify by Polylines



Effect diagram after Classify by Polylines

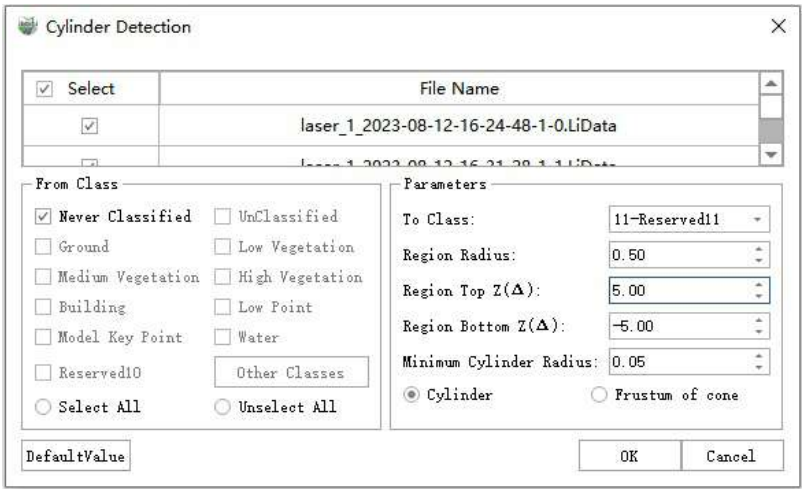
Cylinder Detection

Feature Description: Select seed points within the three-dimensional cylindrical range, perform cylinder detection, set the category of the points within the cylinder as the target category, and write it into the additional attribute GroupID.

Steps

1.Click on the **Cylinder Detection** button.

2.The **Cylinder Detection Dialog Box** pops up.



Cylinder Detection Dialog Box

Parameter Description:

- **Selection**: Select the point cloud file to participate in the detection.
- **Source Category**: Set the point category for detection.
- **Parameters**:
  - **To Class**: The category to which the detected set of points inside the cylinder should be written.
  - **Region Radius**: Used in conjunction with seed points, perform cylinder detection within a three-dimensional cylindrical range formed with the seed point as the center and the region radius as the radius.

> This parameter should be set based on the size of the cylinder to be detected, which should be equal to or greater than the size of the resulting cylinder.

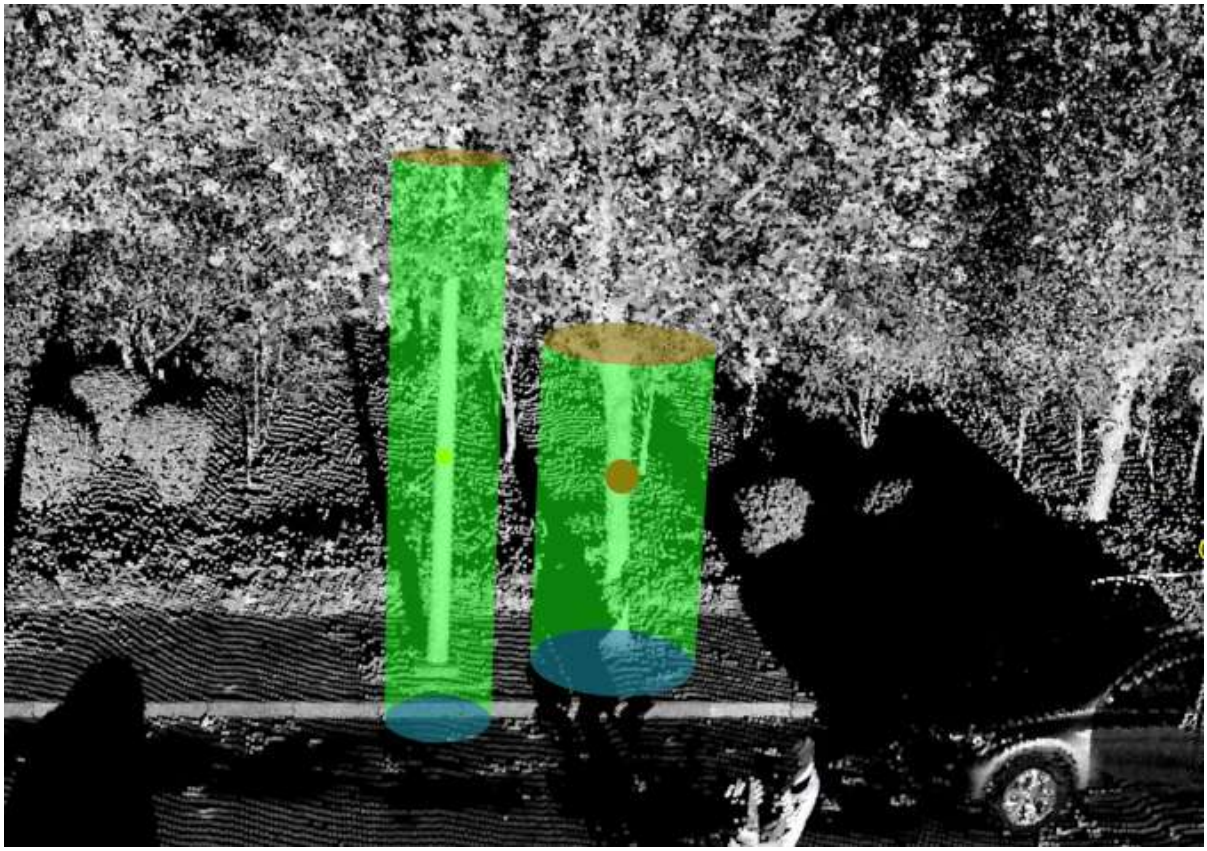
  - **Region Top Z(Δ)**: The relative position of the upper surface of the three-dimensional cylindrical range to the seed point. Please note that there are positive and negative values, where positive values indicate being located above the seed point, and negative values indicate being located below the seed point.
  - **Region Bottom Z(Δ)**: The relative position of the lower surface of the three-dimensional cylindrical range to the seed point. Please note that there are positive and negative values, where positive values indicate being located above the seed point, and negative values indicate being located below the seed point.

- **Minimum Cylinder Radius**：If the detected cylinder's radius is smaller than the set radius, discard the cylinder.

> This parameter is used to filter out small cylinders.

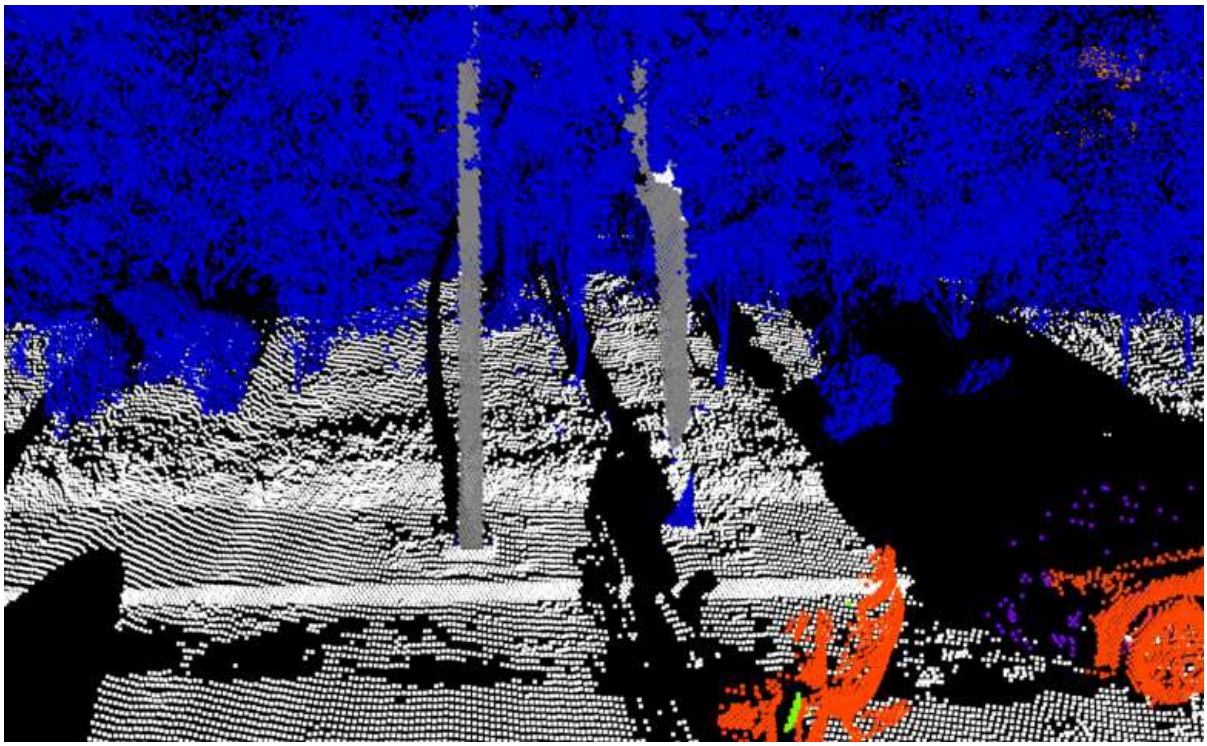
- **Cylinder, Frustum of cone**：Cylinder representation involves fitting based on a cylindrical shape, while frustum representation involves fitting based on a frustum (a truncated cone) shape.

> This parameter is used to avoid overfitting.



3.Use the **left mouse button** to **click** in the point cloud to select seed points. Multiple seed points can be selected.

> Use the **right mouse button** to go back one seed point.

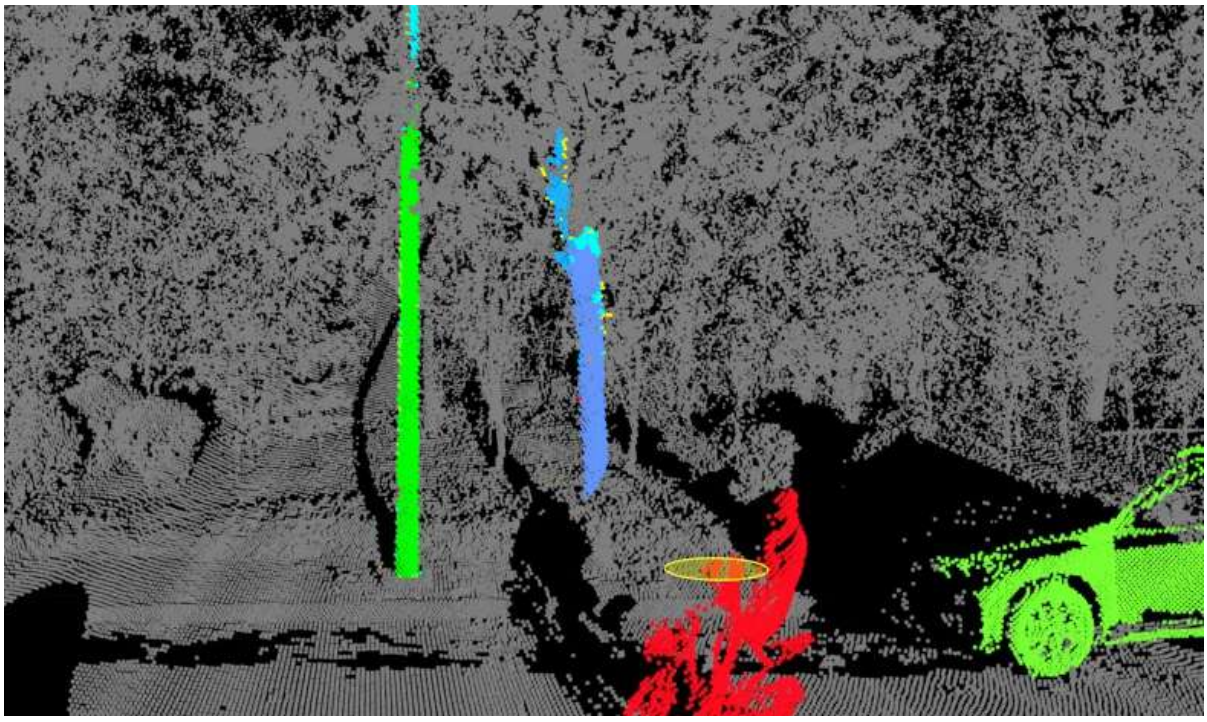


Seed Point Selection

> Each three-dimensional cylinder can be adjusted in size.
>
> The seed point represents a larger-sized region, indicating the currently adjustable size for the three-dimensional cylinder.

4.Click **Confirm** to start cylinder detection.

> The cylinder results are stored in the additional attributes with the name GroupID. After the detection ends, the cyclic additional attribute display is automatically activated.



Result Display

Classify by Region Grow

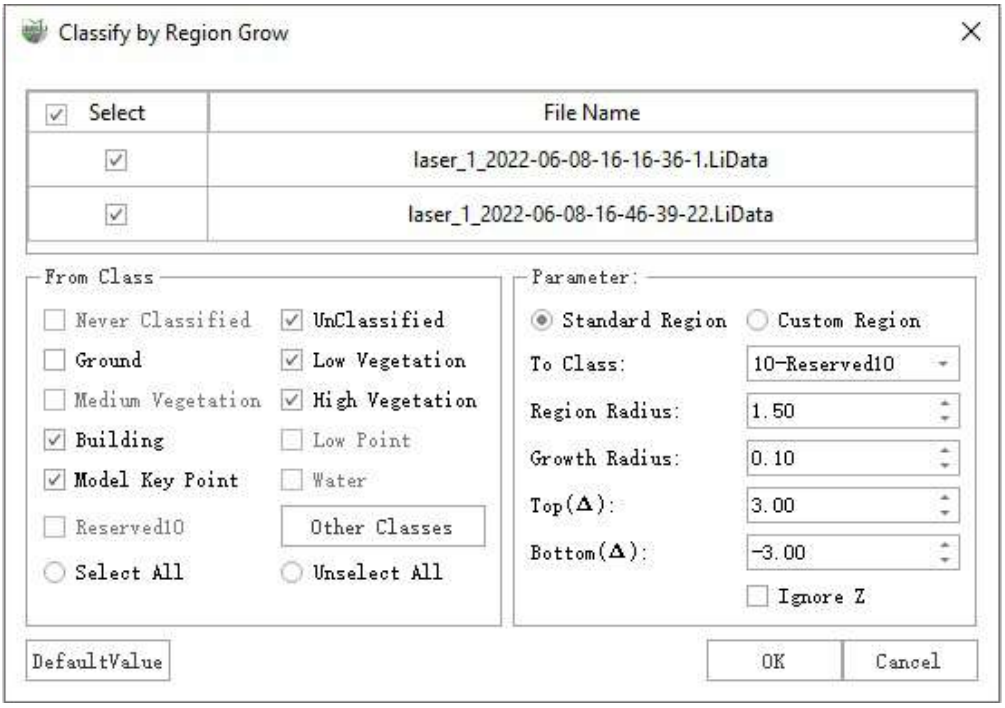
Function Description：Select seed points and perform region growing within the vicinity of the seed points, setting the category of the resulting point set to the specified category and writing it into the additional attribute GroupID.

> It is strongly recommended to perform ground filtering in advance to remove the majority of ground points.

Steps

1.Click the **Classify by Region Grow** button in the Classification panel.

2.The **Classify by Region Grow** dialog will appear.



Classify by Region Grow Dialog

Parameters：

- **Select**：Select the point cloud files for classification.
- **Source Category**：Set the point category for detection.
- **Parameters**：
  - **To Class**：The category to which the region growing results should be written.
  - **Region Radius**：Combined with seed points, perform calculations within a three-dimensional cylindrical range with the seed point as the center and the region radius as the radius.

    > Adjust based on the size of the object to be grown to prevent it from being too large or too small.

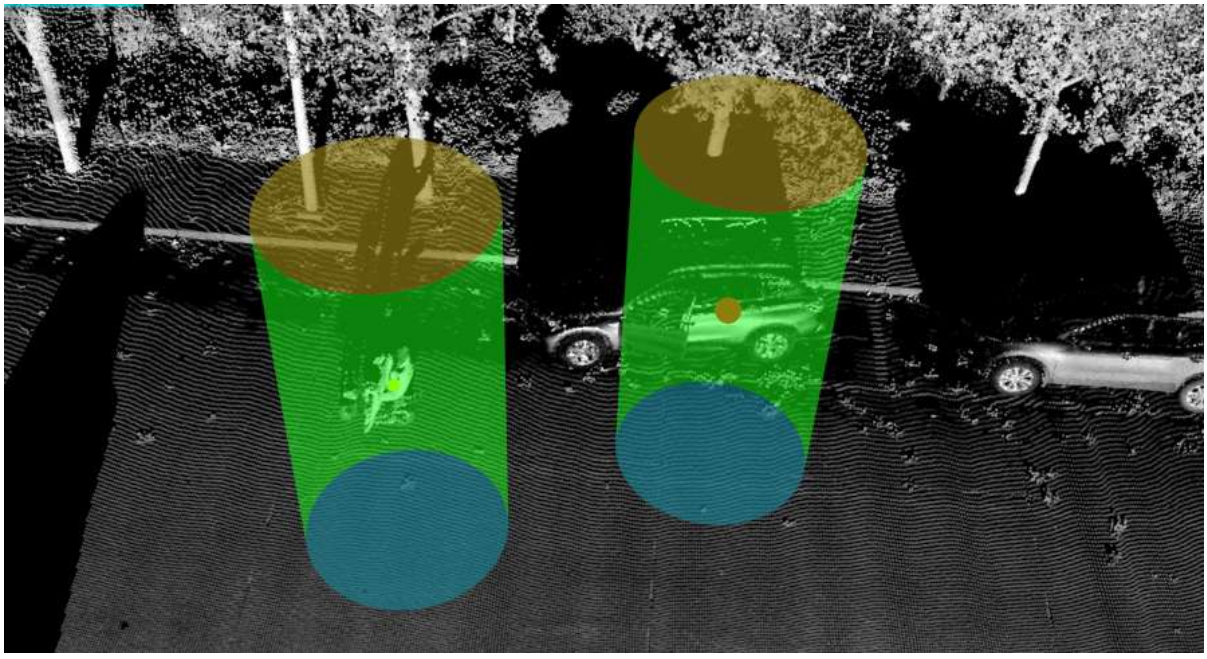
  - **Growth radius**: The minimum distance for region growing, points separated by more than this set distance will not undergo growth.

- **Above (Δ)**: The relative height difference between the upper base of the three-dimensional cylindrical range and the seed point. Note that there are positive and negative values, where positive values indicate being located above the seed point, and negative values indicate being located below the seed point.
- **Below (Δ)**: The relative height difference between the lower base of the three-dimensional cylindrical range and the seed point. Note that there are positive and negative values, where positive values indicate being located above the seed point, and negative values indicate being located below the seed point.
- **Ignore Z**: Checking this option indicates that the upper and lower bases of the three-dimensional cylinder have infinite height, rendering the Above (Δ) and Below (Δ) parameters non-configurable.

> When the 'Ignore Z' option is selected, the Above (Δ) and Below (Δ) parameters cannot be set.

3.Use the **left mouse button** to **click** in the point cloud to select seed points. Multiple seed points can be selected.

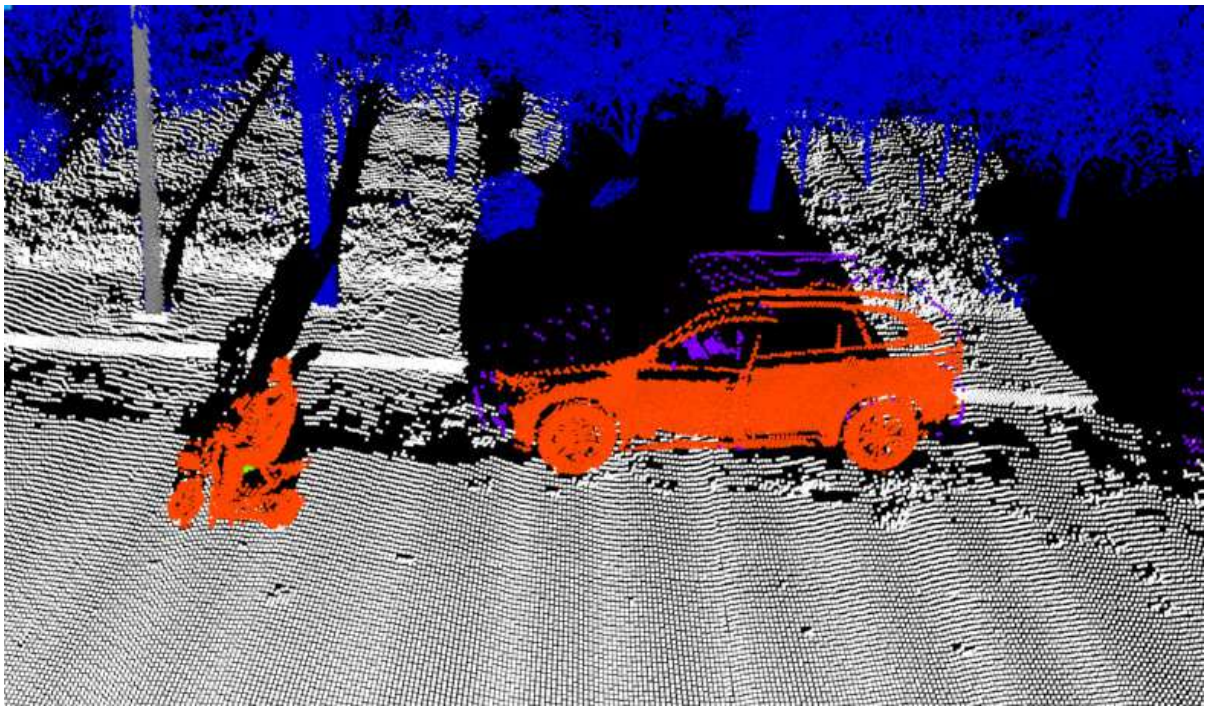
> Use the **right mouse button** to go back one seed point.



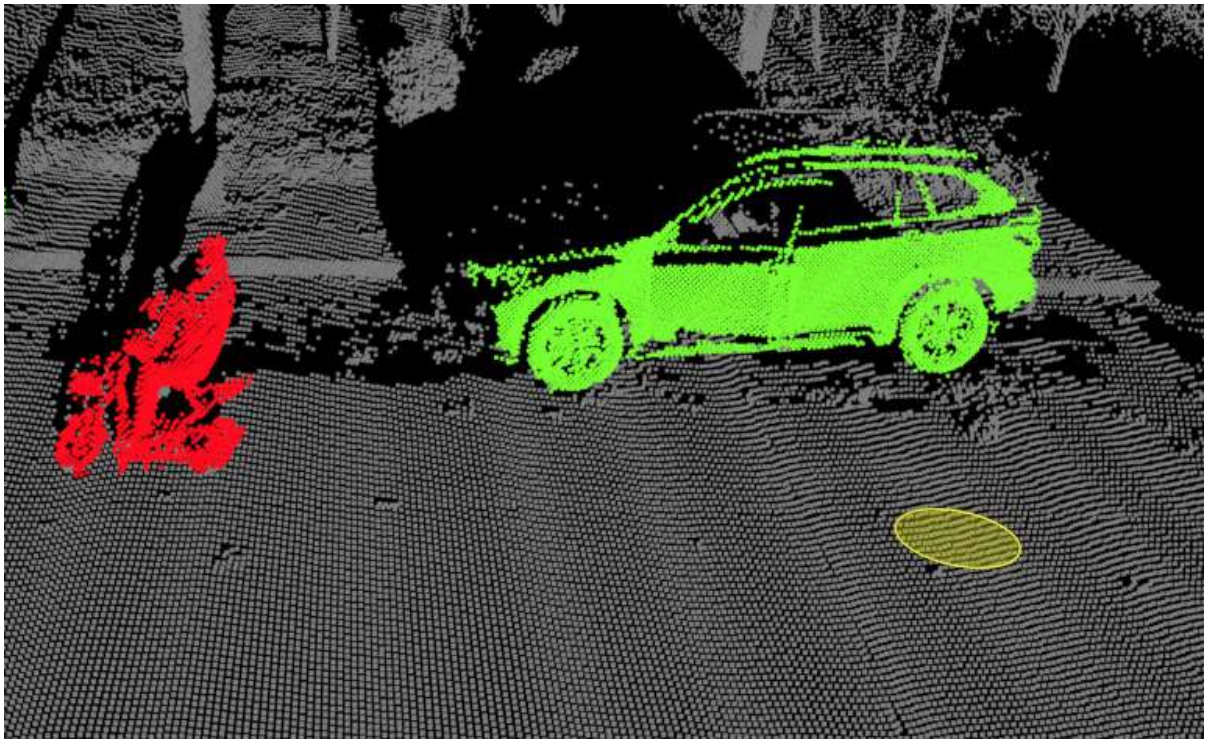
Seed point selection

> Each three-dimensional cylinder can be adjusted in size.
>
> The seed point represents a larger-sized region, indicating the currently adjustable size for the three-dimensional cylinder.

4.Click 'OK' to start the growth.



Displaying the classification results



Displaying the GroupID results

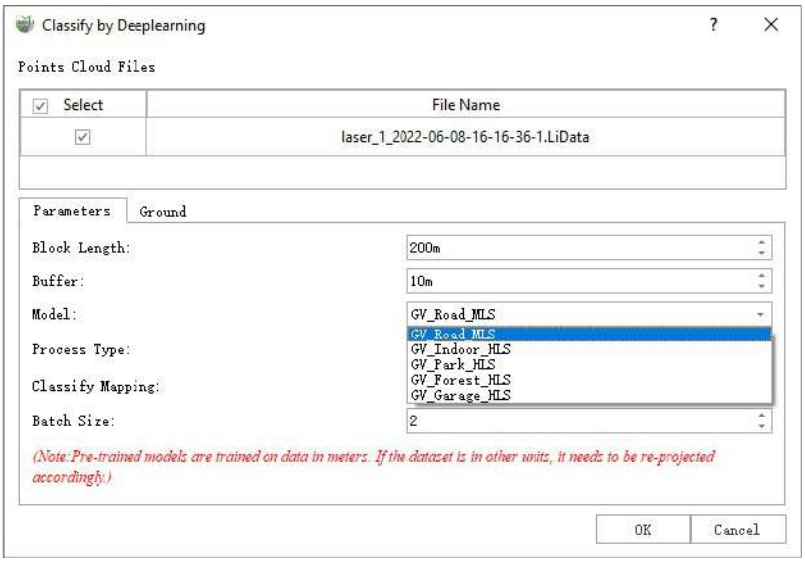
> In the above image, the GroupID results show that each individual grown from a seed point has a different GroupID

Classify by Deeplearning

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

Steps

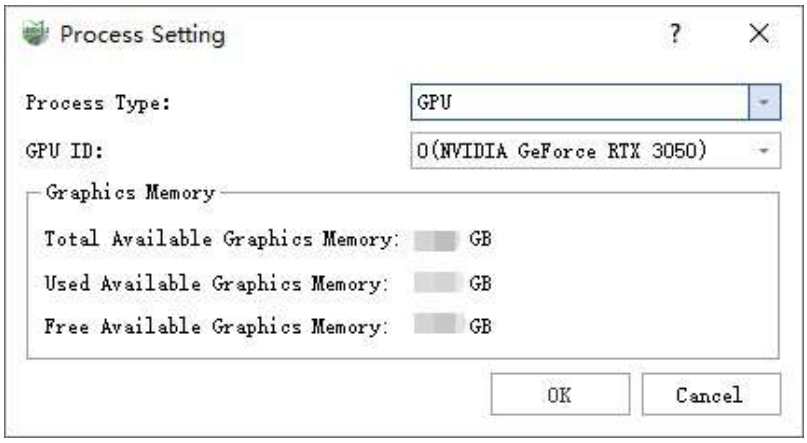
1.Click **Classify by Deeplearning** Button, pop up the **Classify by Deeplearning dialog box**



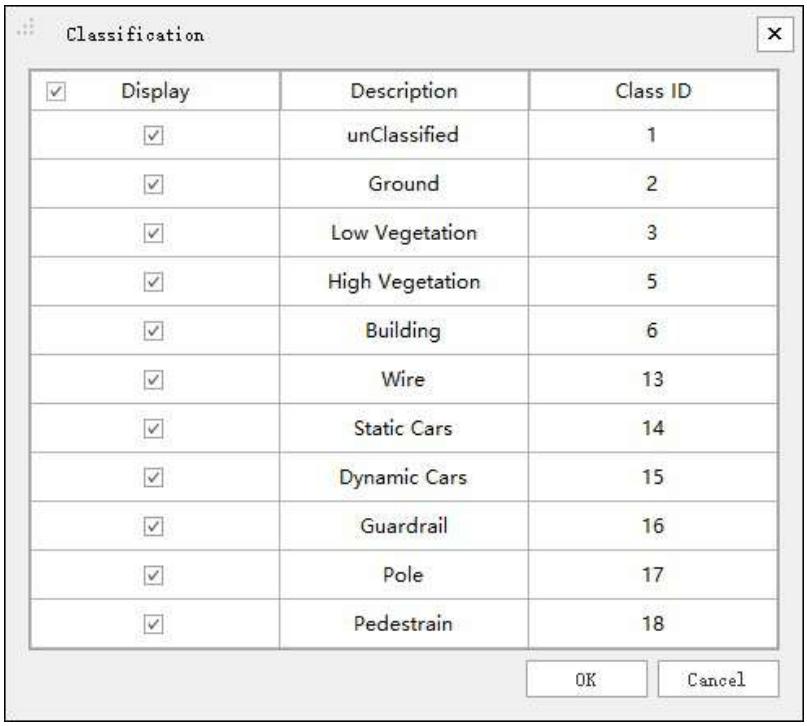
> Note: Pre-trained models are trained on data in meters. If the dataset is in other units, it needs to be re-projected accordingly.

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 five models by default:
  - **GV_Road_MLS** model is a dataset trained using a larger urban road scene and supports unClassified, Ground, Low Vegetation, High Vegetation, Building, Wire, Static Cars, Dynamic Cars, Guardrail, Pole, Pedestrain, a total of 11 categories.
  - **GV_Indoor_HLS** model for SLAM data indoor scene classification.
  - **GV_Park_HLS** model for SLAM data park scene classification.
  - **GV_Forest_HLS** model is suitable for forestry scene data.
  - **GV_Garage_HLS** model is suitable for indoor parking lot scene data. Point clouds with similar scenes can use this model.
  - **GV_Railway** model is suitable for classifying railway scenes.
- **Process Type**: There are two processing modes: GPU and CPU, the default is CPU mode, if the computer graphics card performance is high, you can switch to GPU mode on your own, GPU efficiency will be about twice as high as the CPU, and if the computer has more than one graphics card, you can choose the right graphics card on your own, here it is highly recommended to use NVIDIA graphics card for GPU mode classification.

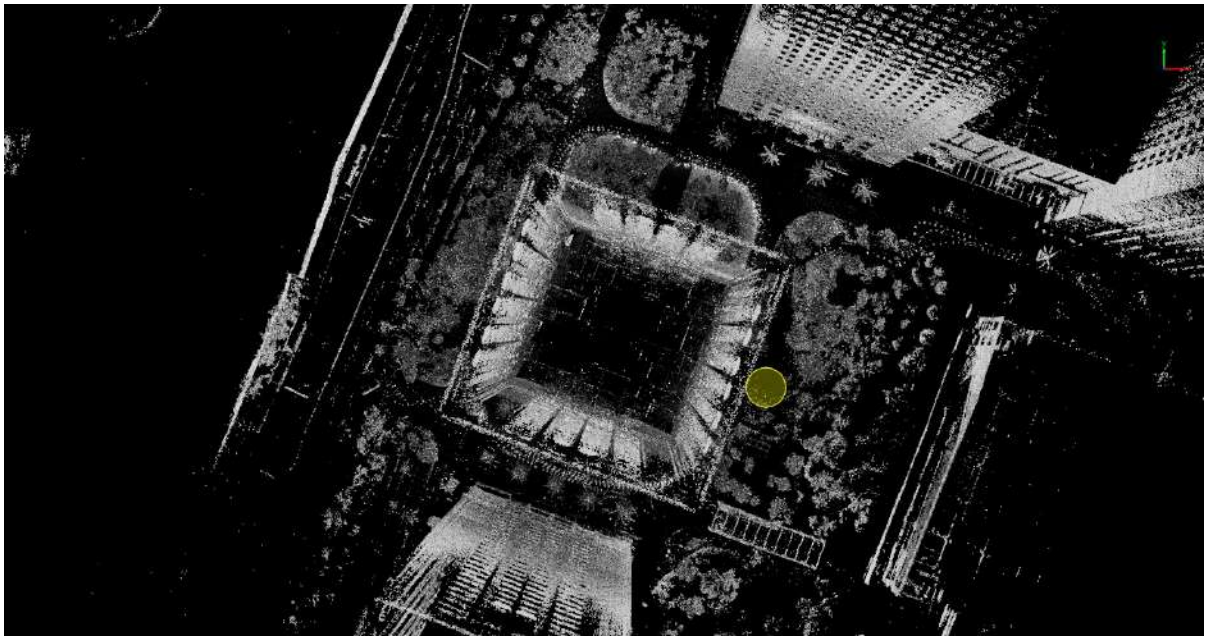


- **Classify Mapping**: Map the category and map the category to the corresponding category label.

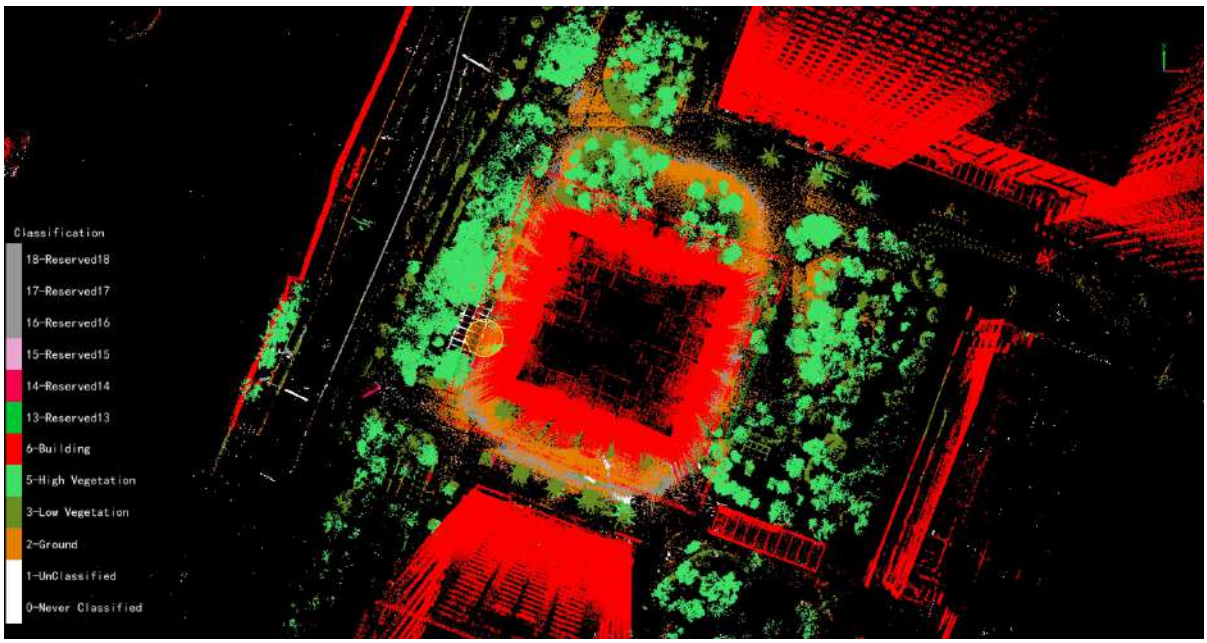


Here you can set your own categories and labels according to user needs, such as: The corresponding class ID for unClassified is 1.

- **Batch Size**: Indicates the number of point cloud samples processed each time, the higher the computer memory and display memory, the larger the set value, and the faster the overall processing speed. Software already set default value is set according to the performance of the computer. It is recommended that the user change the value by himself not more than 8.
- **Ground Optimization**: Default checked. During the deep learning classification process, ground optimization is performed to improve the classification accuracy of ground points. Refer to Classify Ground Points for specific parameter details.



Before Classify by Deeplearning



After Classify by Deeplearning

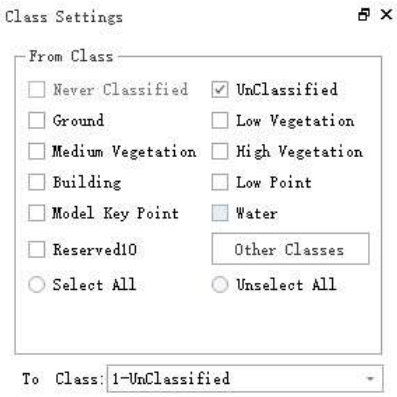
Classify by Manual Editing

Class Settings

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

Steps

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



Parameter description:

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

Classify Selection

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

Steps

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

Parameter description:

1. **Polygon**: Select a polygon area.
2. **Rectangle**: Select a rectangular area.
3. **Spherical**: Select a spherical area.
4. **circle**: Select a circle area.

5. **Online**: Select an online area.

6. **offline**: Select an offline area.

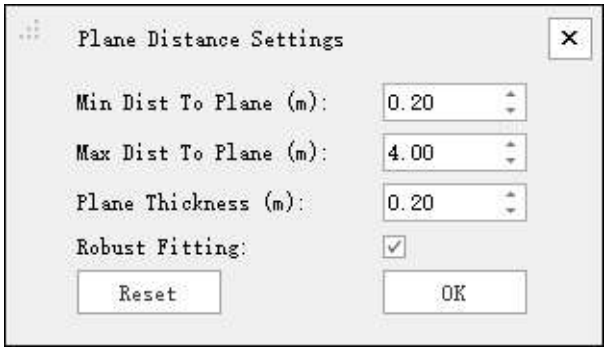
7. **Plane**: Select a plane area.

8. **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.

9. **Under the plane**: Select objects under the plane by selecting at least three plane points; it can select moving objects on the ground/road.

10. **Plane distance setting**: Set the selected area on the plane.

11.Click Plane Distance Settings to pop up the **Plane Distance Settings dialog box**.



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

12. **Unselect**: Reverse the selection in the area.

13. **cancel**: Deselect.

14. **Save**: Save the selected point cloud area.

15. **exit**: Exit editing mode.

Introduction of Custom Deep Learning Classification

The custom deep learning classification feature allows you to classify uncategorized point clouds using deep learning classification models. It also provides the capability to train your own deep-learning classification models. With this feature, you can manually edit a small amount of data within the same batch and use the edited results to train the model, which can then automatically process a large amount of data.

> The "Deep Learning Classification" feature already enables the classification of point clouds into specific categories such as ground points, buildings, vegetation, etc. The "Custom Deep Learning Classification" opens up numerous parameter settings and options, allowing you to classify point clouds into any desired categories. It includes both fully supervised and weakly supervised methods, where fully supervised requires complete annotation of the point cloud data for training, while weakly supervised only requires a small amount of annotated point cloud data, thus significantly improving production efficiency.

This feature supports two main workflows:

（1）**Training Sample Creation -> Model Training**

（2）Classification Using an Existing Model

- Install
- PointCloud Full Supervised Labeling and Weak Supervised Labeling Process
- Preparing Point Cloud Training Data
- Training Point Cloud Classification Model
- Classifying Point Cloud Using Trained Model
- Report

Install

Hardware Preparation

Configuration Requirements

- **CPU**：Recommended Intel® Core™ i5/i7; eight-core sixteen-thread processor, single-core processing performance of 4GHz and above.
- **RAM**：16GB and above.
- **Display Adapter**：Recommended independent NVIDIA graphics card, with no less than 8GB of video memory, and computing power greater than or equal to 5.0. The computing power of the graphics card can be queried on the official NVIDIA website.

System Requirements

Microsoft Windows 7 (64-bit), Microsoft Windows 8 (64-bit), and Microsoft Windows 10 (64-bit).

System Configuration

Before running the deep learning model, it is recommended to check your computer system and install the latest drivers for your graphics card.

Install Graphics Card Driver

The minimum required driver version is **465.89**. It is recommended to install the latest graphics card driver as much as possible, which can be downloaded on the NVIDIA Driver Download page.

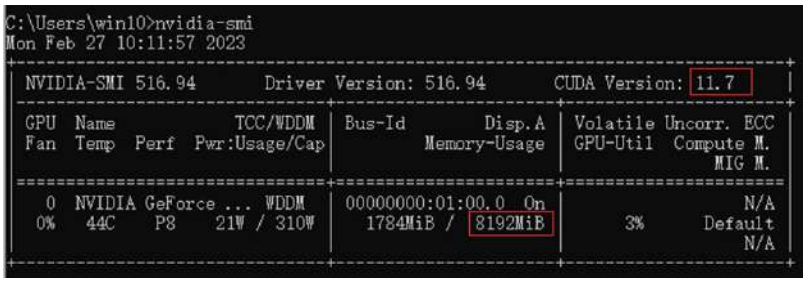
Check Graphics Card

The process of using deep learning will occupy a lot of memory of your computer's graphics card. To perform the deep learning process, it is recommended to use an NVIDIA GPU with at least 8 GB of dedicated memory and a minimum support of CUDA 11.3. To check the availability of the graphics card, you can use the NVIDIA SMI program and the NVIDIA Control Panel. The specific check 6/20/2023 3:47:41 PM teps are as follows:

Method one:

（1）Use the **win+r** combination key to open the Windows runtime interface, and enter **cmd** to open the command prompt window.

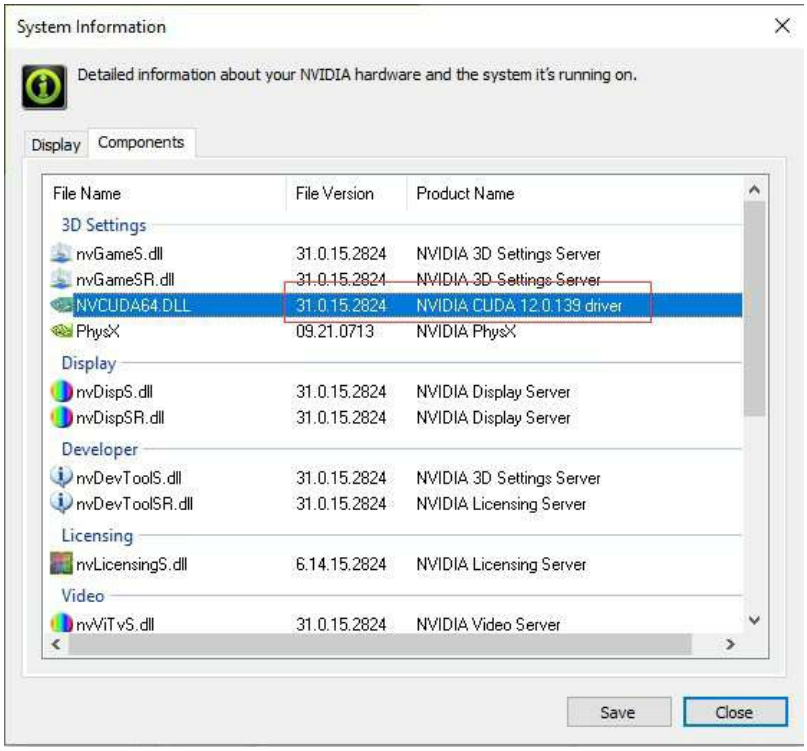
（2）Enter the **nvidia-smi** command in the command prompt window and press Enter. The maximum dedicated GPU memory and the highest supported CUDA version of your computer are displayed (if not displayed, please refer to **Install Graphics Card Driver** tutorial Install Graphics Card Driver). The information displayed may vary depending on your computer. In this example, there are 8192MB (8 GB) of dedicated memory in the GPU, and the highest supported CUDA version is 12.0.



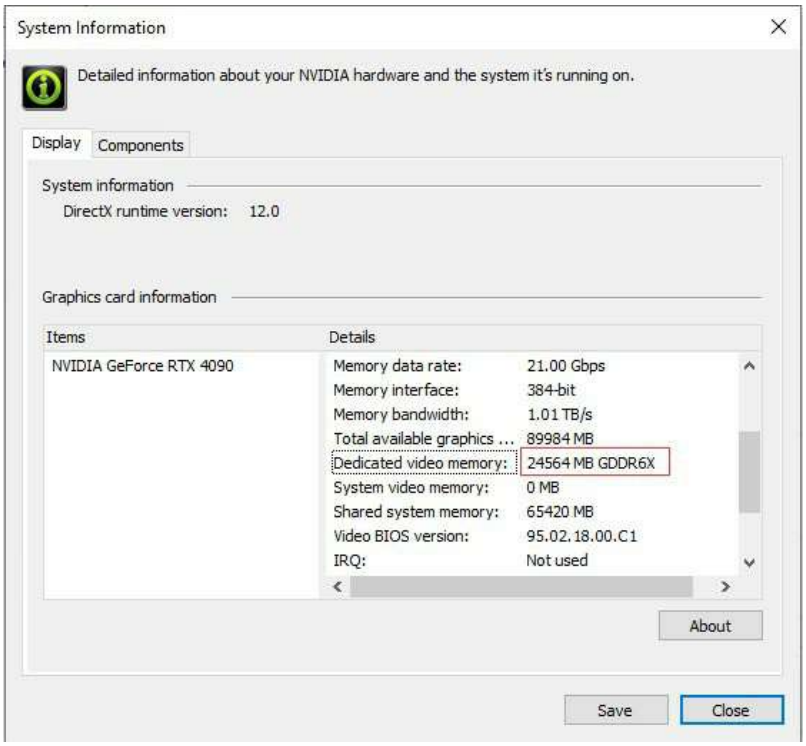
> Note: If there are multiple graphics cards, record the name of the best graphics card, and select the best graphics card for calculation in the deep learning module. Currently, only single graphics card calculations are supported.

Method two:

（1）Right-click on the desktop to open the **NVIDIA Control Panel**. Select **Help**->**System Information**->**Components**, and check the highest CUDA version supported by the current graphics card driver. As shown in the example in the figure, the highest supported CUDA version is 12.0.



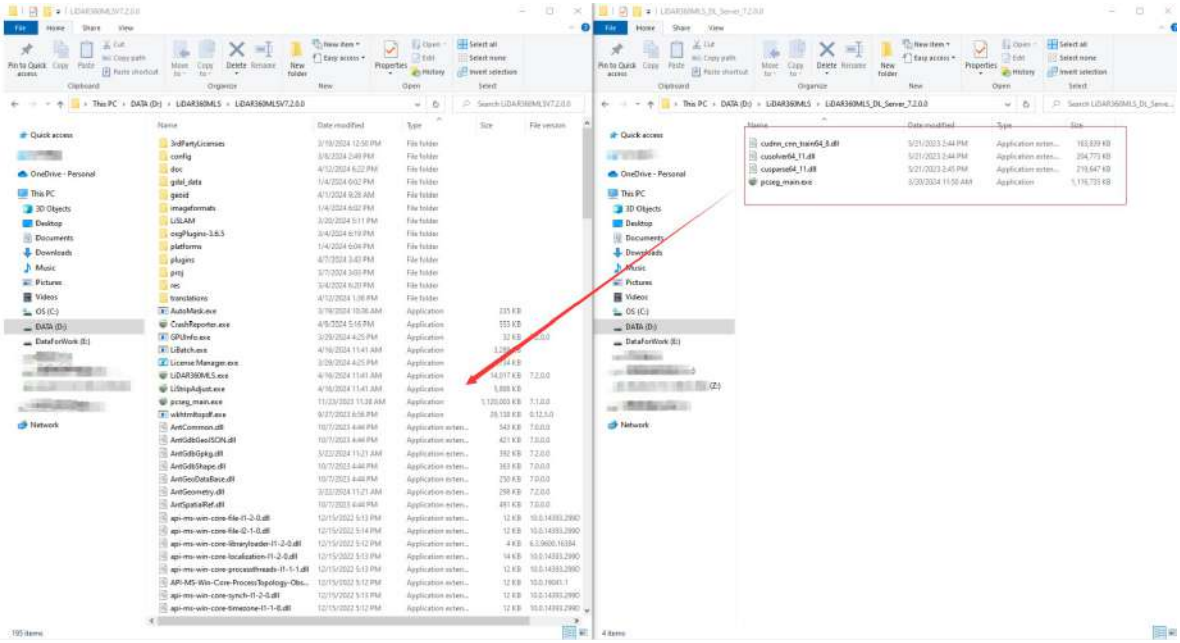
（2）Select "Help->System Information->Display" to check the maximum dedicated GPU memory of the current graphics card. As shown in the example in the figure, the dedicated memory is 24564MB (24G).



Software Preparation

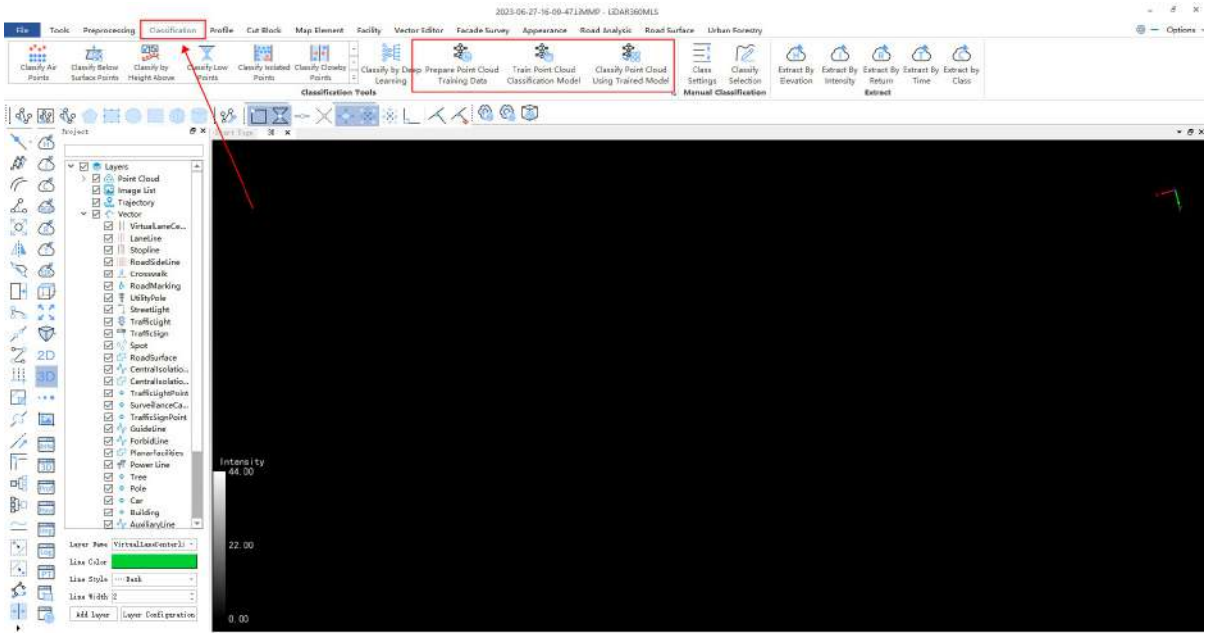
In order to use the Custom Deep Learning Classification function, it is necessary to install the deep learning training module extension on top of the LiDAR360MLS installation.

1）Firstly, visit the website to download the latest version of the extension package and unzip it after downloading. Followingly, copy all the files into the software root directory.（For example D:\LiDAR360MLS\LiDAR360MLS_DL_Server_7.2.0.0）.



2）Right-click on the Lidar360MLS software shortcut on the desktop, select "Open File Location" to locate the software installation directory, enter the "7.2.0.0" folder, and unzip the 4 files extracted in step 1) into the Extract the four files extracted in step 1 to the "7.2.0.0" directory. If you are prompted "You need to provide administrator privileges to copy to this folder", please click Continue. 3）Open the LiDAR360 MLS software and click

Classification.Then check whether the Deep Learning Classification module is activated as shown below. If activated successfully, you can use this module for Custom Deep Learning Classification.



Point Cloud Fully Supervised Labeling And Weakly Supervised Labeling Process

Function Description: This process allows you to classify uncategorized point clouds by labelling the point cloud data for training purposes. The process firstly introduces the concepts of fully supervised and weakly supervised learning, and then covers the basic operations for point cloud classification, as well as finally explains the labelling methods for fully supervised and weakly supervised learning. The fundamental principle of point cloud classification is to label the training data with the classes you want the model to learn while leaving the remaining points uncategorized.

Concepts

1.Fully Supervised Training: Fully supervised training refers to the use of point cloud data with complete labeled information during the training process. Each point in the point cloud data needs to have a specific class label.

2.Weakly Supervised Training: Weakly supervised training refers to the training process where only partial labeled information is available. In the context of this module, weakly supervised learning means that only a small portion of the points have explicit class labels, while the remaining points are uniformly classified as uncategorized. For example, for a tree, only a part of the points need to be classified as a tree, while the rest can be classified as uncategorized.

Steps

1.Classification Processing:

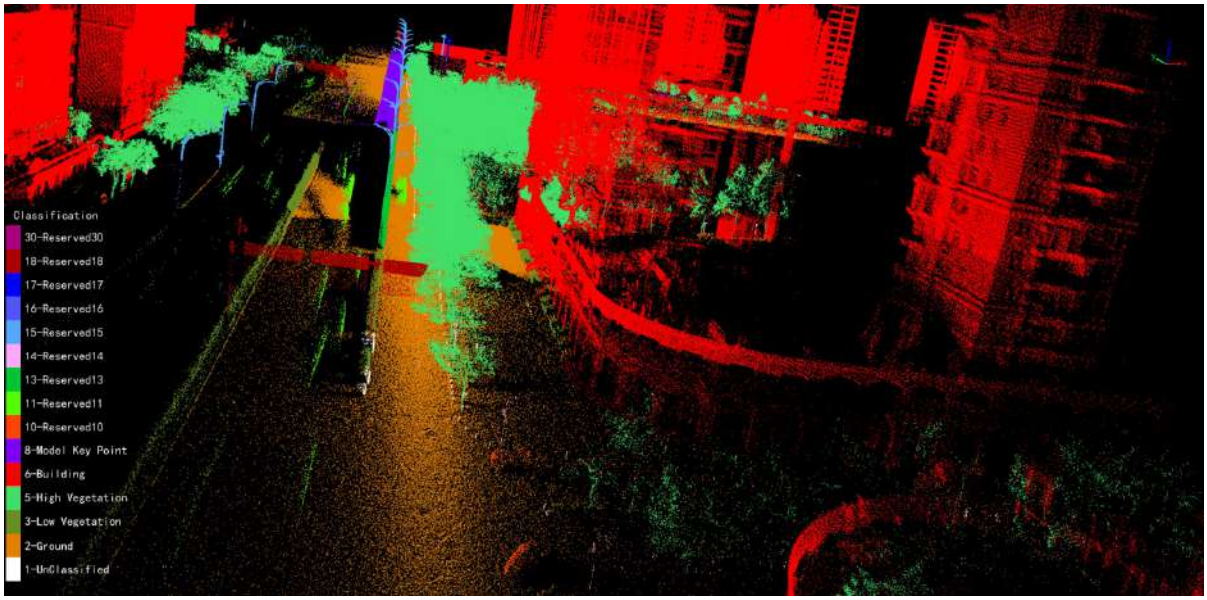
- **Classification using the Classification Module**：Refer to the Manual Editing Classification documentation;
- **Classification using the Profile Module**：Refer to the Manual Classification in Profile documentation;

2.Fully Supervised Data Labeling: When training point cloud data by PCSS and SPV-PCSS, data with full labelling is required. Point cloud data can be labeled using the built-in classification feature of LiDAR360 MLS software (refer to Step 1).

（1）Principles for Labeling:

> 1）Each point of each object must be labeled.
>
> 2）Carefully select points when dealing with boundaries. If points from other objects are classified as the current object's category, it will have a significant impact on training.
>
> 3）Only label the classes of interest. For example, if you want to classify trees and buildings in point clouds during inference, you only need to label trees and buildings, while leaving the rest of the points uncategorized.

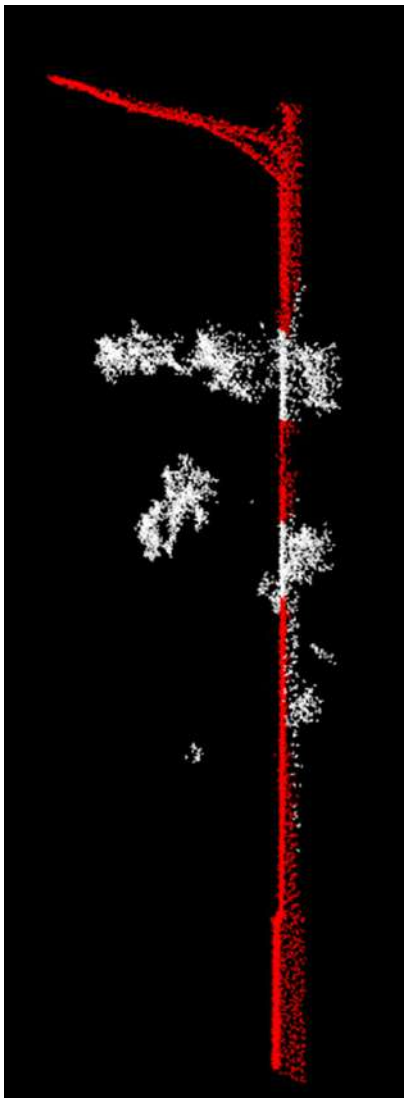
（2）The labeled point cloud should have the interested objects labeled with their respective classes, while the remaining points should be labeled as uncategorized, as shown in the example below.



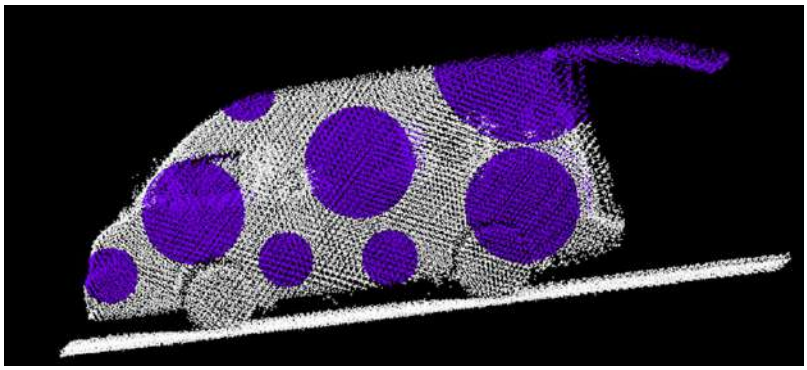
3.Weakly Supervised (WS-PCSS) Data Labeling:

> 1. When using the WS-PCSS algorithm for training, you can use point cloud data with weak labels, which greatly reduces the labelling workload and improves production efficiency. The following describes the labeling method for weakly supervised learning;
> 2. Point cloud data can be labelled by using the built-in classification function of LiDAR360 MLS software, following the same labelling process as described in step 1 and 2. However, fully labelling is not required for weakly supervised learning. Only partial points need to be labelled with their respective labels. The following principles should be followed when labelling:

1）Detailed subdivision of the object is not necessary during labelling. Only a portion of the points for each object needs to be labelled. For example, in the image below, difficult-to-label area can be left unlabeled, and only the target parts need to be labelled.



2）The labelled points should be evenly and uniformly distributed in the target object. For example, in the image below, the labelled points should cover all the characteristic parts of the object as far as possible.

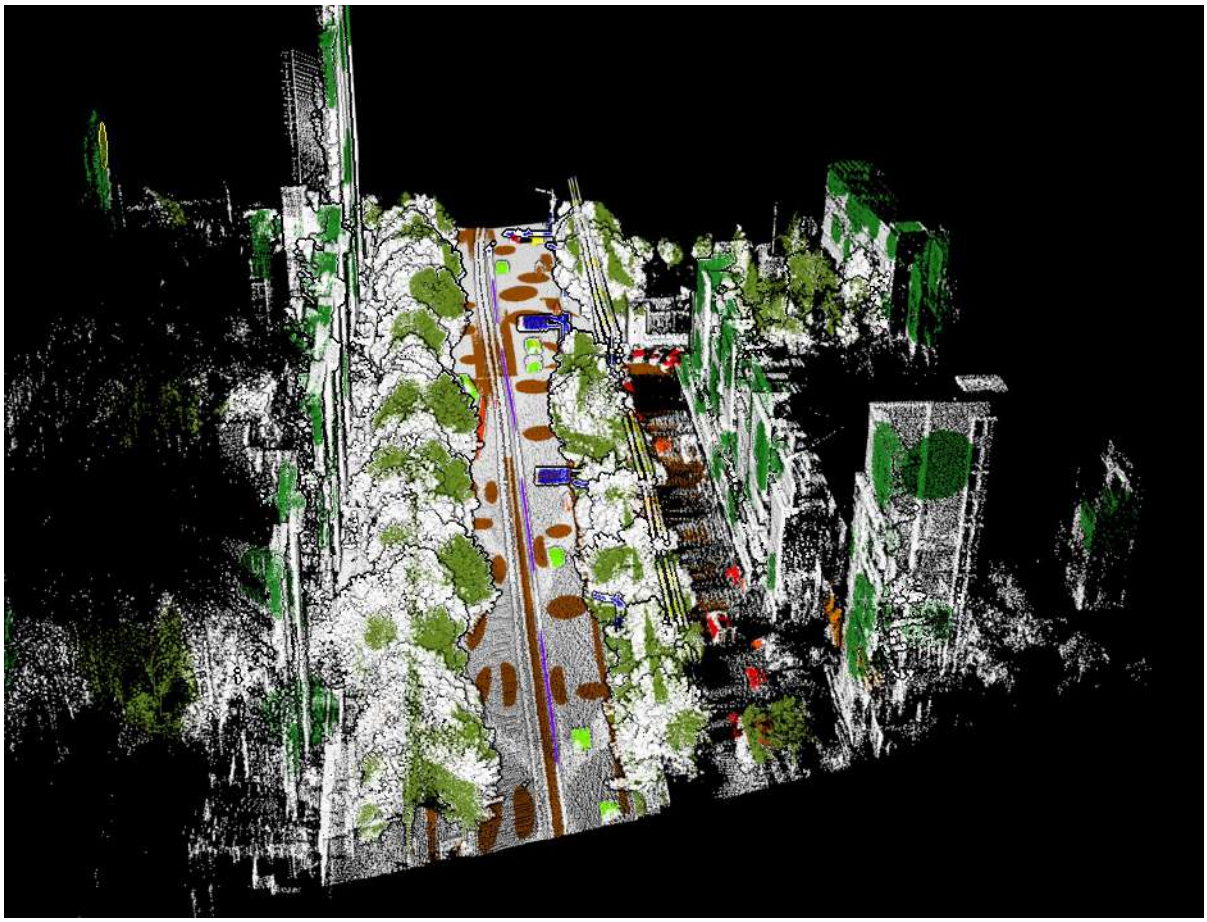


3）You can label less point cloud, but mislabelling or over-labelling should be avoided.

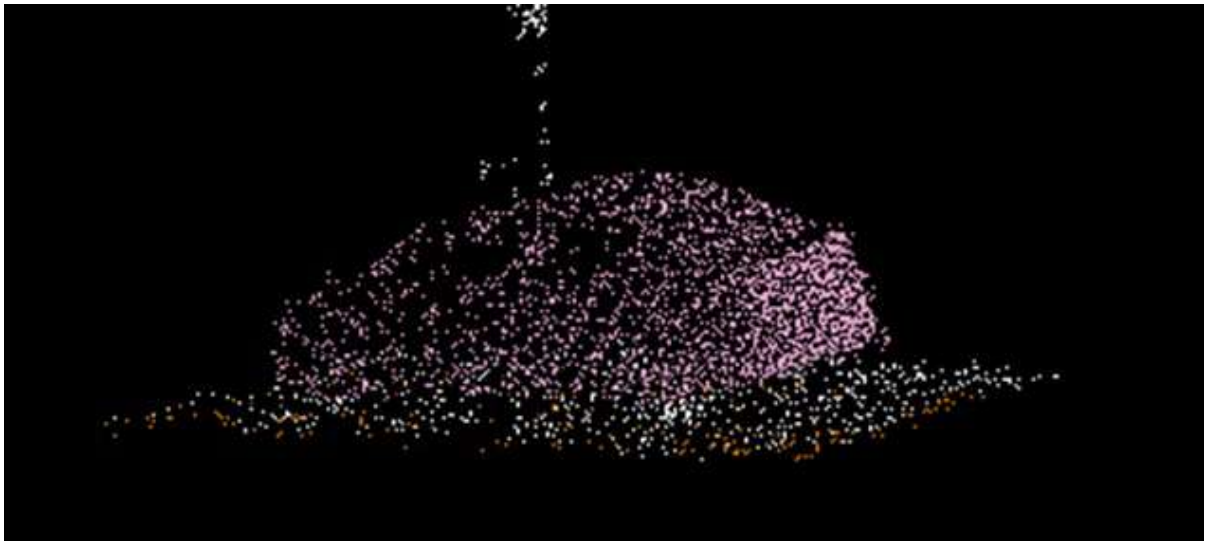
Mislabelled

Correctly labelled

4）Each object in the scene should have labelled points as much as possible. As shown in the image below, every object in the scene should have labeled points, even if only 1% of the points are labelled.



5）When two objects are adjacent (or have overlapping area), both objects should have some labelled points. In the example below, both the ground and the car should have some labelled points because they are adjacent at the boundary.



6）The class with the smallest label index in the labelled data will be ignored during training. Therefore, at least one of **Created Points**, **Unclassified**, or **Unclassified Points** should be present in each LiData data.

Prepare Point Cloud Training Data

Function Description: Prepare data for training the point cloud deep learning model.

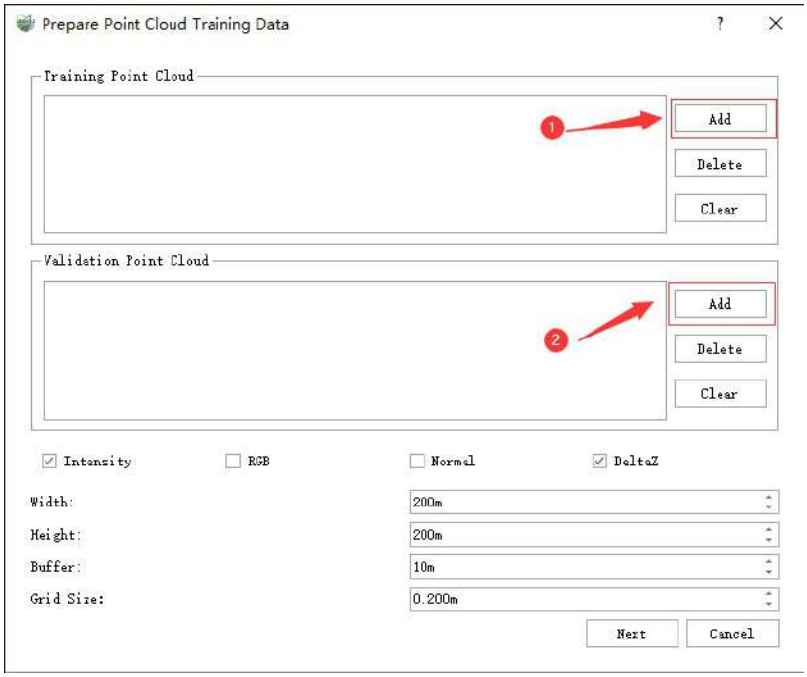
Steps

1.**Data Preparation**：The data format supported by this function is different from that supported by LiDAR360 MLS and this function only supports data in LiData (*.LiData) format. Specific requirements are as follows:

- **Training Data**：Data with class labels. If there is no available labelled data, you can use the point cloud classification function of LiDAR360 MLS to manually/automatically classify the point cloud data. For the methods of point cloud manual/automatic classification, refer to Point Cloud Full Supervised Labeling and Weak Supervised Labeling Process. In theory, the more labelled data involved in training, the better the quality of the trained model. We recommend using data with a range of at least 100m*100m for training to achieve good results. If multiple data are used for training, make sure that the categories of classification in each data are consistent. If not, it will have a negative impact on the quality of model training.
- **Validation Data**：Data with class labels. This data will be used as the "standard answer" for point cloud classification and used to calculate accuracy metrics such as the accuracy of the model classification. The classes of this data should be consistent with the training data.

2.**Data Preprocessing**

1）Firstly, click on the **Prepare Training Data** button in the classification tool, and use the **Add** function in **Training Point Cloud** and **Validation Point Cloud** to add corresponding data. This also includes **Delete** and **Clear** functions, which can delete specific files in the list and clear the file selection list.



2）Select the feature information contained in the processed point cloud data. This function supports the following four features.

- **Intensity**：The intensity feature of the point cloud.
- **RGB**：Color feature.

- **Normal Vector**：The normal vector feature of the point cloud.
- **Relative Elevation**：Elevation feature relative to the ground.

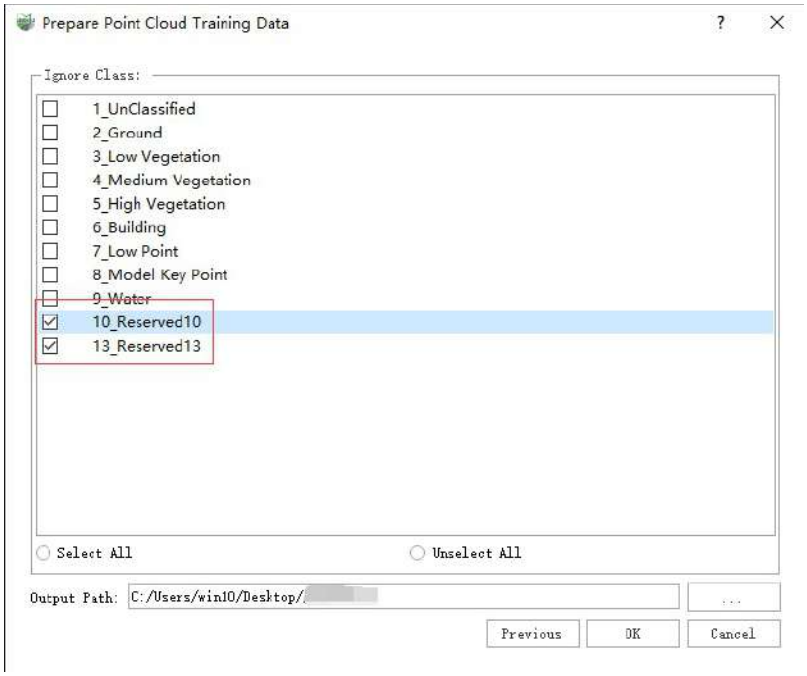
> Note：>It is recommended to keep **RGB**, **Relative Elevation**, and **Intensity** features. If there are no **Intensity** and **RGB** features in the data, the corresponding options cannot be selected. Even if the original data does not have **Vector** and **Relative Elevation**, they will be automatically generated later.

3）Select **Slice Size**

- **Width** and **Height**：Indicate the width and height of the point cloud data after slicing. The larger these two values are, the larger the amount of slice data and the slower the file reading speed. If the amount of slice data is too small, it results in not getting enough data for each training, which affects the final accuracy. Setting an appropriate Width and Height can improve the file reading speed during training.
- **Buffer**：The overlap length of the slice point cloud. Setting the overlap length can avoid the impact of the edge part of the point cloud on the training and validation results, which is generally set to 10 by default.
- **Voxel Size**：The size of the voxel sampling grid. The larger this value is, the sparser the point cloud used in training. Too sparse will affect the final accuracy. The smaller this value is, the denser the point cloud used in training and the computation increases. For data of indoor scenes, it is generally set to **0.02-0.04**m by default, and for data of outdoor scenes, it is generally set to **0.05-0.2**m by default.

4）Click **Next Page**, and the **Ignore Categories** box that pops up shows the category information in the selected data. Here, you can select the classes to be ignored and map them to class 0 so that ignore this class in the subsequent training steps. Only the classes that need to be classified should be retained, and the unrelated classes should be selected and ignored. At **Output Path**, you can choose the data output folder, which is saved in the same directory as the current project by default. After clicking **OK**, the software starts processing data and users can view the data processing progress at the progress bar at the bottom of the software.

> Note:
>
> 1. When using the WS-PCSS method, there must be Never Classified, or UnClassified categories. For specific explanations, refer to Point Cloud Full Supervised Labeling and Weak Supervised Labeling Process.
> 2. If there are "Never Classified" and "UnClassified" categories at the same time, the two categories will be automatically mapped to category 0 during the data preparation stage, and then participate in training.


Prepare Point Cloud Training Data
Ignore Class:
1_UnClassified
2_Ground
3_Low Vegetation
4_Medium Vegetation
5_High Vegetation
6_Building
7_Low Point
8_Model Key Point
9_Water
10_Reserved10
13_Reserved13
Select All
Unselect All
Output Path:
C:/Users/win10/Desktop/
...
Previous
OK
Cancel

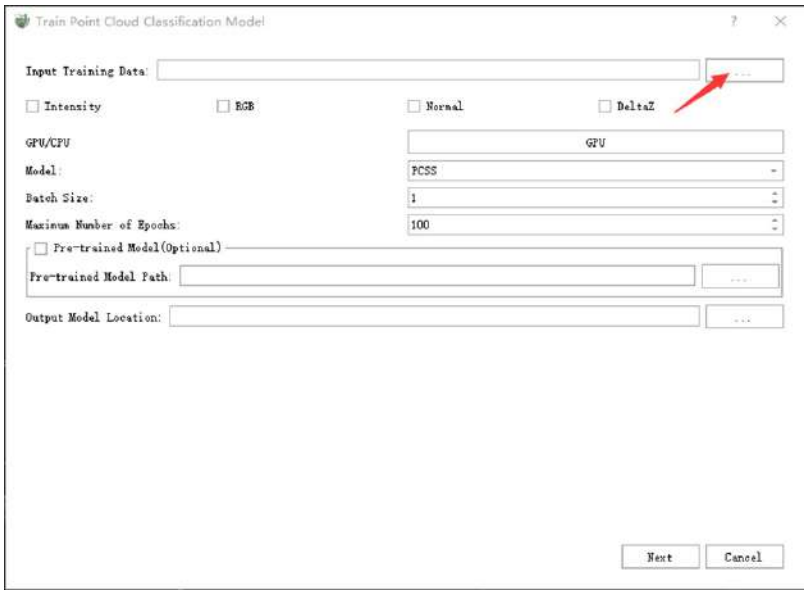
Training Point Cloud Classification Model

Feature Description: Train a deep learning model using the generated point cloud training data.

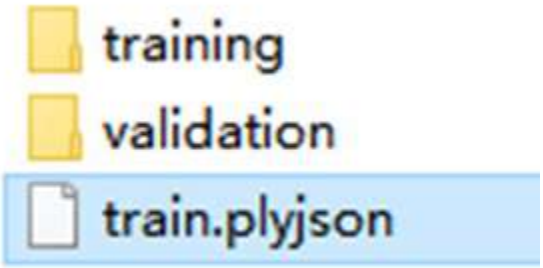
Steps

1.Training process and parameter introduction

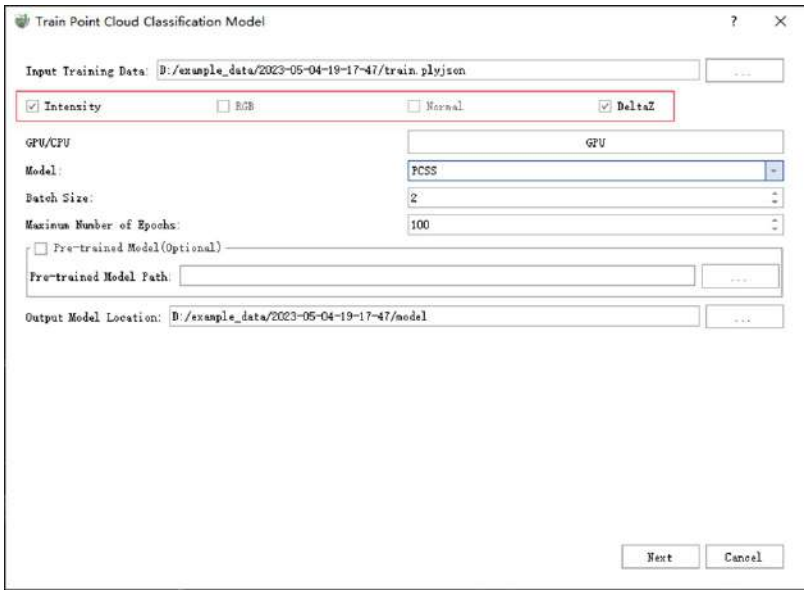
1）Firstly, click the **Train Point Cloud Classification Model** button.



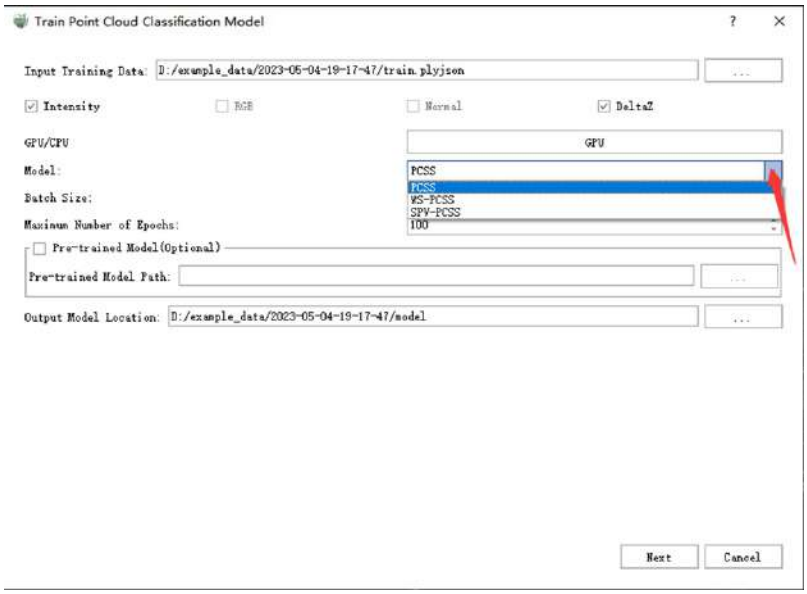
2）Click the button in the figure to select the training file generated in the previous step. You only need to select **train.plyjson**.



3）Select the features involved in the training. All features included in the data are checked by default. Features that are not checked during data preparation will be grayed out.

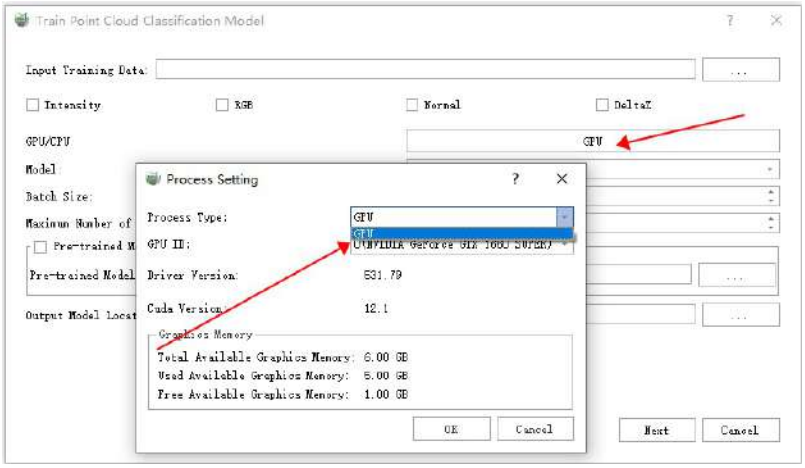


4）As shown in the figure, you can specify the GPU to be used for training in the drop-down menu, and this function only supports GPU for training.



5）**Training Model Selection**：The software currently provides three deep learning method options. The characteristics of each algorithm are shown below. Users can choose according to specific needs.

- **PCSS**： A fully supervised point cloud semantic segmentation model with high accuracy. It is suitable for classification tasks that require high accuracy, but its speed is relatively slow. It requires fully labelled point cloud data and the training cycle is relatively long.
- **SPV-PCSS**：A fully supervised point cloud semantic segmentation model is suitable for classification tasks that focus on large-scale objects. The accuracy is relatively high and the inference speed is faster, but it requires fully labelled point cloud data and the training cycle is moderate.
- **WS-PCSS**：A weakly supervised point cloud semantic segmentation model, which can use less labelled point cloud data for training. Each point cloud data only needs to be labelled more than 1% of the labels, but the accuracy is lower than the fully supervised model. Its training and inference speed is slightly slower. However, this method can effectively reduce the workload of labelling.



6）**Batch Size**：This is the number of data inputs for each training session. For example, if the batch size is 2, then two groups of data are trained in the same batch. Theoretically, the larger the batch size, the higher the accuracy of the results obtained. It is recommended not to exceed 8. The upper limit is limited by the size of the video memory. According to different video memory sizes, the recommended batch size is as follows. When the **memory or video memory is insufficient, you can appropriately reduce this parameter**. In addition, it should be noted that the larger the ratio of data-labeled points to total points, the more video memory the WS-PCSS algorithm uses.

	PCSS	WS-PCSS(1%)	SPV-PCSS
8GB	2	1	2
10GB	2	2	2
11GB	2	2	2
12GB	2	2	2
16GB	4	2	4
24GB	4	2	4

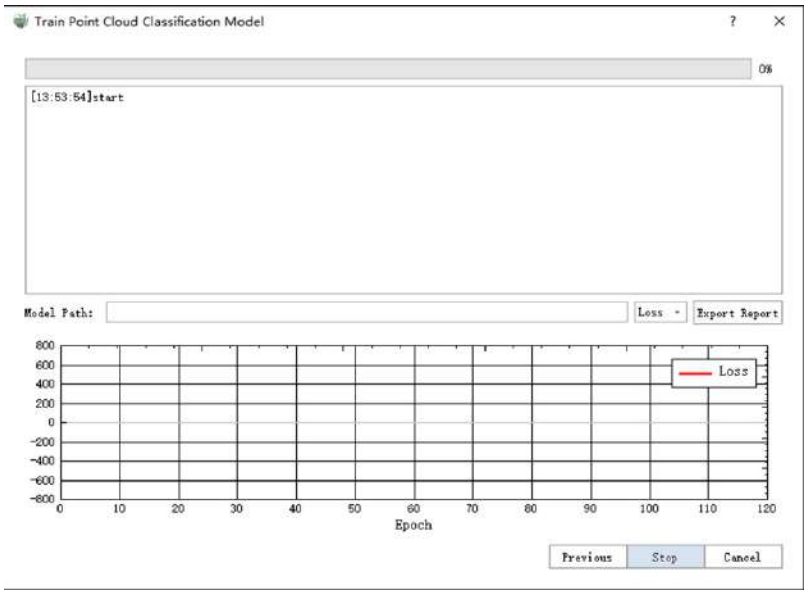
7）**Maximum Epoch Number**：The maximum training period. This parameter set needs to be adjusted according to the trend of loss and IoU. The specific numerical setting is best when the value of Loss no longer decreases and IoU no longer increases. The default is 100. **Training is stopped when the average IoU does not improve in the validation set for 10 consecutive epochs**.

8）**Pre-trained Model:**：This is an optional feature. Its function is to select a pre-trained model, which can be used in two situations:

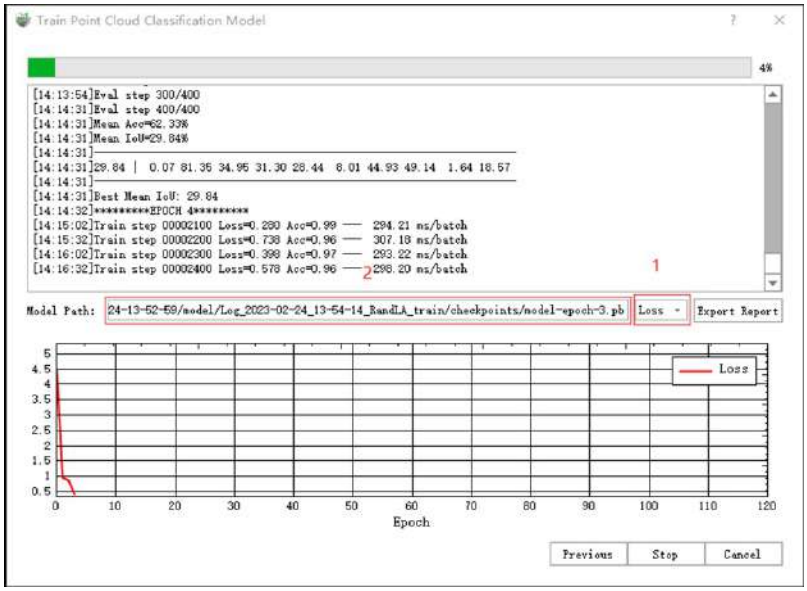
- If the training is unexpectedly interrupted, there is no need to retrain. You only need to use the model file saved at the time of interruption as a pre-trained model for training.
- When you already have a trained model and have new labelled data, you do not need to merge the original data and retrain. You only need to train based on the original model, which can effectively save time. However, the classes of the training data, the selected features, and the algorithm name need to remain consistent between the two training sessions.

9）**Model Output Path**：The output folder of the trained model. Saved in the project file by default.

10）After clicking **Next**, go to the training interface. The progress bar of the training process is displayed at the top of the interface and the training log is displayed in the box below.



11）After clicking **Start**, the training begins. You can select the change curve you want to view from Loss, mIoU, and mAcc in the dropdown menu in the 1 place in the figure. After the training stops, you can click Export Report to print the training report. The report file is automatically saved in the project file. The trained model file is saved in the box marked 2 in the figure.

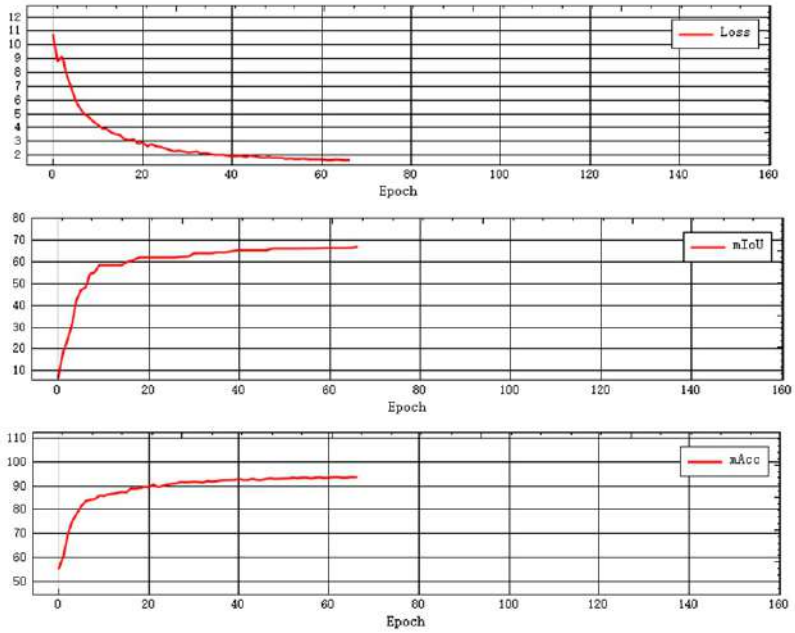


12）During the training process, you can observe the change in the curve to analyze the training effect.

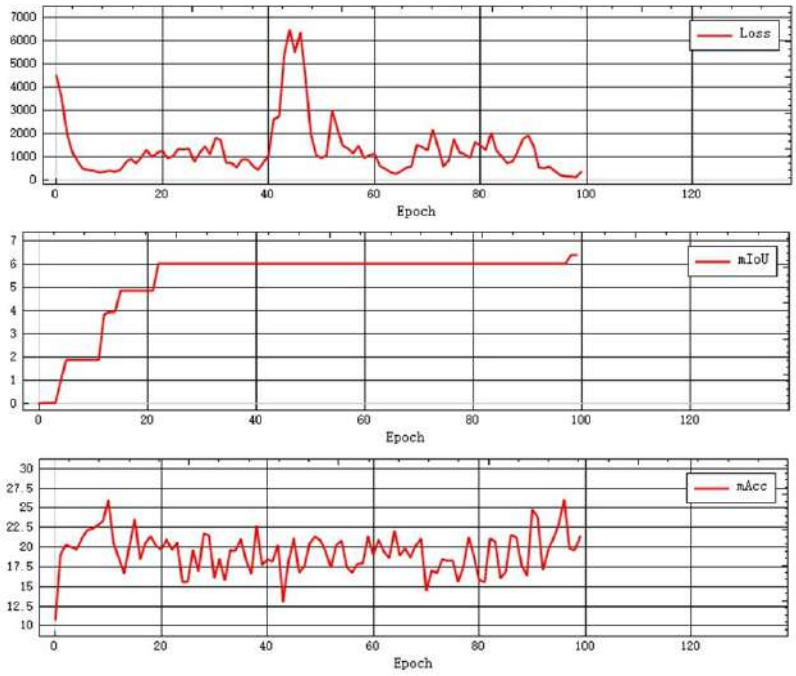
- Normally, Loss will decrease with the increase of Epoch and tend to stabilize after dozens or hundreds of Epochs. If this characteristic is not met or the Loss curve oscillates, it indicates that there is a problem with the training. Please stop the training, check the data and retrain.
- Normally, the mAcc value will increase with the increase of Epoch and finally stabilize. If this characteristic is not met, it indicates that there is a problem with the training. Please stop the training, check the data and retrain.
- Normally, mIoU will gradually increase and stabilize after a certain epoch. If mIoU does not stabilize in the end, it means that the model still has training potential. Please continue training using the load pre-trained model method until mIoU stabilizes. If mIoU does not increase, it indicates that there is a problem with the training. Please stop the training, check the data and retrain.

The figure below provides samples of the three curves for users to refer to.

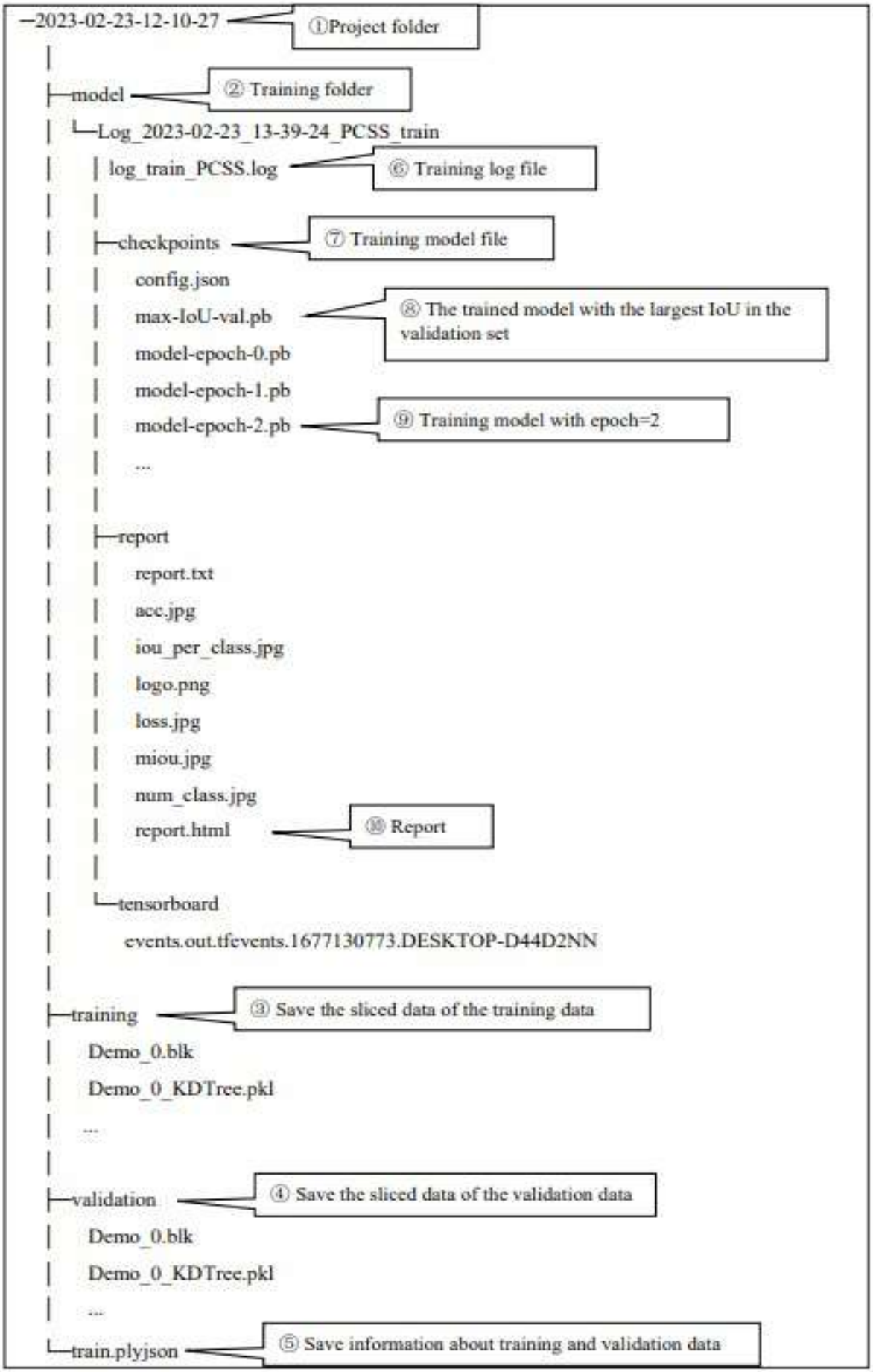
Normal curves:



Examples of problematic curves:



2.Training Project File Structure:


2023-02-23-12-10-27
①Project folder
model
② Training folder
Log_2023-02-23_13-39-24_PCSS_train
log_train_PCSS.log
⑥ Training log file
checkpoints
⑦ Training model file
config.json
max-IoU-val.pb
⑧ The trained model with the largest IoU in the validation set
model-epoch-0.pb
model-epoch-1.pb
model-epoch-2.pb
⑨ Training model with epoch=2
...
report
report.txt
acc.jpg
iou_per_class.jpg
logo.png
loss.jpg
miou.jpg
num_class.jpg
report.html
⑩ Report
tensorboard
events.out.tfevents.1677130773.DESKTOP-D44D2NN
training
③ Save the sliced data of the training data
Demo_0.blk
Demo_0_KDTree.pkl
...
validation
④ Save the sliced data of the validation data
Demo_0.blk
Demo_0_KDTree.pkl
...
train.plyjson
⑤ Save information about training and validation data

The table above shows an example of a project folder structure:

① **2023-02-23-12-10-27**：Project folder is automatically generated in the data directory based on the processing time during the data processing stage.

② **model**：It stores the intermediate results of the training.

③ **training**： It stores the sliced data for training.

④ **validation**： It stores the sliced data for validation.

⑤ **train.plyjson**：It saves the relevant information of the training and validation data.

⑥ The Model folder stores the training result files. Multiple training results for the same dataset are saved in this folder. The example shows the training folder "Log_2023-02-23_13-39-24_PCSS_train". The **log_train_PCSS**.log file stores the training log data.

⑦ **checkpoints** folder： It stores the trained model files for this training.

⑧ **max-IoU-val.pb**：The trained model with the maximum validation IoU.

⑨ **model-epoch-.pb**：The trained models corresponding to each epoch.

> Note: The model files for PCSS and WS-PCSS are in .pb format, while the model file for SPV-PCSS is in .pt format.

⑩ The report folder stores the files related to the report, including the report.html file in HTML format.

> Note: The project folder mentioned here refers to the project folder of the training process.

3.Model Deployment: The trained models are saved in the model folder. You can copy this folder to another computer with an Nvidia GPU and use LiDAR360 MLS software for inference based on the trained model. Refer to the Classify Point Cloud Using Trained Model for the specific inference process. Please refer to the installation instructions in the module to check if your device supports the inference.

Classify Point Cloud Using Trained Model

Feature Description: Use the generated point cloud training data to train a deep learning model.

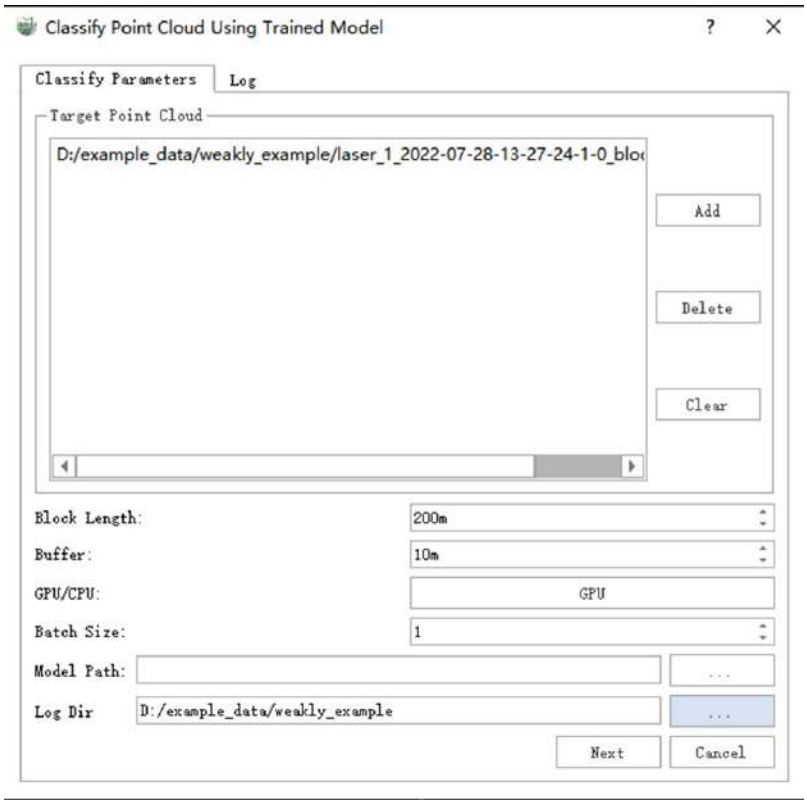
Steps

1.**Data Preparation**：Before using the inference function, you need to prepare the following data:

- **Model**：The model is trained using the custom deep learning classification feature. It can be found in the project directory -> model -> Log*2023-02-14_10-15-29**_train(specific training directory)* -> .pb/.pt. The model data is obtained after the training is complete.
- **Inference Data**：The point cloud data that needs to be processed and unclassified, i.e., inference data, in the format of **.LiData**.

2.**Inference**

1）Click the **Classify Point Cloud Using Trained Model** button, the setup interface pops up. Select the data to be classified in the **Target Point Cloud** box, which should be in.LiData format. This box has the same **Add**, **Delete**, and **Clear** functions as the training module. Click the Add button to add data to be classified.



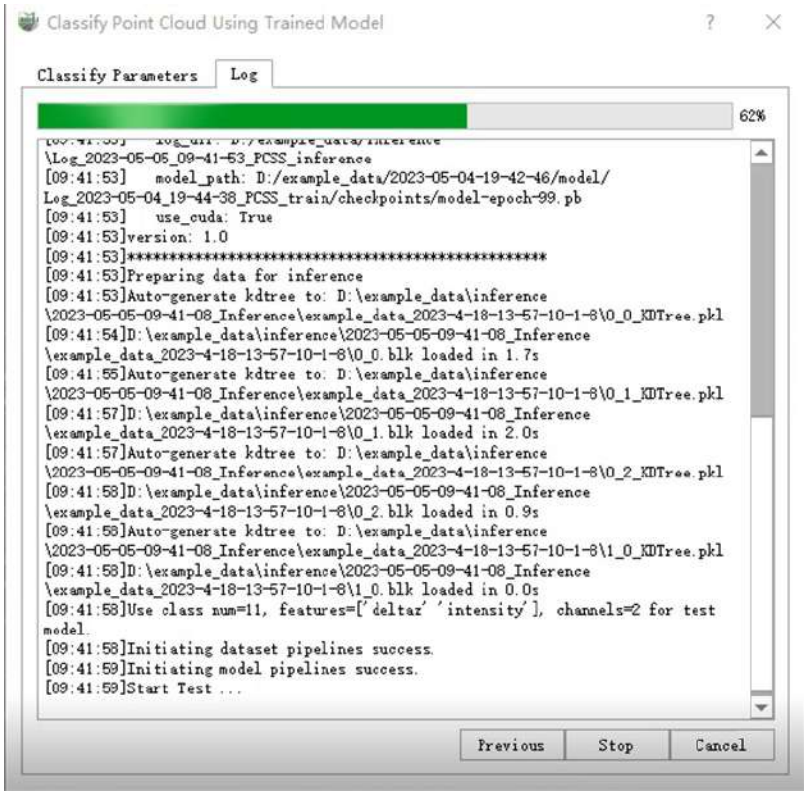
- **Chunk Length**：The slice length of the data to be classified.
- **Buffer**：The overlap length between adjacent slices during slicing. This is usually left at the default. These two values should be set consistently with the training.
- **GPU**：Click the GPU button in the picture to select the GPU to be used for inference. Currently, only single GPU computing is supported.
- **Batch Size**：The same as in training. The image below shows the recommended values.
- **Unclassified Threshold**：Set the unclassified threshold here to distinguish unclassified points. It is usually

set to 0.1 by default.

	PCSS	WS-PCSS(1%)	SPV-PCSS
8GB	4	1	4
10GB	6	1	4
11GB	6	1	6
12GB	8	1	8
16GB	8	2	8
24GB	8	2	8

- **Model Path**: Select a model file that has been trained and is of high quality. If the user has no special needs, it is recommended to select max-IoU-val.pb/max-IoU-val.pt. If you are focusing on a specific category, it is recommended to select the file corresponding to the epoch with the highest IoU for that category based on the training report.
- **Log Path**: The storage path of the inference log. By default, it is stored in the project folder at the same level as the inference data.

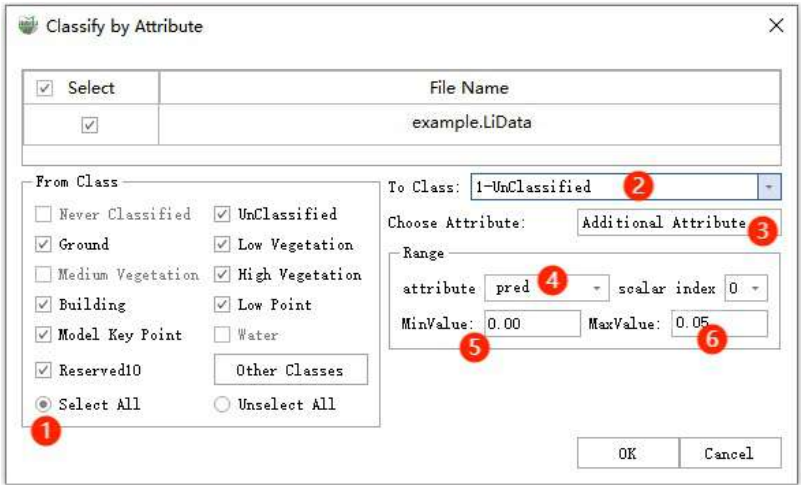
2）Click **Next** to go to the inference interface. From top to bottom, there are a progress bar and a log window. Click **Start** to start inference. After the inference is finished, the selected.LiData file will be replaced.



Note：

1. During inference, the point cloud slice files and log files for the inference are automatically saved in the current project directory. For example, 2023-02-23-14-32-01_Inference and Log_2023-02-23_14-32-24_PCSS_inference save the point cloud slice files and the inference log files respectively.
2. The inference result automatically adds the classification information to the originally selected.LiData file. If there is category information, it will automatically be replaced.

3.Extract unclassified points: Since every point in the point cloud after inference has a specific category, there are no unclassified points. If you need to distinguish unclassified points, you can use **Classification** -> **Classify by Attribute** to select the point cloud to be classified and classify according to the pred attribute in the order shown in the figure. The minimum value of the unclassified **pred** attribute is generally set to 0, and the maximum value is set between 0.05 and 0.1.



Report

Function Description: This function allows you to view the basic information of the current training and the training results.
Below are the details in the report.

1.**Project Detail**: This table records the basic information of the current training project.

For example：

1.Project Detail

Generated Time	2023-02-20_11-43-13
Pattern	Train
Model	RandLA
Optimizer	Adam
Criterion	WeightCrossEntropy

Function Description

- **Generated Time**: The time when the report was generated, corresponding to the project file name.
- **Pattern**: The mode, including training and inference. Currently, only the training report is printed.
- **Model**: The trained model, including PCSS, WS-PCSS, SPV-PCSS.
- **Optimizer**: The optimizer used during training.
- **Criterion**: The loss function used during training.

2.**Dataset**

For example：

2.Dataset

Grid Size	0.2
Ignored Labels	[]
Number Classes	11
Train Data Directory	[redacted]\training
Validation Data directory	[redacted]\validation

Function Description

- **Grid Size**: The voxel grid size is used for sampling the training point cloud.
- **Ignored Labels**: The label indices to be ignored during training.
- **Number of Classes**: The number of classes involved in training.
- **Train Data Directory**: The directory path for the training data files.
- **Validation Data Directory**: The directory path for the validation data files.

3.**Train**

For example：

3.Train

Whether to Use GPU	True
GPU Ids	[0]
Batch Size	4
Max Train Epoch	100
The Number of Steps per Epoch	500
Whether to Use Pre-trained Model	False
Pre-trained Model Path	

Function Description

- **Whether to Use GPU**: Whether to use GPU for training.
- **GPU Ids**: The IDs of the GPUs to be used.
- **Batch Size**: The batch size used during training.
- **Max Train Epoch**: The maximum number of training epochs, indicating the number of times the training data is looped during training.
- **The Number of Steps per Epoch**: The number of steps per epoch in training.
- **Whether to Use Pre-trained**: Whether to use a pre-trained model.
- **Pre-trained Model Path**: The file path of the pre-trained model.
- **Label Ratio**: The ratio of labelled points to all points in the training data. (This information is only displayed when training with the WS-PCSS model).

4.**Feature Whether to Use Delta**

For example：

4.Feature

Whether to Use Delta z	True
Whether to Use Intensity	True
Whether to Use RGB	True
Whether to Use Normal	False

Function Description

- **Whether to Use Delta z**: Whether to use the delta z feature in training.
- **Whether to Use Intensity**: Whether to use the intensity feature in training.
- **Whether to Use RGB**: Whether to use the RGB color feature in training.
- **Whether to Use Normal**: Whether to use the normal vector feature in training.

5.**Result**

For example：

5.Result

Completed Training Epochs	10
Final Loss	0.362
The Path of Trained Model	[redacted]\Log_2023-05-17_10-24-00_SPV-PCSS_train\checkpoints
The Max Validation Intersection over Union (m IoU)	59.76
Mean Precision	59.19
Mean Recall	62.84
F1	0.60

Function Description

- **Completed Training Epochs**: The number of epochs completed in the training process for which the report is printed.
- **Final Loss**: The final value of the loss function.
- **The Path of Trained Model**: The file path where the trained model is saved.
- **The Max Validation Intersection over Union (mIoU)**: The maximum average intersection over union (IoU) on the validation set.
- **Mean Precision**: The average precision at the epoch with the maximum IoU.
- **Mean Recall**: The average recall at the epoch with the maximum IoU.
- **F1**: The average F1 score at the epoch with the maximum IoU.

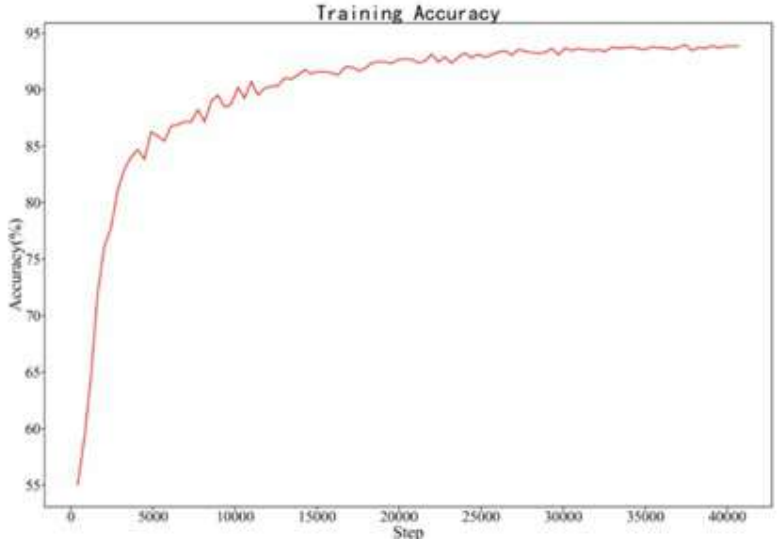
6.**Training Loss**：This graph represents the trend of loss value (loss) during training as the number of steps increases (each epoch contains a certain number of steps). The x-axis represents the step number, and the y-axis represents the loss value. Normally, the loss value decreases as the number of steps increases and

eventually stabilizes. If the loss does not exhibit this characteristic, it indicates that there may be issues with the training. Please check the data and consider retraining.

For example：

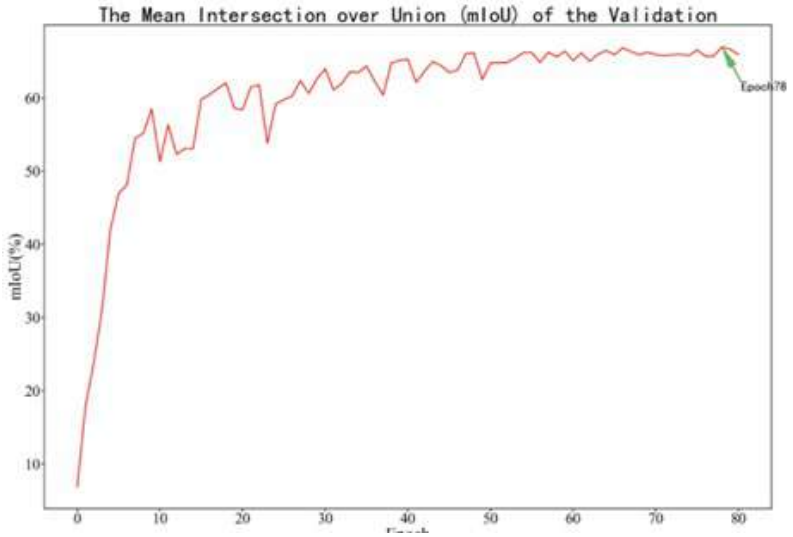
7.**Training Accuracy**：This graph represents the trend of prediction accuracy of the sampled point clouds during training as the number of steps increases. The x-axis represents the step number, and the y-axis represents the prediction accuracy. Normally, the accuracy increases as the number of steps increases and eventually stabilizes.

For example：



8.**The Mean Intersection over Union (mIoU) of the Validation**：This graph represents the trend of mIoU (Mean Intersection over Union) during the evaluation process as the number of epochs increases. The x-axis represents the epoch number, and the y-axis represents the mIoU value. Normally, the mIoU increases gradually and starts to stabilize at a certain epoch. The arrow in the graph points to the epoch with the highest mIoU value and indicates the corresponding epoch and model name in the model folder. If the mIoU does not stabilize, it means that the model still has the potential for further training. In that case, you can continue training by loading the pre-trained model until the mIoU stabilizes.

As shown in the figure, the maximum mIoU is achieved at Epoch 78.



9.**The Max Intersection over Union (IoU) and Epoch of each Class**：This table indicates the epoch at which the maximum IoU (Intersection over Union) occurs for each class during the validation process, along with other corresponding information. The first row of the table represents the class name, epoch, and mIoU for each corresponding class. The second row in the table indicates that the class "Never Classified" has a maximum IoU of 25.62 at epoch 41 and the average IoU for participating classes is 65.20 followed by the IoUs of other categories. If you are particularly interested in a specific class during the classification task, you can choose the model file corresponding to the epoch with the highest IoU of that class for inference.

As shown in the figure,

9.The Max Intersection over Union (IoU) and Epoch of each Class

The Max Intersection over Union (IoU) and Epoch of Each Class

Class	Epoch	mIoU(%)	IoU for each Class(%)										
			Never Classified	UnClassified	Ground	Low Vegetation	Medium Vegetation	High Vegetation	Building	Low Point	Model Key Point	Water	Reserved10
Never Classified	41	65.20	24.55	91.44	59.98	94.65	83.88	71.41	77.55	77.57	55.30	25.66	55.20
UnClassified	57	65.20	23.79	91.70	59.82	95.02	83.95	68.30	77.13	76.51	55.61	27.78	51.53
Ground	54	65.20	17.69	89.69	61.00	95.18	84.62	72.27	76.20	76.01	52.25	32.02	55.33
Low Vegetation	70	65.43	24.27	91.46	59.01	95.22	85.01	71.56	76.16	77.50	55.29	30.98	51.53
Medium Vegetation	70	65.43	24.27	91.46	59.01	95.22	85.01	71.56	76.16	77.50	55.29	30.98	51.53
High Vegetation	22	59.64	10.14	89.62	53.15	93.65	71.37	77.61	64.36	70.99	41.11	18.48	48.54
Building	37	63.02	13.26	87.60	59.52	93.97	81.63	71.11	78.38	76.21	55.40	29.56	46.63
Low Point	32	62.50	17.02	89.19	52.97	93.24	80.83	72.31	74.74	78.46	51.41	28.74	48.55
Model Key Point	48	65.20	18.38	90.68	56.08	94.55	82.68	73.04	71.53	77.92	58.34	28.51	46.09
Water	39	63.02	15.31	89.76	44.41	92.74	82.99	73.50	70.10	77.40	54.01	35.22	50.14
Reserved10	54	65.20	17.69	89.69	61.00	95.18	84.62	72.27	76.20	76.01	52.25	32.02	55.33

10.**The Precision and Epoch of each Class**：This table indicates the epoch at which the maximum precision occurs for each class during the validation process, along with the precision value for each class. As shown in the figure:

The Max Precision and Epoch of Each Class

Class	Epoch	Precision(%)	Precision for each Class(%)							
			Grounnd	Building	Natural	Building	Utility_line	Pole	Car	Fence
Ground	9	23.48	84.14	4.52	64.72	7.24	4.10	11.86	5.71	5.55
Building	9	23.48	84.14	4.52	64.72	7.24	4.10	11.86	5.71	5.55
Natural	19	39.19	61.56	1.35	88.60	21.31	56.99	63.40	11.97	8.37
Building	10	34.51	61.37	0.99	39.01	88.75	20.13	52.83	5.15	7.87
Utility_line	14	39.24	62.35	1.88	45.57	54.39	81.41	56.25	4.30	7.74
Pole	17	38.08	57.76	1.65	42.68	60.06	38.07	78.37	17.54	8.54
Car	7	26.92	67.77	1.90	29.30	52.28	2.32	7.42	51.53	2.83
Fence	16	41.75	60.88	2.08	46.31	57.01	52.88	66.77	29.46	18.57

11.**The Recall and Epoch of each Class**：This table indicates the epoch at which the maximum recall occurs for each class during the validation process, along with the recall value for each class. As shown in the figure:

The Max Recall and Epoch of Each Class

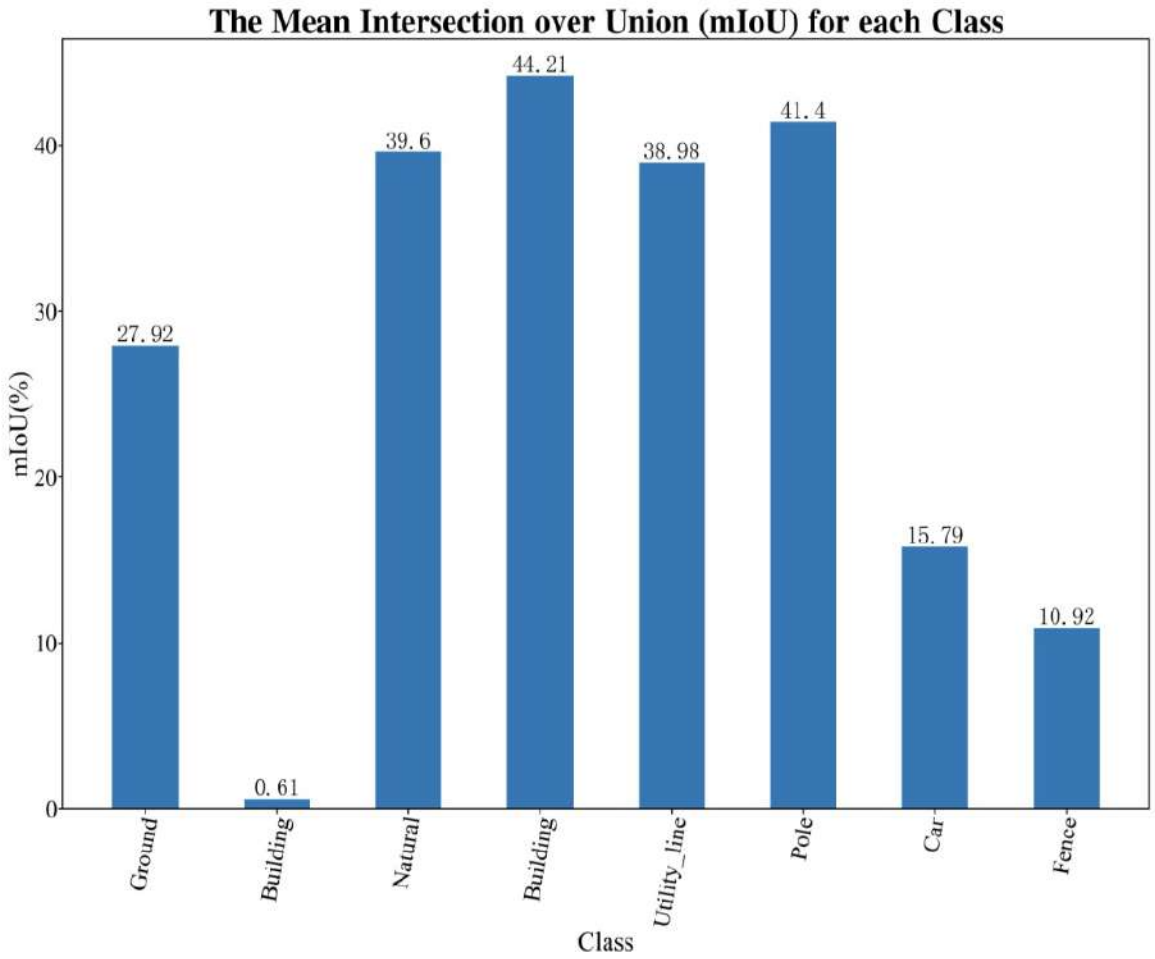
Class	Epoch	Recall(%)	Recall for each Class(%)							
			Ground	Building	Natural	Building	Utility_line	Pole	Car	Fence
Ground	18	0.41	34.36	0.86	68.21	69.90	50.99	53.75	26.55	27.29
Building	9	0.35	6.57	21.33	49.76	10.00	71.26	64.31	2.86	57.24
Natural	16	0.42	34.03	0.86	73.22	66.32	59.72	52.14	25.40	20.97
Building	15	0.35	30.51	0.97	69.52	70.71	47.72	47.91	11.40	4.25
Utility_line	9	0.35	6.57	21.33	49.76	10.00	71.26	64.31	2.86	57.24
Pole	7	0.28	4.62	2.68	20.85	25.01	70.19	79.79	17.70	3.50
Car	13	0.44	33.24	7.83	61.62	53.31	55.69	55.62	62.63	20.16
Fence	9	0.35	6.57	21.33	49.76	10.00	71.26	64.31	2.86	57.24

12.**The F1 and Epoch of each Class**：This table indicates the epoch at which the maximum F1 score occurs for each class during the validation process, along with the F1 score value for each class. As shown in the figure：

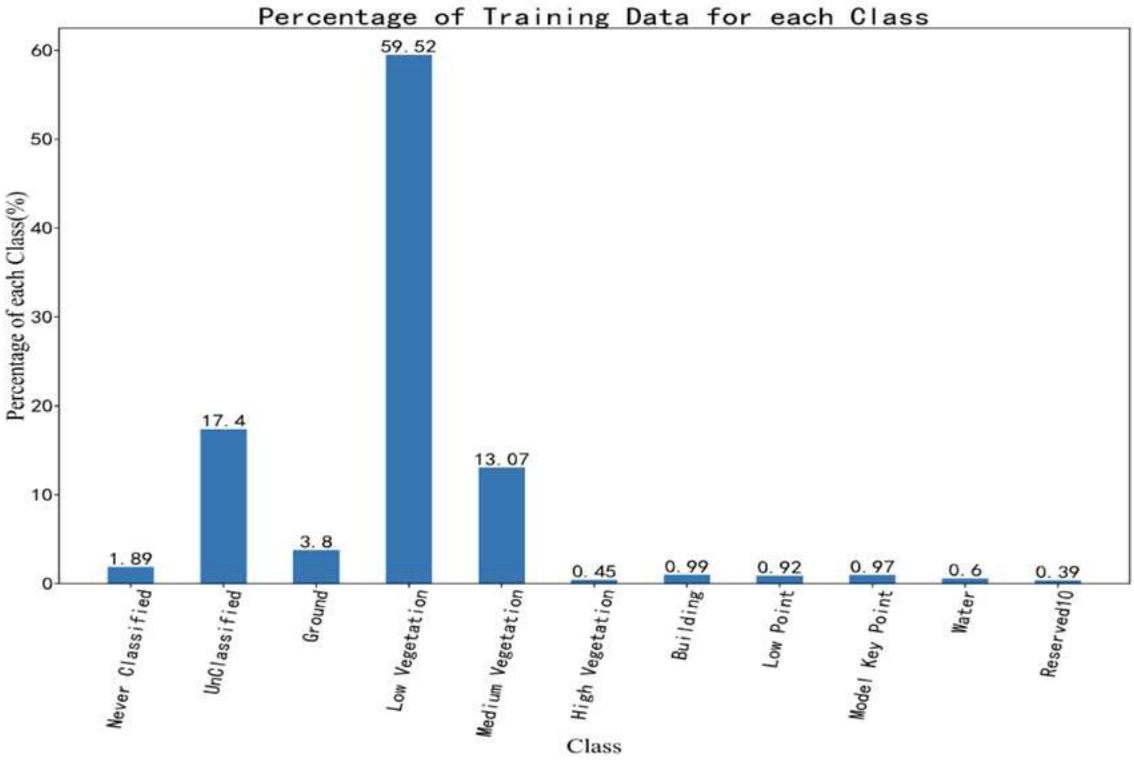
The Max F1 and Epoch of Each Class

Class	Epoch	F1	F1 for each Class							
			Ground	Building	Natural	Building	Utility_line	Pole	Car	Fence
Ground	16	0.41	0.44	0.01	0.57	0.61	0.56	0.59	0.27	0.20
Building	9	0.16	0.12	0.07	0.56	0.08	0.08	0.20	0.04	0.10
Natural	19	0.35	0.42	0.01	0.76	0.32	0.52	0.60	0.14	0.05
Building	13	0.40	0.42	0.04	0.51	0.62	0.55	0.59	0.38	0.10
Utility_line	15	0.35	0.41	0.01	0.72	0.38	0.58	0.59	0.10	0.04
Pole	19	0.35	0.42	0.01	0.76	0.32	0.52	0.60	0.14	0.05
Car	13	0.40	0.42	0.04	0.51	0.62	0.55	0.59	0.38	0.10
Fence	16	0.41	0.44	0.01	0.57	0.61	0.56	0.59	0.27	0.20

13.**The Mean Intersection over Union (mIoU) for each Class**：This graph represents the IoU for each class when the maximum mIoU is achieved. The x-axis represents the class, and the y-axis represents the mIoU value. The closer the value is to 100, the more accurate it is. As shown in the figure:



14.**Percentage of Training Data for each Class**：This graph represents the percentage of each class in the total number of training points. The x-axis represents the class, and the y-axis represents the percentage. Ideally, when all classes are evenly distributed, the training results are the best. Therefore, it is advisable to ensure class balance to improve classification accuracy. As shown in the figure:


Percentage of Training Data for each Class
Percentage of each Class(%)
60
50
40
30
20
10
0
59.52
17.4
13.07
1.89
3.8
0.45
0.99
0.92
0.97
0.6
0.39
Never Classified
UnClassified
Ground
Low Vegetation
Medium Vegetation
High Vegetation
Building
Low Point
Model Key Point
Water
Reserved10
Class

Profile

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

- Select Profile Region
  - Fixed Buffer
- Measure Tools
  - Move Selected Section
  - Rotate Selected Section
  - Expand Selected Section
  - Switch View
  - Manual Classification

Profile and Measuring Tool

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

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

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

Profile and Select Tool

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

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

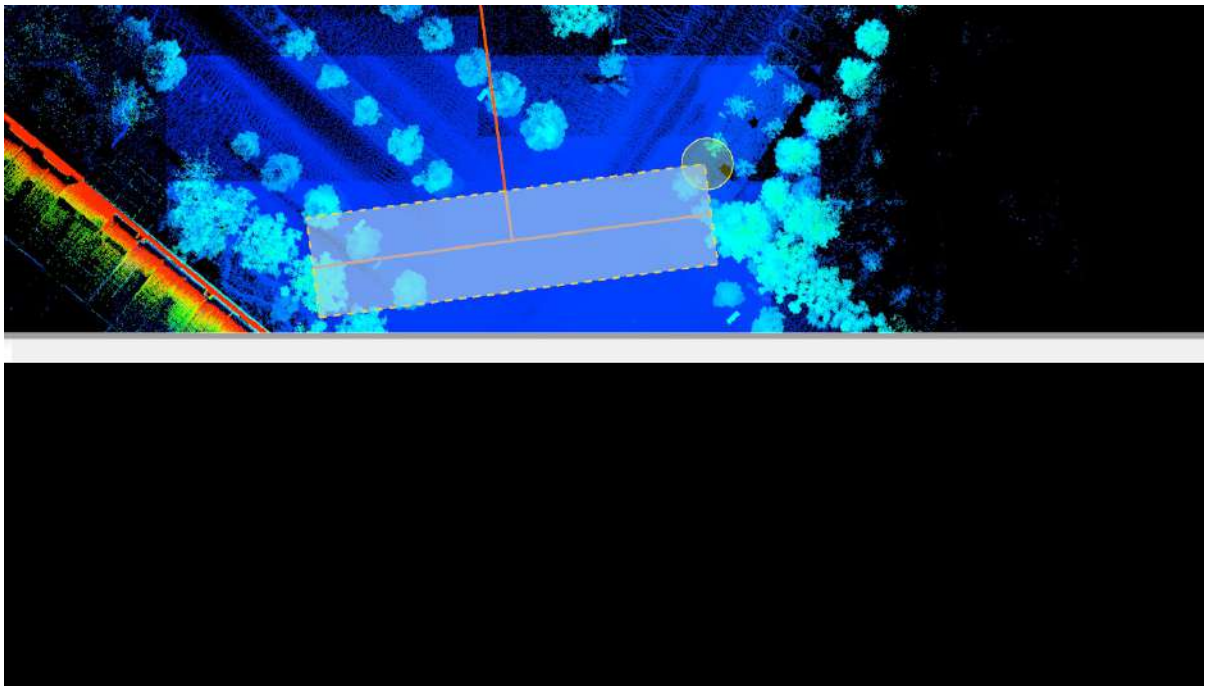
Select Profile Region

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

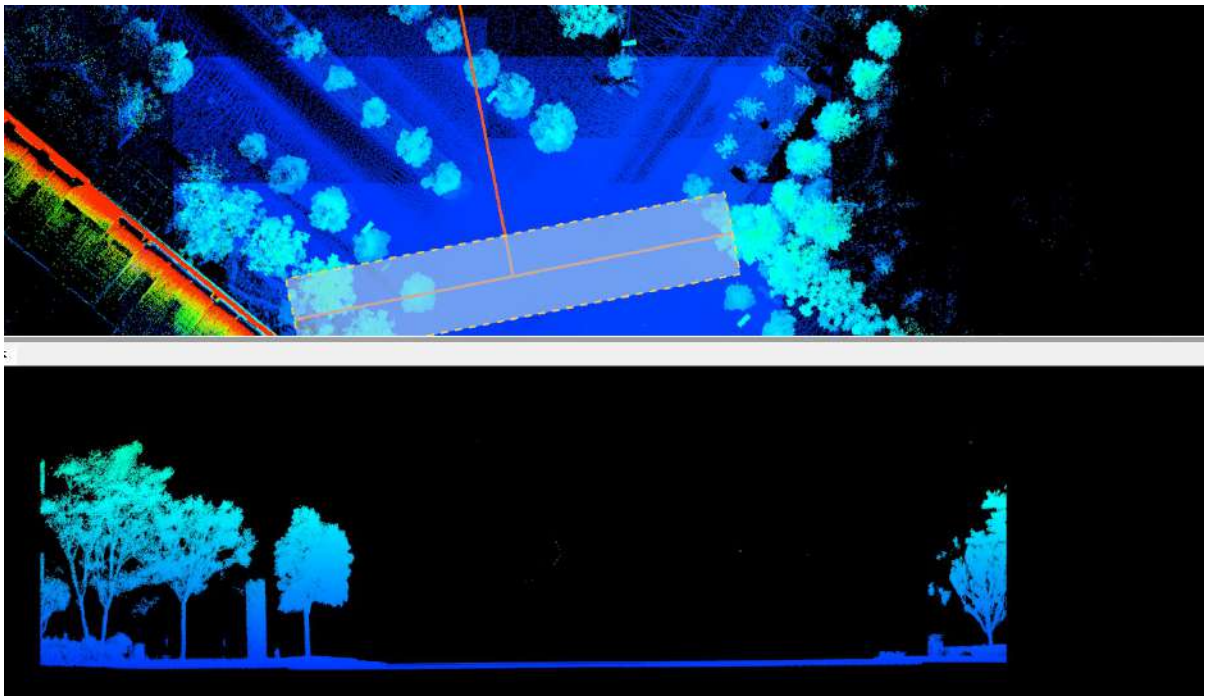
Steps

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

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



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



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

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

Fixed Buffer

Move Selected Section

Rotate Selected Section

Expand Selected Section

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

Fixed Buffer

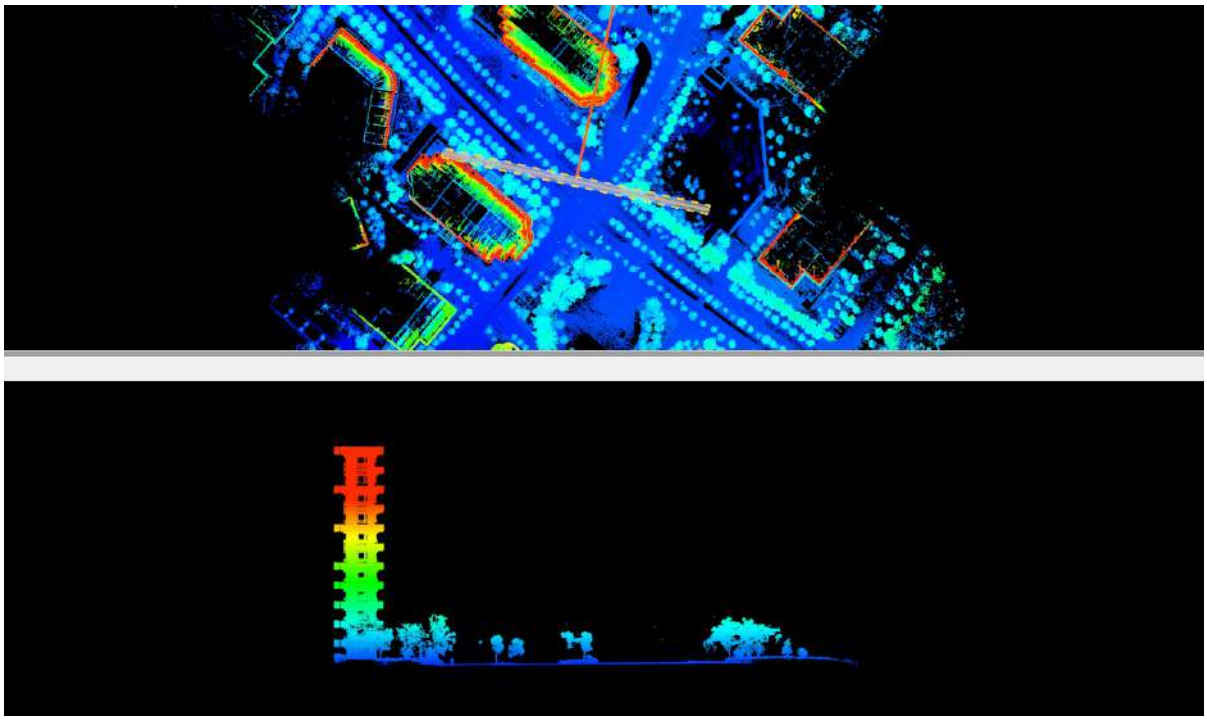
Function Description: Fixed rectangular width is supported to use.This function can help users to fix the size of the buffer area.

Steps

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

2.Click the **Buffer** button.

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



Measure Tools

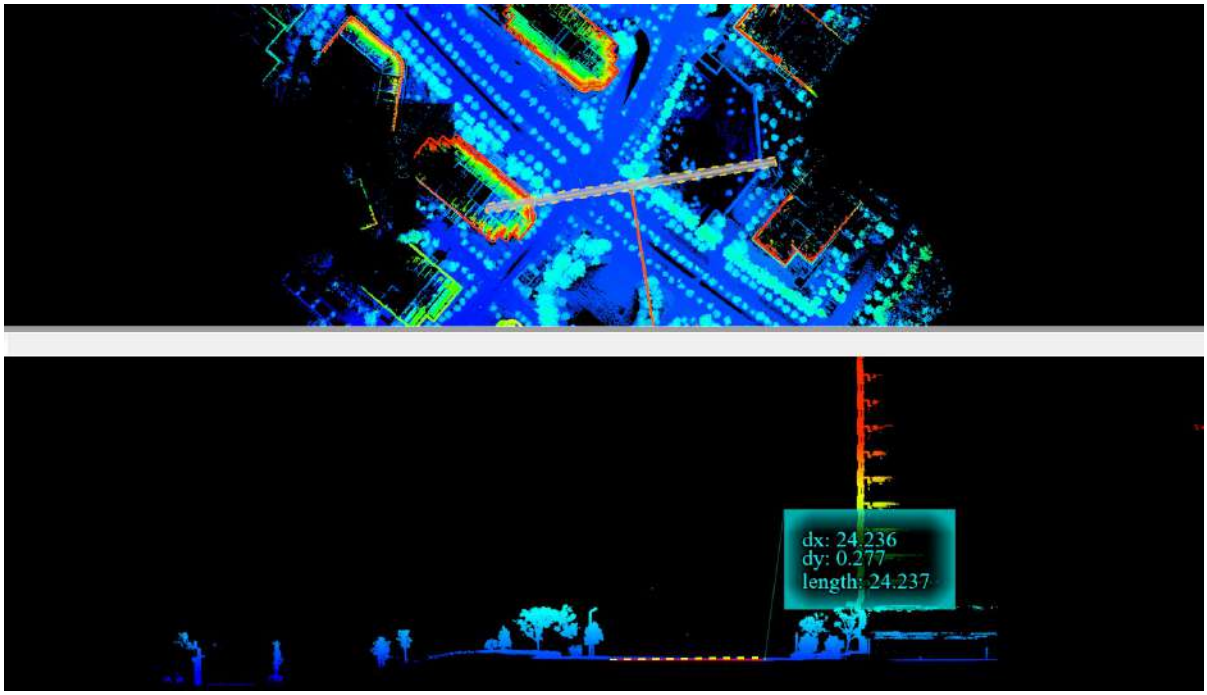
Function Description：Orthogonal projection is as default for profile window, and is convenient to calculate the horizontal and vertical distance.

Steps

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

2.Left-click to select the first point.

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



Move Selected Section

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

Steps

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

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

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

Rotate Selected Section

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

Steps

1.Click

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

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

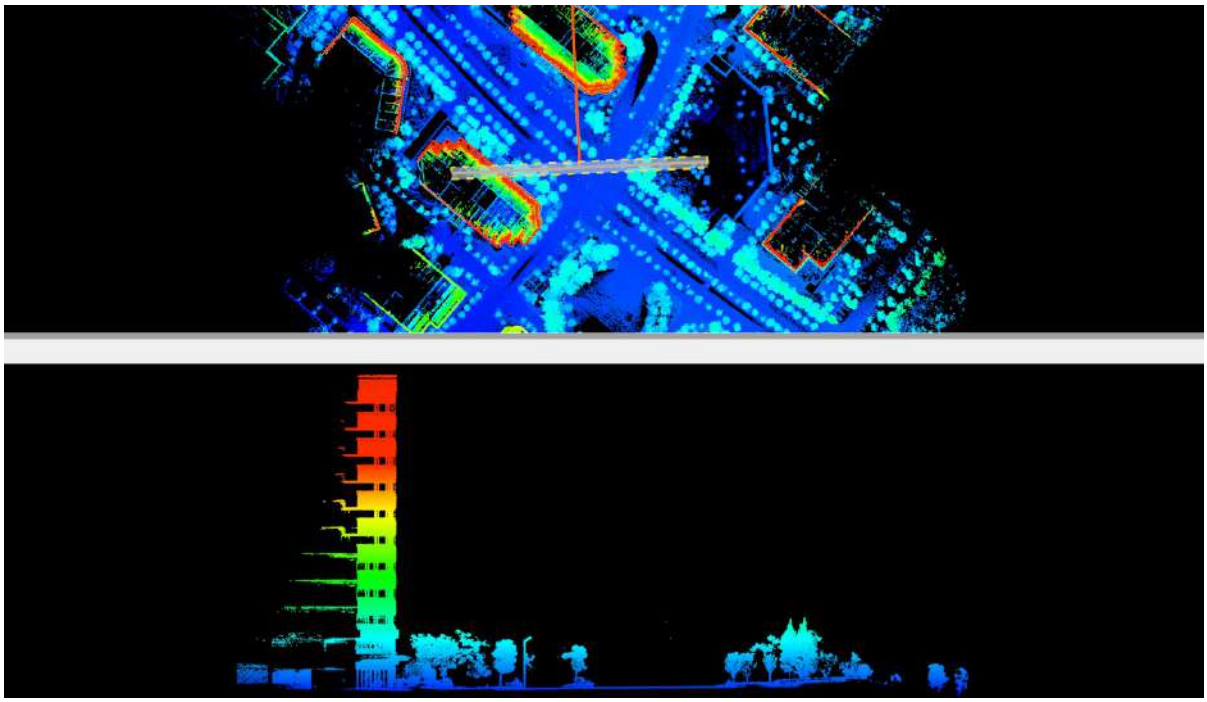
Expand Selected Section

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

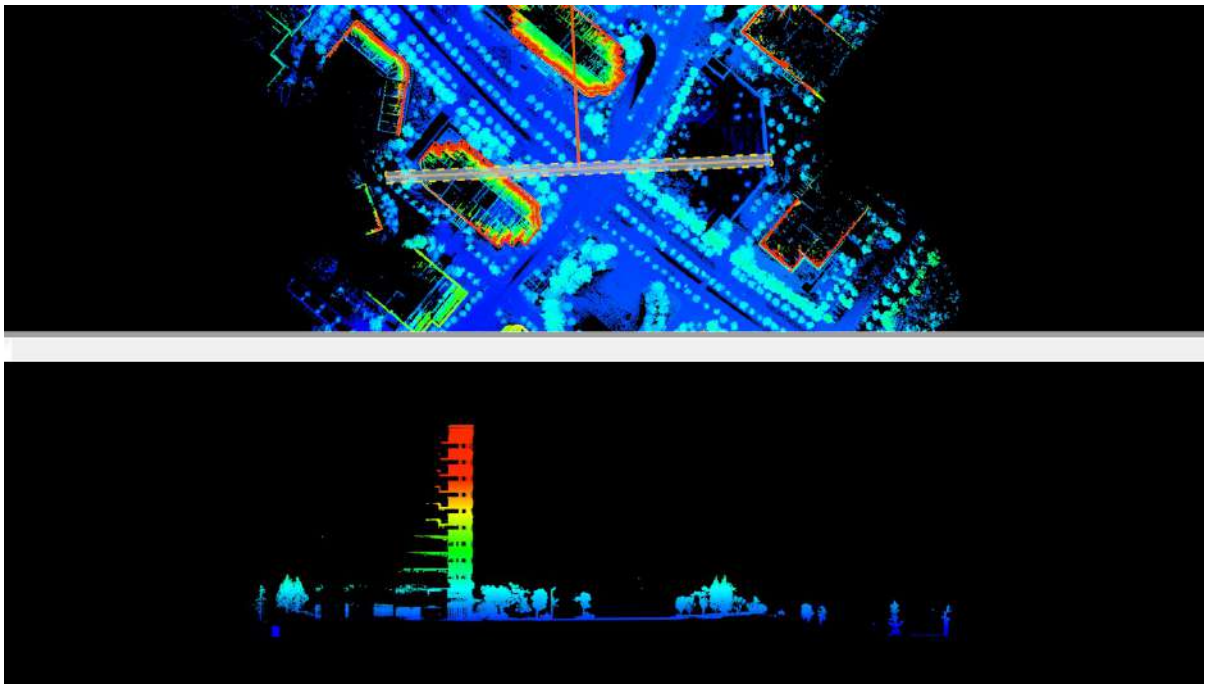
Steps

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

Before expanding：



After expanding:



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

Switch View

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

Steps

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

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

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

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

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

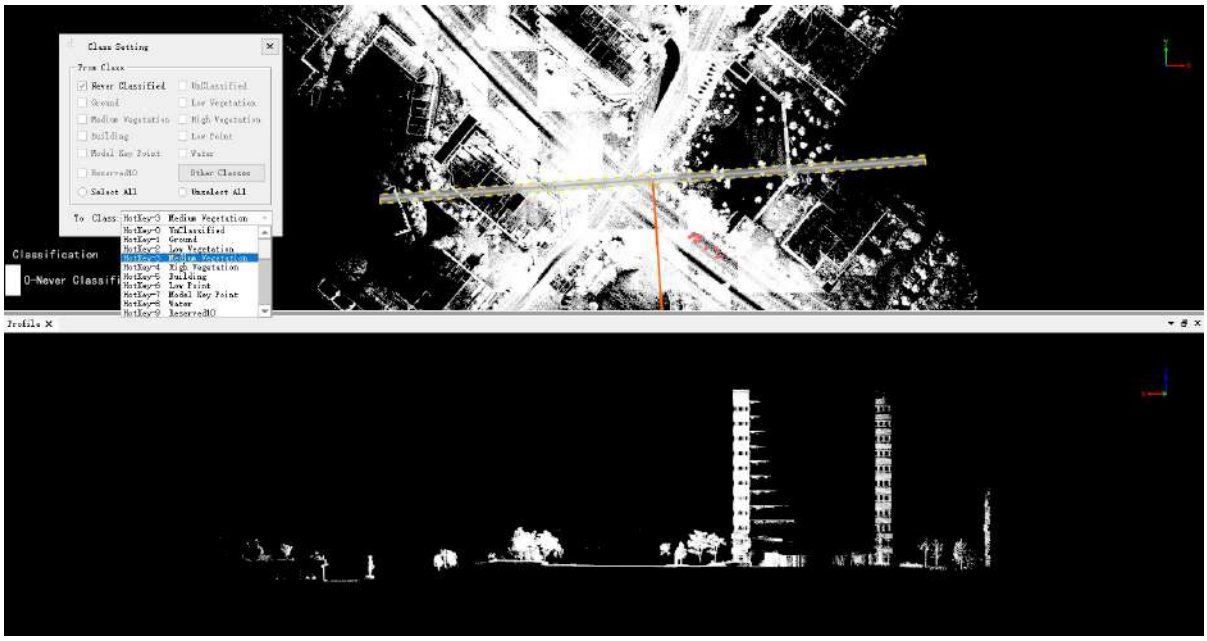
6.(optional) Click button to start viewing at any angle. By default, the section scene cannot be rotated. After activating this button, you can observe the section point cloud at any angle.

Manual Classification

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

Steps

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



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

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

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

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

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

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

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

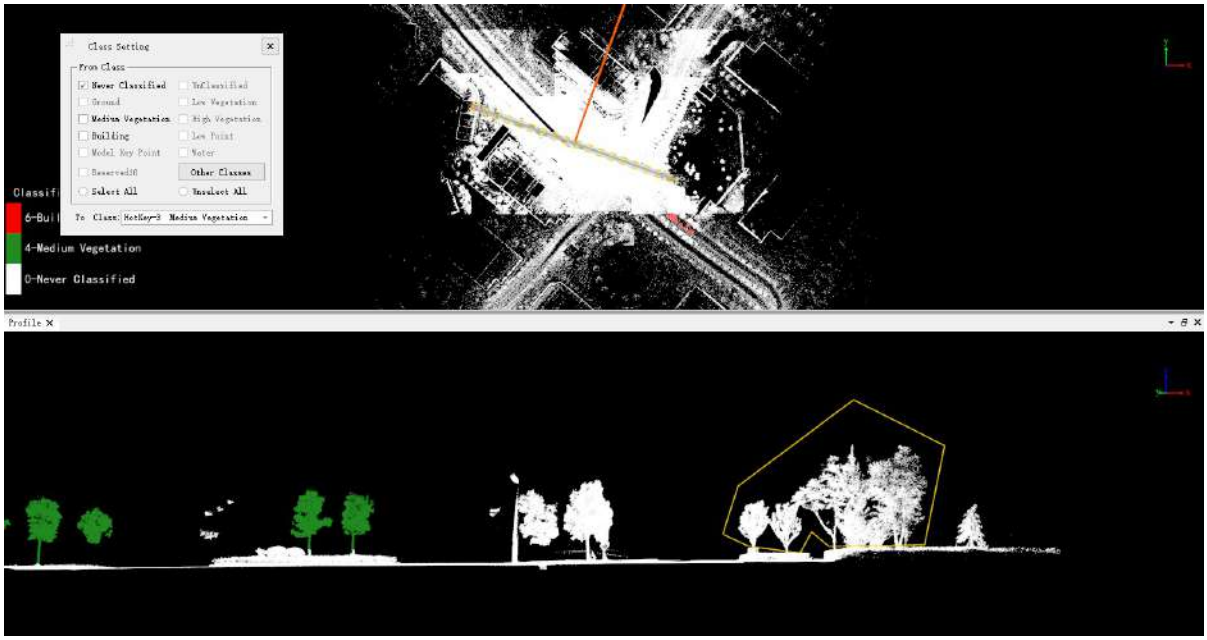
3.Adjust the display mode to display by category, and you can clearly see the point cloud selected in the previous step.

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

5.Click to save the profile.

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

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



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

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

Panorama Measure Tools

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

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

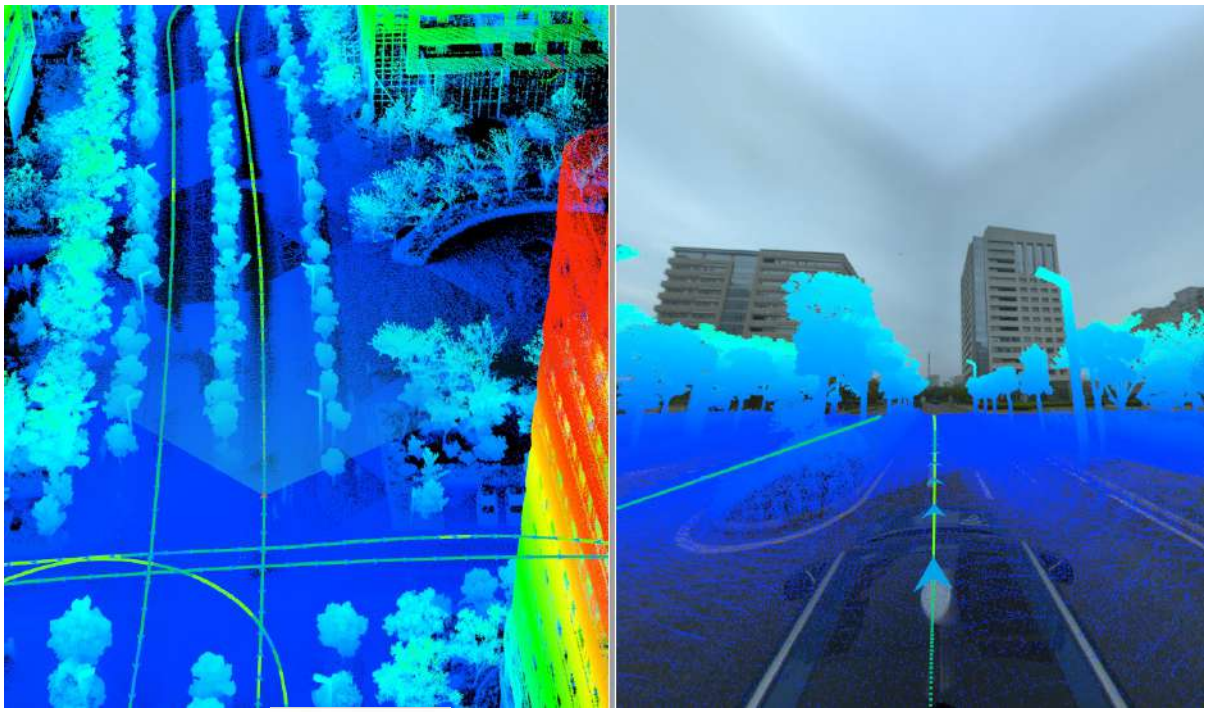
- Point Cloud and Panoramic Image Roaming
- Panorama Measurements Based on Point Cloud DepthInterpolation
- Panorama Measurements Based on Forward Intersection
- Result Panel
- Setting Dialog
- Update EXIF
- Blur Plates And Face

Point Cloud and Panoramic Image Roaming

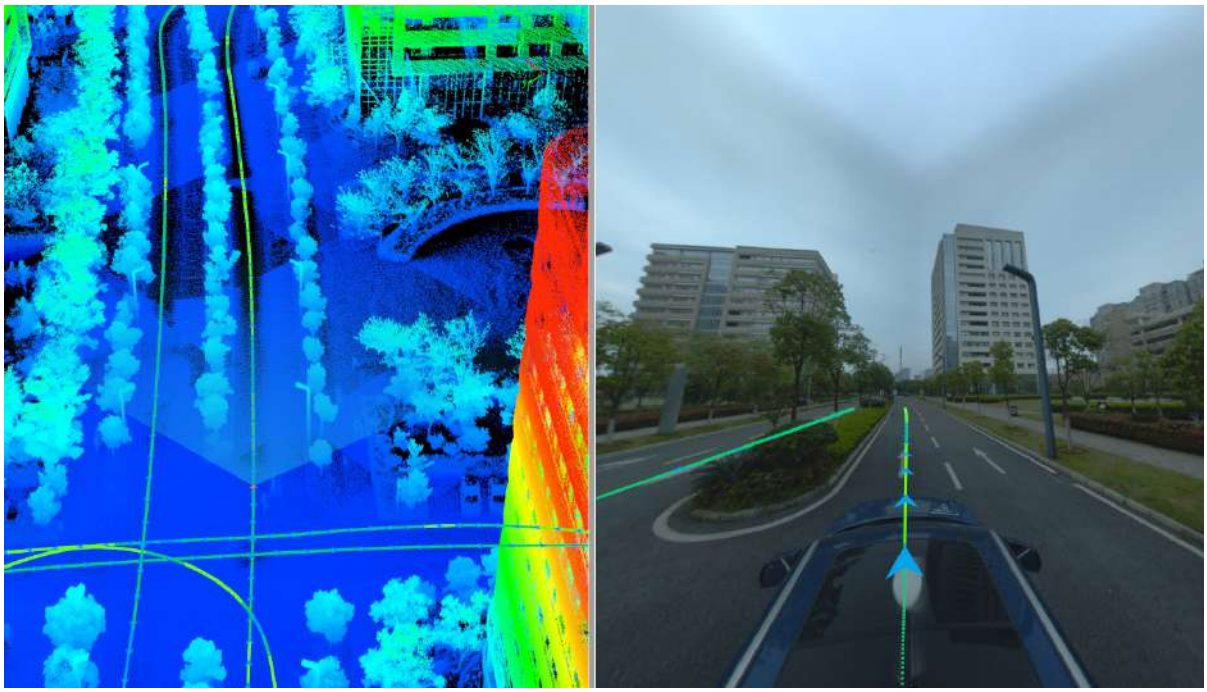
Function Description：The software supports point cloud and panoramic image roaming. The original camera file must be added when creating a new project before the menu bar will be activated.

Steps

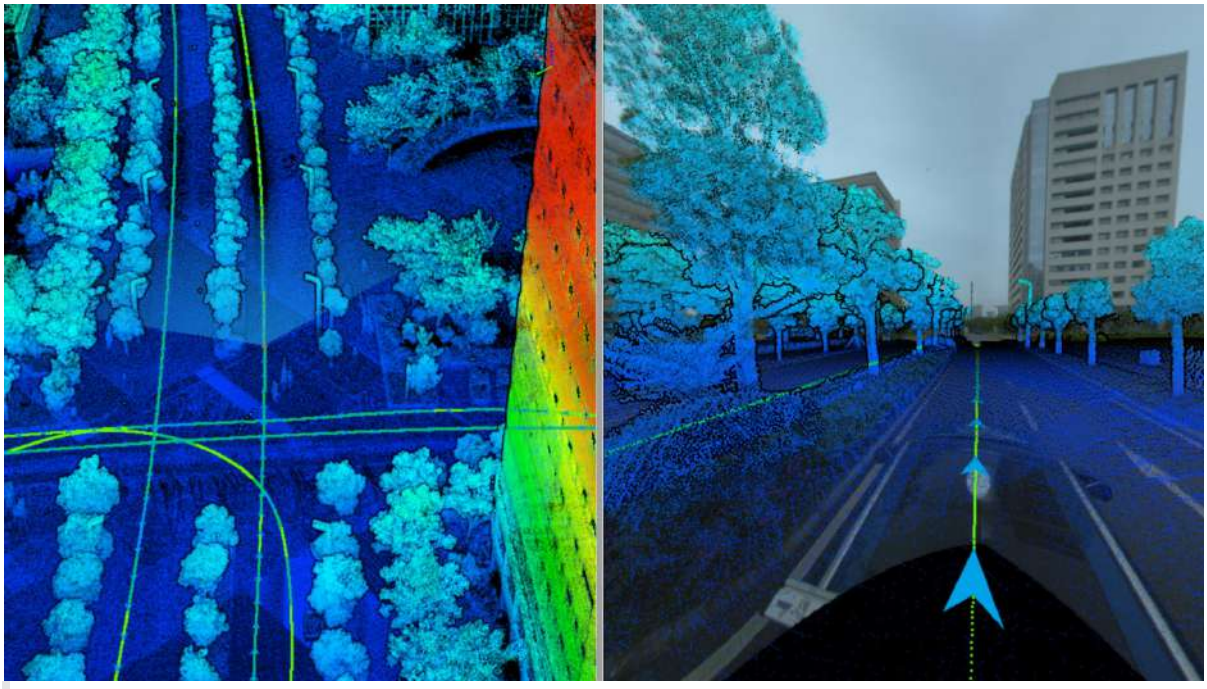
1.After a new project is created, a 3D window and a panoramic window will be opened. The point cloud data will be displayed in the 3D window and the panoramic window at the same time, and the panoramic image will only be displayed in the panoramic window. You can choose to show or hide the corresponding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



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



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

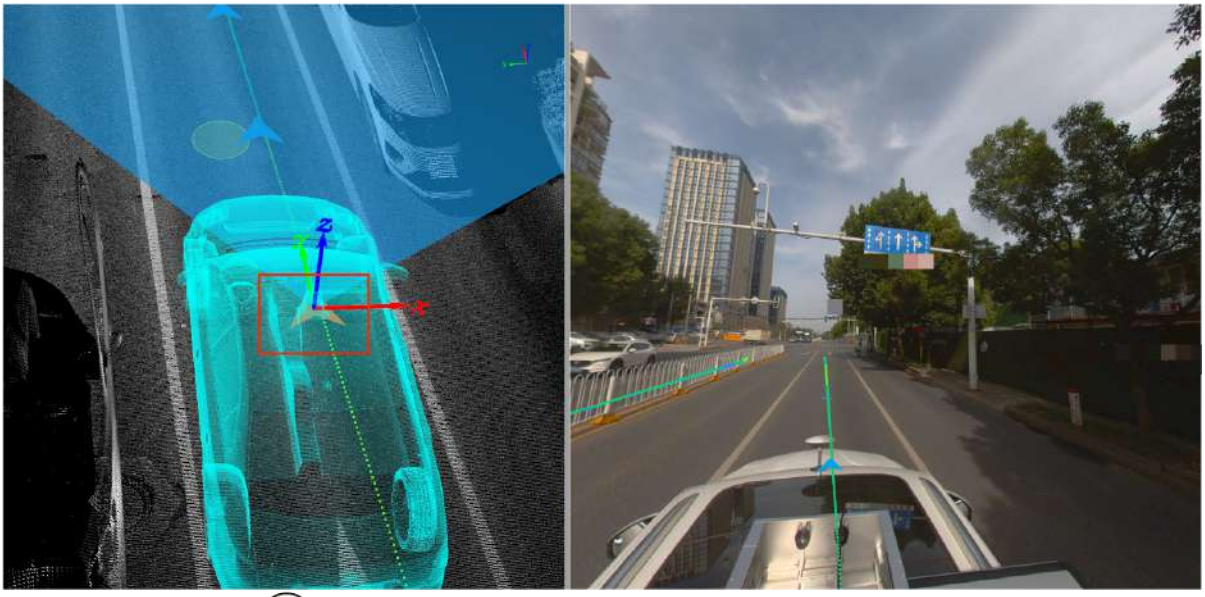


Note:

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

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

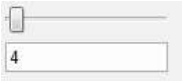
4.Click the **Select Frame** button to select the image exposure position on the trajectory in the 3D window or the panoramic window (the default is the blue triangle arrow symbol, and the orange after clicking) to jump to the selected image position.



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

> Note: Shortcut of next frame and previous frame:

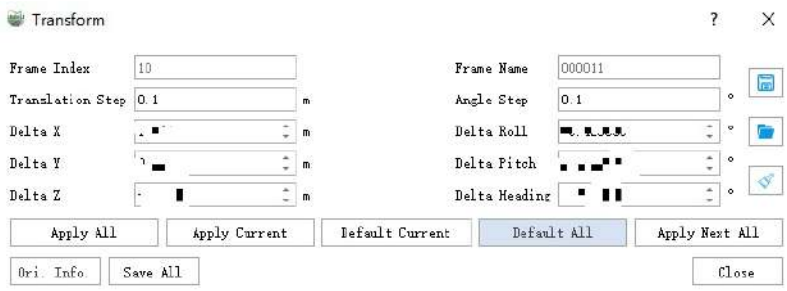
Shortcut key	Function
PageUp	Next Frame
PageDown	Previous Frame

6.Drag the slider 

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

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

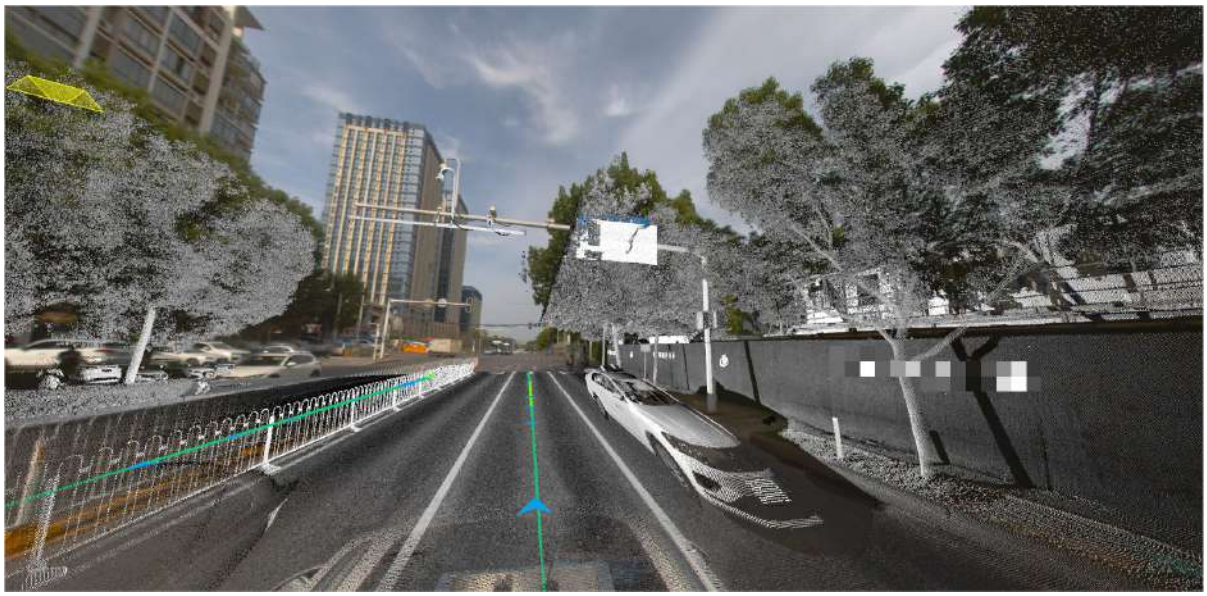
8.Click **Transform** button, the transformation is mainly to adjust the pose of the image in real time and superimpose the point cloud perfectly.



Description

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

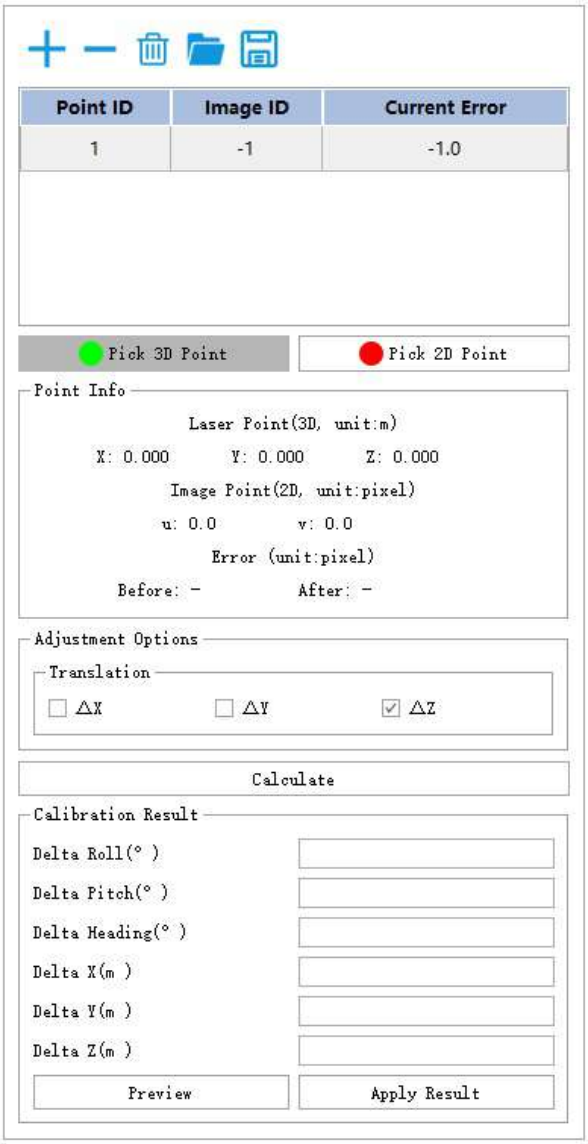
9.Due to the placement error between the panoramic image camera and the laser, there is still a certain level of deviation between the plane image and the point cloud data, which cannot be superimposed perfectly, as shown in the following figure:



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

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

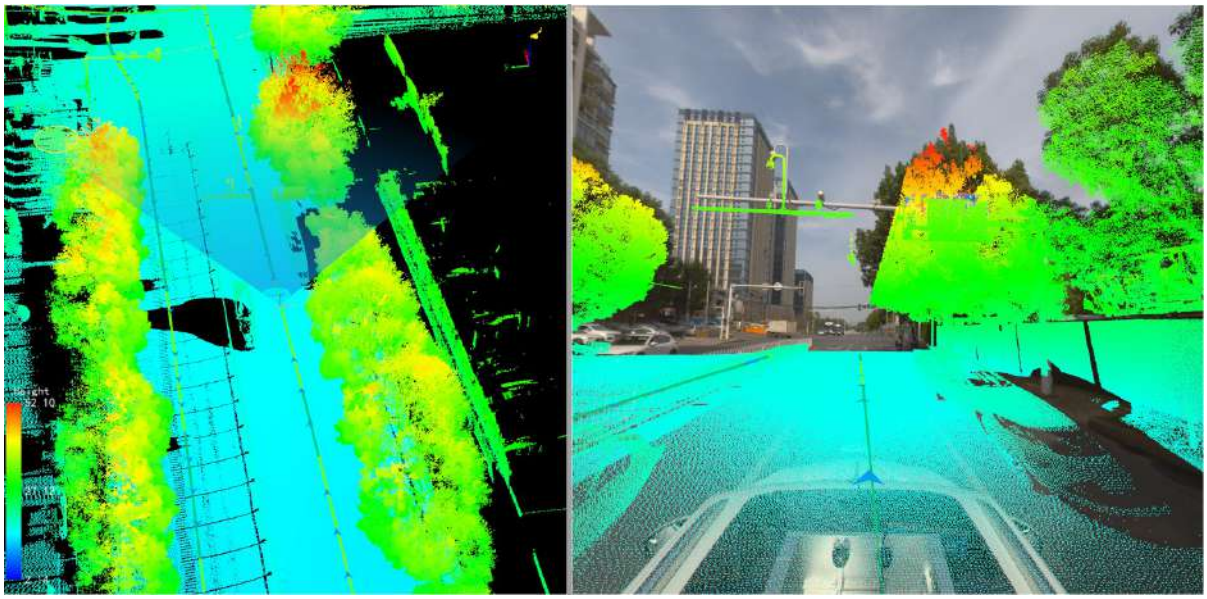
1.Switch to the menu page, and click **Calibration** button, the calibration window will pop up on the right side of the user interface.



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

The specific steps are:

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

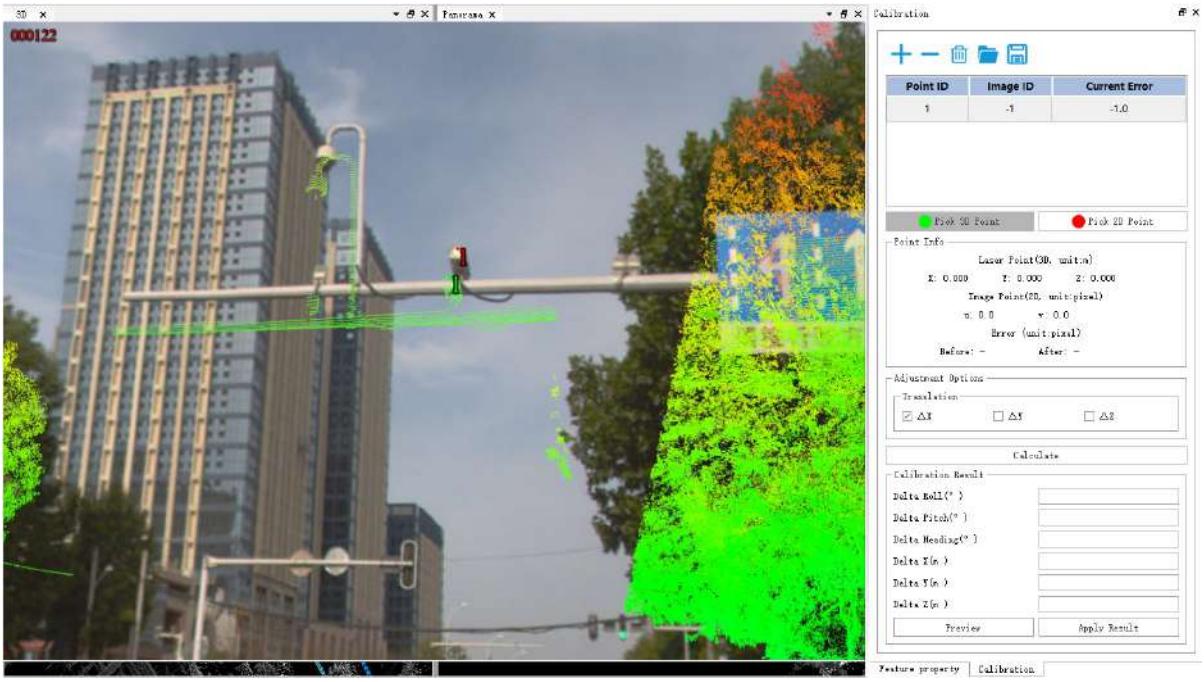


- Click the *Add Point* button at the top of the calibration window to add a row to the point list.
- Click the *Pick 2D Points* button to select image points in the image window. The top of the picture is a magnifying glass, which can enlarge the position of the mouse.

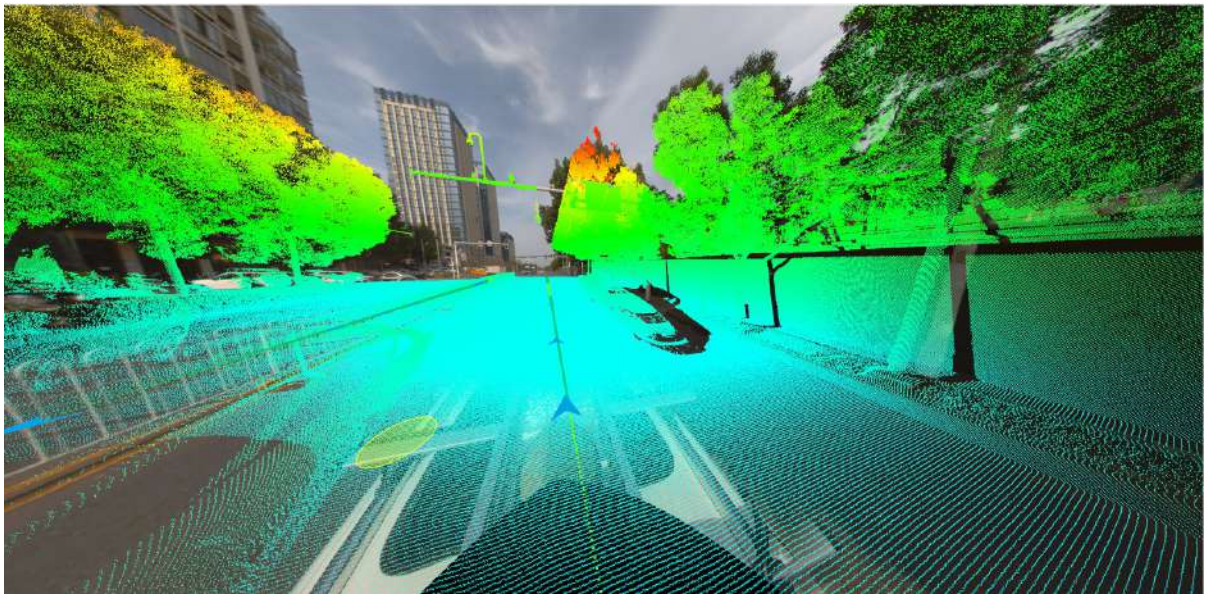
- Click the *Pick 3D Point* button to select the laser point corresponding to the image point in the image window.
- Repeat the above three steps until enough point pairs are selected on the current image frame (four point pairs are recommended).
- Select the corresponding image frames in the other three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

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

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



After preview:



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

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

Panorama Measurements Based on Point Cloud Depth Interpolation

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

The main functions are listed below:

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

Pick Point (Depth Interpolation)

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

Steps

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

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

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

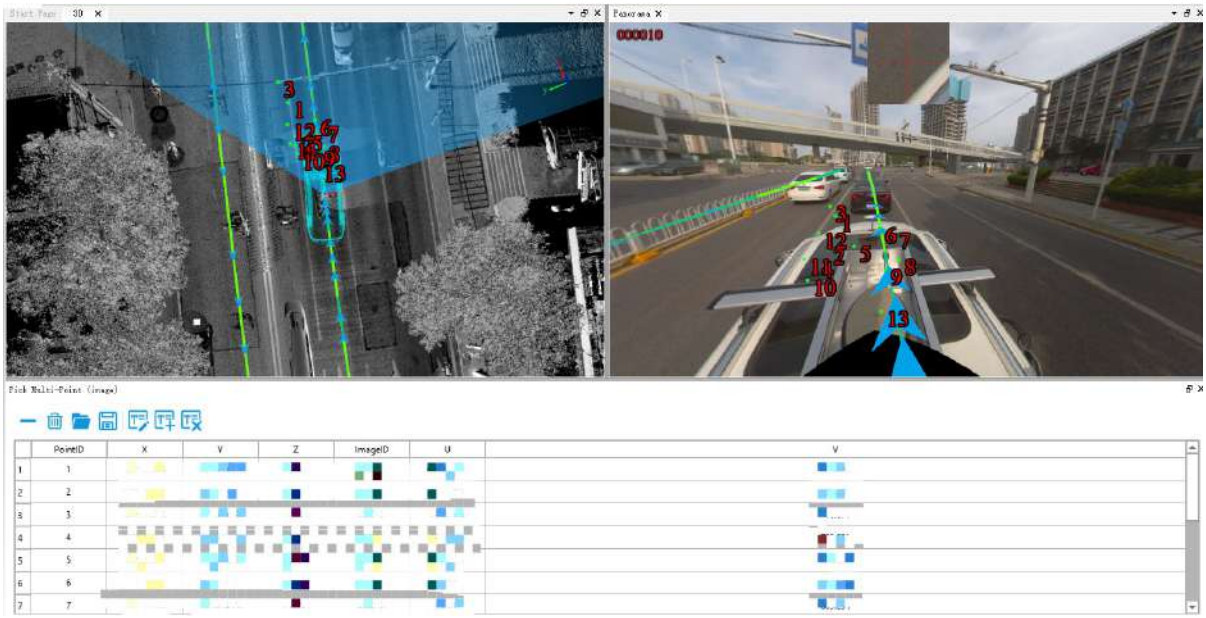
Multi Pick Point (Depth Interpolation)

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

Steps

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

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

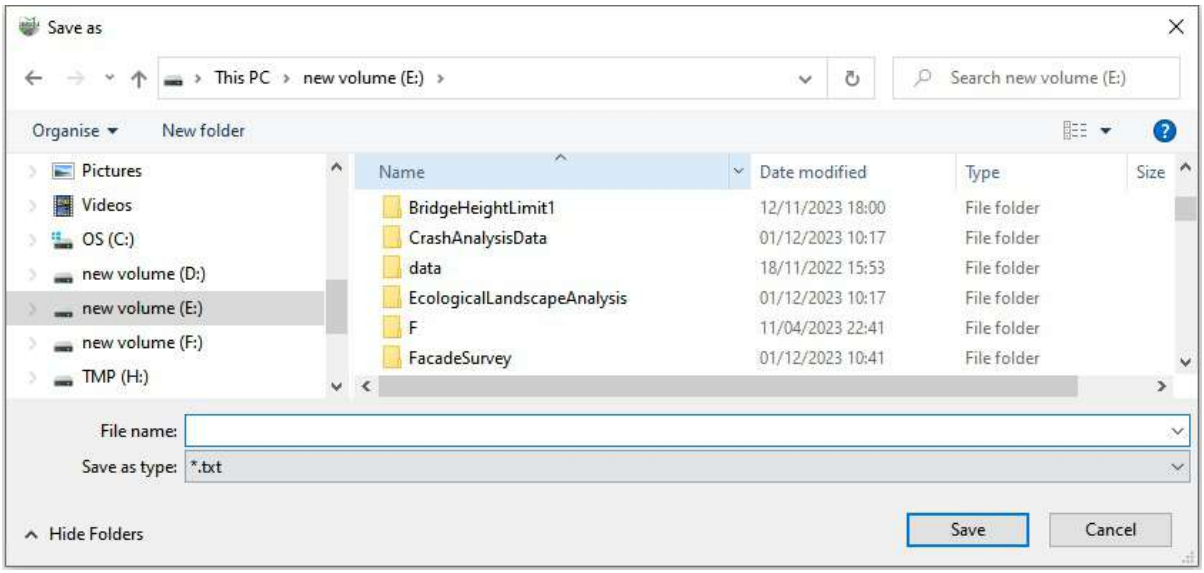


Parameter settings

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

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

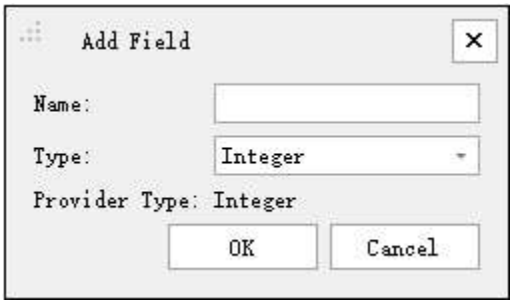
3.The selected point set is exported in *.txt* format. Click the button to save the coordinates and other attribute information as a *.txt* file.



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

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

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



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

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

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

Length Measurement (Depth Interpolation)

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

Steps

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

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

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

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

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

Height Measurement (Depth Interpolation)

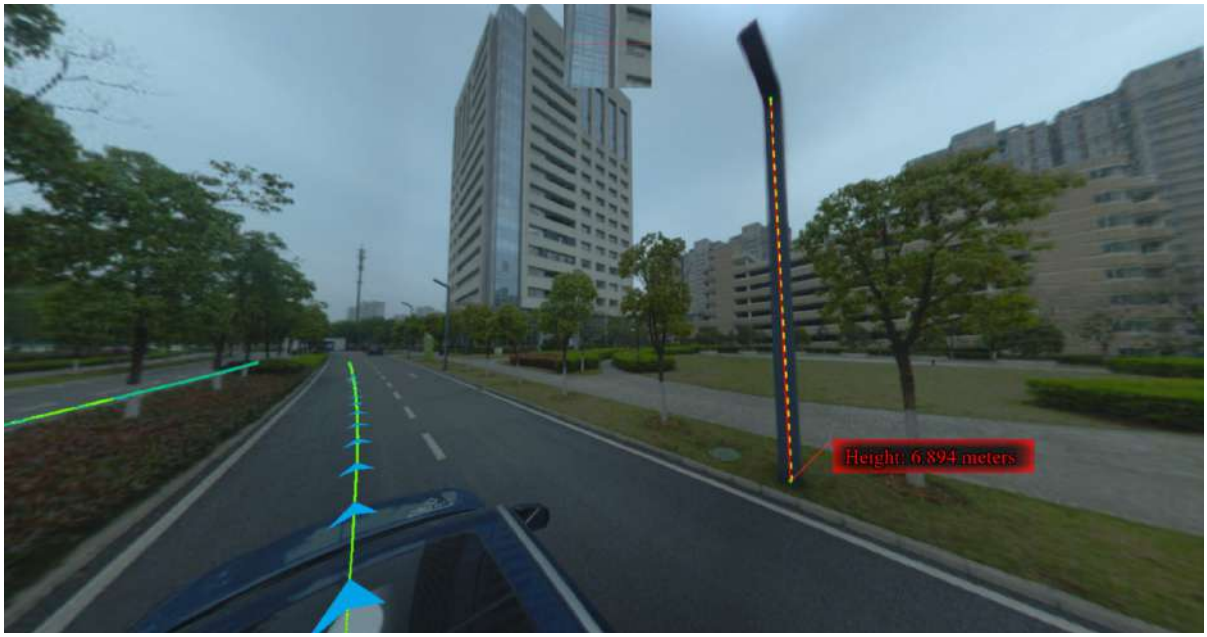
Function Description：This tool is applicable to point cloud data and works on panorama data. It allows the user to calculate the relative height difference between two points in panorama window.

Steps

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

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

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



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

Angle Measurement (Depth Interpolation)

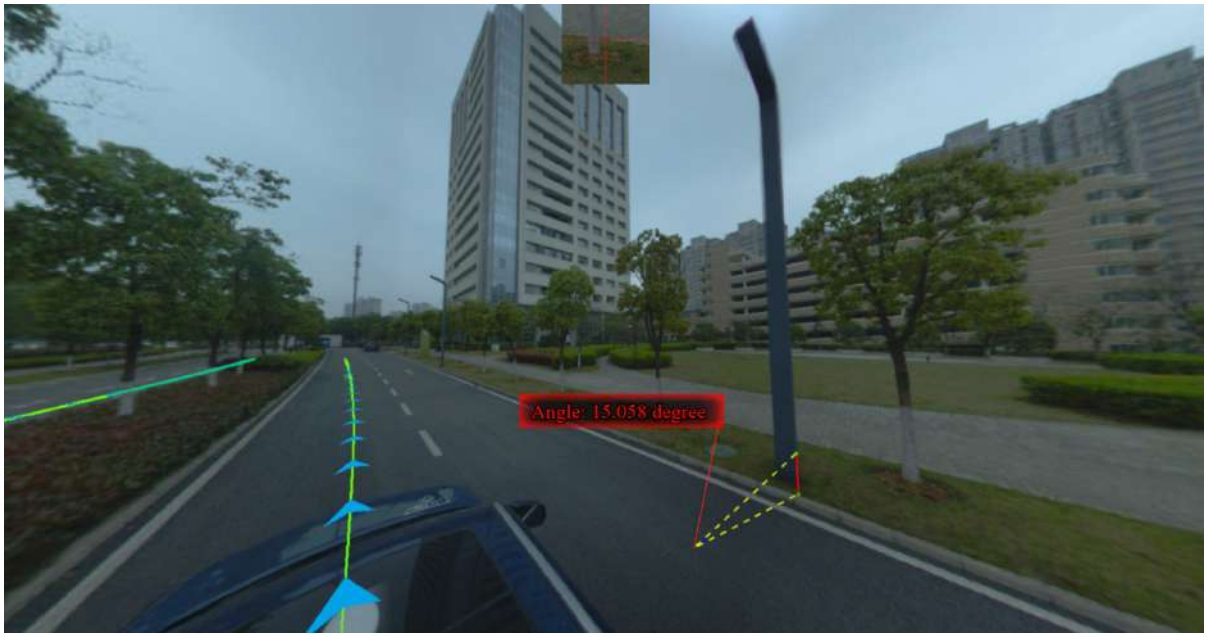
Function Description：The Angle measurement tool works on panorama data and allows the user to select the measurement point, and calculates the angle of pitch between two points in Panorama view.

Steps

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

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

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



Panorama Measurements Based on Forward Intersection

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

The main functions are listed below

- Pick Point
- Length Measurement

Pick Point (Forward Intersection)

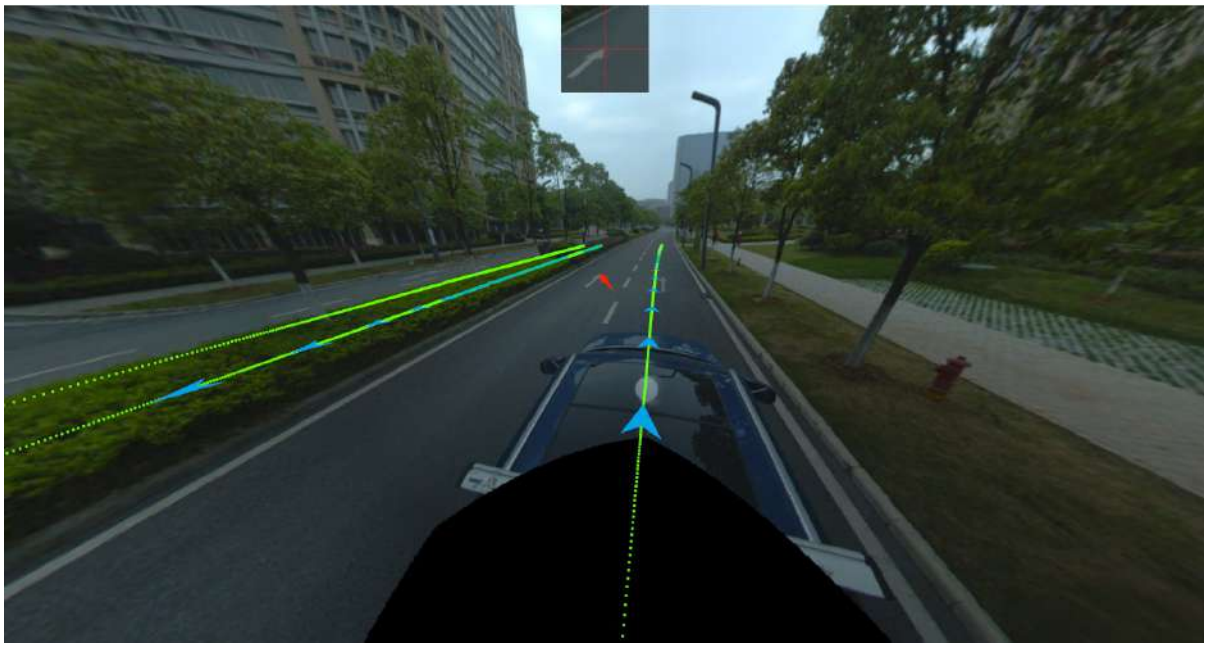
Function Description: This tool is applicable to panoramic data and allows the user to query attributes that contain position information.

Steps

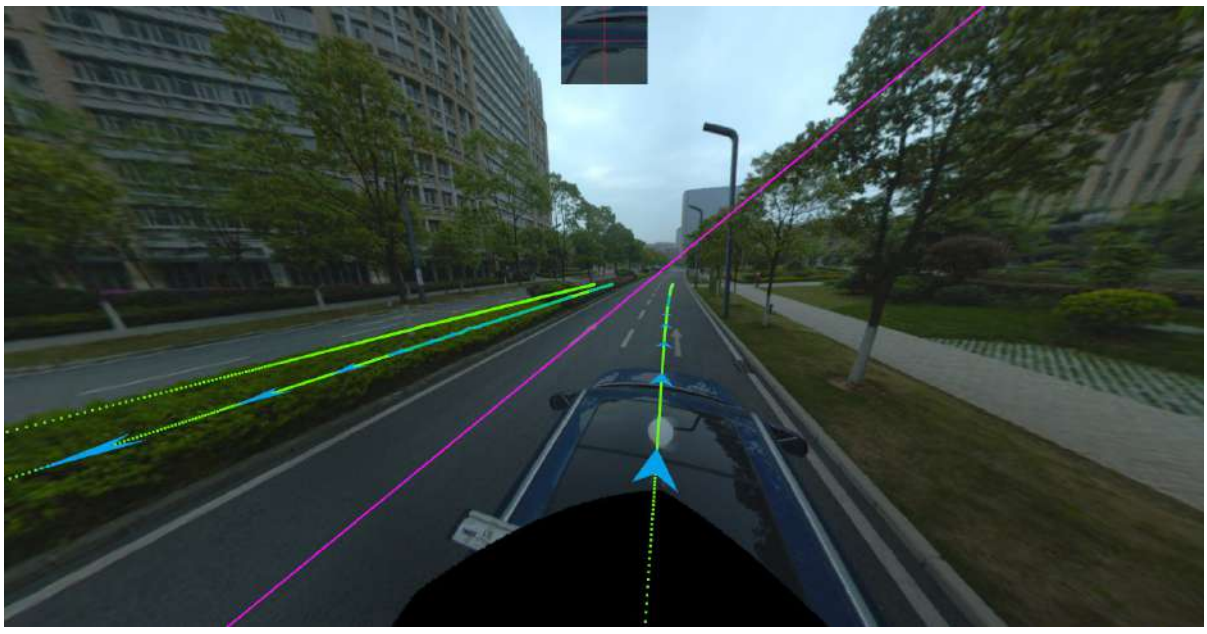
1.Click Pick Point(stereo) button via Panorama Measurement mode.

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

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

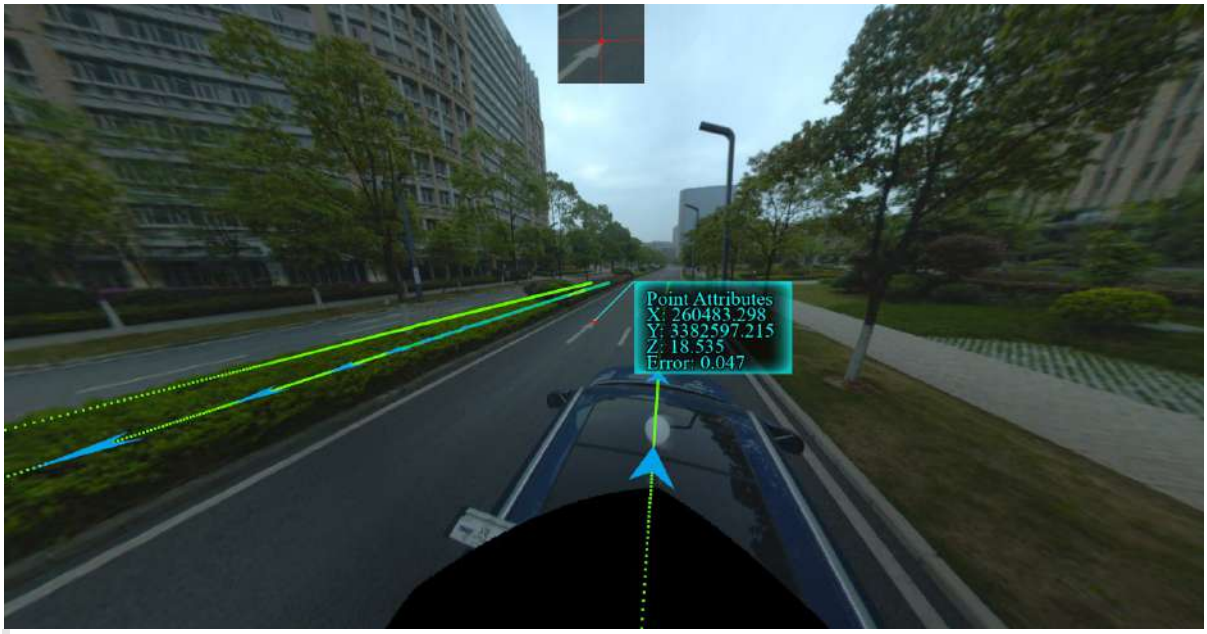


The panorama window will switch to the second frame image automatically and the auxiliary line will display：



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

Select the corresponding point on the second frame image:



Note:

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

Length Measurement (Forward Intersection)

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

Steps

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

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

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

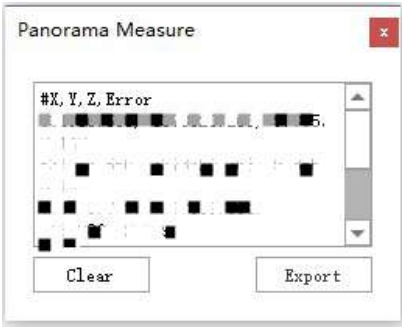
4.Right-click to go back to the previous point during the measurement.

Result Panel

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

Steps

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



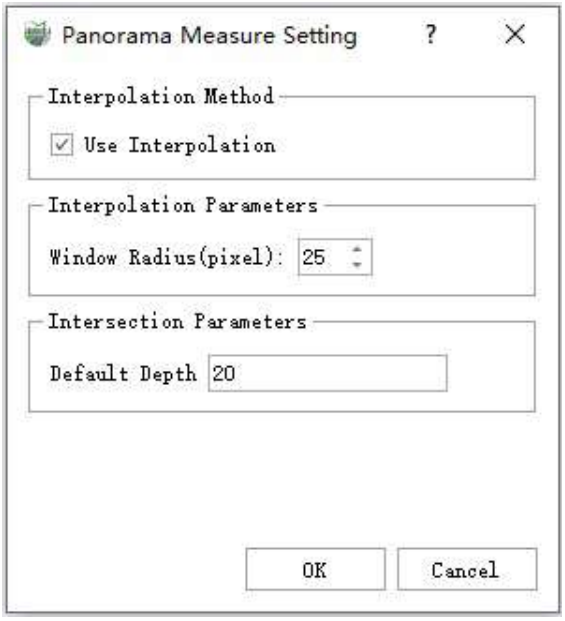
2.Click Clear button to clear Result Panel.

3.Click Export button to export measured points list.

Dialog Setting

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

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



Parameters Description

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

Update EXIF

Function Description：Add latitude and longitude information to the images in the current project's panoramic image list.

Steps

1.Click the **Update EXIF** EXIF button to open the dialog box

2.If you select "OK," latitude and longitude information will be directly added to the images in the image list of the current project.

3.If you select "Cancel," the EXIF export function will be exited.

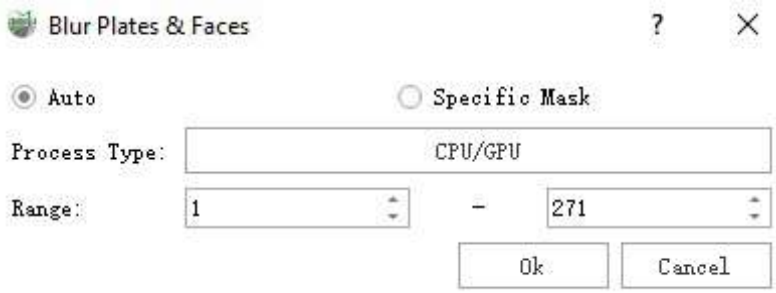
> Note: In step 2, the latitude and longitude coordinates of the photos are derived from the absolute coordinate projection in the image list file. If the coordinate system of the current project is undefined or defined incorrectly, it may lead to export failure or incorrect export results.

Blur Plates And Face

Function Description：De-privatise images within the current project image list, such as licence plates and faces.

Steps

1.Click the **Blur Plates And Face** button to bring up a dialogue box:

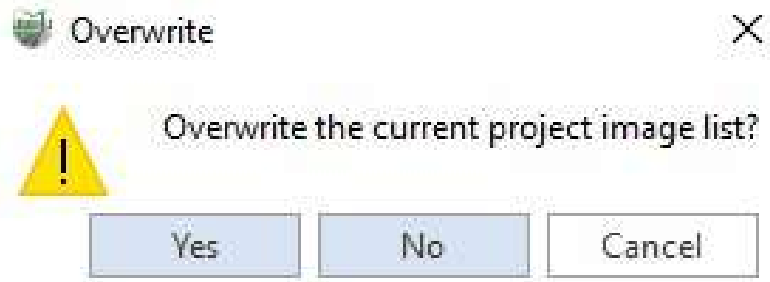


Parameter descriptions

- **Processing method**：There are two ways of processing, automatic means AI-based way to fully automatic image de-privacy, de-privacy location for the face and licence plate; designated area means you can enter a mask image, all images will be mask range of all the private
- **Processing Type**：This option can be set in the automatic mode, it refers to the use of GPU or CPU processing, when the computer performance is better, it is recommended to use the GPU mode
- **Mask**：This option can be set in the specified way, you need to enter an external mask image, the general mask image should be the same as the resolution and size of the image to be processed, the position of the mask needs to be set to black, do not need to be set to the position of the mask is set to white, as follows

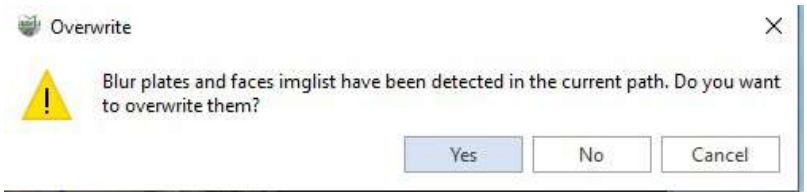
- **Range**：This option can be set in both ways to specify the number of pictures to be processed automatically. The value box on the left side of the window is the starting picture number and the value box on the right side is the ending picture number, and the default range is from the first to the last picture.

2.After selecting the range of images that need to be de-privatized in the pop-up dialogue box, click OK to select whether to overwrite the images within the current project:



3.If you select "Yes", the image columns in the current project will be overwritten directly, otherwise the software will automatically create an image list file of the same name with the suffix "_Blur" within the current image file, and a new folder for storing the image pictures will be created in the same naming The folder where the images are stored will also be created with the same naming.

4.If you select "No", i.e. not to overwrite the images in the current project, if the current image list has already been processed by the de-privacy function, the software will prompt you whether to overwrite it or not, if you need to overwrite it, the image list that has been processed by the de-privacy function in the last time will be overwritten.



Planar Image

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

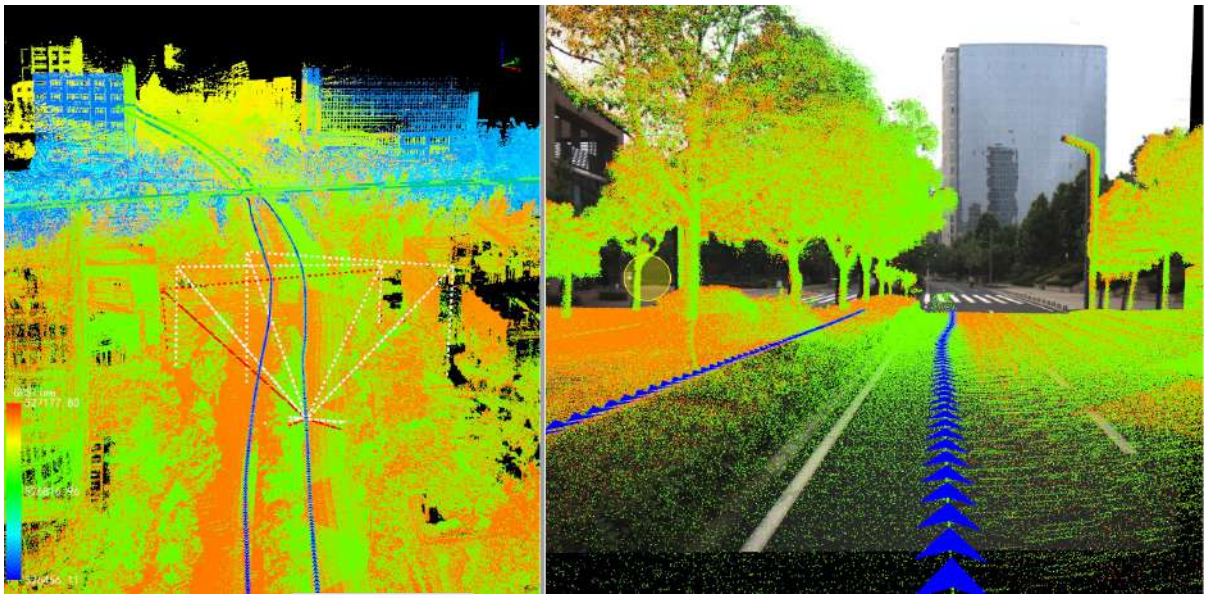
- Point Cloud and Image Roaming
- Image Measurement
- Image Undistorting
- Setting Dialog
- Update EXIF
- BlurPlates And Face

Point Cloud and Image Roaming

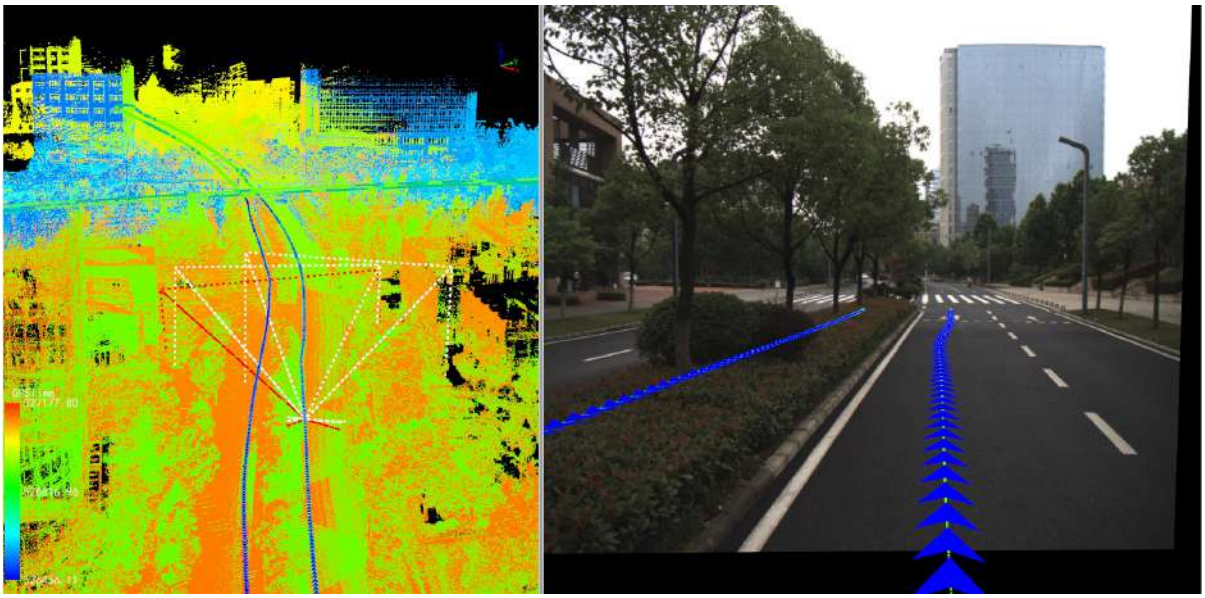
Function Description: The software supports point cloud and image roaming. The original image camera file must be added when creating a new project, and then the menu bar will be activated.

Steps

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

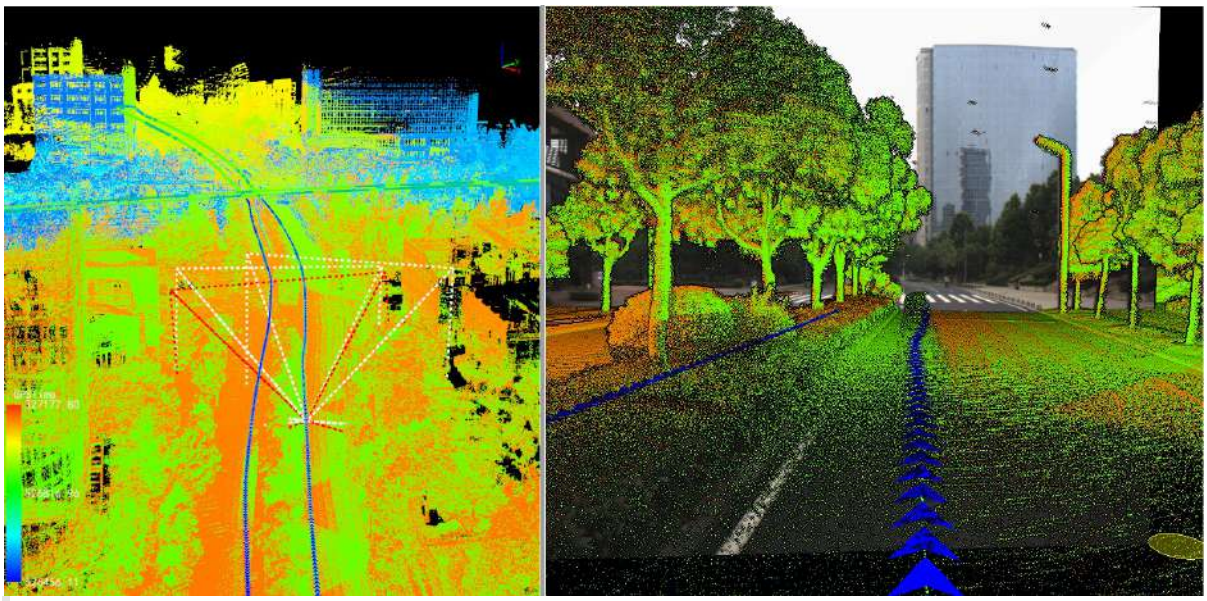


2.In the menu bar uncheck ☐ Show Point Cloud , the user can browse only the image data in the image



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

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

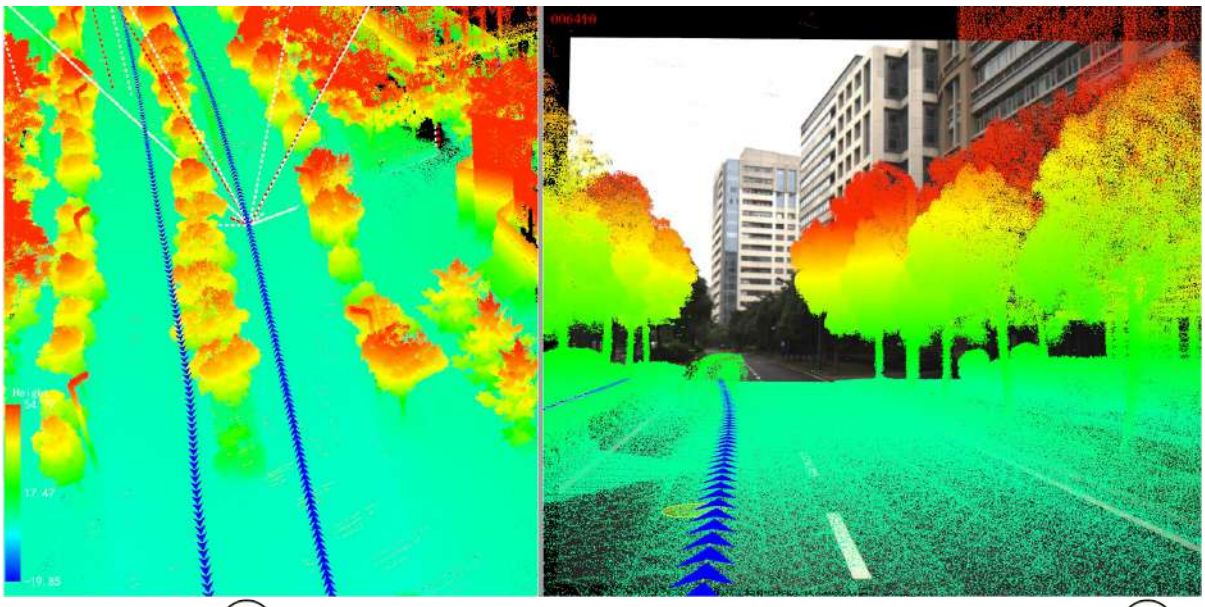


Note:

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

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

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

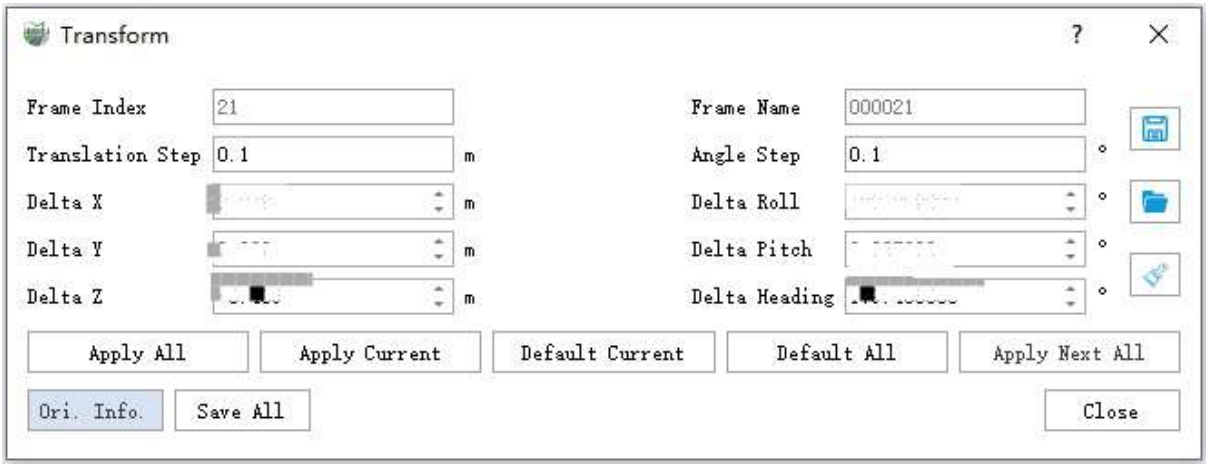


6.Click **Next Frame** button jumps to the next frame of the current image. Click on **Previous Frame** button jumps to the previous frame of the current image.

7.Drag the slider to switch different image positions or directly input the number of image frames.The step size is how much to jump each time during automatic roaming or when going to the next/previous frame. The speed multiplier adjusts the roaming speed during automatic roaming, with a minimum of 1 and a maximum of 10.

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

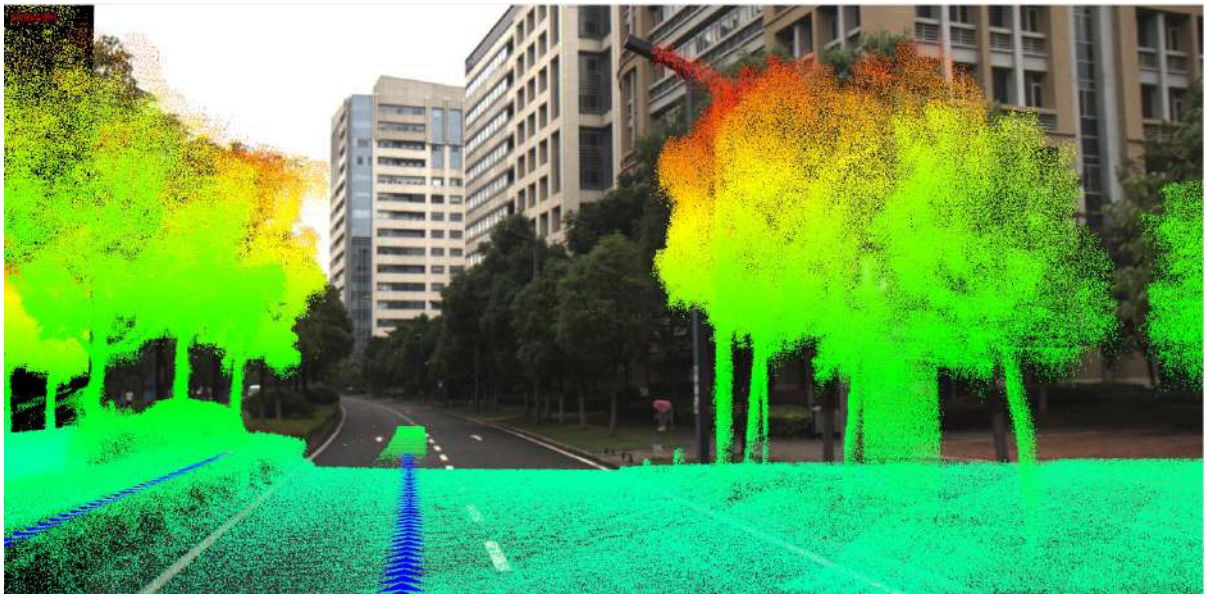
9.Click **Transform** button, the transformation is mainly to adjust the pose of the image in real time and superimpose the point cloud perfectly.



Illustrate

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

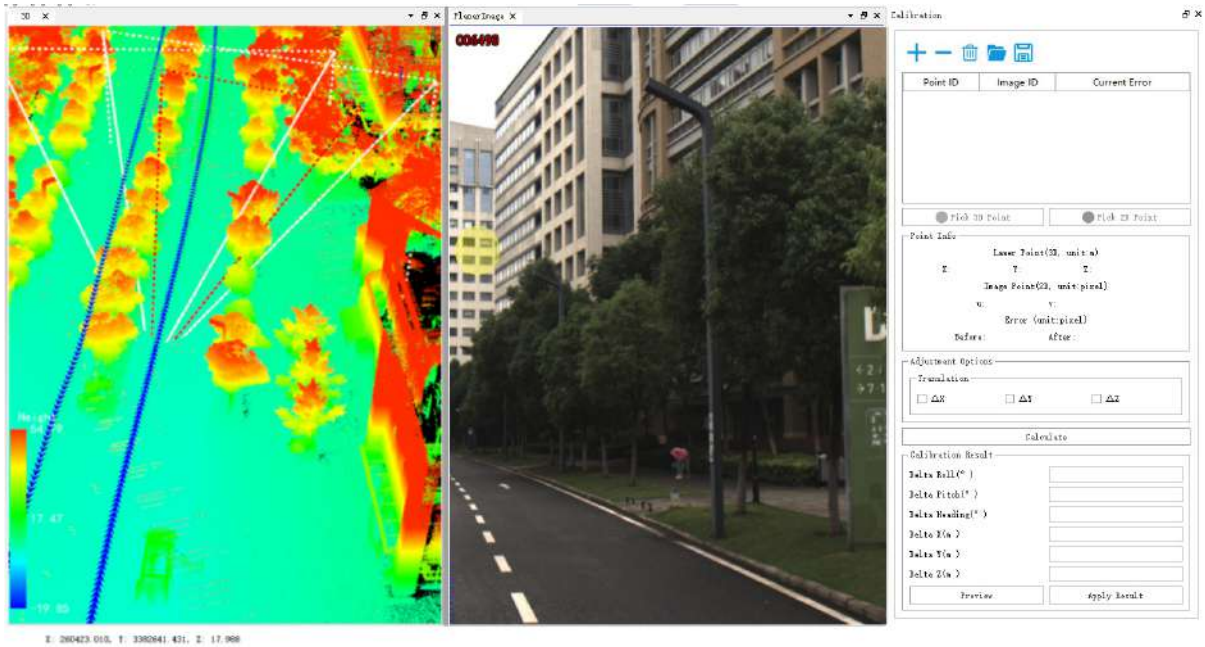
10.Due to the placement error between the image camera and the laser, there is still a certain deviation between the plane image and the point cloud data, which cannot be superimposed perfectly, as shown in the following figure:



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

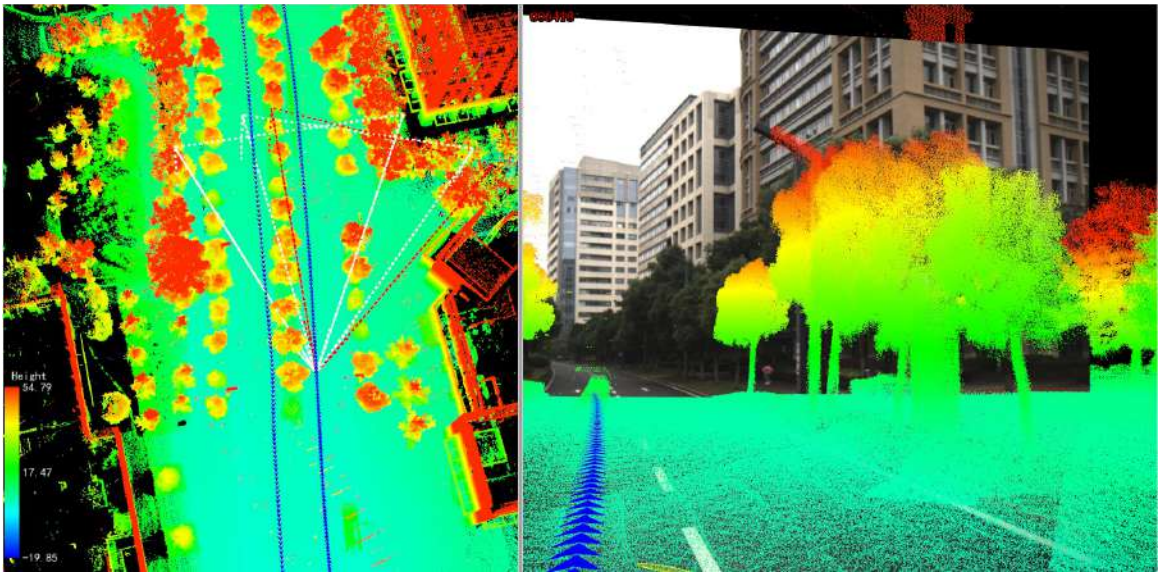
> Note: Set the point cloud display depth by adjusting the menu bar Image->Image Display->Distance, so as to hide the distant point cloud and facilitate the observation of the area of interest. This little trick can also be used in the calibration point selection process below.

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

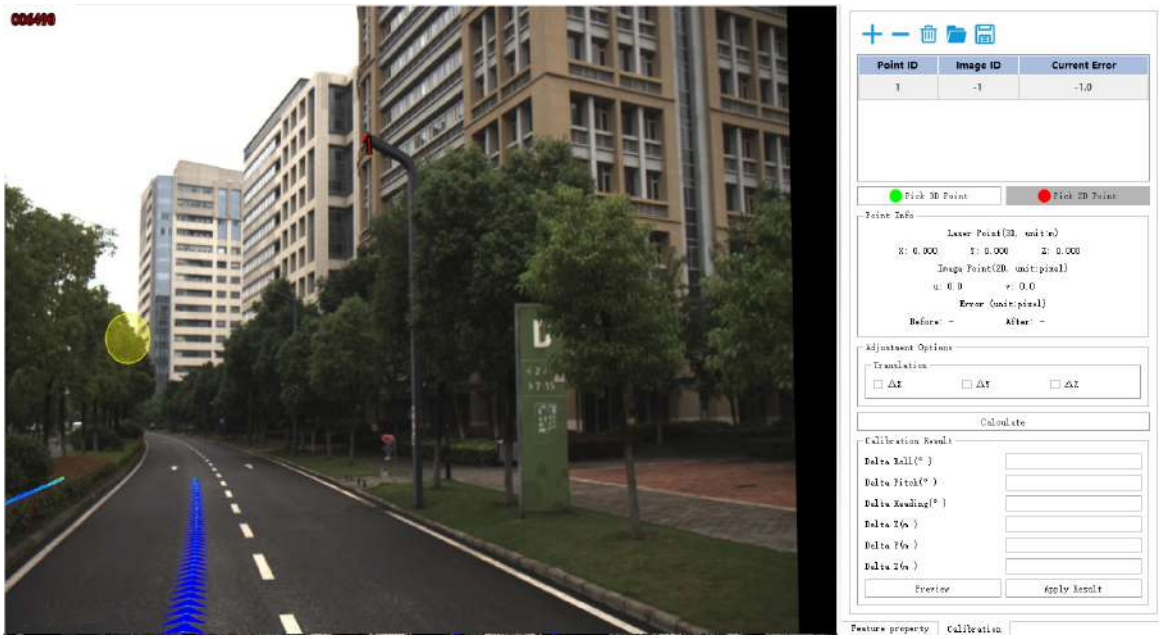


2.Add calibration point. It is recommended to select camera exposure points in multiple directions of the scene, and then select multiple point pairs at each exposure point, at least 4 point pairs need to be added (for example, 4 point pairs are selected at the four exposure moments of the east, south,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

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



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

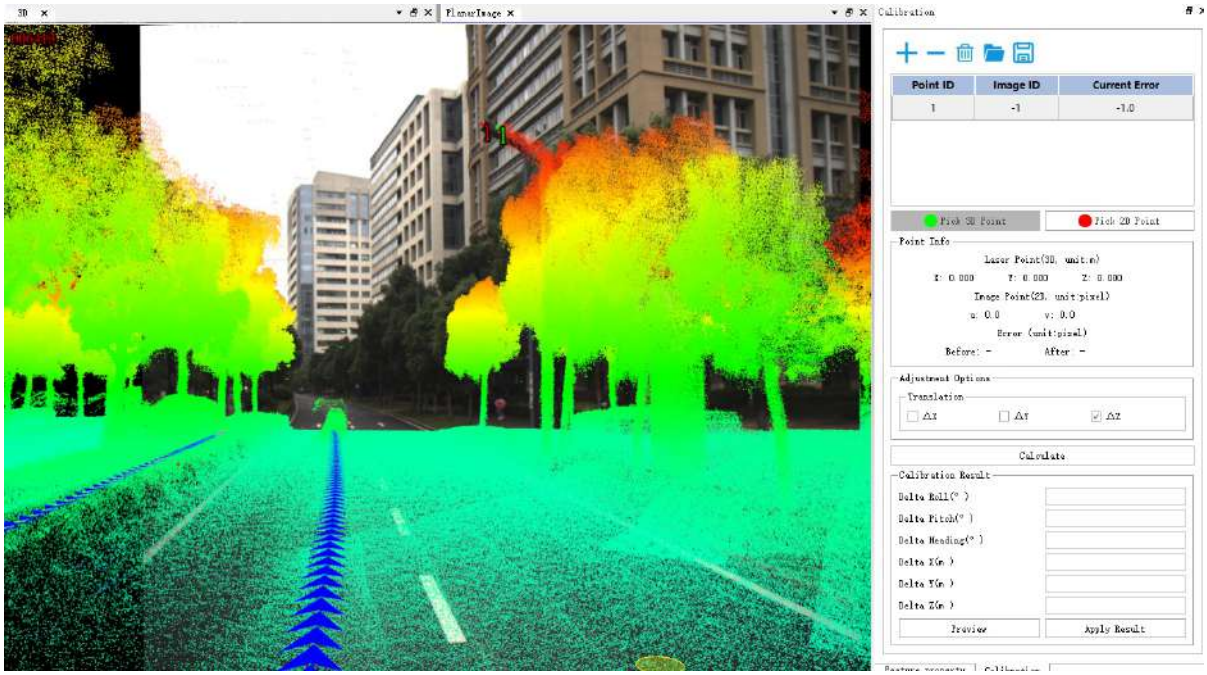


- Click the *Pick 3D Point* button to select the laser point corresponding to the image point in the image window.
- Repeat the above three steps until enough point pairs are selected on the current image frame (four point pairs are recommended).
- Select the corresponding image frames in the other three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

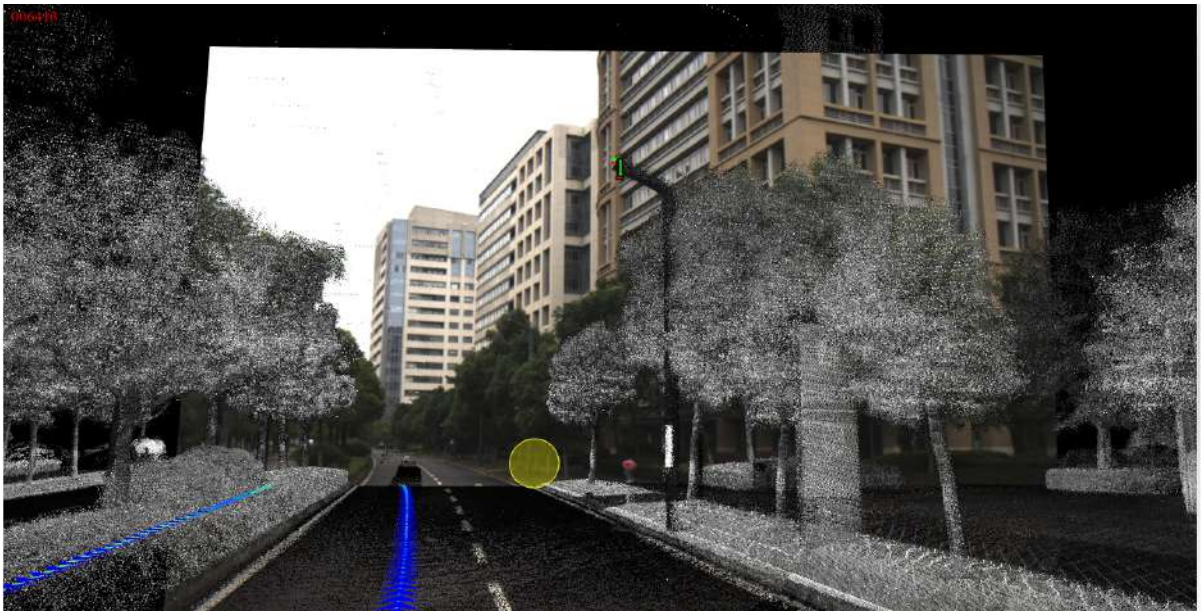
3.Click the *Calculate* button to start the calibration calculation.

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

Before preview:



After preview:



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

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

Image Measurement Based on Point Cloud Depth Interpolation

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

The main functions are listed below:

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

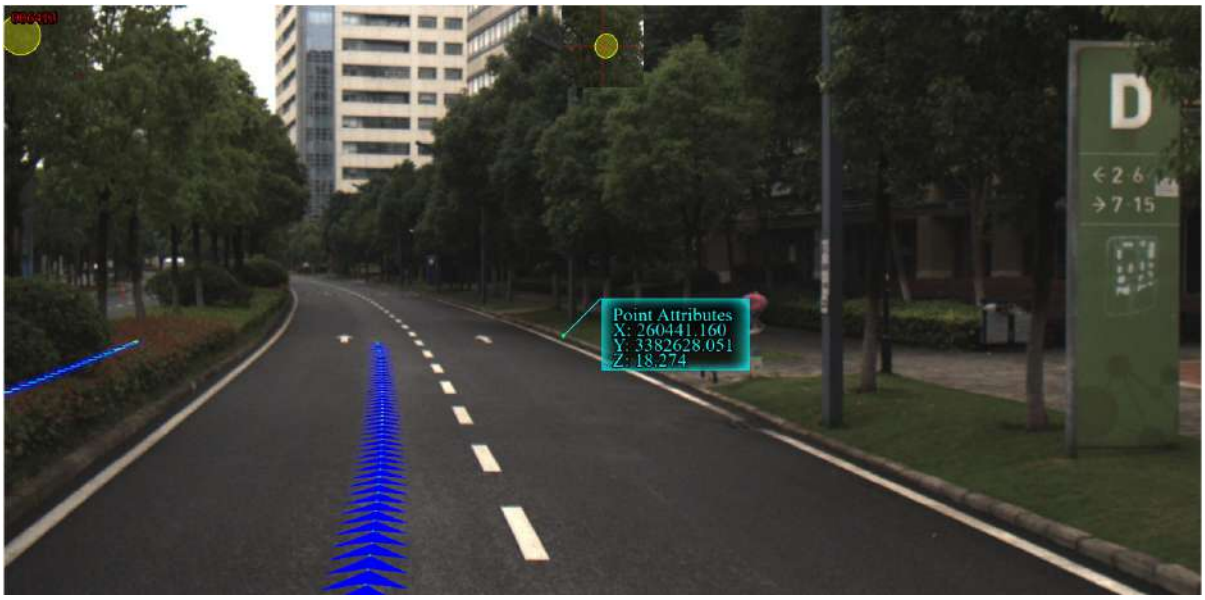
Pick Point (Depth Interpolation)

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

Steps

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

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



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

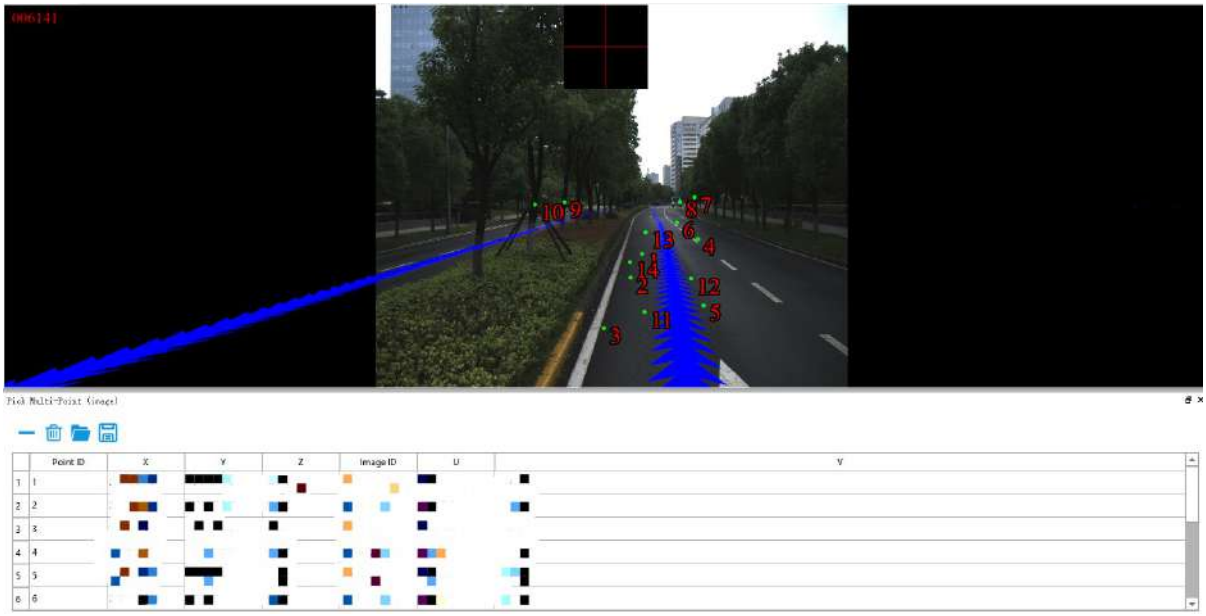
Multi Pick Point (Depth Interpolation)

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

Steps

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

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

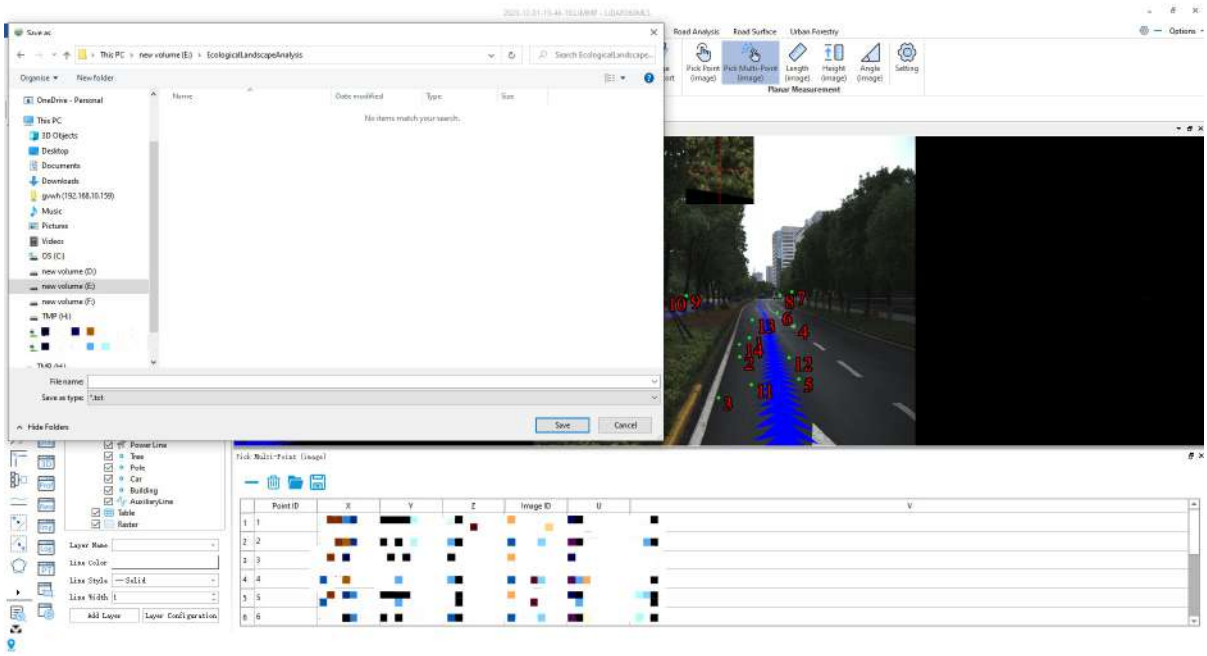


Parameter settings

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

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

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



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

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

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

Length Measurement (Depth Interpolation)

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

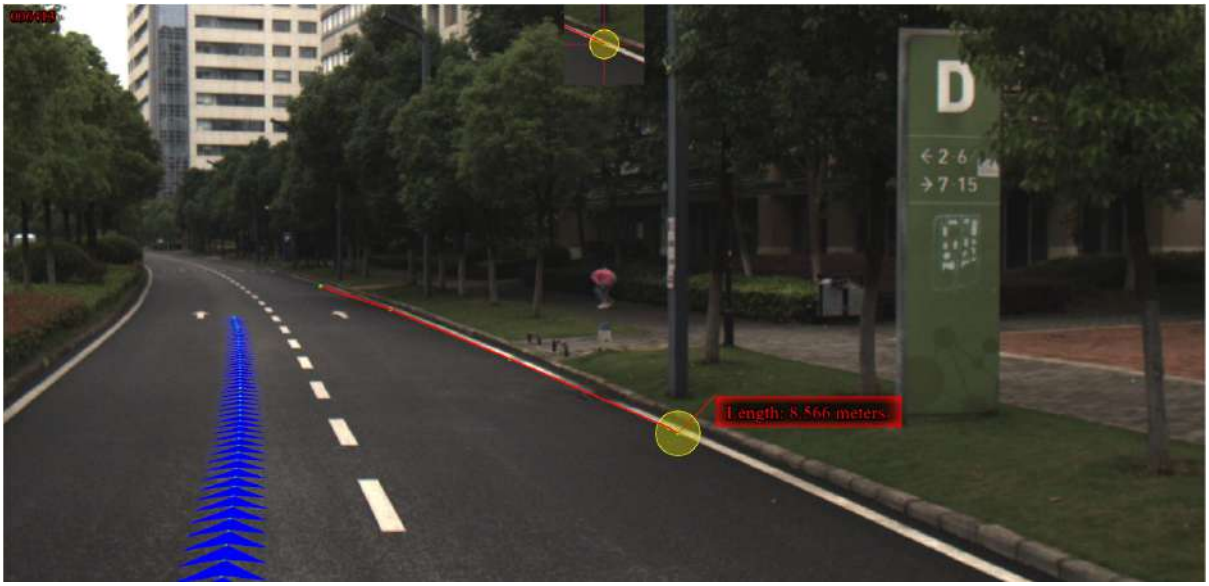
Steps

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

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

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

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



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

Height Measurement (Depth Interpolation)

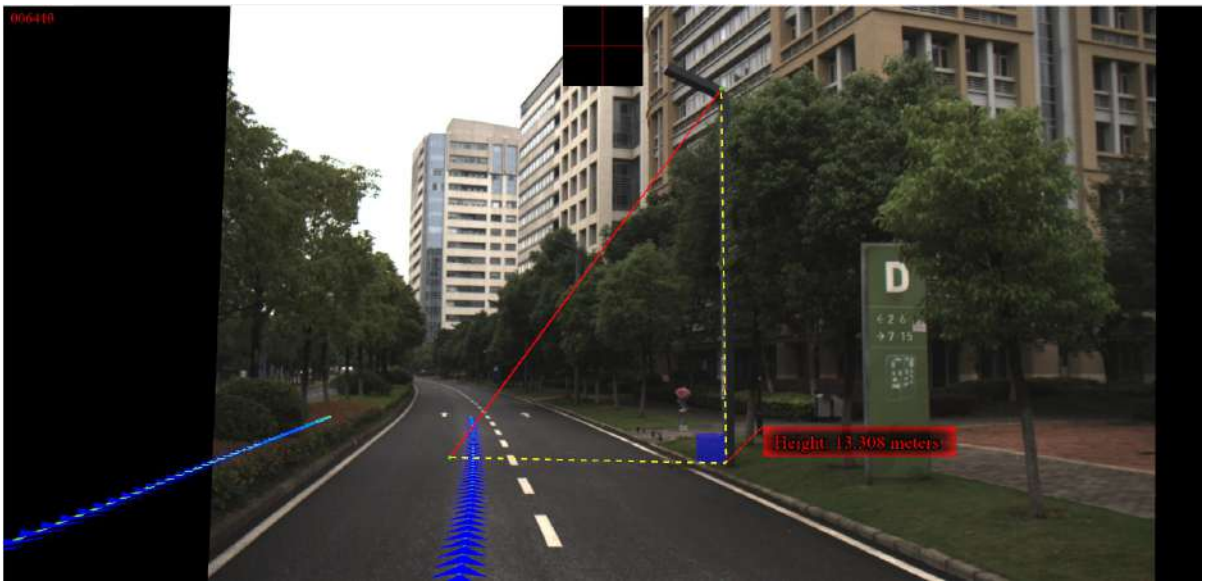
Function Description：This tool is applicable to point cloud data which calculates the relative height difference between two points in PlanarImage window.

Steps

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

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

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



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

Angle Measurement

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

Steps

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

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

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

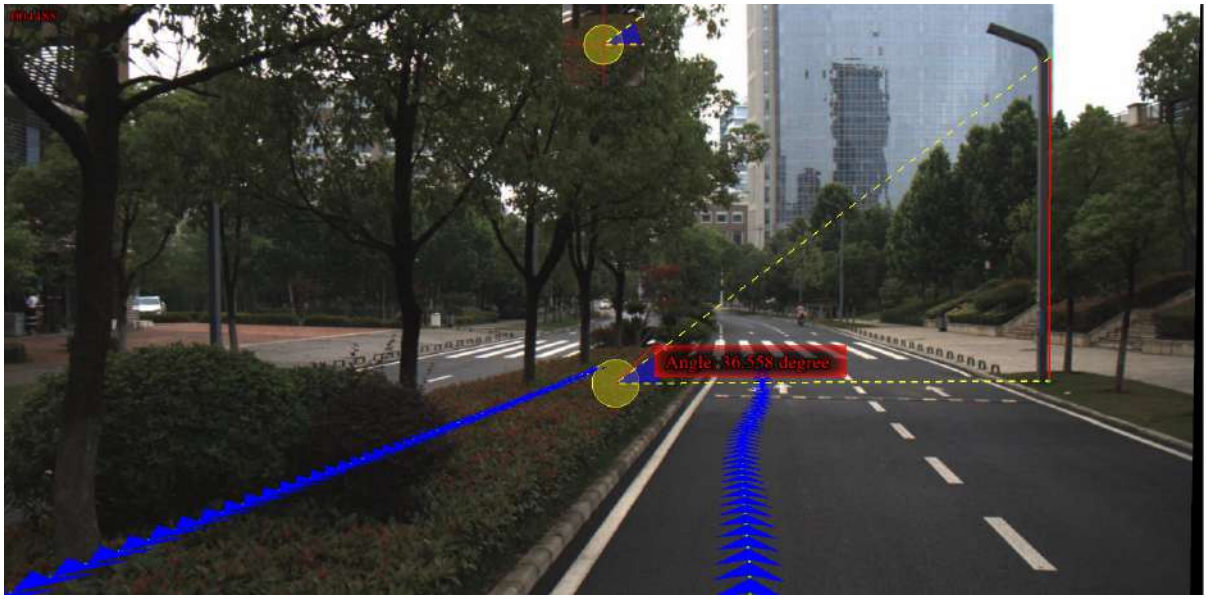


Image Undistorting

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

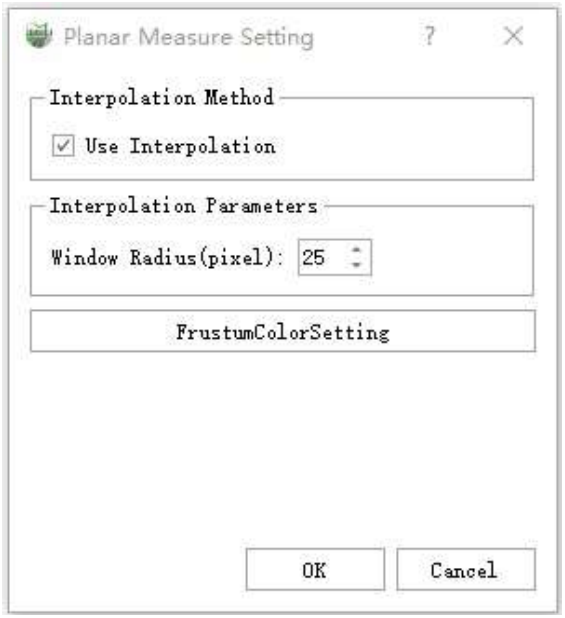
1.Image distortion is due to the lens manufacturing precision, where the deviation of the assembly process will introduce distortion, resulting in distortion of the original image. The distortion of the lens is divided into two categories: radial distortion and tangential distortion.

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

Dialog Setting

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

Click the **setting** button.



Parameters Introduction

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

Preprocessing

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

Trajectory

Segmentation

Calibration

Boresight

GCP Adjustment

Strip Adjustment

Colorization

Trajectory

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

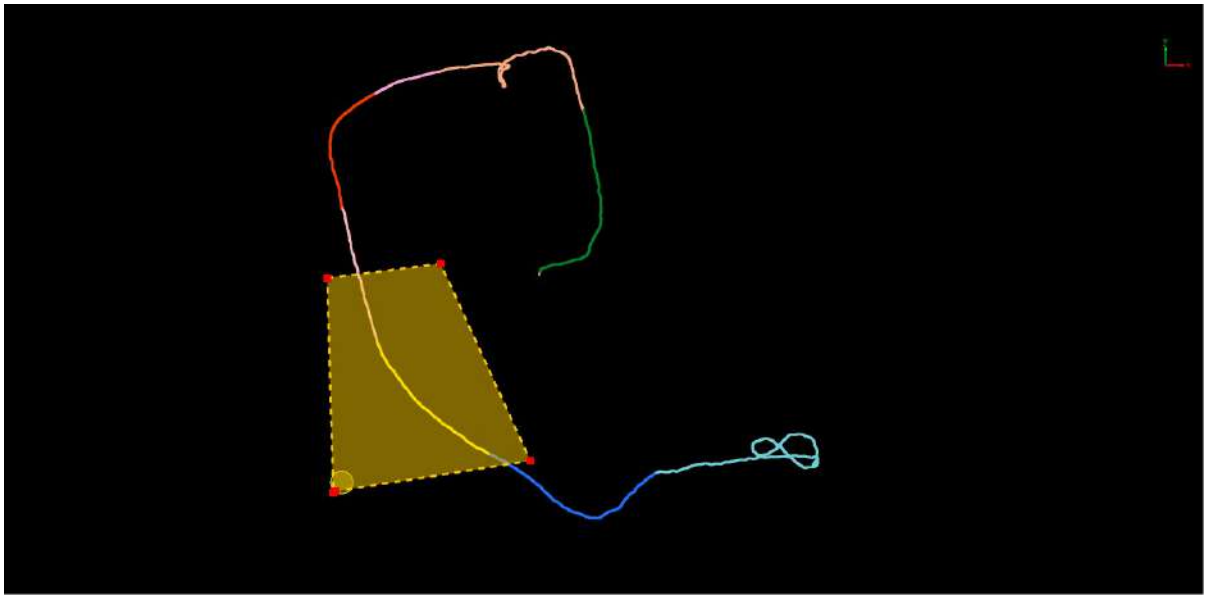
Trajectory Segmentation

Split by Polygon

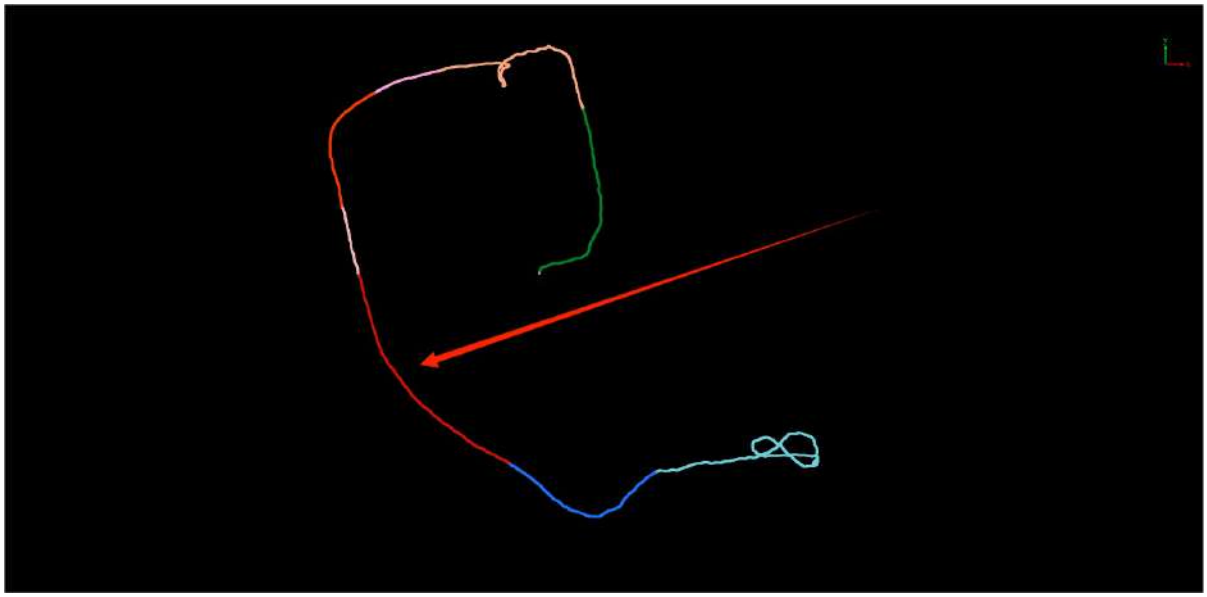
Steps

1.Click **Draw Polygon** button.

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



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

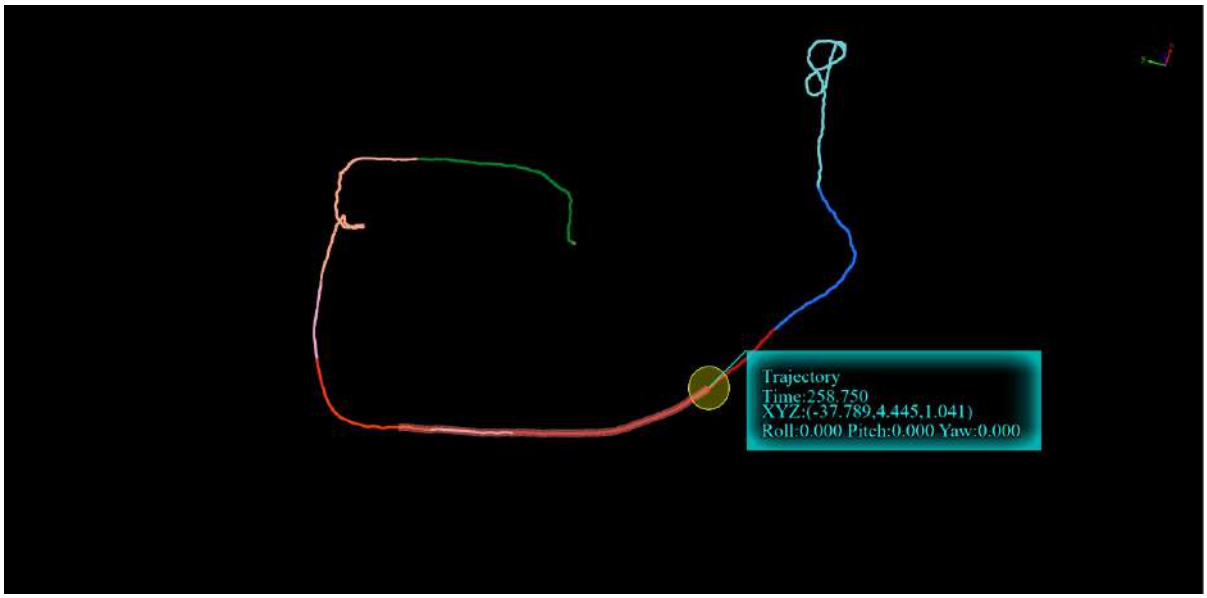


Select on Trajectory

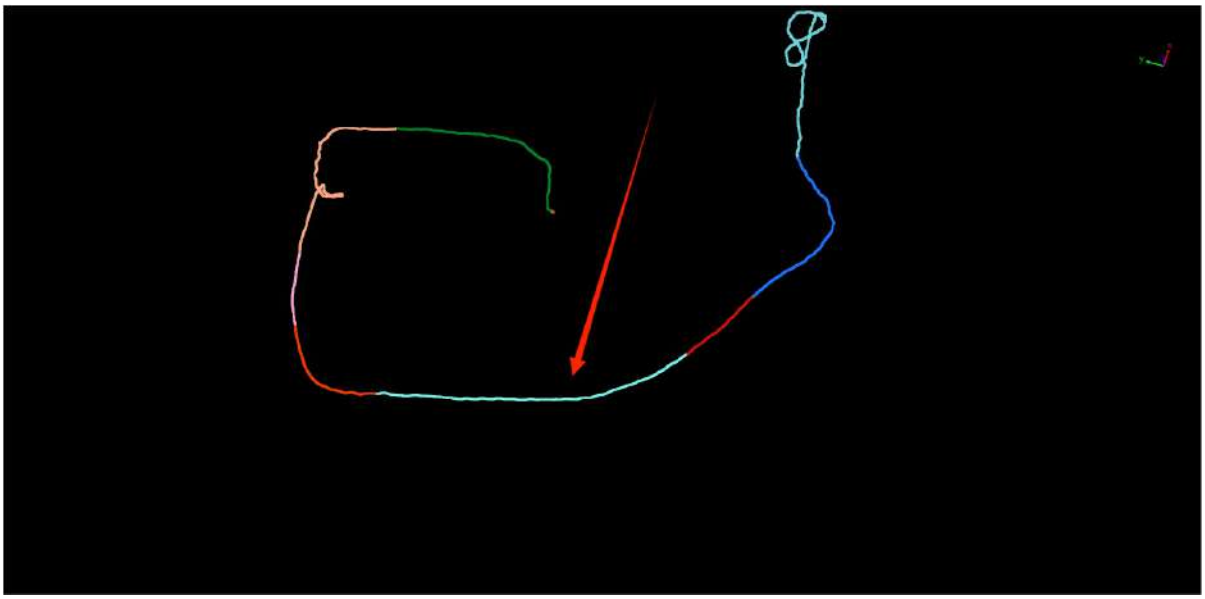
Steps

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

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



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



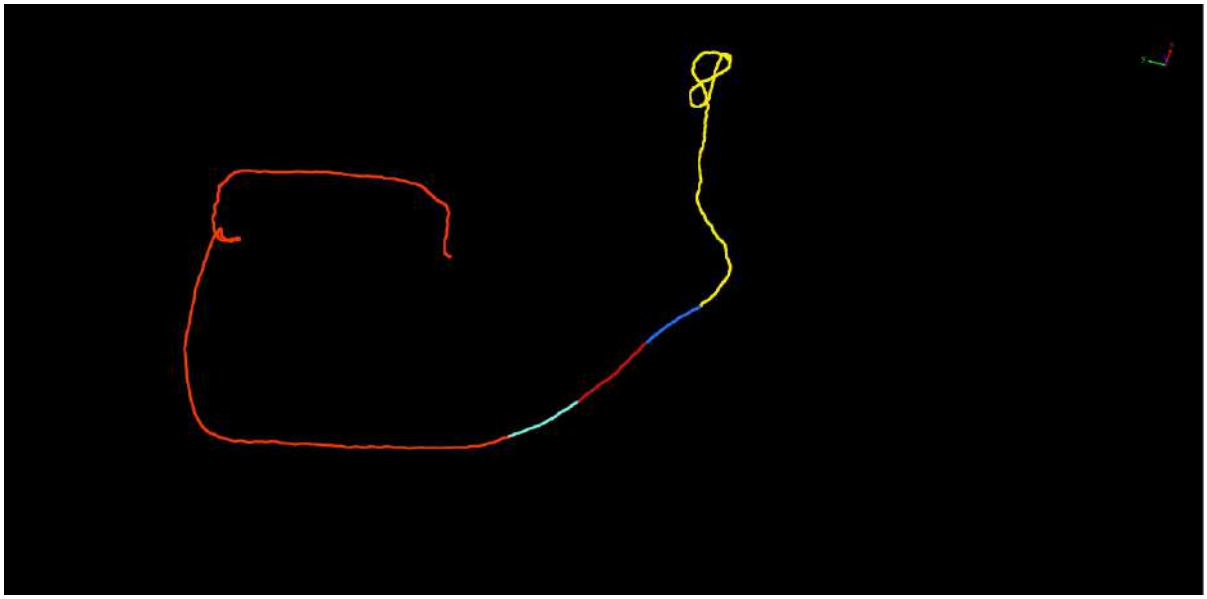
Invert Select on Trajectory

Steps

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

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

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

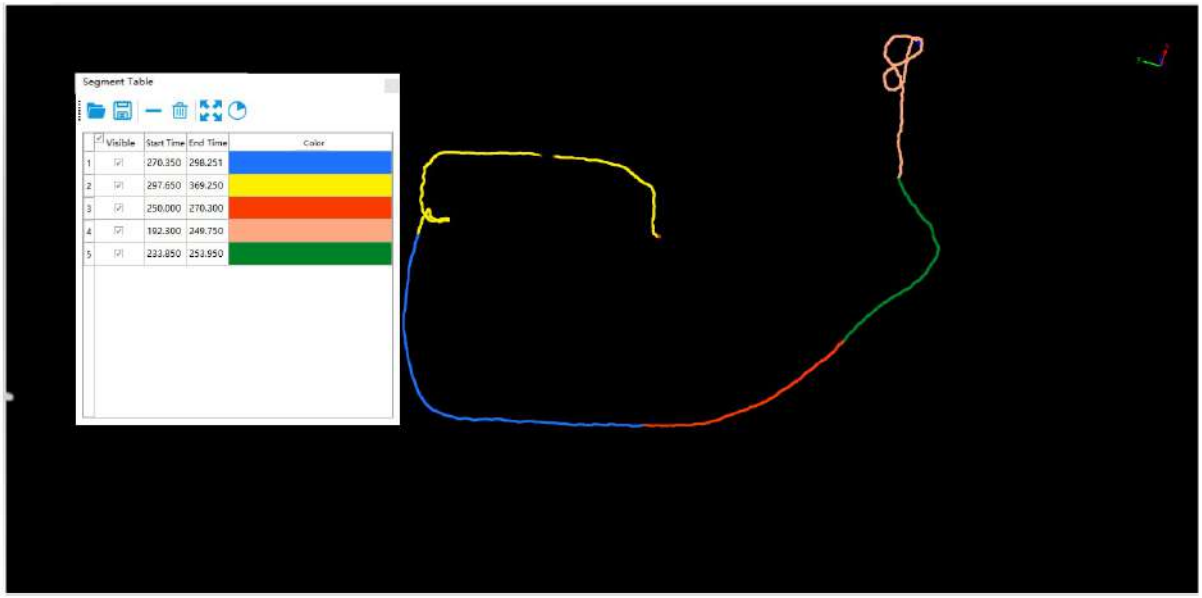


Segment Table

Function Description The table records detailed information about trajectory segments segmented using the above two methods, including the visibility of the trajectory segment, start and end times, and color.

Steps

1.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.



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

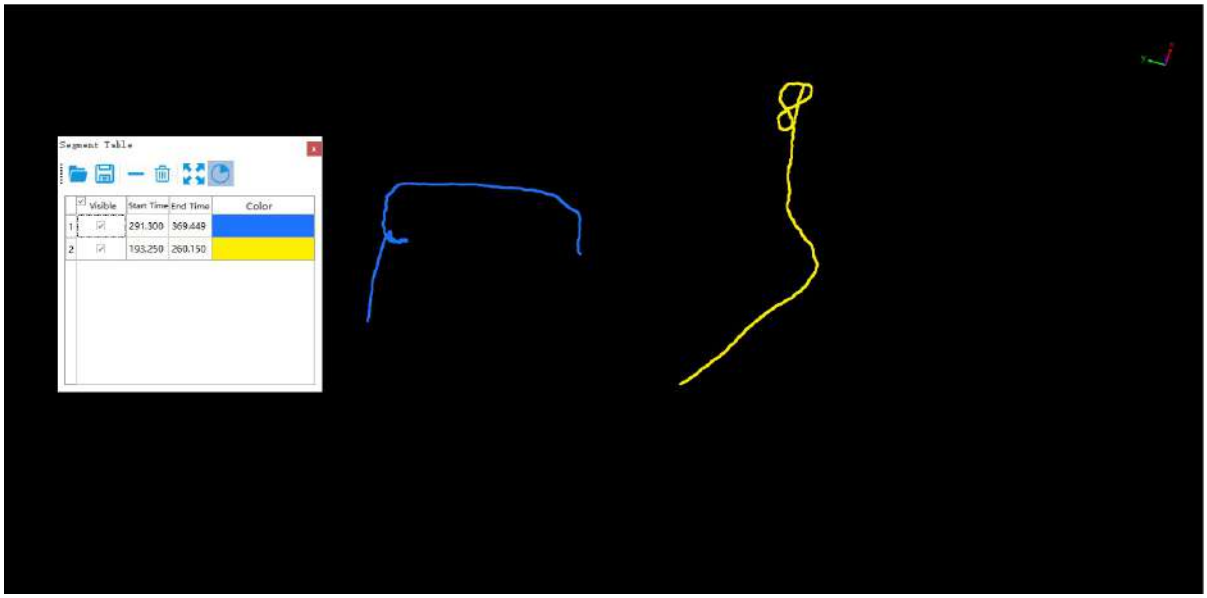
3.Open File Click the **Open File** button, and a pop-up window reads the saved track segment from the specified location on the hard disk, and the reading format is *.xml*.

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

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

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

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

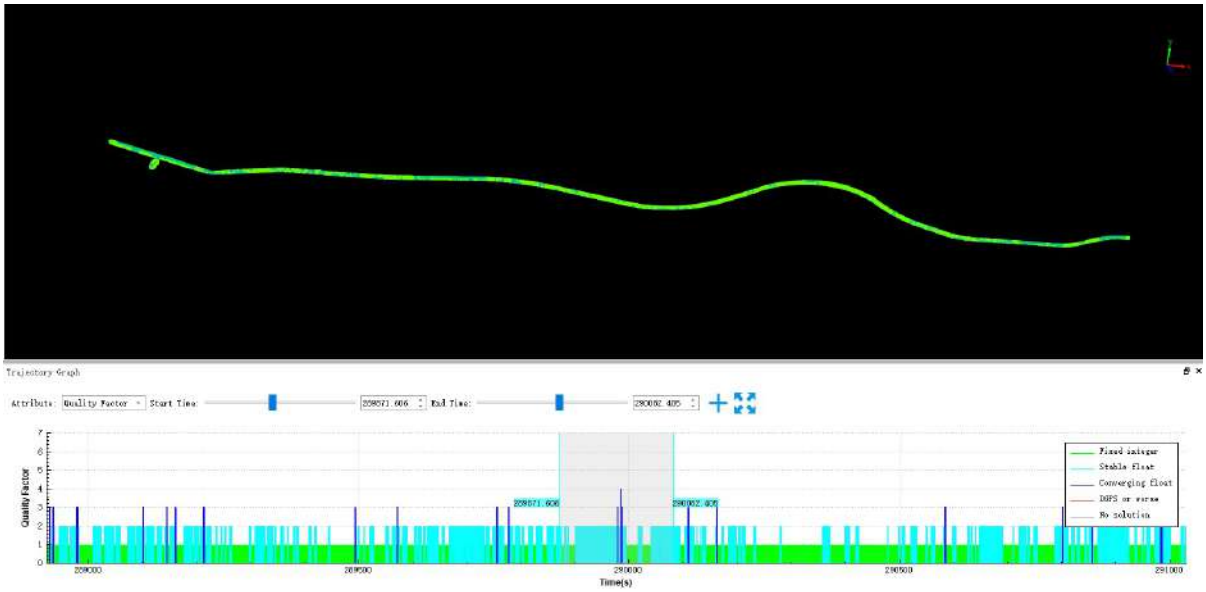


Trajectory Graph

Function Description：The trajectory chart displays trajectory segmentation information for each moment. Through the trajectory chart menu bar, you can adjust display properties, add new trajectory segments, and change the display range of trajectories.

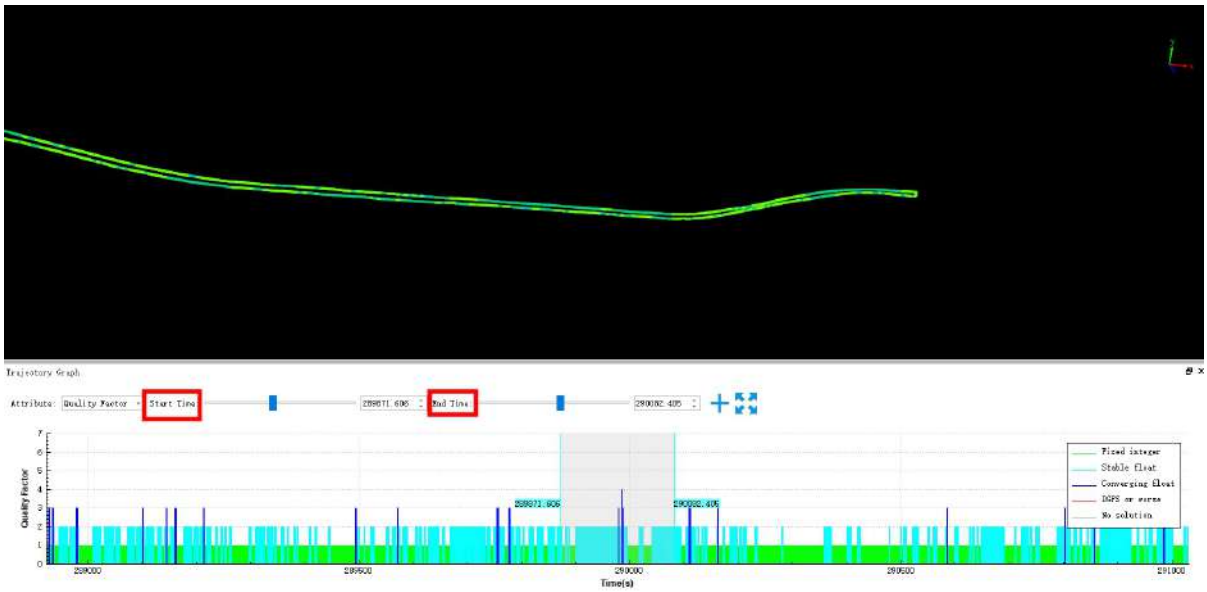
Steps

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.

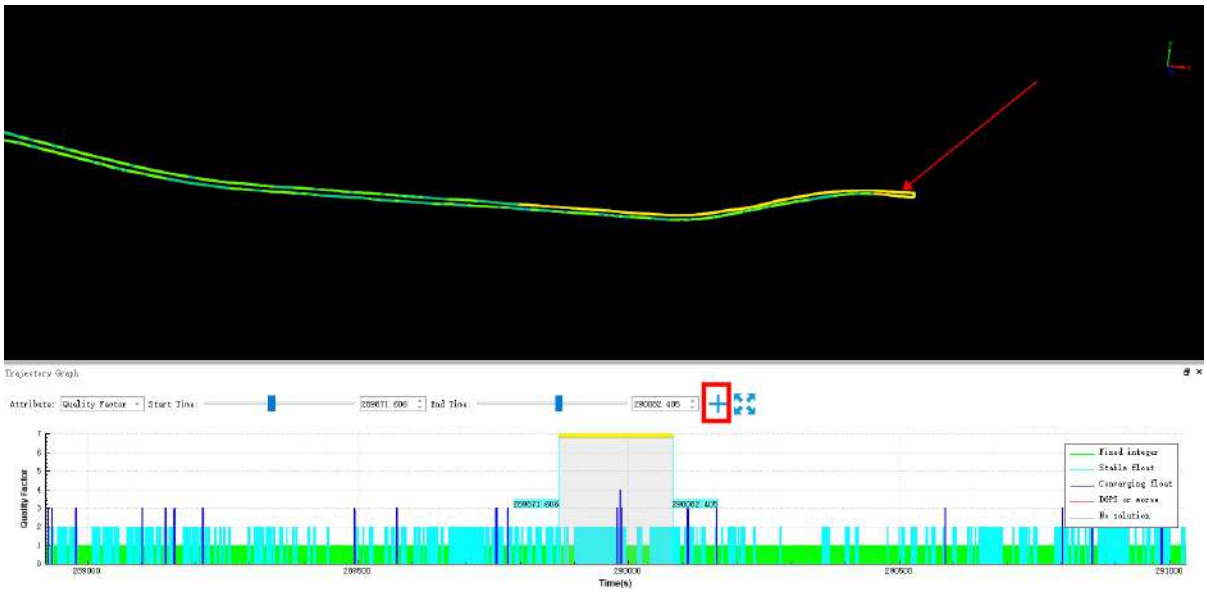


2.Adjust Display Properties：Click the selection box after the **Attribute** tab to adjust the information displayed in the segmented status chart. You can choose to display by Height, display by Quality Factor, display by Roll and Pitch, display by Heading, and display by Velocity.

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



Select the start and end time



Add track segment

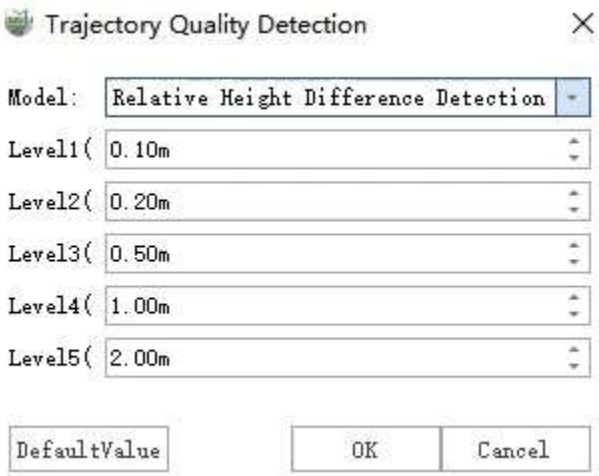
4.Change the Display Range of the Trajectory Graph：The trajectory graph displays the track segmentation information at all times by default. Use the mouse wheel to slide up and down on the track map to zoom in and out of the display range of the track map, click **Full Extent** button to restore the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

Trajectory Quality Detection

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

Steps

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



2.**Elevation Difference Detection on Return to Area**：Used to traverse the area back and forth in two passes, detecting points with a relative elevation difference greater than a threshold.

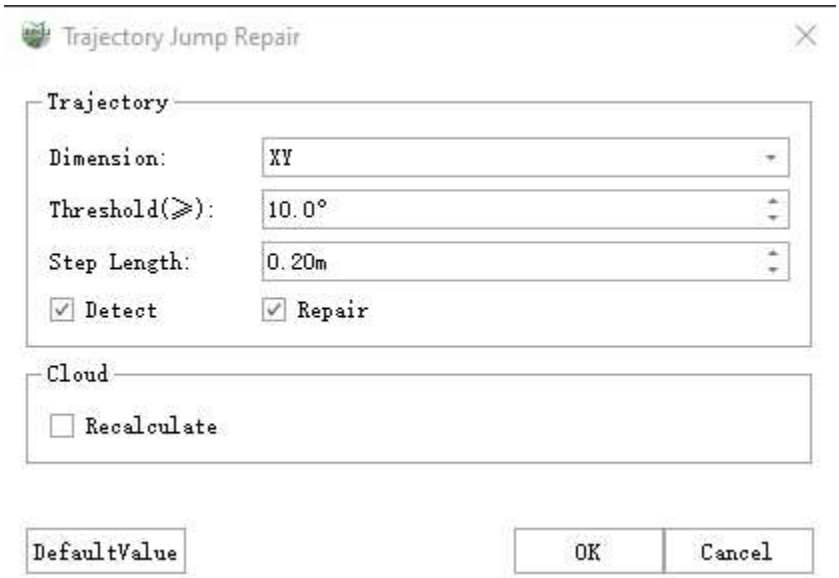
- **Level Parameters**：Set five levels, displaying different colors for abrupt changes in different levels.

Trajectory Jump Repair

Function Description: Detect and correct trajectory segments with horizontal/elevation discontinuities within the trajectory.

Steps

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

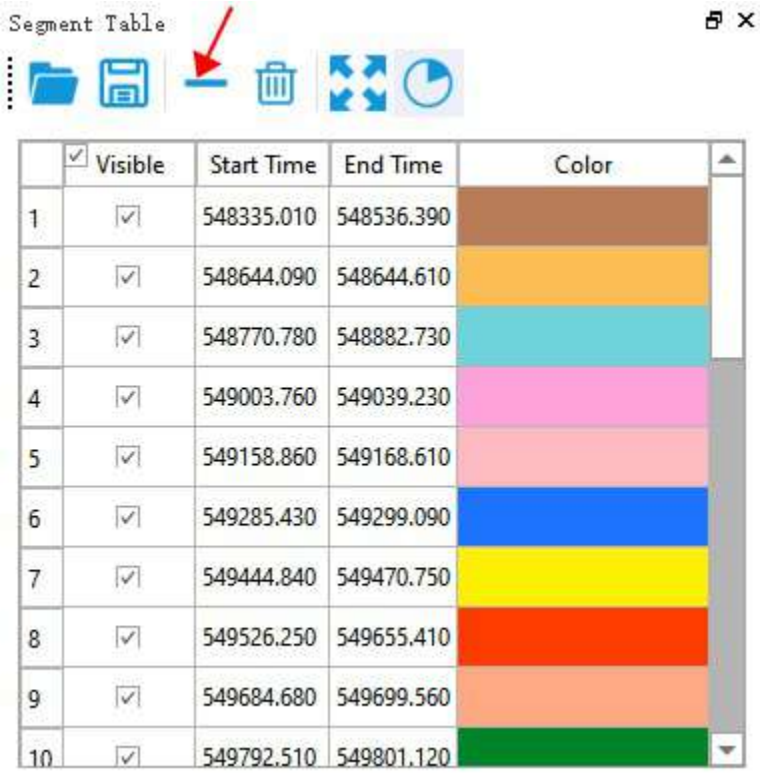


Function Description:

- **Dimension**: Divided into two dimensions, the XY plane and the Z elevation, it is necessary to select the corresponding dimension for separate detection and correction.
- **Threshold**:
  - Under the XY dimension, the threshold represents the change in the plane angle along the forward direction. The smaller the angle, the more detected discontinuities.
  - In the Z dimension, the threshold represents the height difference threshold of trajectory segments when detecting elevation changes along the forward direction. The smaller the height difference, the more detected elevation changes.
- **Step Length**:Trajectory segment length when detecting along the forward direction, typically recommended as default.
- **Detect**：If only 'Detect' is selected, clicking OK will only run trajectory detection for the current dimension.
- **Repair**：If 'Repair' is selected, it will repair the detected discontinuous segments based on the current dimension's detection.
- **Recalculate Cloud**：Based on the repaired trajectory, recompute the point cloud.

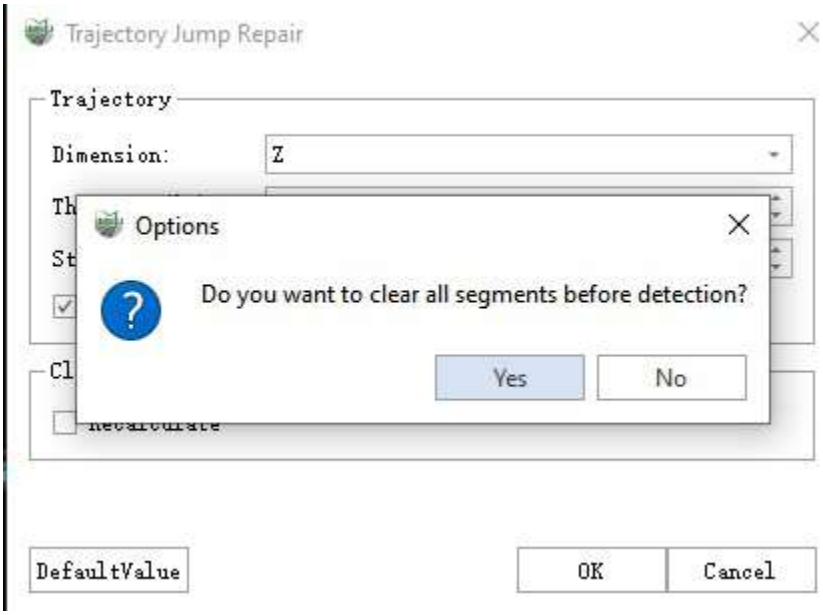
2.If the data exhibits both lateral changes in the XY plane and elevation changes in the Z direction, the recommended processing steps can be as follows:

- a.Select the "XY" dimension, check "Detect", and click OK. Coordinate with the trajectory segment table to remove any redundant detected discontinuity segments.



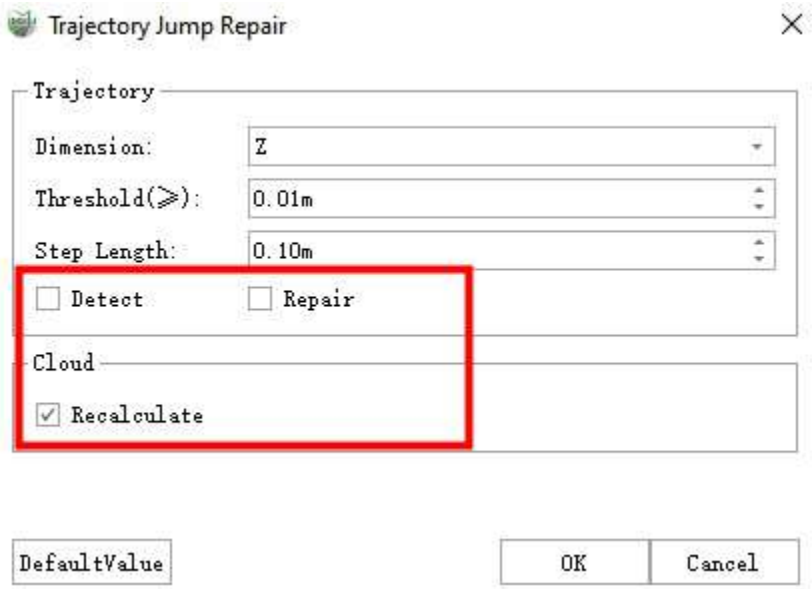
If there are discontinuity segments that were not automatically detected, you can manually select these segments using the trajectory selection feature. After confirming that there are no issues with the discontinuity detection in the "XY" dimension, uncheck "Detect", and check "Repair" to fix the detected discontinuities in the XY dimension.

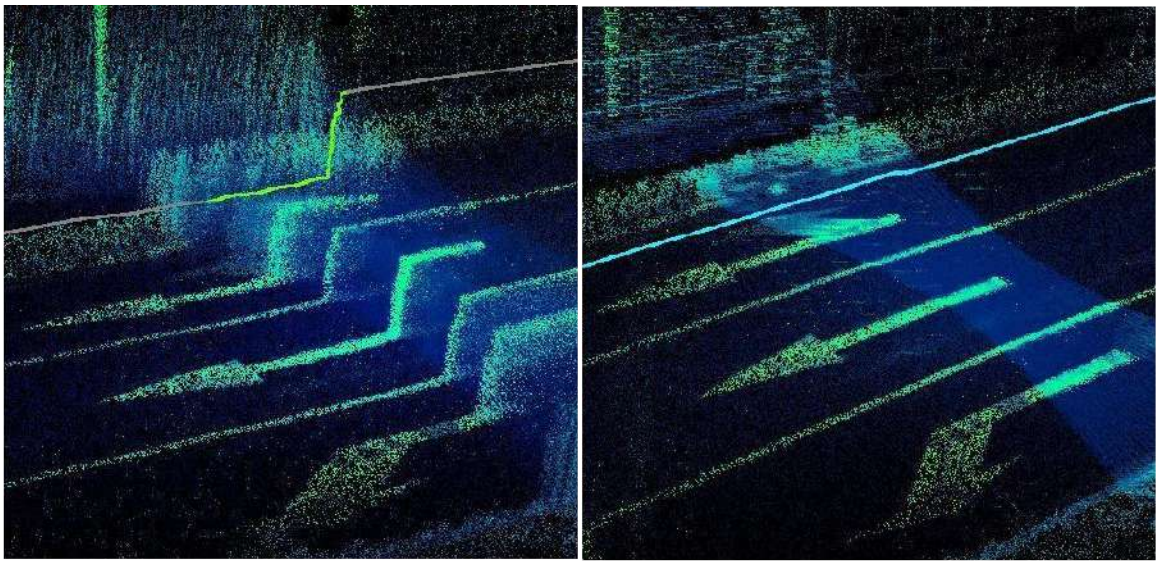
- b.Switch to the "Z" dimension, similarly check "Detect", and click OK. At this point, a prompt will appear asking, "Do you want to clear all segments before detection?" You need to select "Yes".



In coordination with the trajectory segment table, delete any redundant detected discontinuity segments. If there are discontinuity segments that were not automatically detected, you can manually select these segments using the trajectory selection feature. After confirming that there are no issues with the discontinuity detection in the 'Z' dimension, uncheck 'Detect', and check 'Repair' to fix the detected discontinuities in the Z dimension.

- c.When trajectory discontinuity repairs have been separately performed for both the XY and Z dimensions, uncheck 'Detect' and 'Repair', leaving only 'Recalculate' selected. This will recalculate the point cloud based on the repaired trajectories.

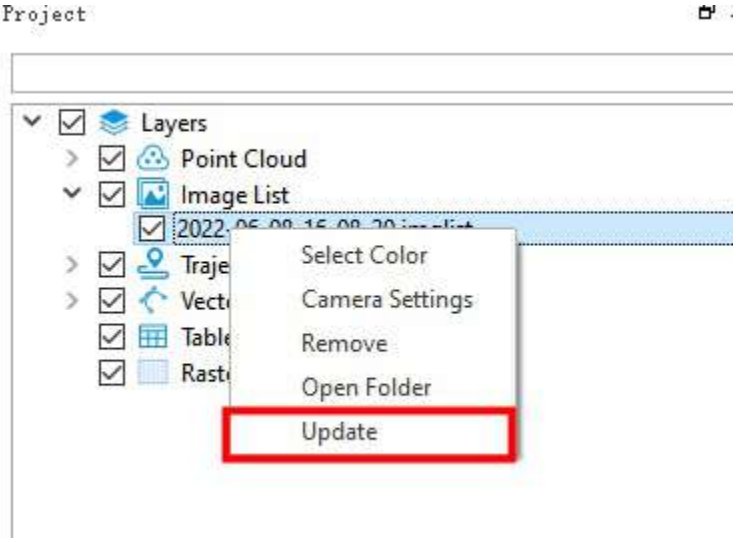




Before Trajectory Jump *After Trajectory Jump*

> Note:
>
> - The trajectory detection and repair for the XY and Z dimensions need to be handled separately. If the data only exhibits discontinuities in one dimension, you can perform detection and repair for that specific dimension and then recalculate the point cloud accordingly.
> - When switching detection dimensions, it is important to ensure that the trajectory segments from the previous detection are cleared.
> - The repaired trajectories will take effect only when recalculating the point cloud. If you do not recalculate the point cloud, the trajectory repairs will be displayed temporarily.
> - If there is image data, it needs to be loaded directly into the software when creating a new project. The update of image data will occur when recalculating the data. If image data was not loaded during trajectory jump detection and repair, the software provides an image file update feature. In the directory tree, under Image List, right-click on the XXX.imagelist file and choose 'Update'. This option

allows the image information to be updated based on the repaired trajectory file.



Point Cloud

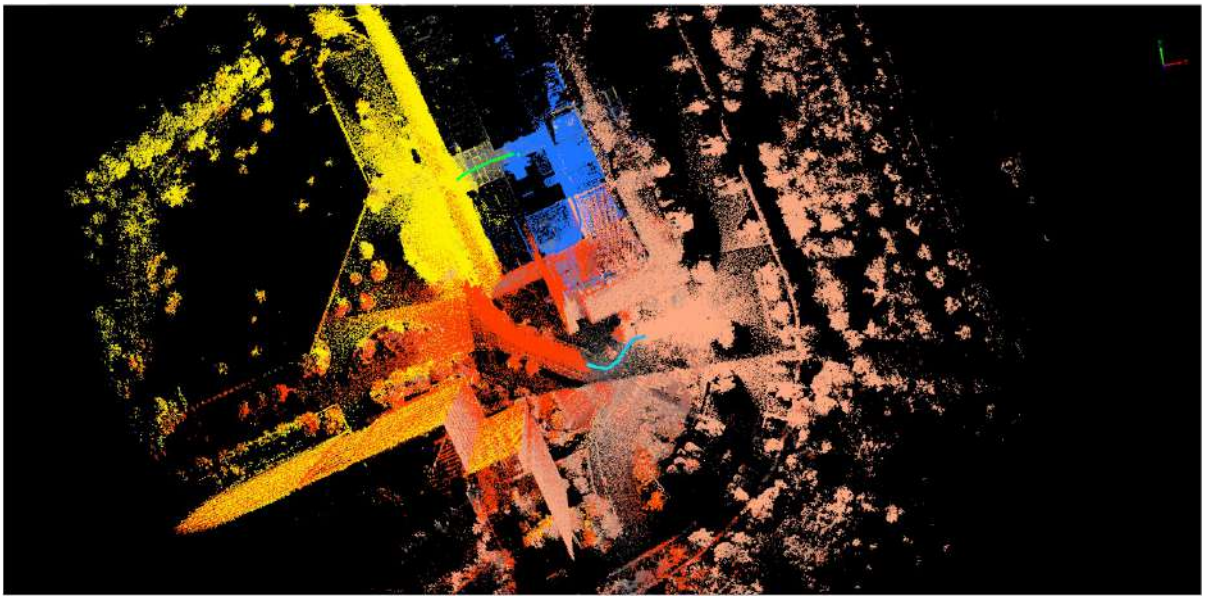
- Colorize by Segments
- Split by Segments

Colorize by Segments

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

Steps

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

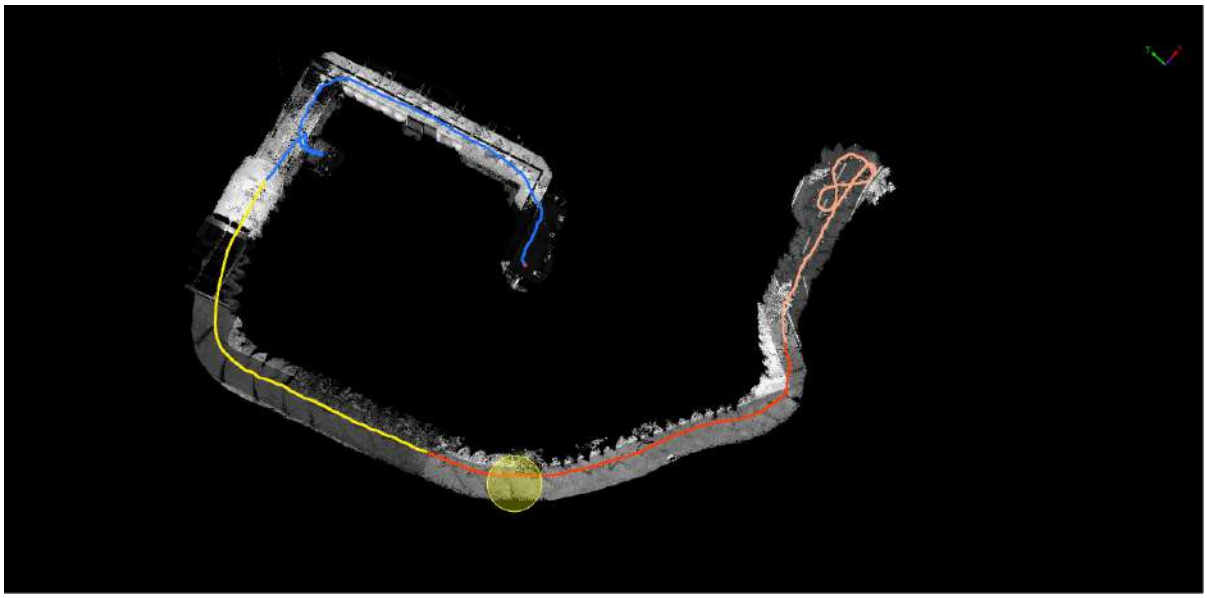


Split by Segments

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

Steps

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



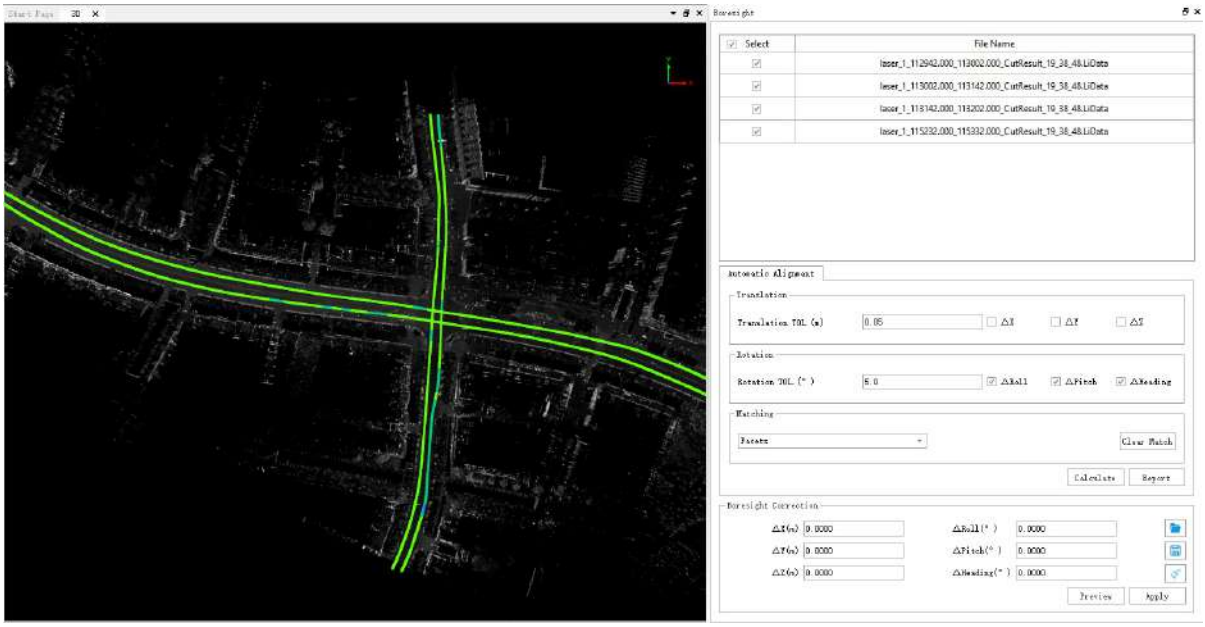
Segmented by track fragment

Boresight Calibration

Function Description: This function is mainly used to correct the placement error of the laser, and the point cloud data will be transformed according to the corrected value. To obtain high-quality point cloud data, the selection of laser placement error correction values is particularly important.

Steps

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

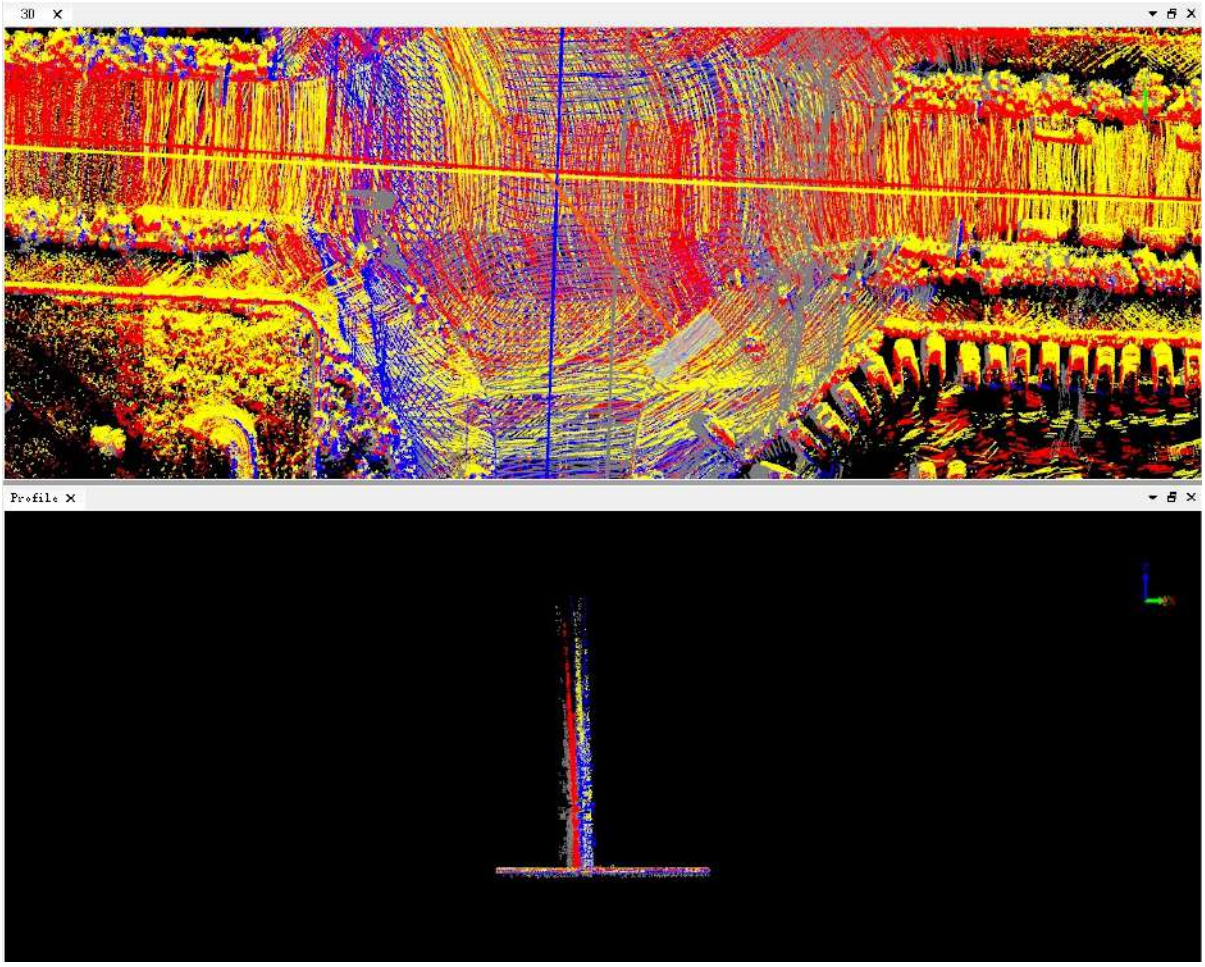


2.Automatic placement error correction：

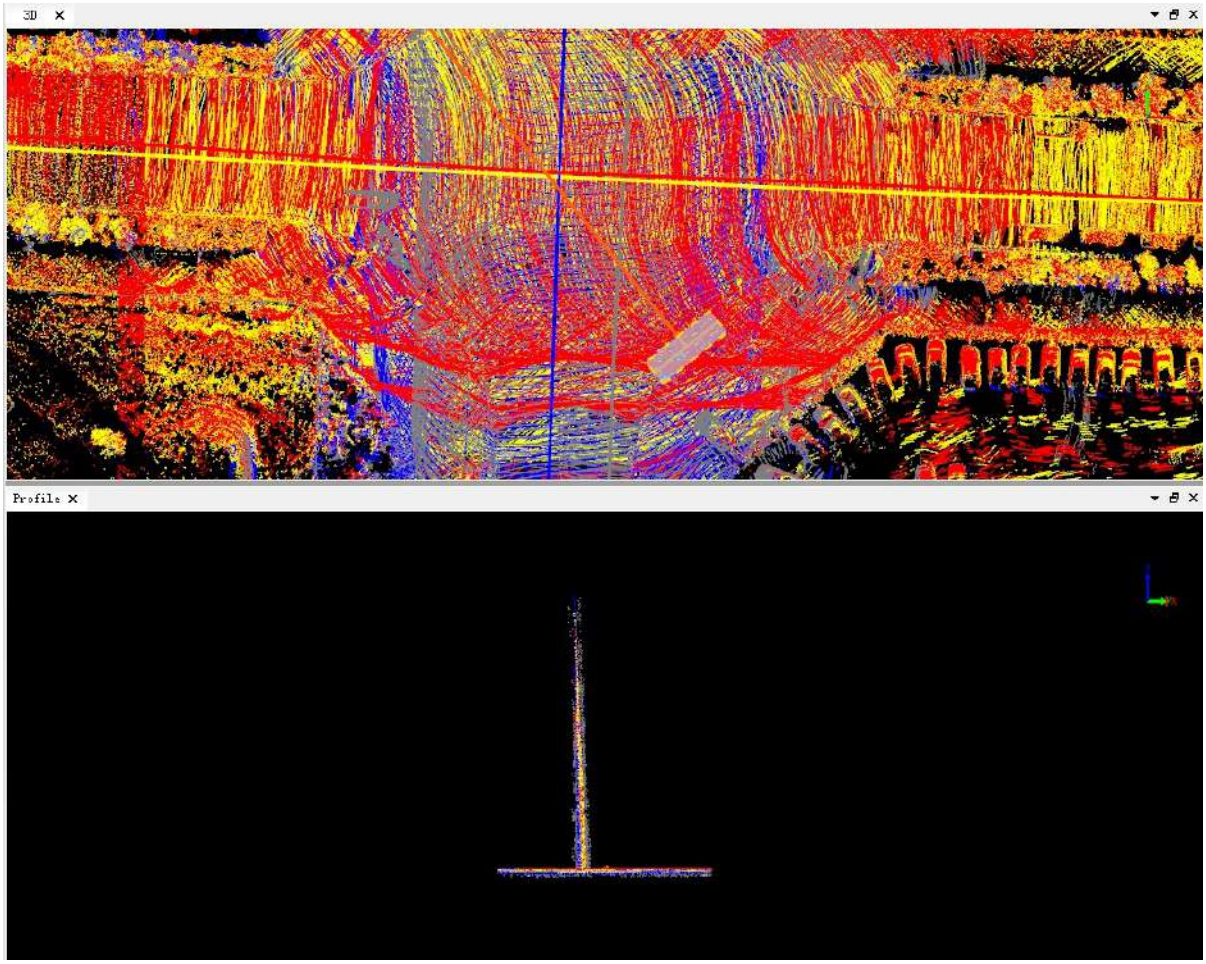
Check the final placement error that needs to be calculated in the placement interface, enter the translation tolerance and rotation tolerance, and Boresight After clicking calculate, the software will automatically calculate the placement error and display the results in the placement error correction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

3.Manual Calculation of Placement Error：

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

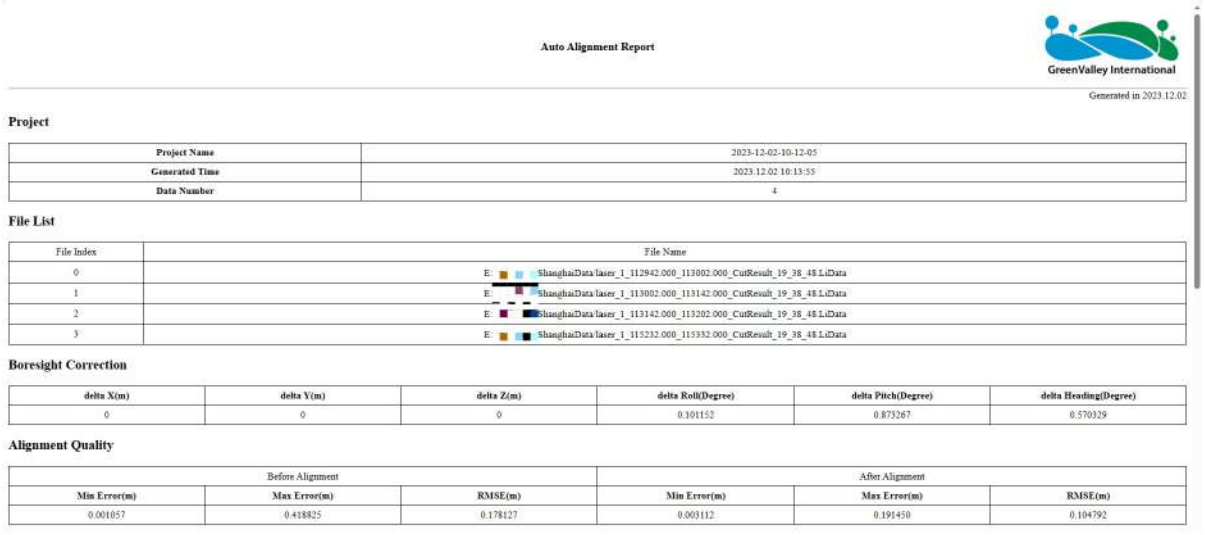


Before correction



After correction

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



Auto Alignment Report

GreenValley International

Generated in 2023.12.02

Project

Project Name	2023-12-02-10-12-05
Generated Time	2023.12.02 10:13:55
Data Number	4

File List

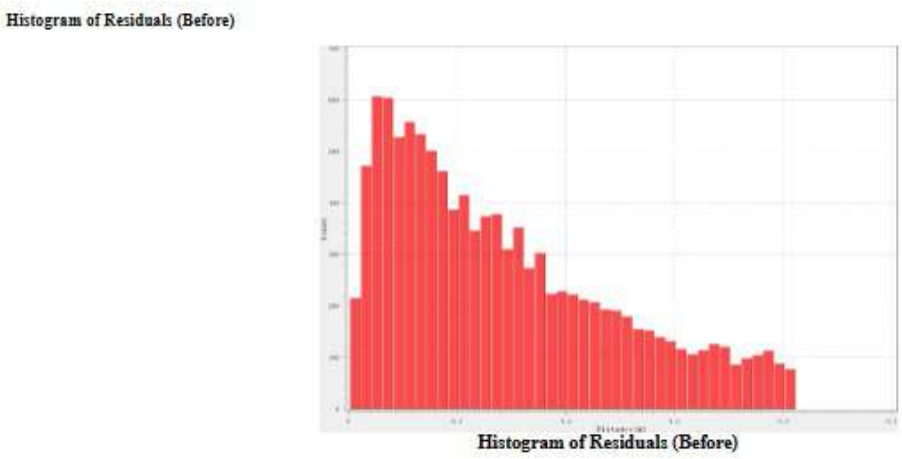
File Index	File Name
0	E: [redacted]ShanghaiData/laser_1_112942.000_113002.000_CutResult_19_38_48.LiData
1	E: [redacted]ShanghaiData/laser_1_113002.000_113142.000_CutResult_19_38_48.LiData
2	E: [redacted]ShanghaiData/laser_1_113142.000_113202.000_CutResult_19_38_48.LiData
3	E: [redacted]ShanghaiData/laser_1_115232.000_115332.000_CutResult_19_38_48.LiData

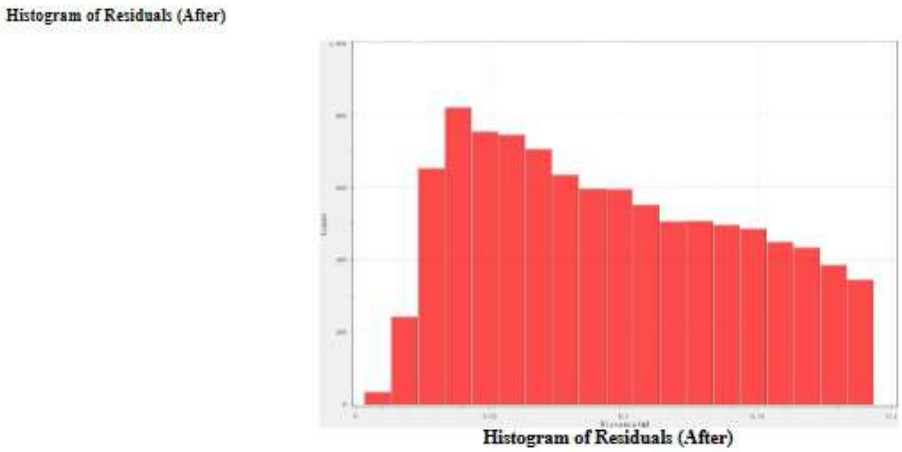
Boresight Correction

delta X(m)	delta Y(m)	delta Z(m)	delta Roll(Degree)	delta Pitch(Degree)	delta Heading(Degree)
0	0	0	0.101152	0.873267	0.570329

Alignment Quality

Before Alignment			After Alignment		
Min Error(m)	**Max Error(m)**	**RMSE(m)**	**Min Error(m)**	**Max Error(m)**	**RMSE(m)**
0.001057	0.418825	0.178127	0.003112	0.191450	0.104792





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

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

Parameter setting

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

matched (relatively robust on data with this feature).

- **Sampled Points**: The sampling point is extracted by analyzing the significance of the data to be matched, and then matched based on the sampling point and its normal vector.

Placement Error Calculation

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

> The software provides two ways to eliminate placement errors：
>
> 1.Manual measurement and inspection.
>
> 2.Automatic adjustment correction.

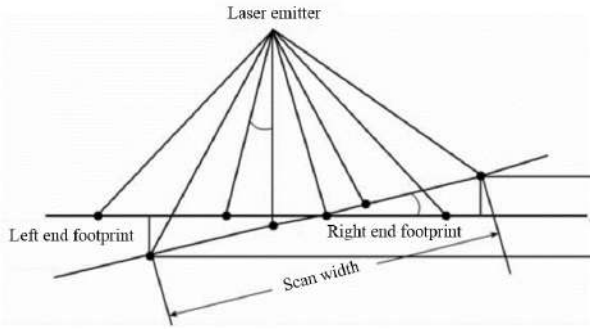
Their principles are described in detail below.

Manual Measurement and Inspection

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

Estimate the Roll Angle Error (ΔRoll)

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



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

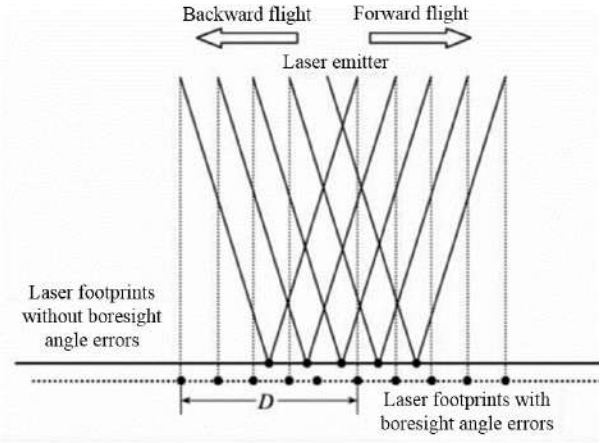
- Open the profile perpendicular to the direction of flight to measure the height difference of the approximate object of the same name.
- Measure the horizontal distance r between the approximate object of the same name and the center line of the two air belts in the 2D view.

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

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

Estimate Pitch Angle Error (ΔPitch)

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



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

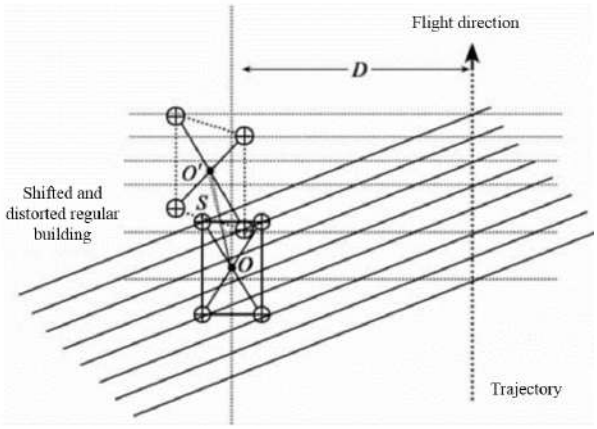
- Open the profile parallel to the flight direction to measure the distance difference D between the center position of the same object along the flight direction.
- Calculate the average flight altitude H based on the trajectory (try to keep the round-trip flight altitude as consistent as possible).

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

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

Estimate Heading Angle Error (ΔHeading)

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



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

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

The estimation formula for heading angle error is as follows:

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

Automatic Adjustment Correction

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

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

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

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

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

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

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

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

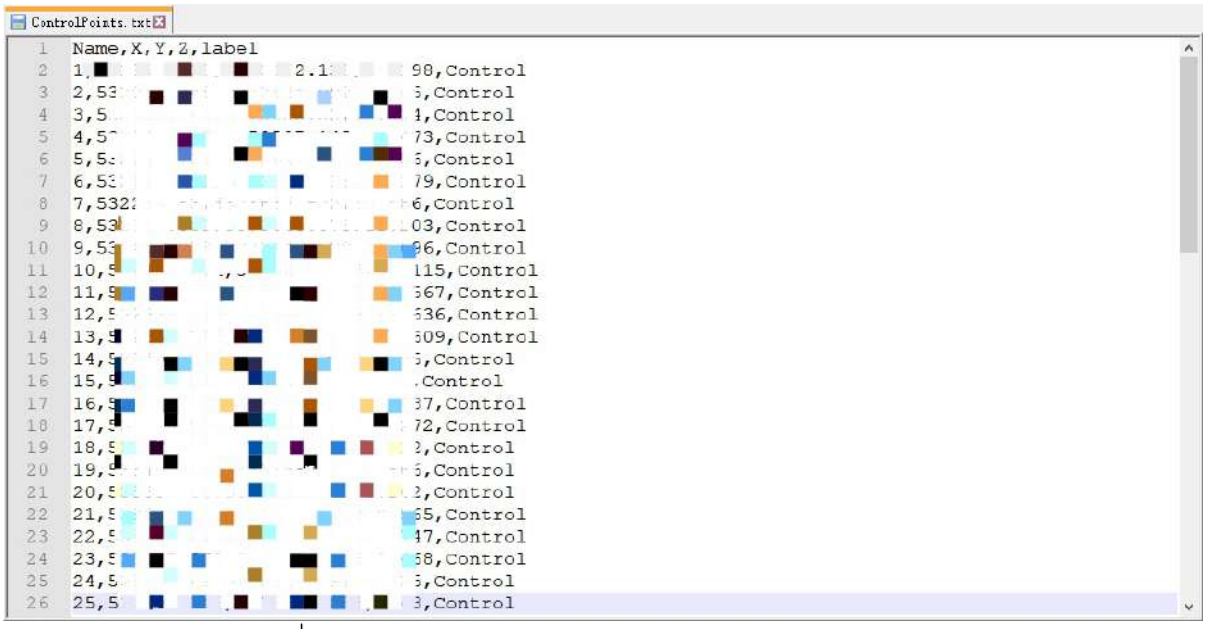
GCP Adjustment

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

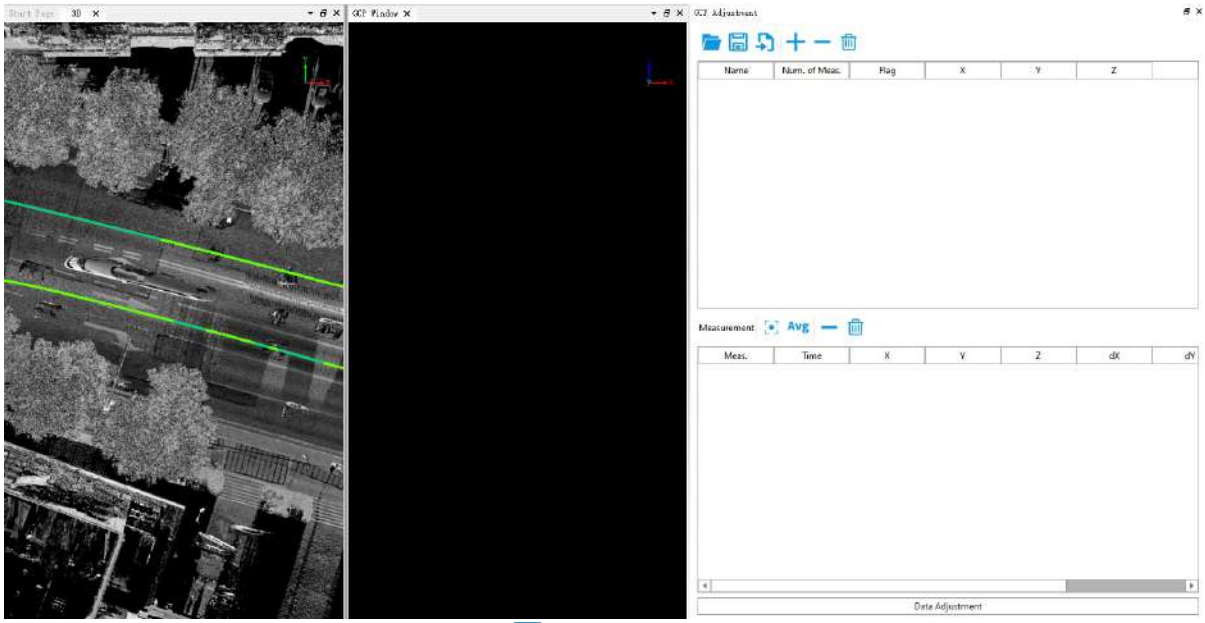
Steps

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

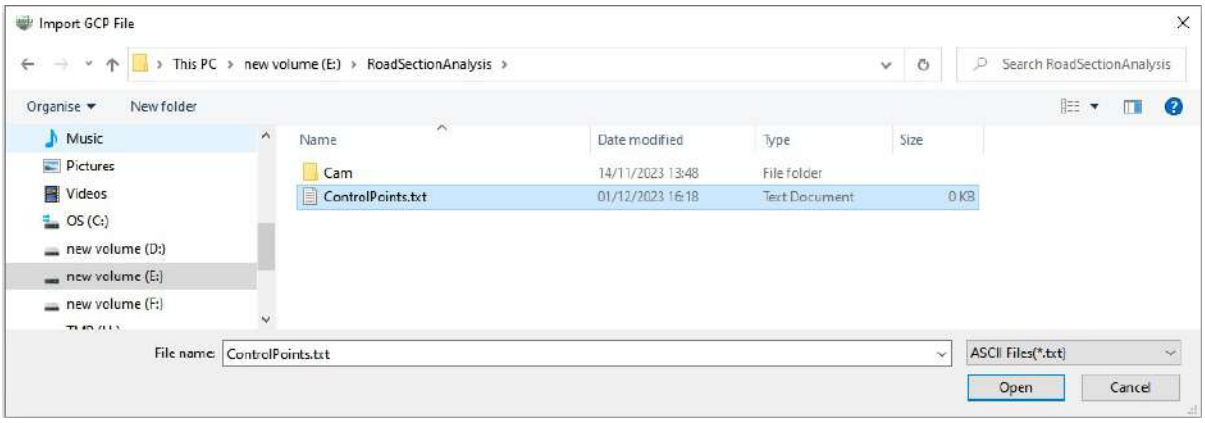
- 1.1. Source data: It contains the original point cloud to be corrected and the corresponding trajectory file. If there is an image file during the data calculation, the image data is also required. Among them, LiDAR360 MLS software can be used to directly open the .ligeo project file solved by Ligeo,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



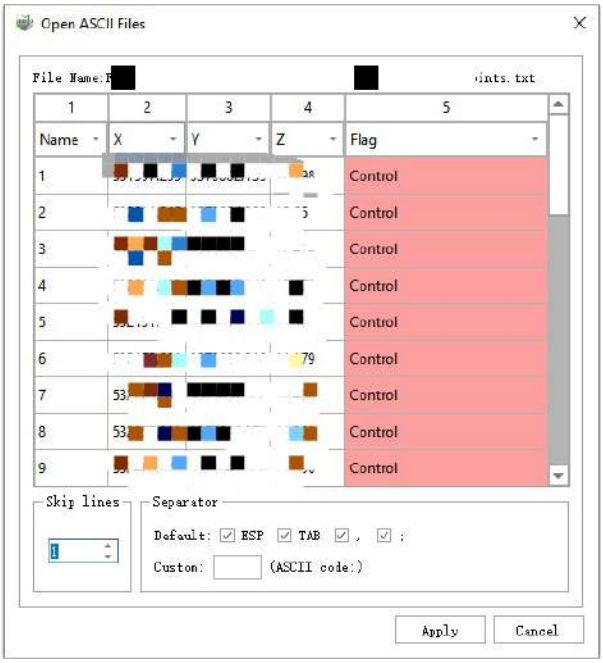
2.Click **GCP Adjustment** button to activate this function, the control point correction dialog box will appear on the right side of the main page, and the GCP window will appear at the same time.



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



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



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

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

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

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

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

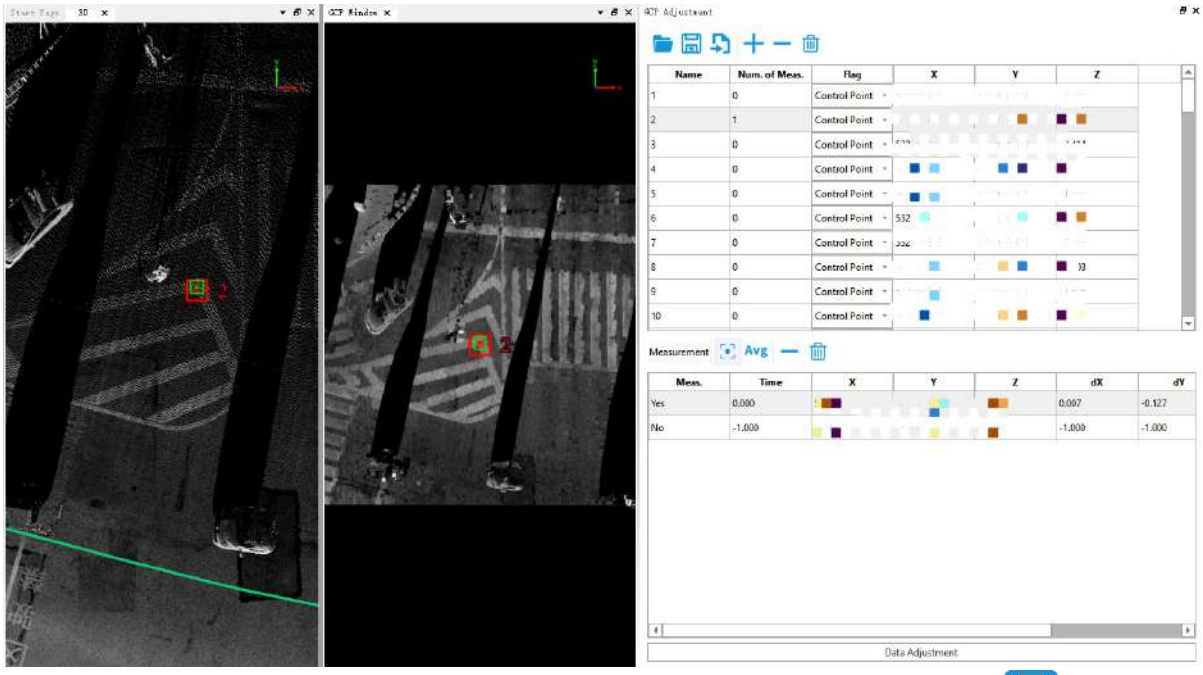
(3) Control point guide dialog:

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

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

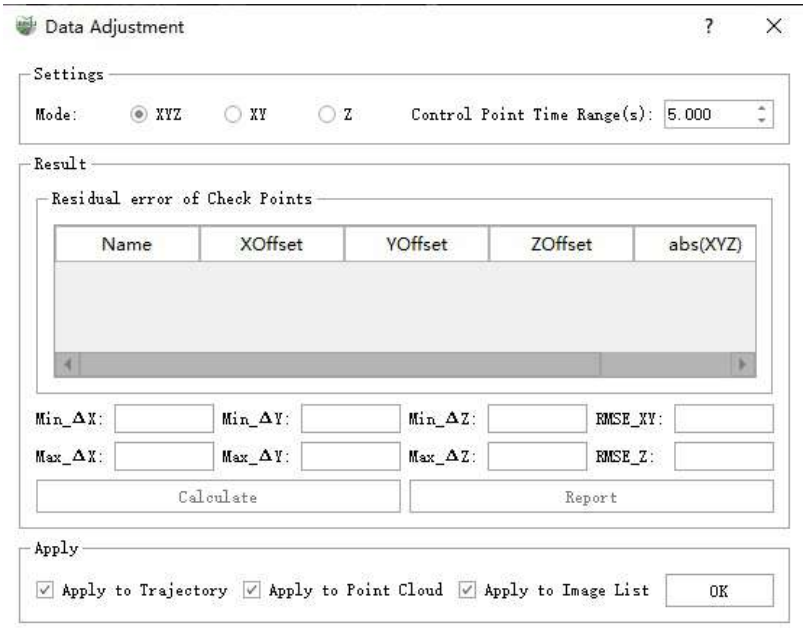
(3.2) The lower part displays the information about each control corresponding point to be punctured, click any record, and then use the mouse to click the point of the point cloud corresponding to the current control point in the GCP window with the left mouse button to puncture:

- Meas.: No means that the flight zone is not constrained by the corresponding control point, Yes means the flight zone is constrained by the current control point.
- Time: The collection time of the current puncture point position.
- X: The east coordinate of the point cloud representing the current puncture point position.
- Y: The north coordinate of the point cloud representing the current puncture point position.
- Z: The point cloud elevation value representing the current puncture point position.
- dx: The offset between the point cloud and the east coordinate of the current control point.
- dY: The offset of the point cloud from the north coordinate of the current control point.
- dZ: The coordinate offset between the point cloud and the current control point elevation.



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

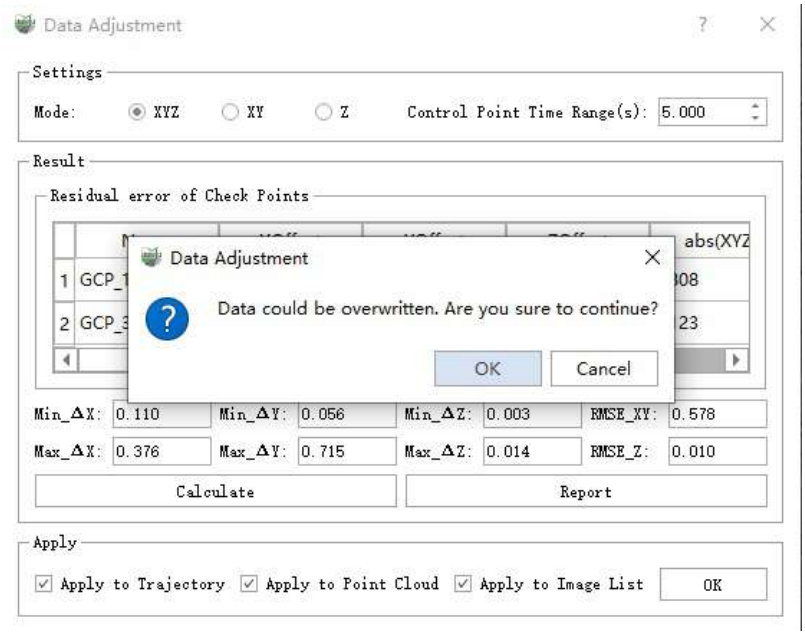
7.Click **Data Adjustment**, and the **Data Adjustment** dialog box will pop up. In this dialog box, set the correction mode, perform the correction calculation, and view the checkpoint residuals. Click the **Calculate** button to calculate the control point correction parameters according to the puncture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



Parameter Description:

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

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



Strip Adjustment

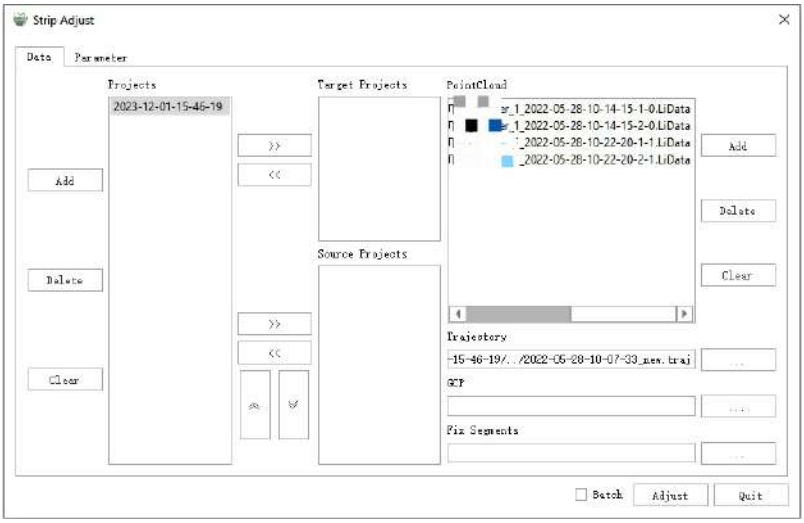
Function Description: The strip adjustment supports the adjustment of the point cloud revisited area within the vehicle data of a single project; supports the adjustment calculation of multiple groups of projects, and aligns the registration project point cloud to the reference to the engineering point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

Parameter Description

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

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

2.**Project Data**

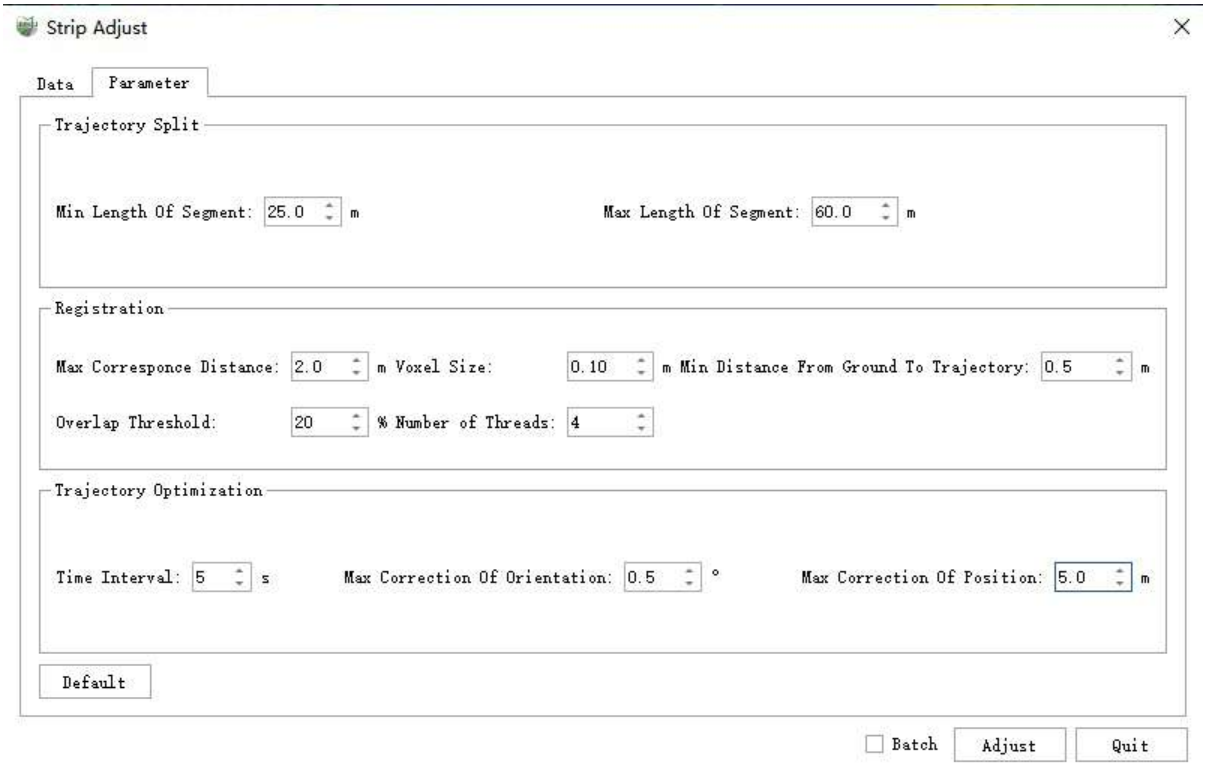


- **Add Project**: Add project files (.LiStreet, .LiMMP) from outside.
- **Delete Project**: Delete the added project.
- **Clear Project**: Clear all projects.
- **Add target project**: Select one in the project bar and click Add to target project.
- **Remove Target Project**: Remove the added project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

the point cloud file.

- **Clear Point Cloud**: Select any target/matching project, clear all point cloud files in the project and set again.
- **Select Track**: Select any target/matching project and reset the track file.
- **GCP**: Select any target/matching project, import the GCP control point file corresponding to the current project, and the file will be saved and generated by saving the puncture points through the Control Point Correction function.
- **Fixed trajectory segmentation**：Select any target/matching project, import the trajectory segment file corresponding to the current project, and this file is generated by selecting part of the trajectory through the Trajectory Segmentation function. The selected trajectory indicates that these data will not be processed during stitching.

3.**Adjustment Parameters**



- **Trajectory Split**:
  - **Min Length of Segment**: The minimum length of the track segment result. It is recommended to set the default value.
  - **Max Length of Segment**: The maximum length of the track segment result. It is recommended to set the default value.
  - It can be adjusted according to the size of the collection area, the open area can be appropriately enlarged, and the shaded area should be reduced. Segmented point cloud is based on trajectory segmentation and provides matching point cloud for segment matching.
- **Registration**:
  - **Max Correspond Distance**: Matched pairs whose distance is greater than this value do not participate in the adjustment. It is recommended to set the default value.
  - **Voxel Size**:In the matching process, set the voxelization sampling interval for matching point clouds. Setting it too large may cause the disappearance of key points, while setting it too small results in a large computational load. It is recommended to use the default settings.
  - **Min Distance from Ground to Trajectory**: Ground points with a distance from the track height less

than this value will be filtered. It is recommended to set the default value.
    - **Overlap Threshold**: Matching pairs with overlap less than the threshold will be filtered. It is recommended to set the default value.
    - **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.
- **SpLine Optimization**:
    - **Time Interval**: SPLINE node time interval, the smaller the interval, the finer the pose correction and the greater the calculation amount. It is recommended to set the default value.
    - **Max Correction of Position**: The maximum correction amount of SPLINE node position. It is recommended to set the default value.
    - **Max Correction of Posture**: The SPLINE overall adjustment takes the trajectory pose corresponding to the SPLINE node as the basic adjustment unit, and adjusts the trajectory pose correction based on the SPLINE assumption to perform adjustment optimization to achieve the corrected trajectory pose splicing point cloud and guarantee the corrected trajectory pose smooth and specific effects. The maximum amount of correction for the attitude of SPLINE nodes. It is recommended to set the default value.

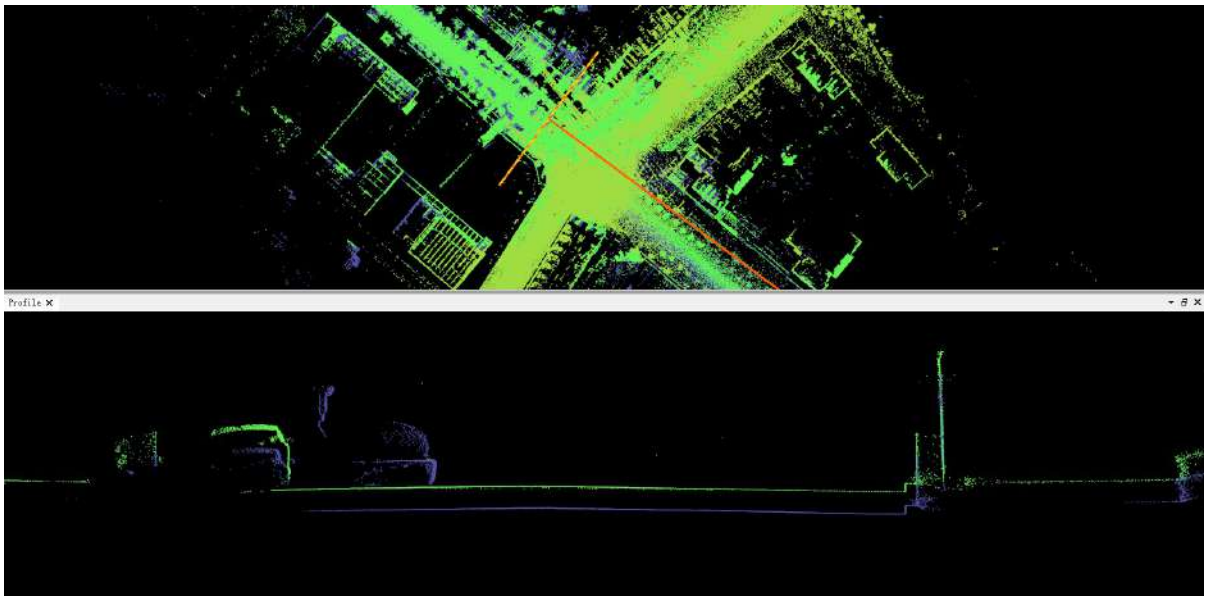
Single Project Strip Adjust

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

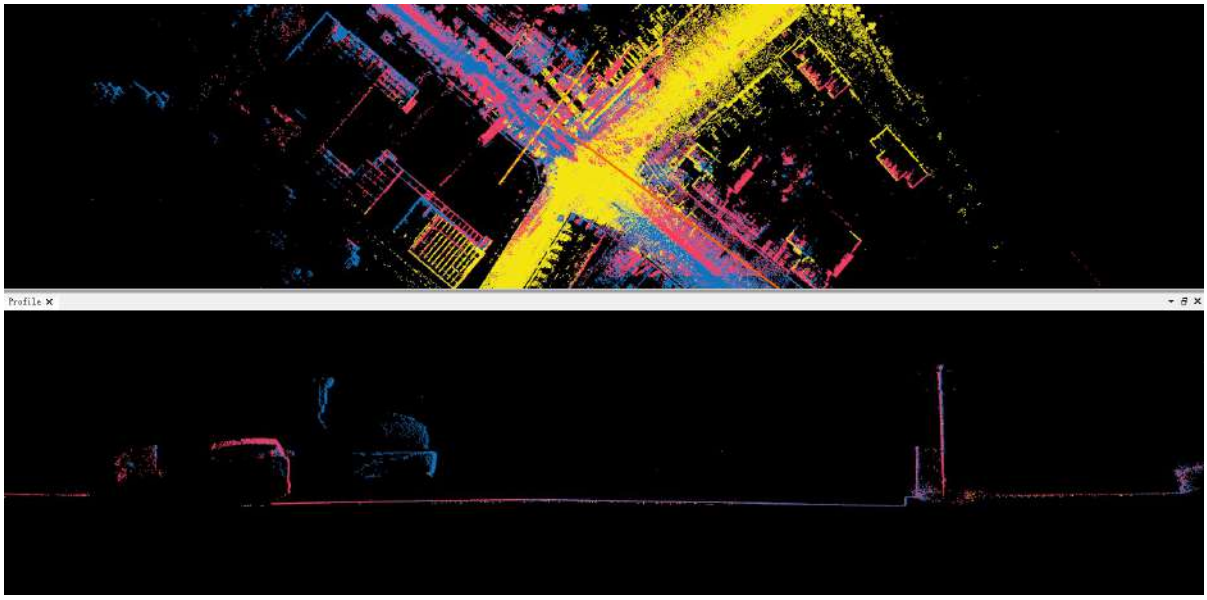
2.Click the function button, in the setting of the air belt splicing project, move the current project to the matching project. If internal splicing of control point parameters is required, you need to pass Control Point Correction in advance to puncture the control point and save it as an .xml file,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

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

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



Before the single-engineering air belt splicing



After the single-engineering air belt splicing

Multi-project Strip Adjust

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

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

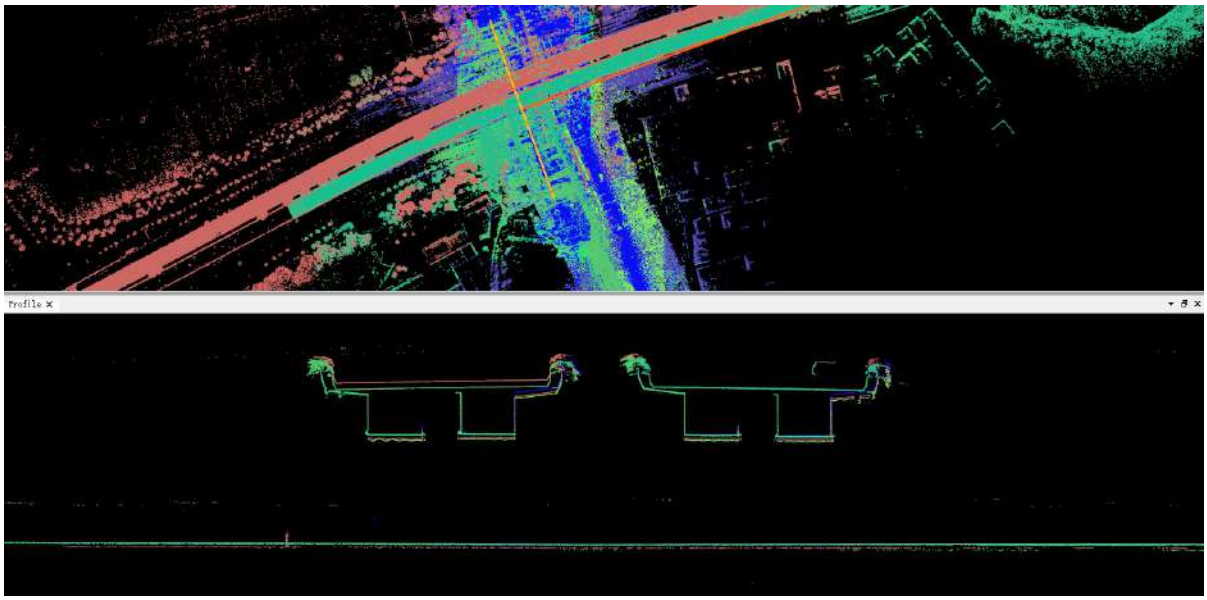
3.According to step 1, it is known that project A can be used as a target project, and BCD needs to be used as a matching project, then in the project list, select project A, and move project A into the project A through the **Add Target Project** button on the left side of the target project list. Then in the project list, select the BCD project respectively, and move the BCD project into the matching project list by pressing the **Add Matching Project**

button on the left side of the matching project. And according to step 1, the splicing sequence is project A←project B←project C←project D. Therefore, you can adjust the order of BCD projects by matching the up and down buttons on the left side of the project.

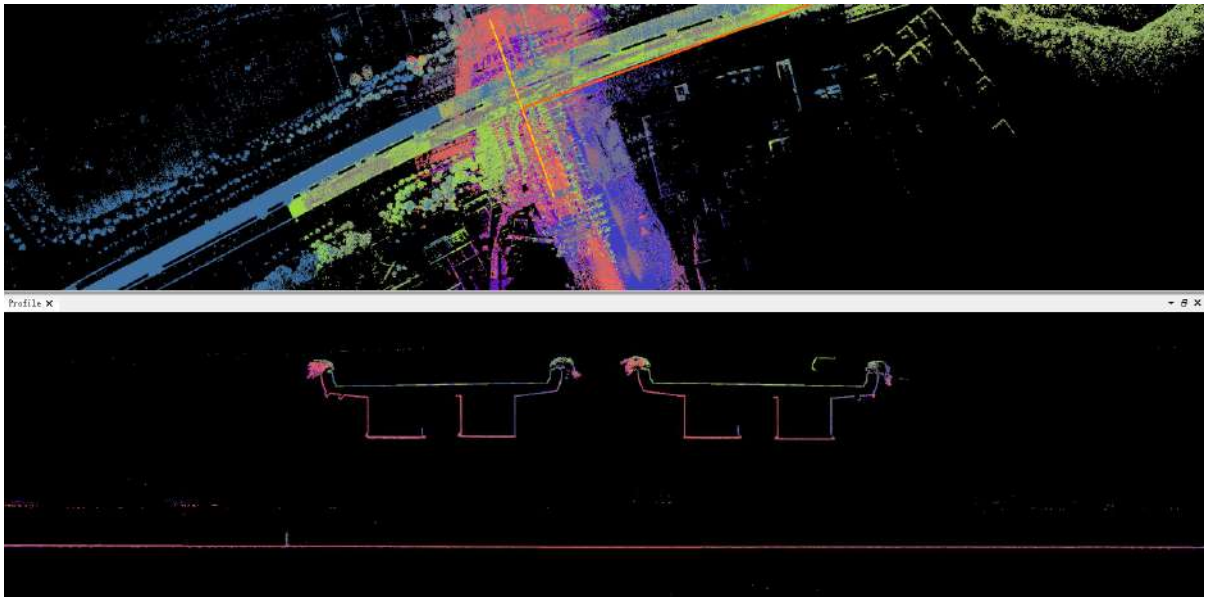
4.Click on each project to view and modify the point cloud, trajectory and control point data of the current project on the right side of the air belt splicing interface. If the control point data is set in a single project, the control points will participate in the adjustment during the process of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

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

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



Before multi-engineering air belt splicing



After the multi-engineering air belt splicing

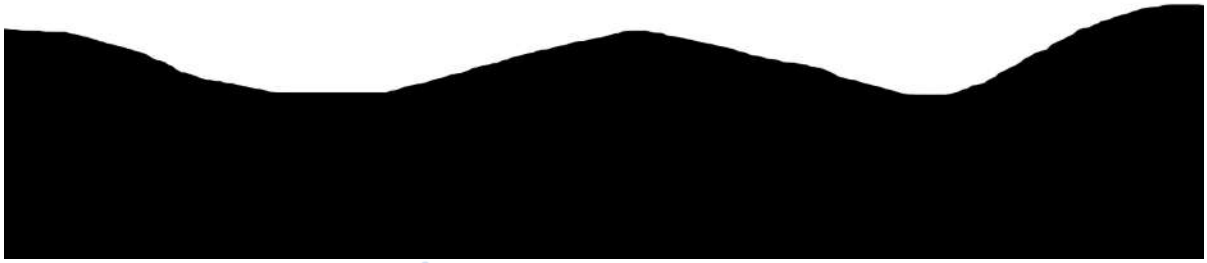
Colorize Point Cloud

Function Description: Point cloud colorization is the process of assigning colors to point cloud data based on the information from images. It currently supports panoramic cameras and single-plane cameras. The following sections provide detailed information about each mode.

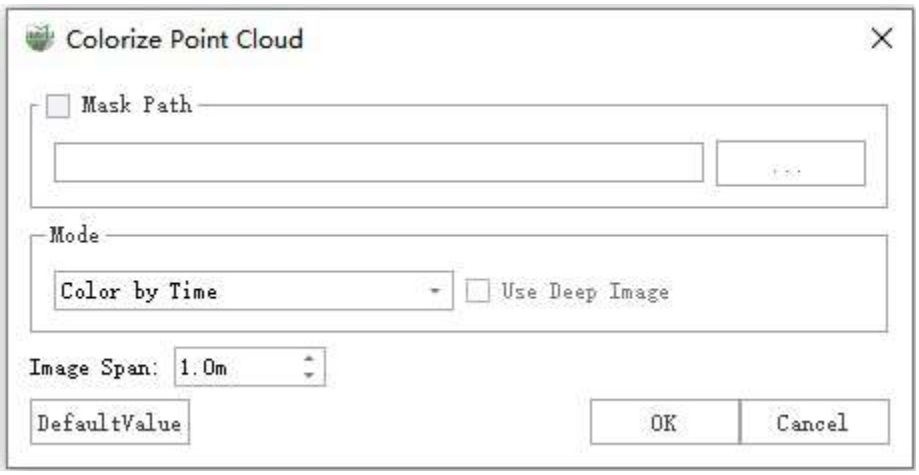
Mask File

1.A mask file is a black-and-white image file. It is generated by filling the occluded areas with black and leaving the visible areas as white based on the occlusion regions where vehicles and equipment are located.

2.During the colorization process, the black regions are not involved in the colorization to avoid assigning incorrect colors to the ground or other objects.



3.Click on the **Colorize Point Cloud** function to bring up a pop-up box:



Parameter Settings

- **Mask File**: Activate this option to use a mask file. You can choose your own mask file. If this option is

unchecked, the colorization will include all areas of the image.

- **Colorization Modes**
  - **Time Colorization**: Assigns colors based on the closest image in terms of time.
  - **Distance Colorization**: Assigns colors based on the closest image in terms of distance.
- **Use Depth Map**: Enabling the use of a depth map can greatly reduce the occurrence of color misassignments, such as assigning streetlight colors to the ground behind. However, this option will increase the processing time and is only available for distance colorization with panoramic images.
- **Photo Interval**: This parameter is used to control the density of the images used for colorization. A larger interval will result in fewer images being involved in the colorization process.

Cut Block

The functions included under the Data Fragmentation menu bar are:

- Subdivision
- Cut Block
- Block Display

Subdivision

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

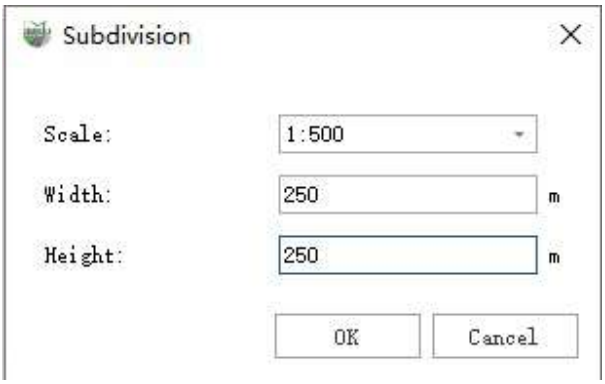
- Point Cloud Subdivision
- Point Cloud Cut Block
- Create Along Trajectory
- Create Along Center Line

Point Cloud Subdivision

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

Steps

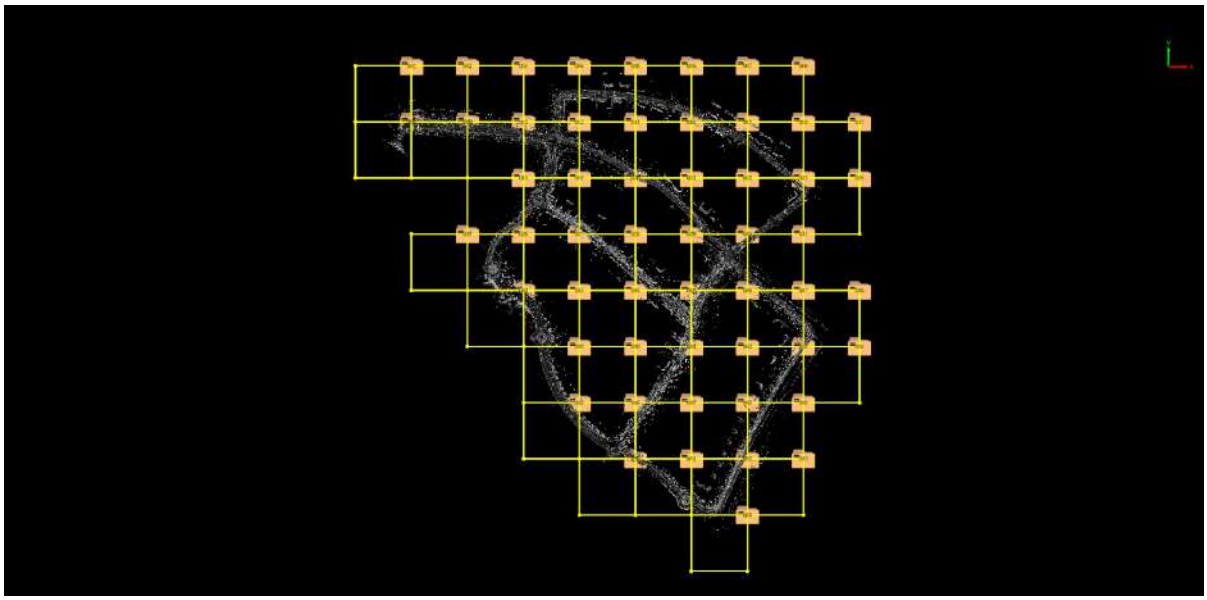
1.Click **Point Cloud Subdivision** button and the following interface will appear.



Parameter Description

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

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

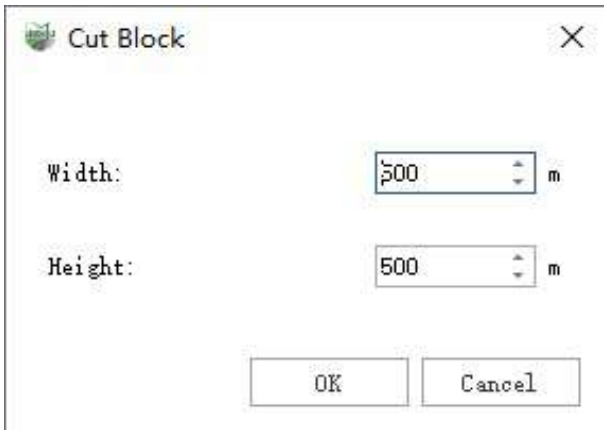


Point Cloud Cut Block

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

Steps

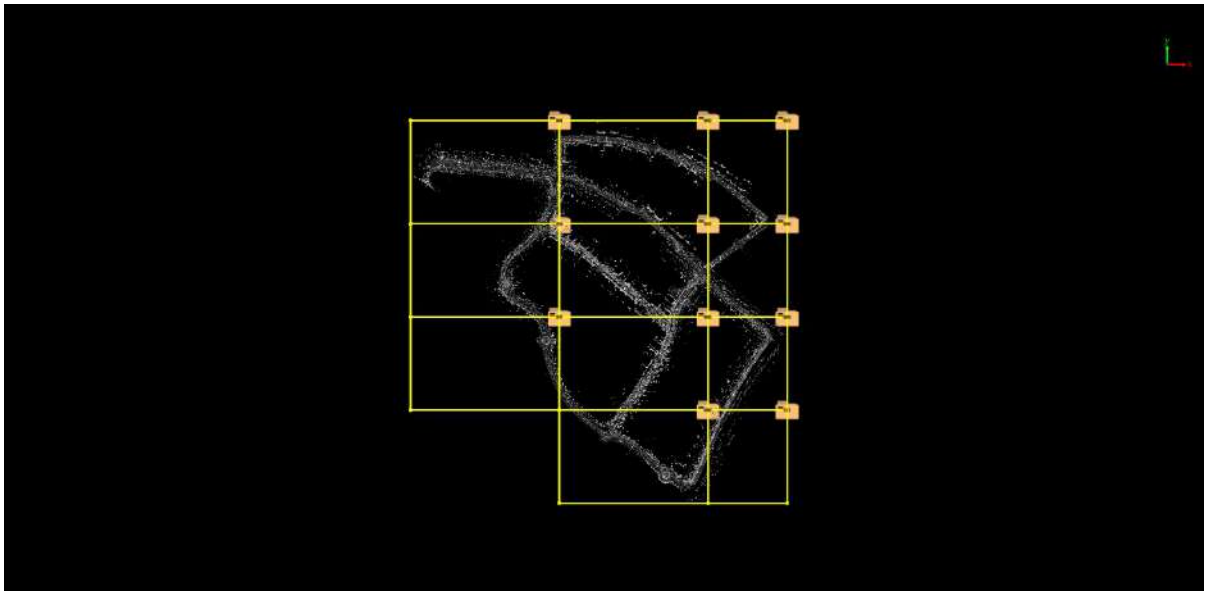
1.Click **Point Cloud Cut Block**, the following interface pops up.



Parameter Description

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

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



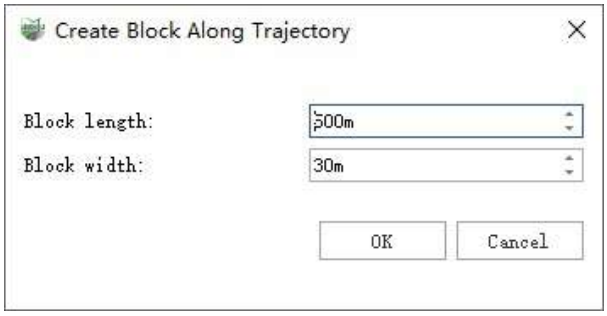
Create Along Trajectory

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

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

Steps

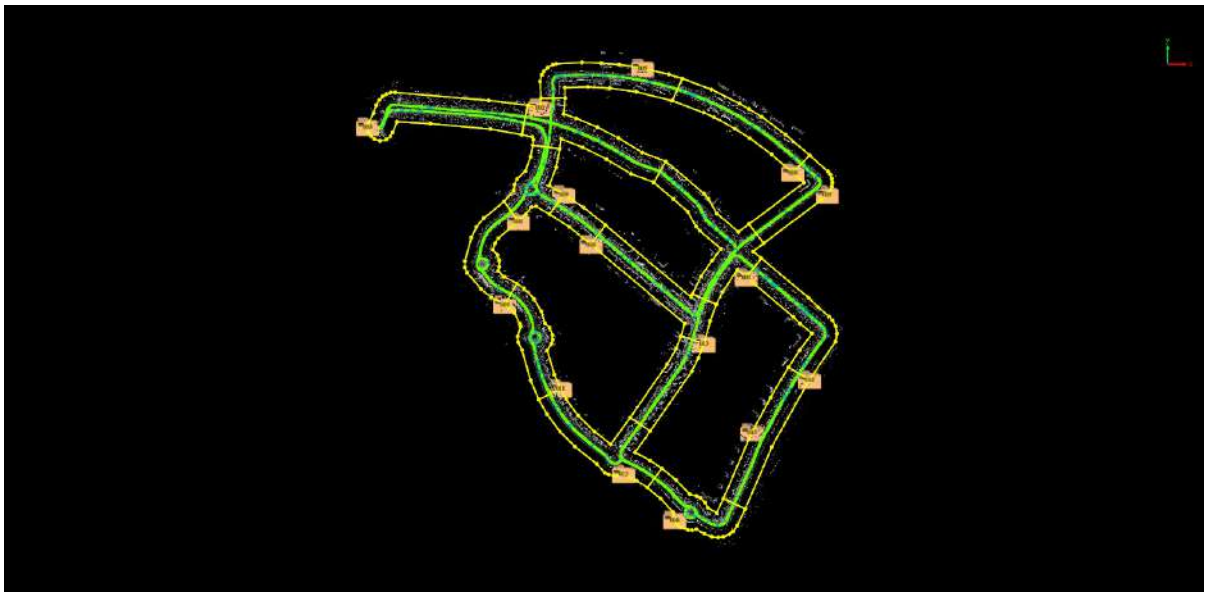
1.Click **Create Along Trajectory** , the following interface pops up.



Parameter Description

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

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



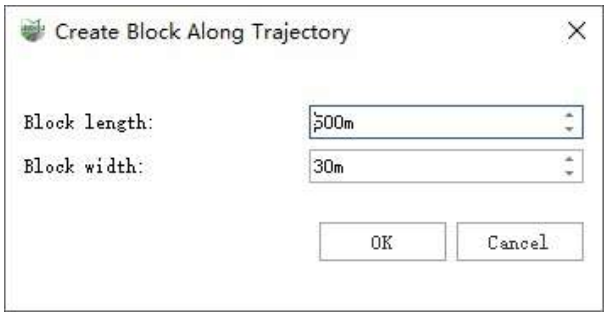
Create Along Center Line

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

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

Steps

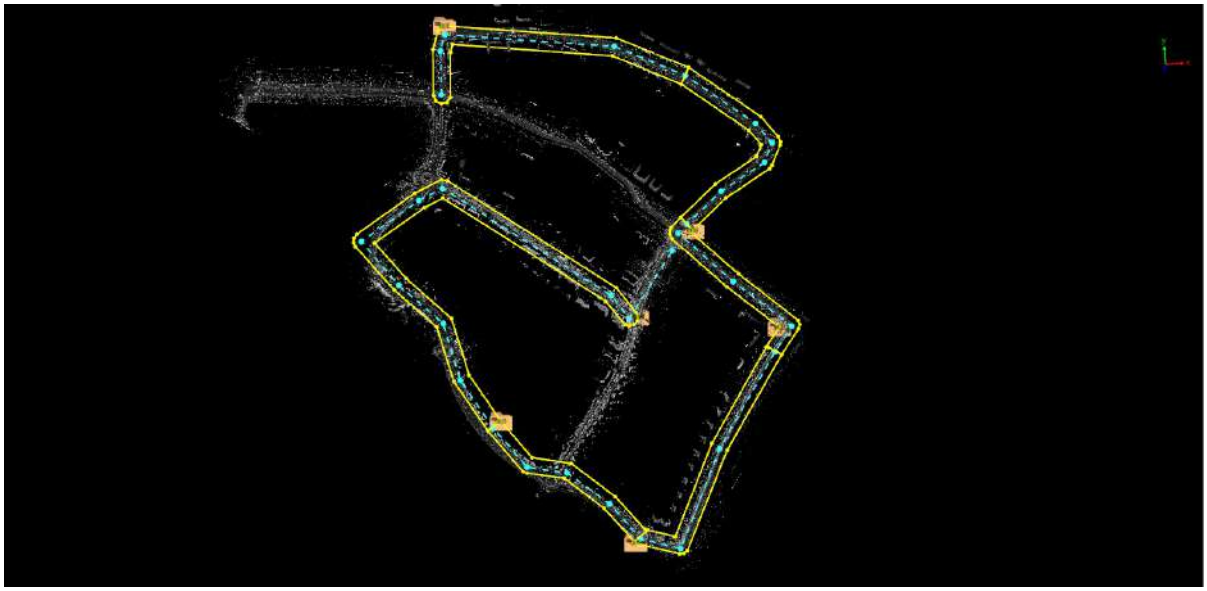
1.Click **Create Along Center Line** , the following interface will appear.



Parameter Description

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

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



Cut Block

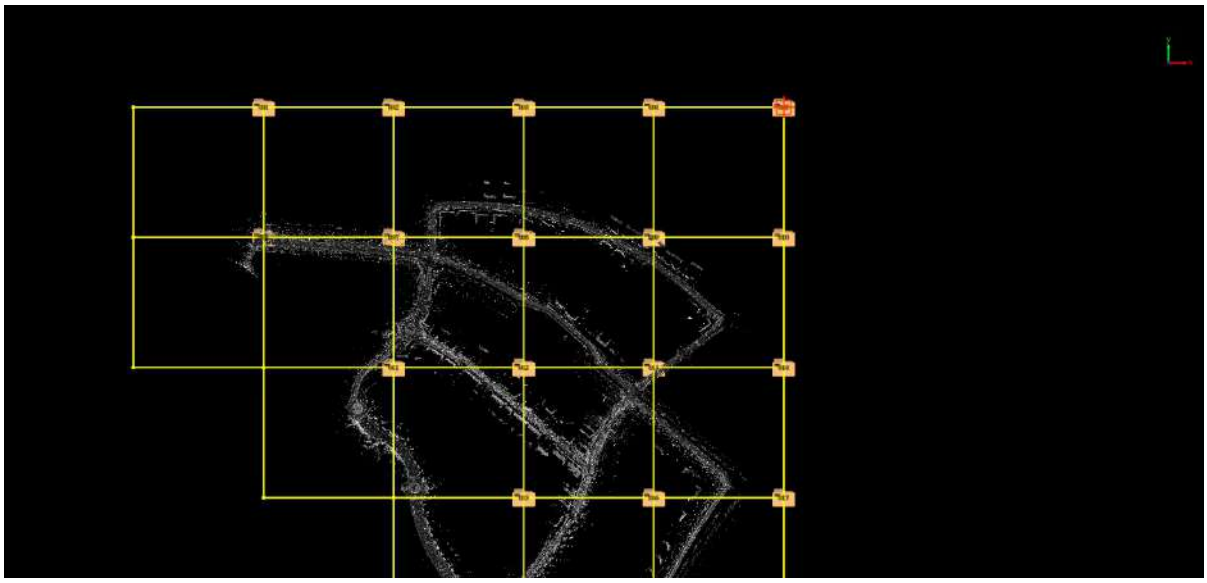
- Cut Block Edit
- Split Point Cloud

Cut Block Edit

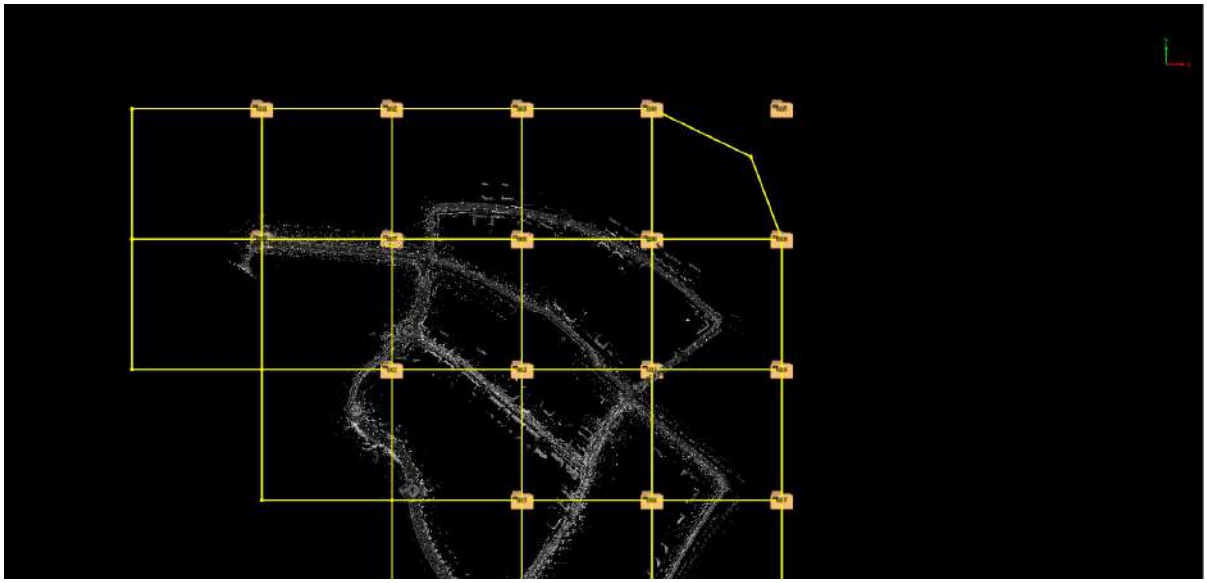
Function Description: This function is used to edit the area block after framing based on the track.

Steps

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

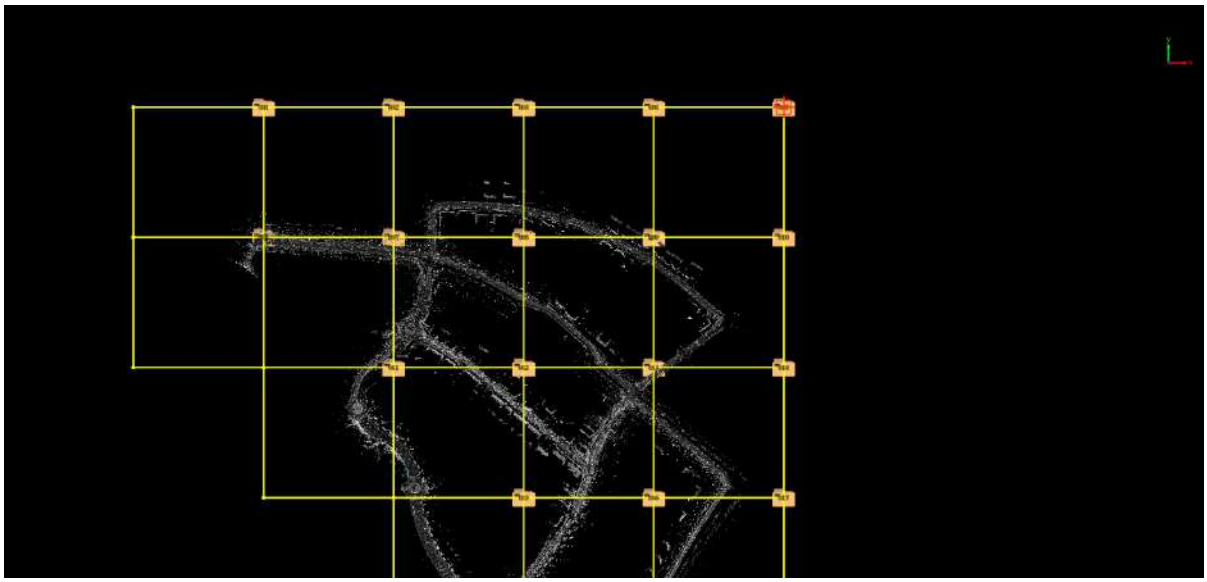


Before editing

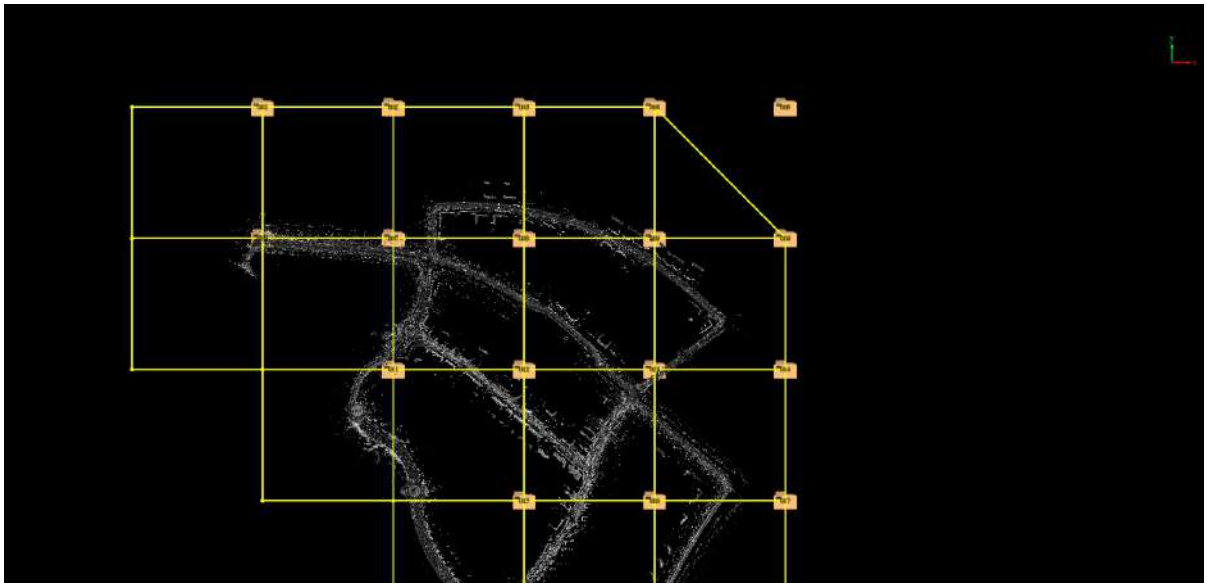


After editing

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

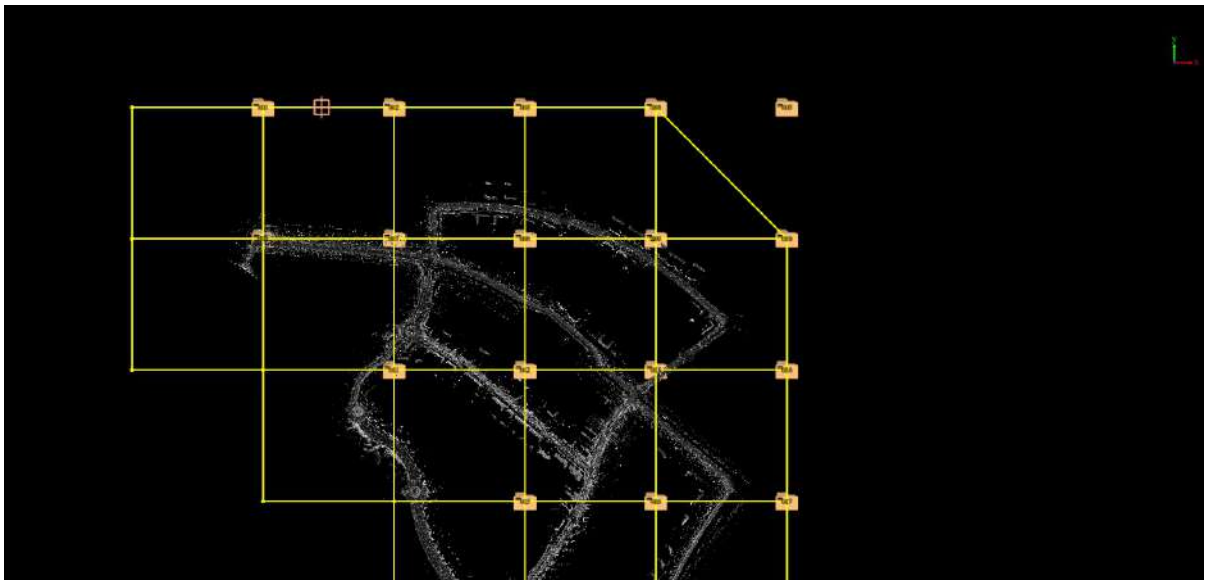


Before editing

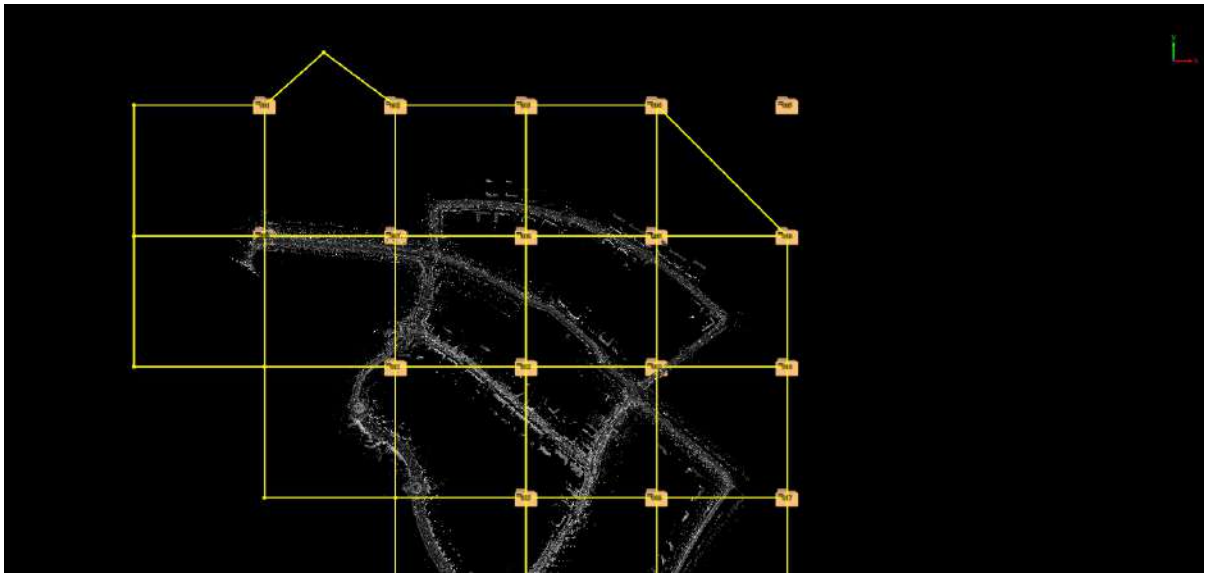


After removing the node

3.**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,

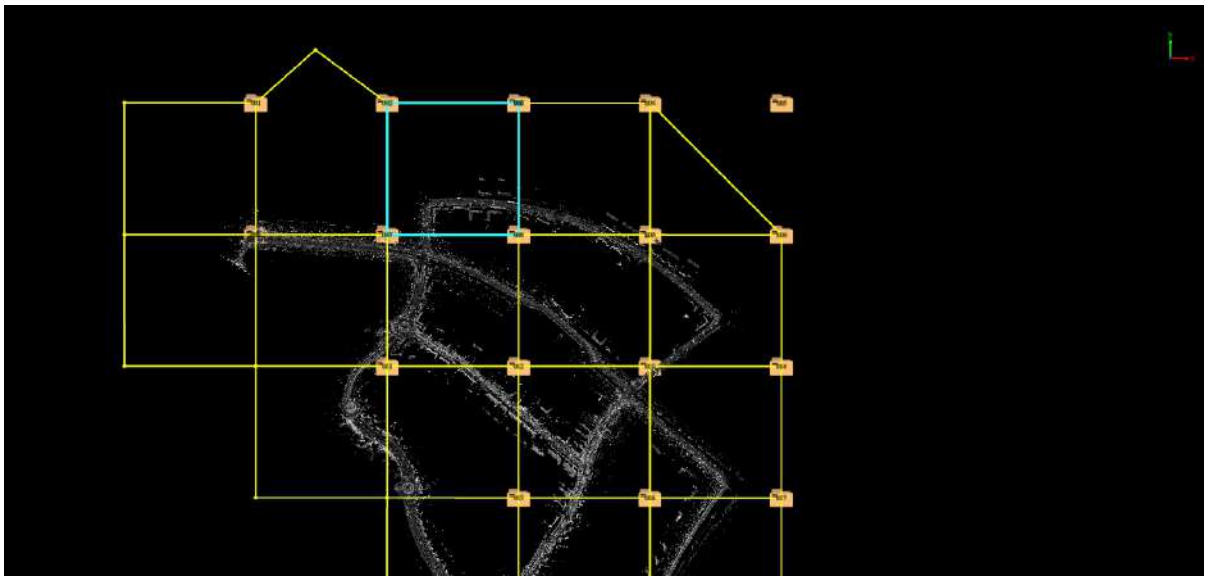


Before inserting a node

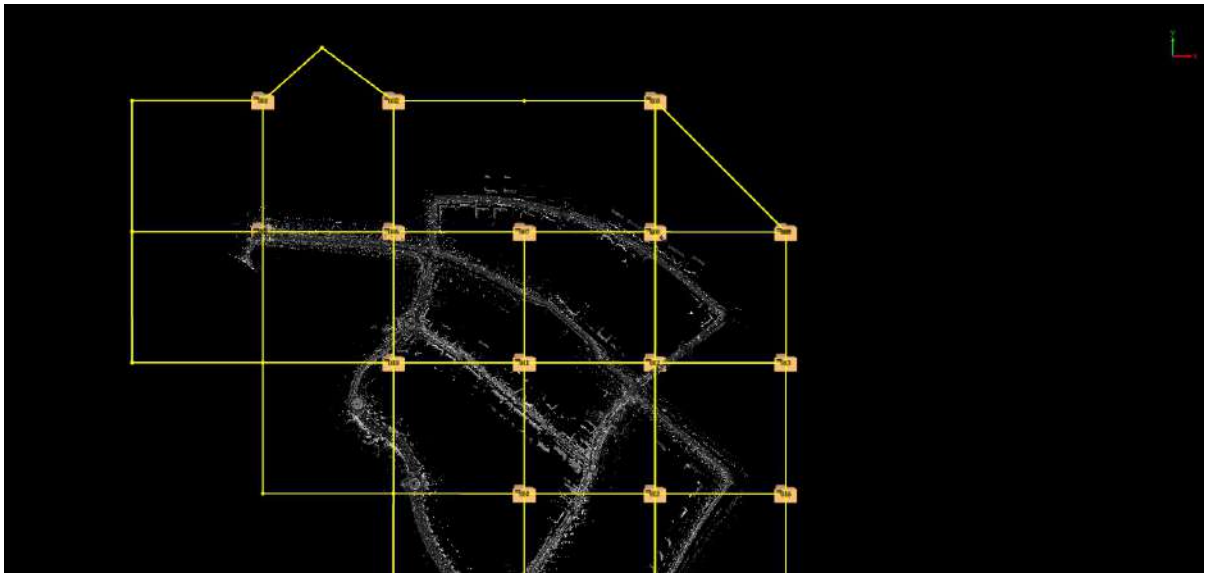


After inserting the node and moving it

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



Before the region blocks are merged



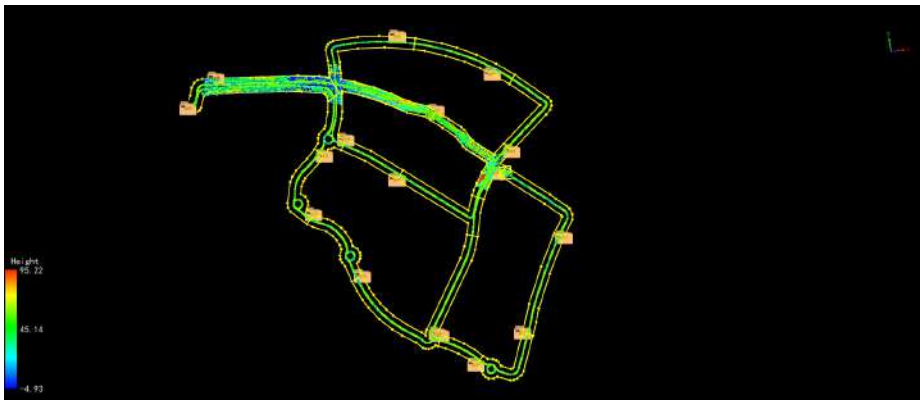
After the region blocks are merged

Split Point Cloud

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

Steps

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



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

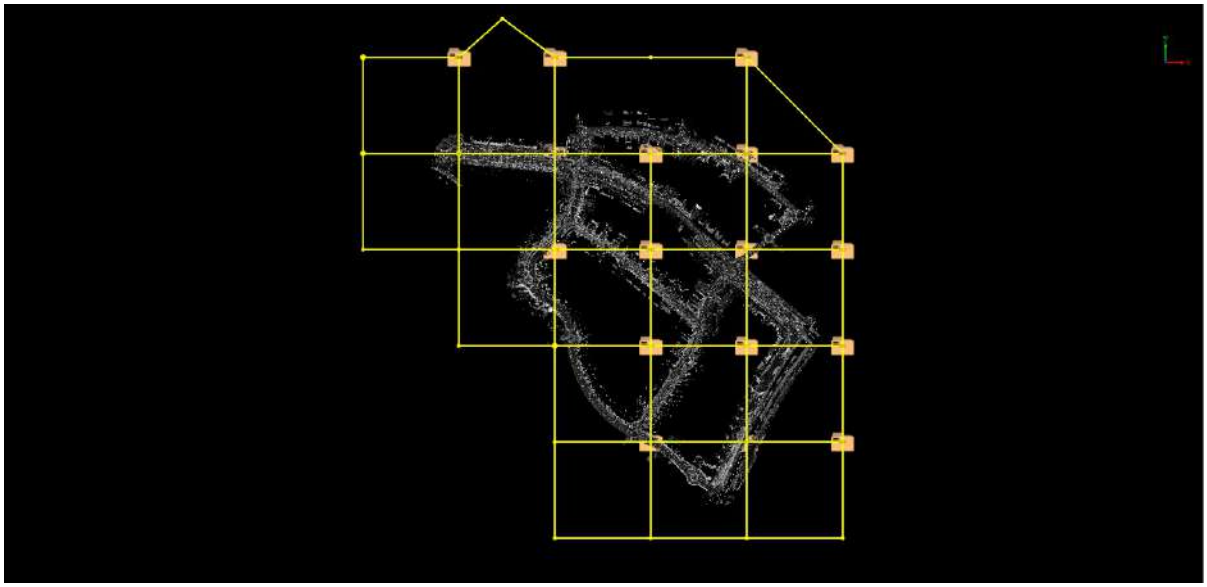
3.Click the **Yes** button to re-segment the point cloud, and click **No** to cancel the point cloud segmentation.

Cut Block Display

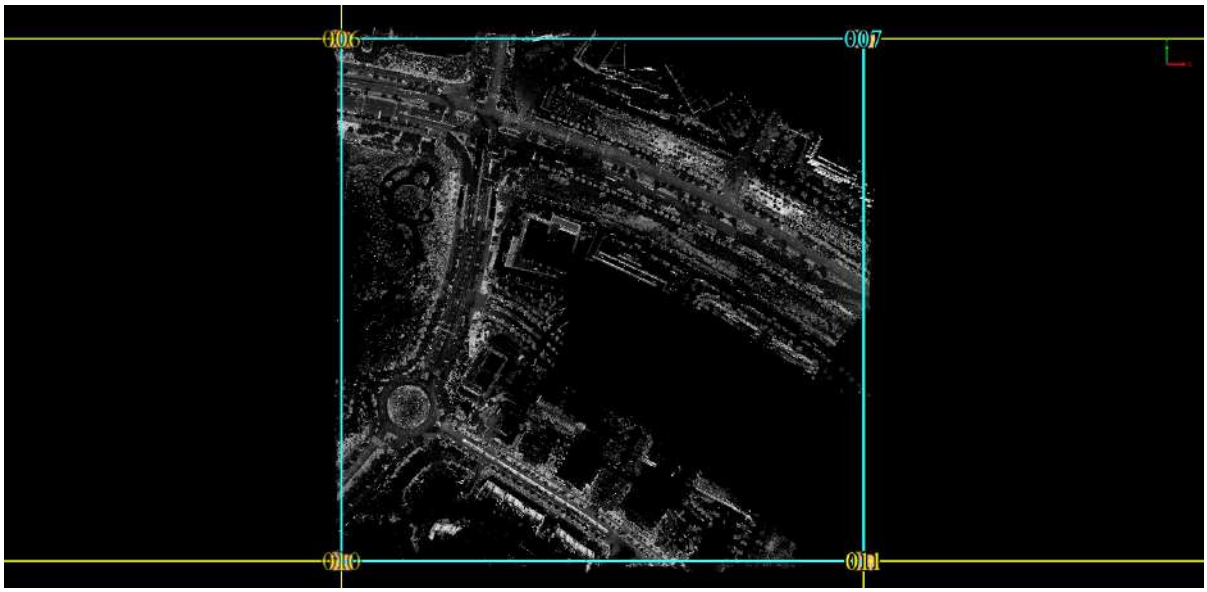
Function Description: This function selects, highlights, jumps, shows and hides each area block after framing, which is convenient for users to adjust and browse the area blocks after framing.

Steps

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

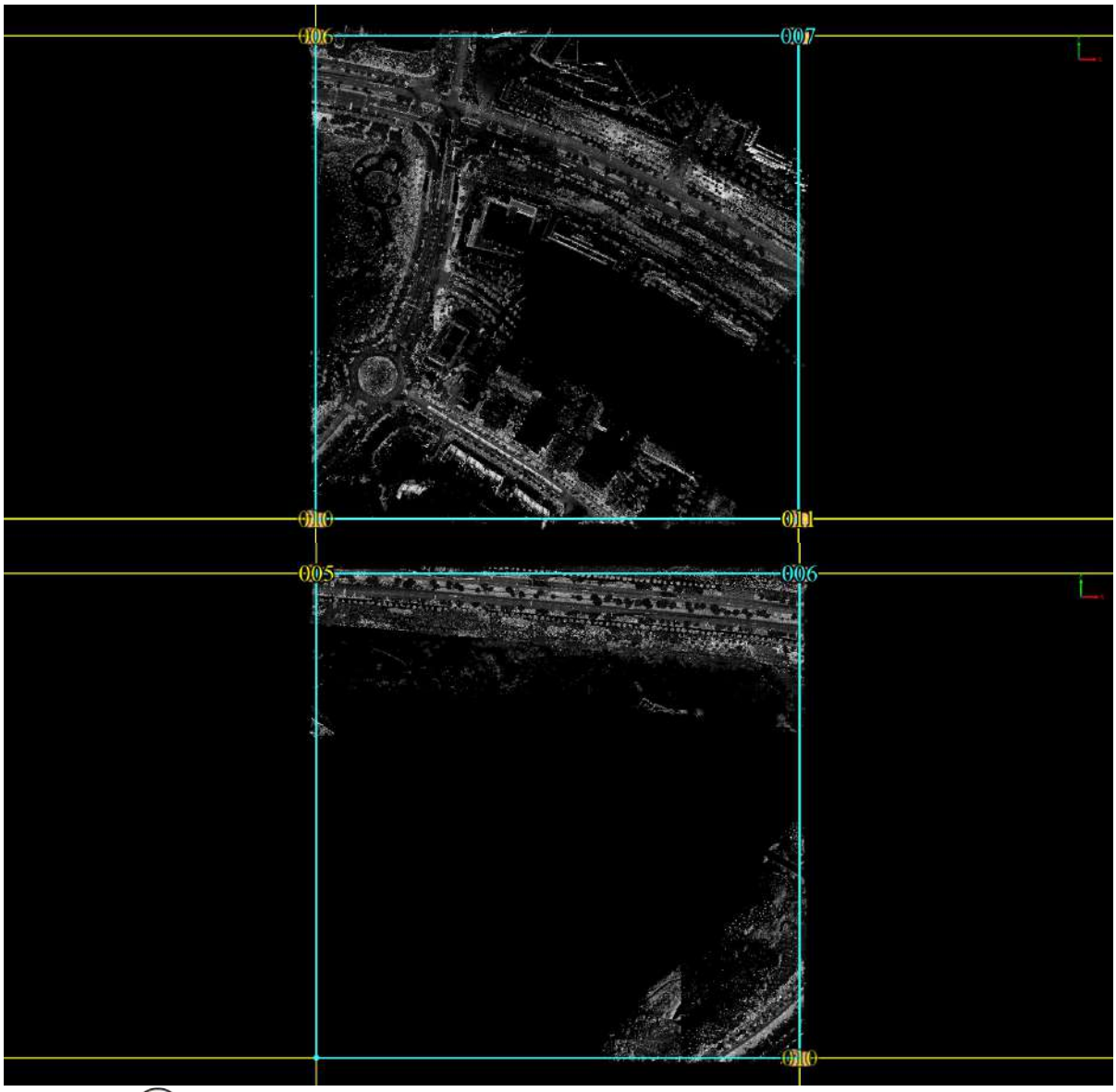


Before frame selection

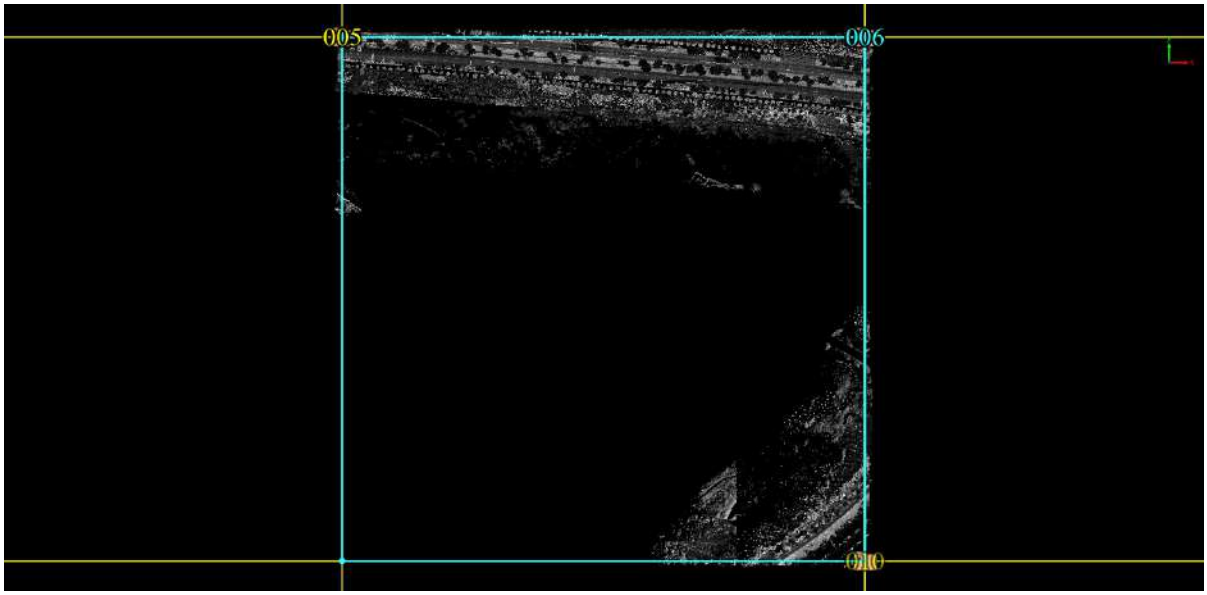


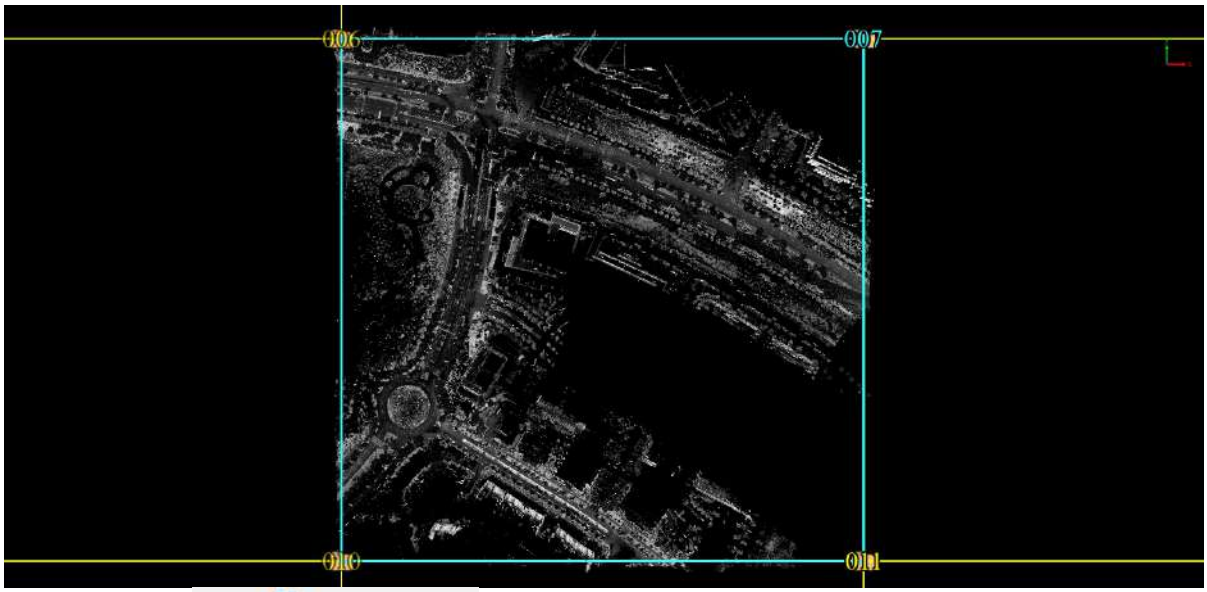
After frame selection

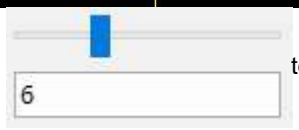
2.Click **Previous** to jump to the previous area block of the current area block number.



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



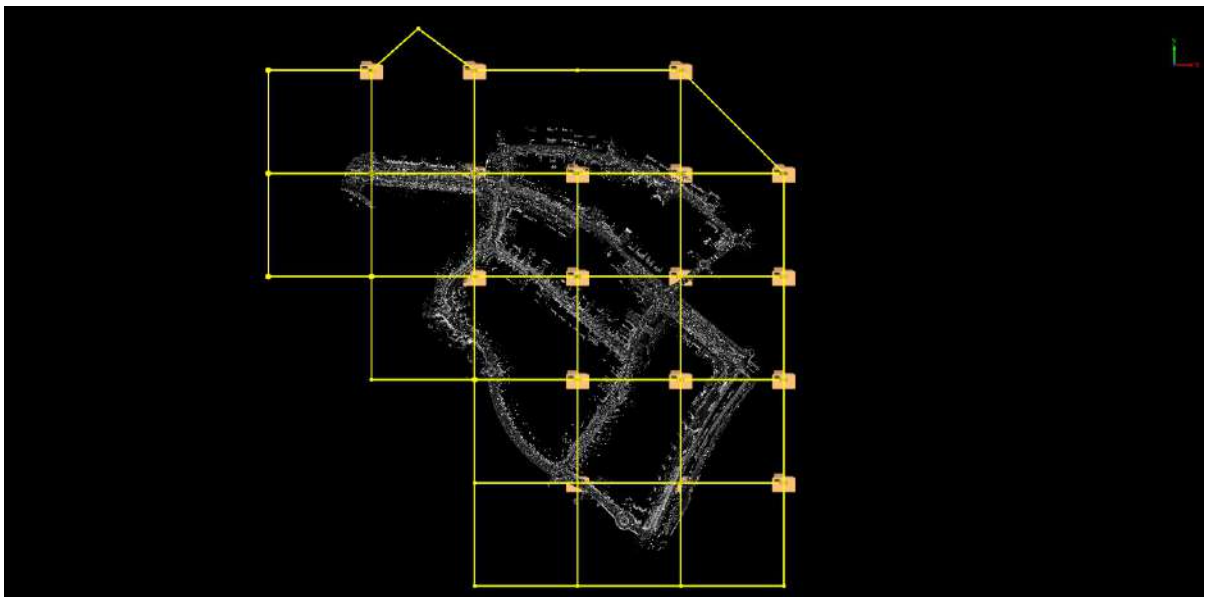


4.Drag the **slider** 

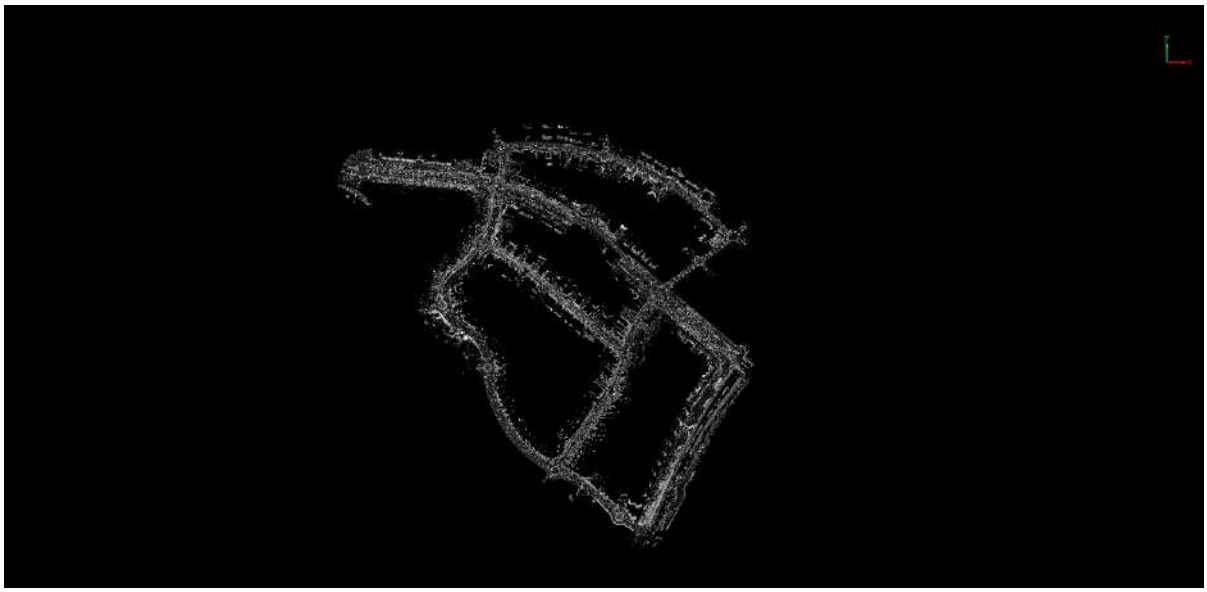
to switch different area blocks to select and highlight;

5.Select

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

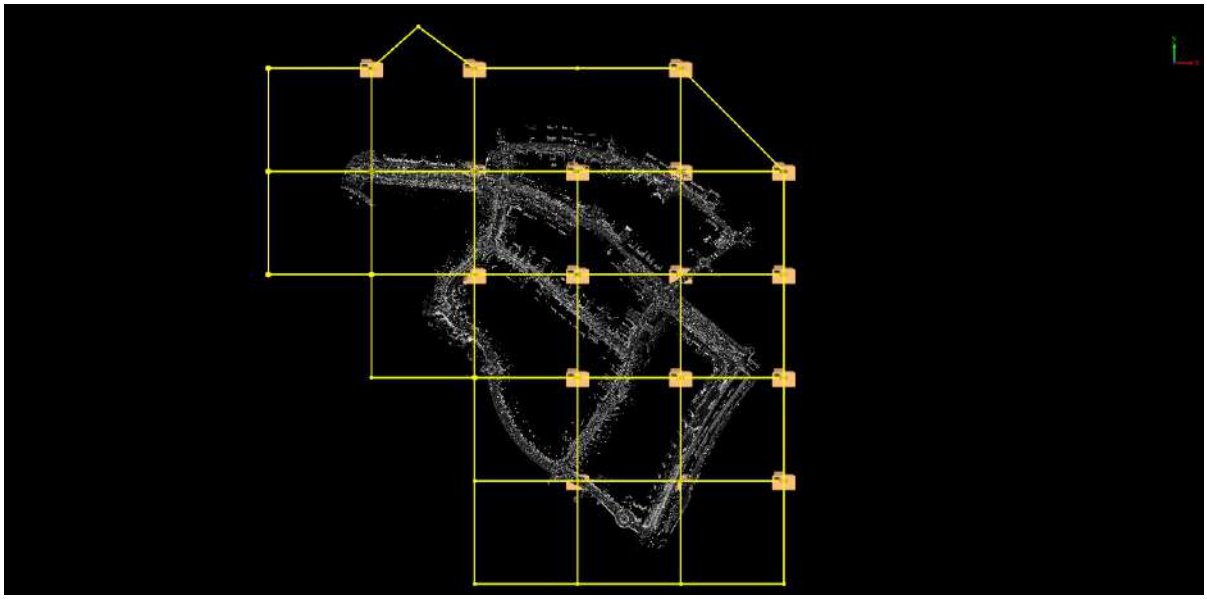


Display area block

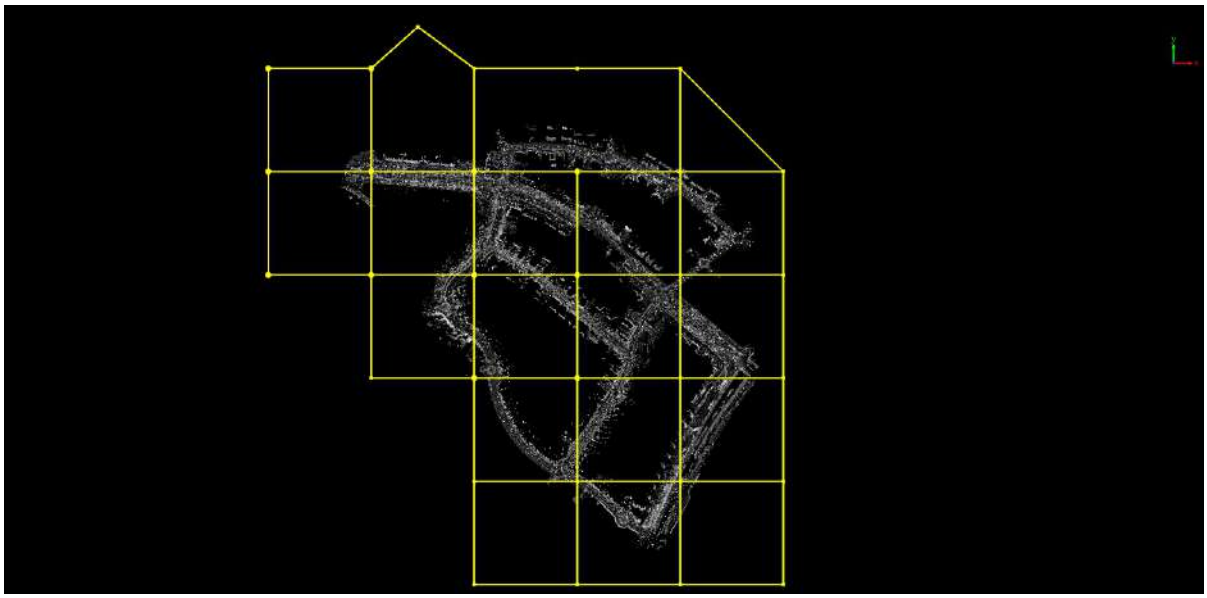


Hidden area block

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



Show labels



Hide labels

Map Features

The Map Features module is mainly used to extract road surface features through manual drawing or semi-automatic detection methods.

It includes the following:

- Road Shape
- Road Surface
- Road Vector Detection
  - Road Marking
  - Road Vector
  - Merge To Lane

Road Shape

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

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

Manual Function

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

Semi-Automatic Functions

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

Add Virtual Lane Centerline

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

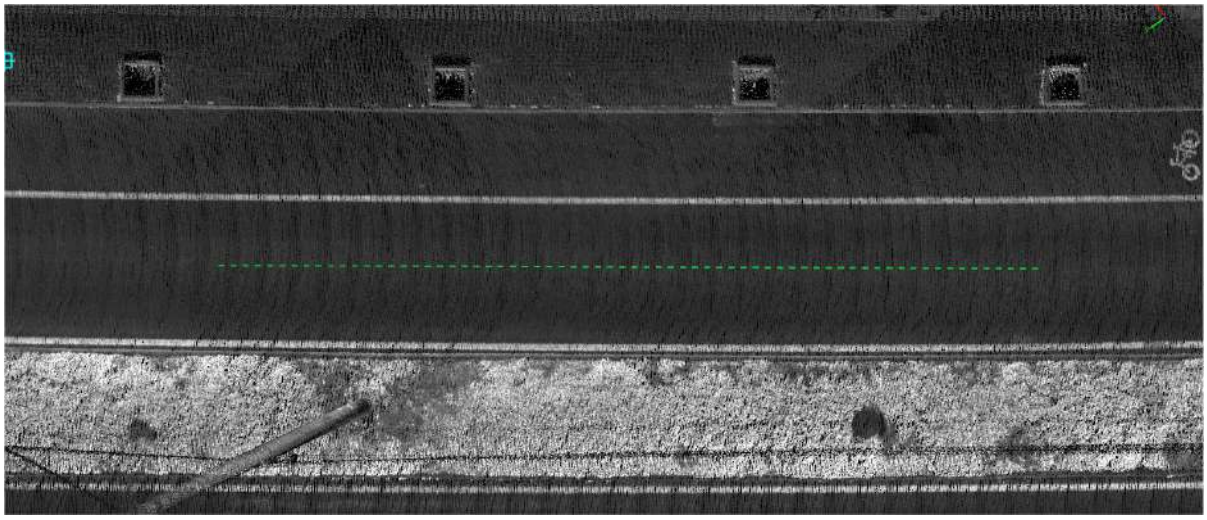
Steps

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

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

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

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



Drawing lane center line

Note:

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

Add Lane SolidLine

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

Steps

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

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

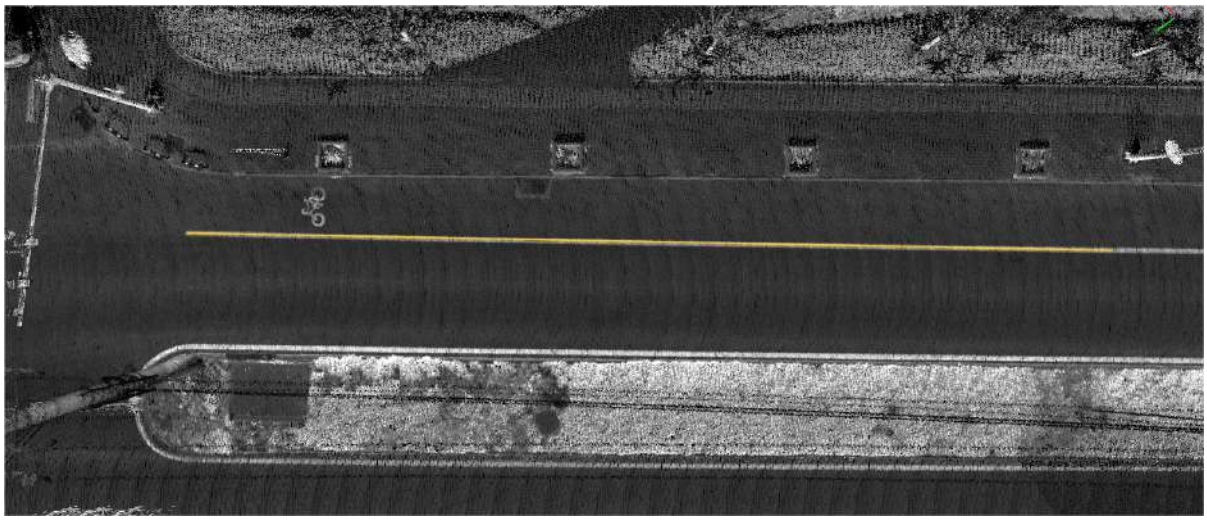
3.Move the mouse, and **left click** in turn to select the node of the Lane Solidline.

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

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

Function	Shortcut key	Description
Tracking Mode	T/t	Press the "T" key briefly to switch to tracking mode, transitioning from regular drawing to tracking mode drawing.
Line, Polygon	S/s	Short press the “s” key to switch the drawing mode from drawing a straight line to drawing an arc. The arc adopts a three-point mode. The first point is the last point before short pressing the “s” key, the second point is the end point of the arc, and the third point controls the radian
Plot Spline Curve	F/f	During the drawing process, if you need to create a Spline curve, you can press the "F" key to switch to Spline curve drawing mode. In this mode, the added nodes will be calculated and added according to the Spline curve.
Drawing Bezier Curves	G/g	During the drawing process, if you need to create a Bézier curve, you can press the "G" key to switch to Bézier curve drawing mode. In this mode, the added nodes will be calculated and added according to the Bézier curve.

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



Lane SolidLine

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

Add Lane DashLine

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

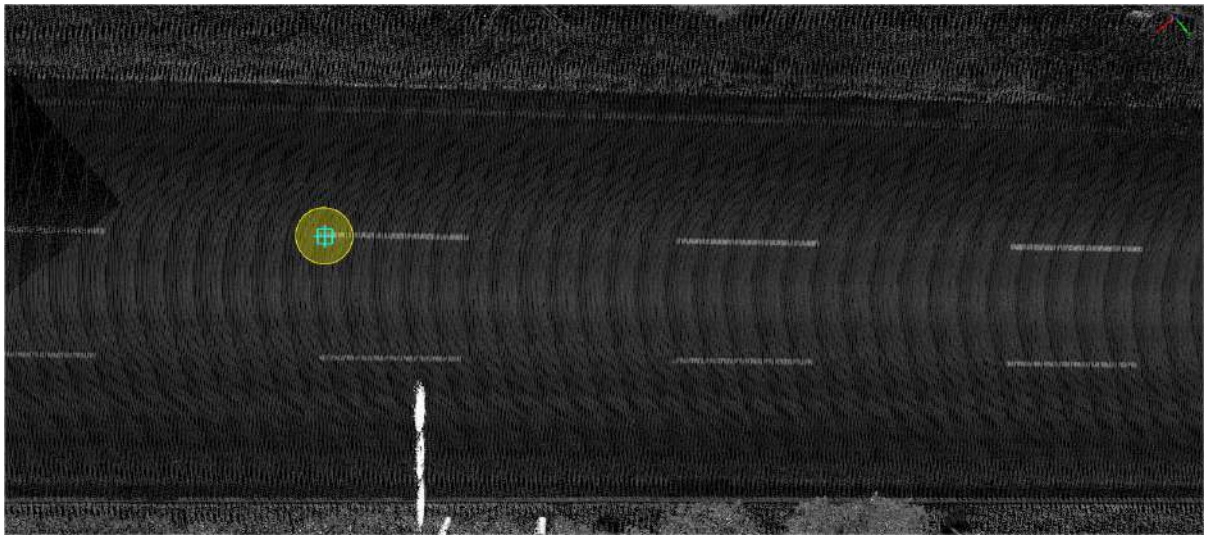
Steps

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

2.3D window tips：

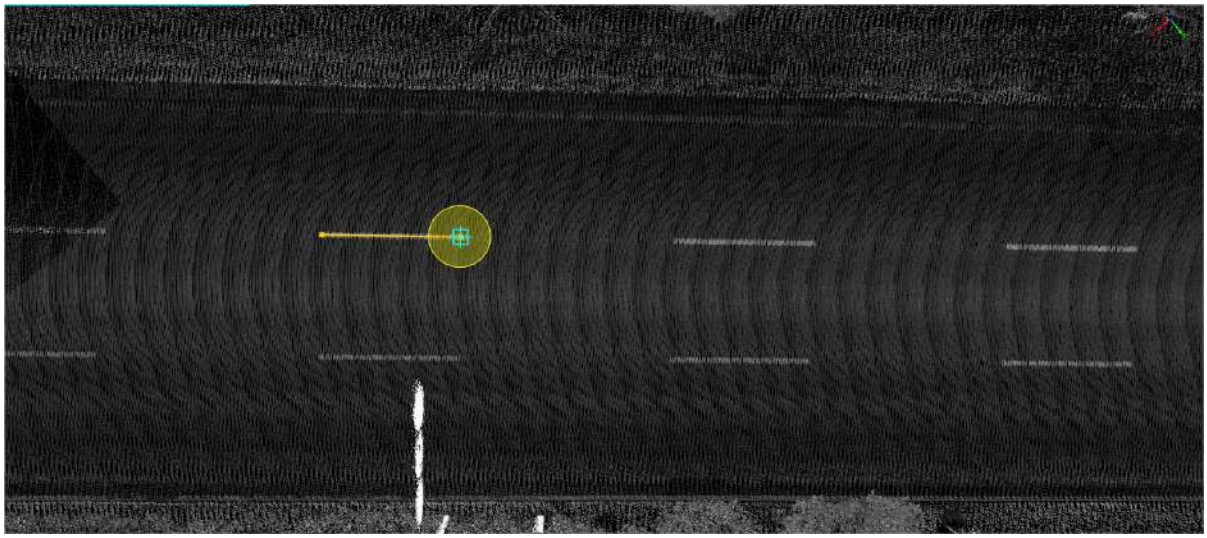
Shortcut name	Function
B	Toggles drawing mode (batch drawing/one by one)
H	Turn off/on the top left corner prompt Right mouse button
Current model	Current model
Right mouse button	Retracts a dashed line and clears the preview vector if in batch drawing mode

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



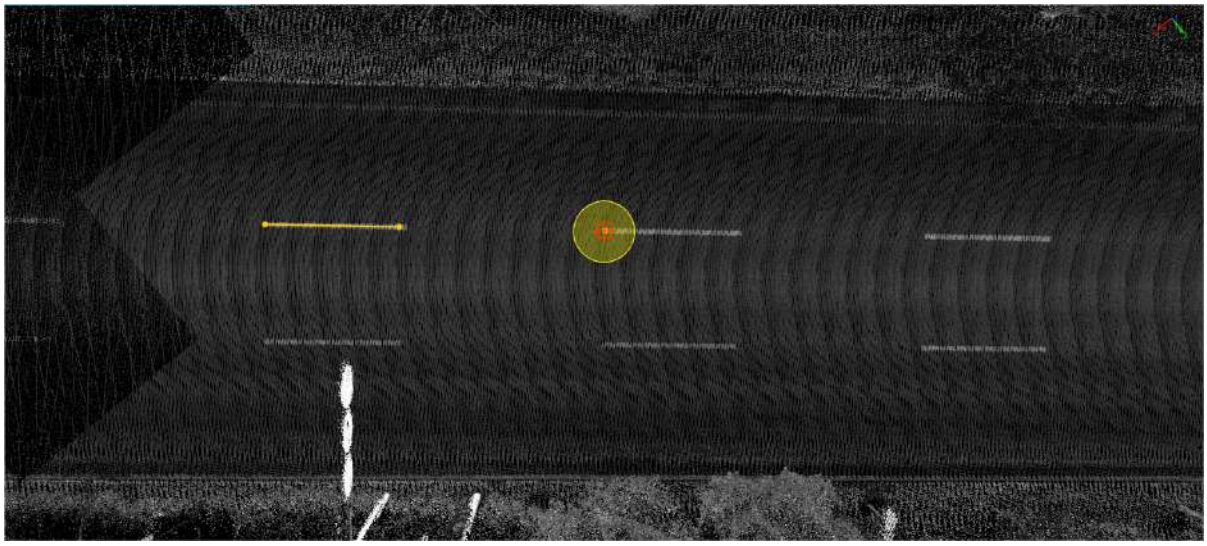
The first key point

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



The second key point

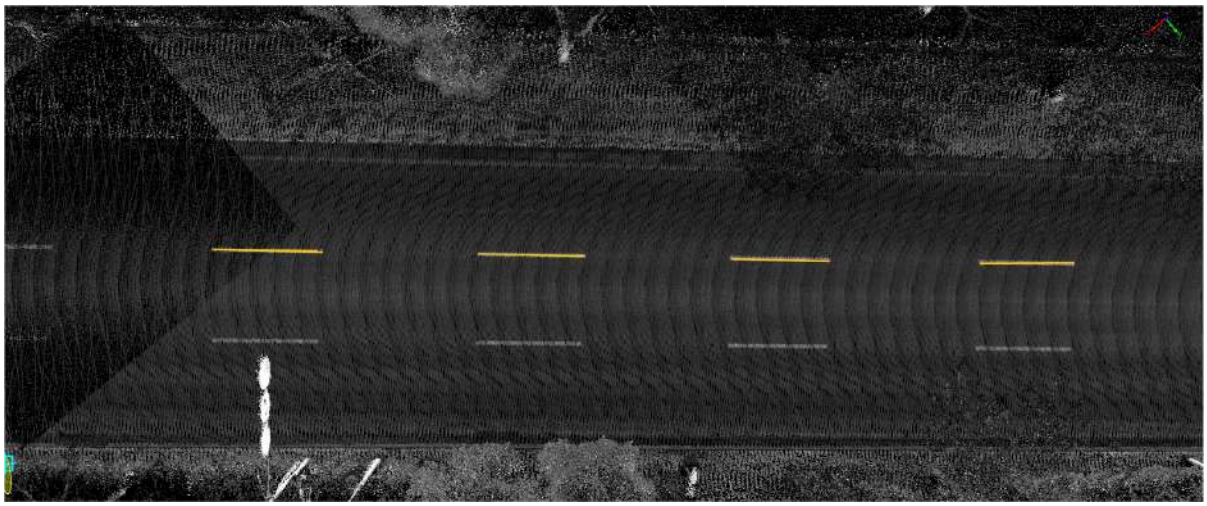
5.Move the mouse, **left click the third key point**, the details are shown in the figure, **used to determine the length between the dashed segments**.



The third key point

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

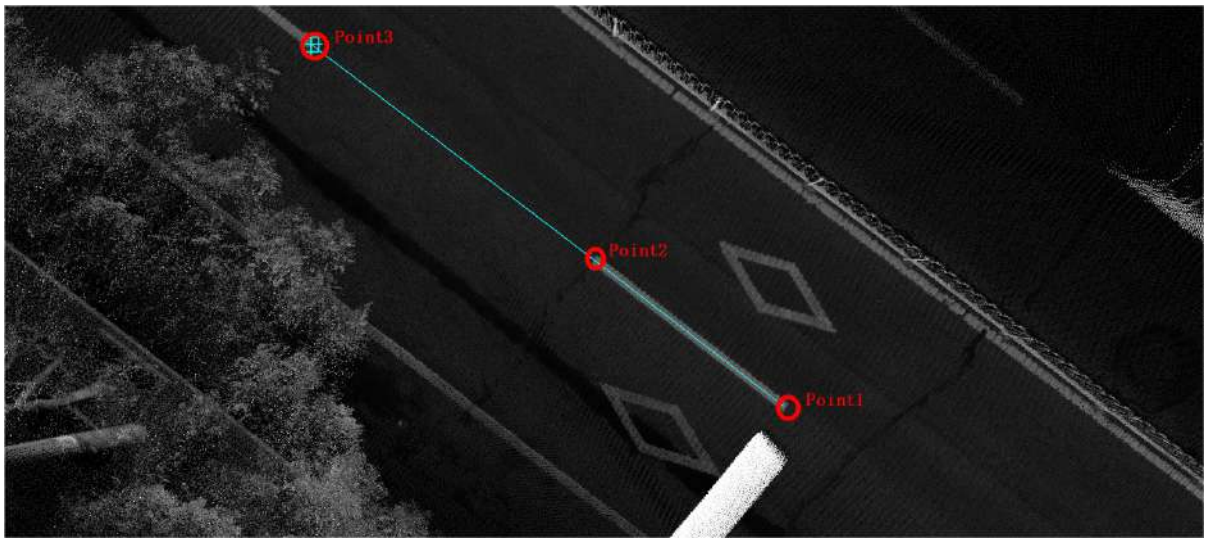
> - When the point cloud is outside the current window, the user can drag the point cloud by **pressing and holding down the right mouse button**.
> - **Press and hold the left mouse button** to adjust the viewing angle.



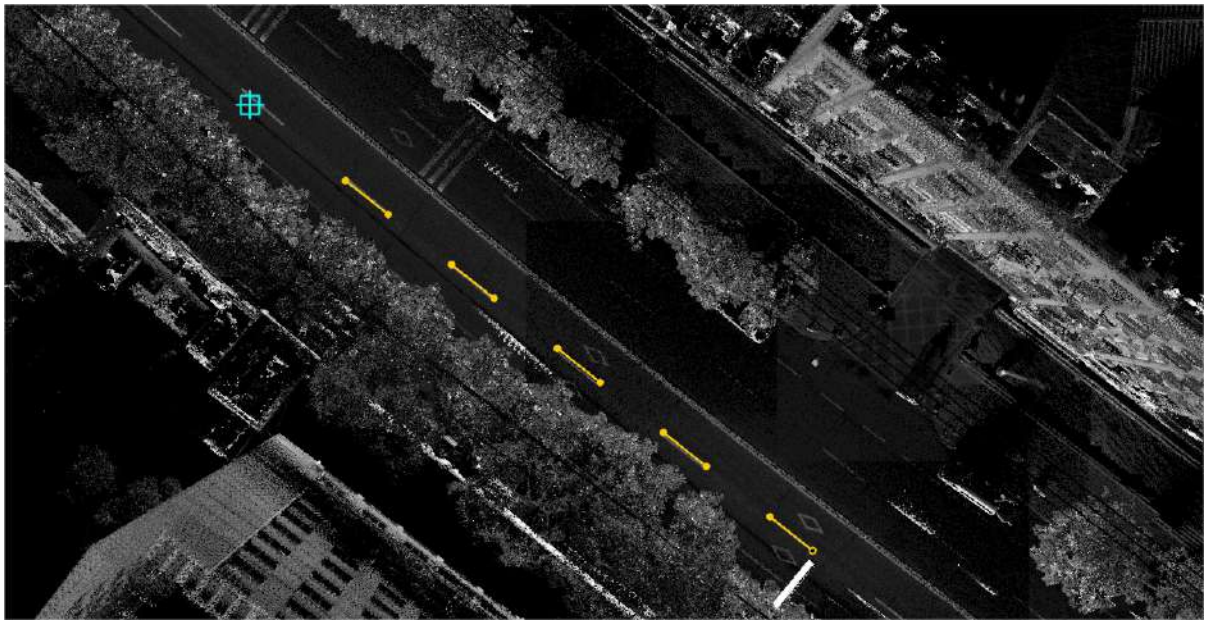
Drawing lane dashed line

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

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



Key points for drawing lane dashed lines (batch mode)



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

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

Add Stopline

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

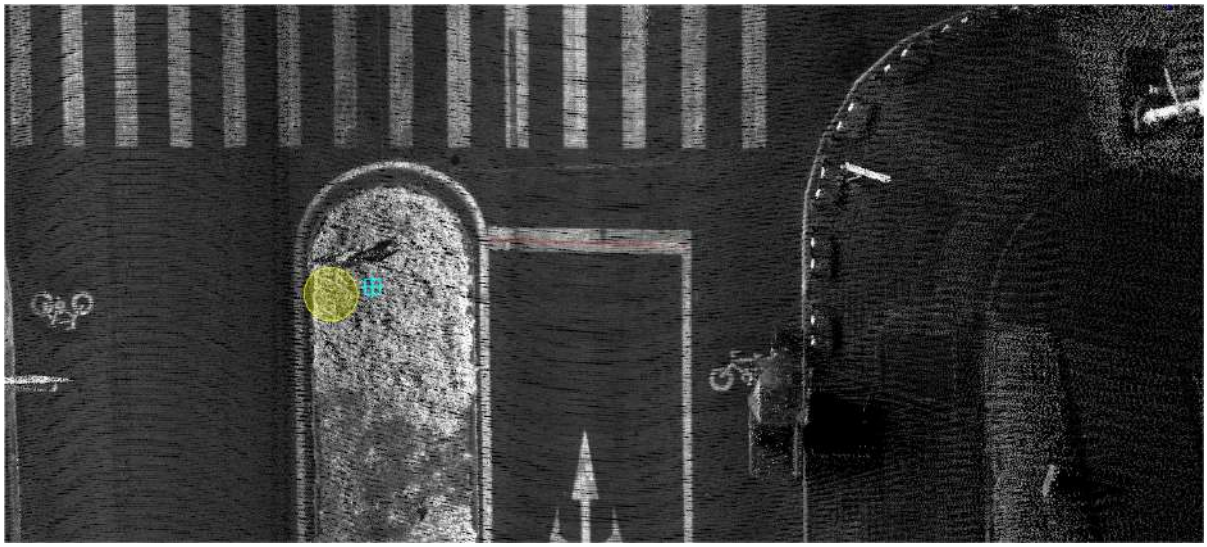
Steps

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

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

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

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



Drawing stop line

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

Add Road Side Line

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

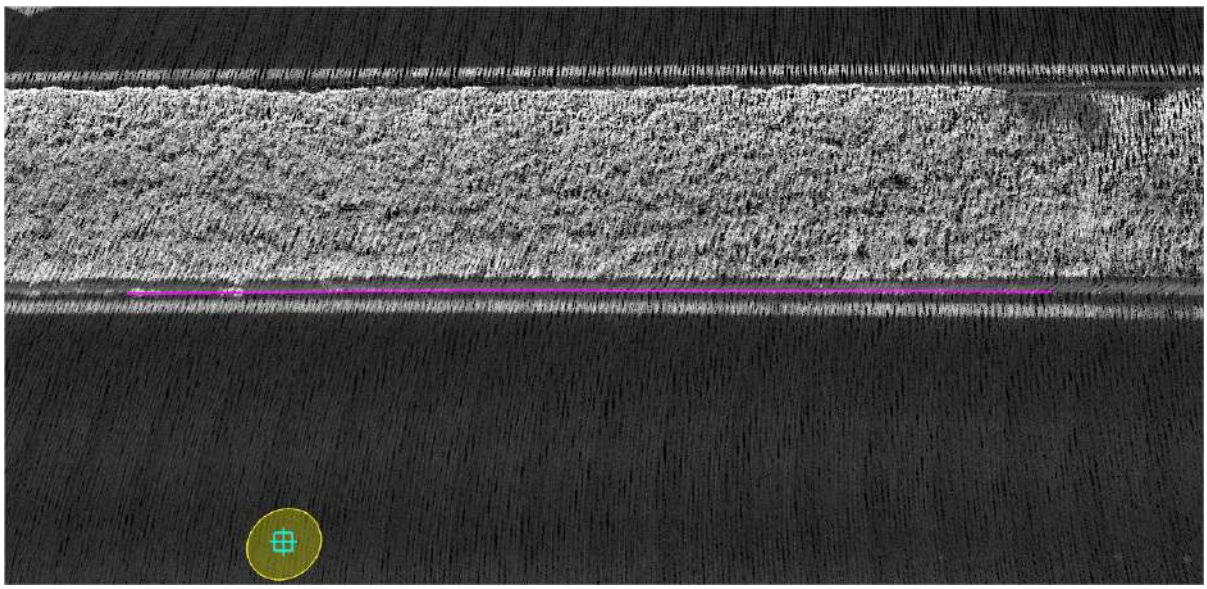
Steps

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

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

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

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



Drawing road side line

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

Generate Virtual Lane Centerline

Function Description：This tool can be used to automatically generate the centerline of the lane by clicking the two lane lines with the left mouse button on the premise that there are two lane lines.

Steps

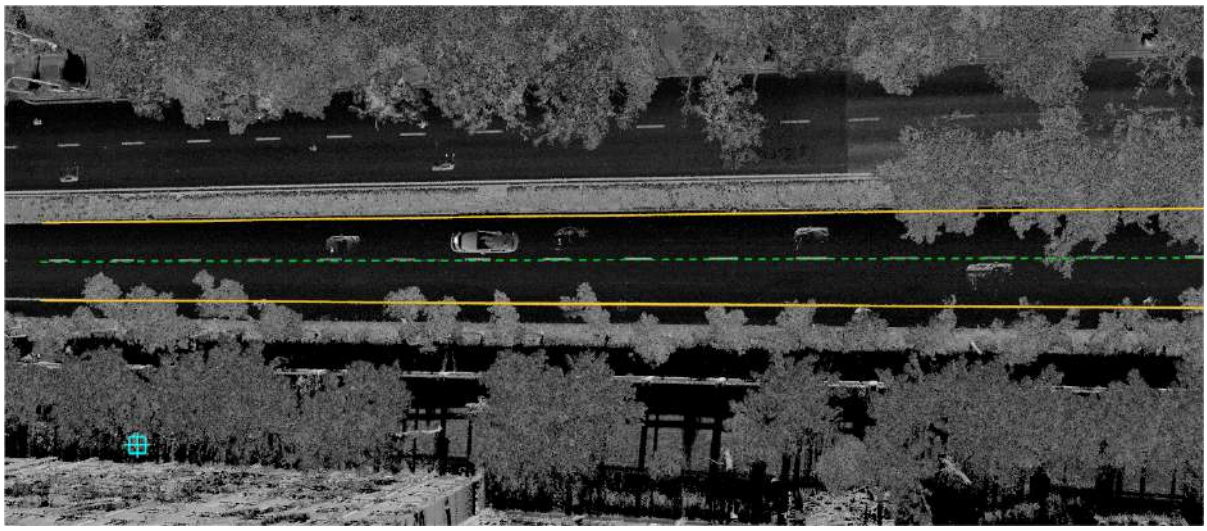
1.Click the **Generate Virtual Lane Centerline** button in the Road Shape panel.

2.**Click with the left mouse button** and click **adjacent** the left lane line and the right lane line in turn to automatically generate the lane center line.

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

Before generating

After generating



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

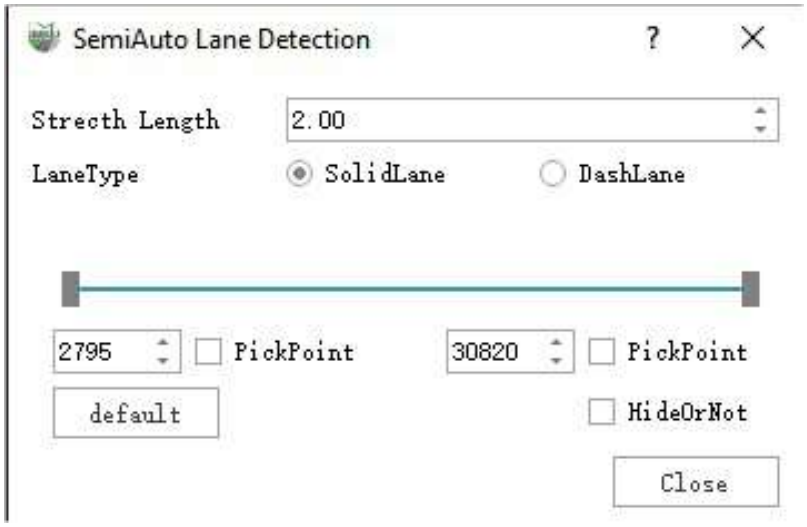
Detect Lane

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

> Note: This function is only effective for the area where lane lines and road surface have high recognition in point cloud intensity.

1.Click **Detect Lane** button in Road Shape panel.

2.The line **Detection Settings dialog box** should pop up:



Solid Lane dialog

Parameter Description:

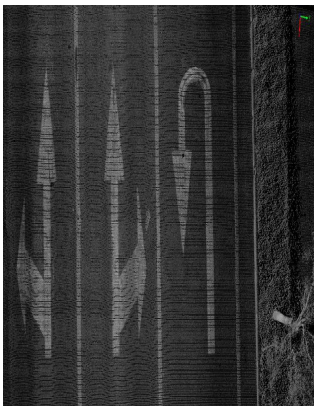
- **Maximum Angle**：The default is 20, which means the deviation angle between the current segment and the previous segment. If this angle is exceeded in a local area, the growth will stop. **Maximum Angle is suggested below 30 degrees**.
- **Maximum Diff Intensity**：The default value is 0.5, which indicates the tolerance value of the intensity difference between the front and rear segments of the solid lane. If the intensity difference is greater than the set value, the growth will stop. **The default value is recommended**.
- **Stretch Length**：The default value is 2 meters. If there is a certain defect in the middle of the line and it can be extended without stopping within the set distance, **the default value is recommended and can be adjusted according to the situation**.
- **LaneType**: The default type is SolidLane, if not, the user can **selects SolidLane**.
- **Intensity Adjustment**: For stretching the intensity by manually selecting the maximum and minimum values where the strength is not easily distinguished.
- **PickPoint**：Sets the minimum and maximum intensity and can be determined by clicking the point cloud with the left mouse button after checking the selection box. The left is the minimum intensity and the right is the maximum intensity.

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

- **Default**: Restores the original maximum and minimum intensity, and the point cloud display will also be linked in real time.
- **HideOrNot**: After adjusting the intensity through the scroll bar, value box, and check box, the point cloud display will be updated accordingly.

> Unchecked: For intensity stretching of the whole point cloud, points below the minimum value are displayed as black, and points above the maximum value are displayed as white.
>
> Checked: The intensity of the whole point cloud is stretched, and the points below the minimum value and above the maximum value are not displayed.

Intensity adjustment effect

		
Raw point cloud screenshot	The minimum value is adjusted to 14, the maximum value is adjusted to 18, and the box is not selected to hide	The minimum value is adjusted to 14, the maximum value is adjusted to 18, and the box selection is hidden

3.3D window tips:

Shortcut name	Function
Space	Interrupt growth
Enter	Confirm growth result
C	Toggles between automatic/manual drawing mode
B	Toggles between batch or one-by-one plotting in manual mode
H	switch off/on the top left corner prompt
Current model	Current model
Right mouse button	Rewind node (only works after interruption, not during growth)

The lane line detection function offers two modes, a semi-automatic mode, where automatic detection is carried out in the "Start + Direction" mode, and a manual mode, which follows the manual drawing mode of the "Draw lane lines" function. Both modes can be switched between during the drawing process by means of C.

4.The initial drawing mode is "semi-automatic". **Click left mouse** on a point on the lane line point cloud as the **starting point**, and then **left mouse click** to select another point on the lane line as the **growth direction**.

5.Automatic lane detection begins from **the starting point along the selected direction**.

> During the growth process, the center of the screen is automatically adjusted to the current growing point; at the same time, the point cloud can be viewed by **angle adjusting, dragging, and zooming** through **left mouse button, right mouse button, and scroll wheel**.

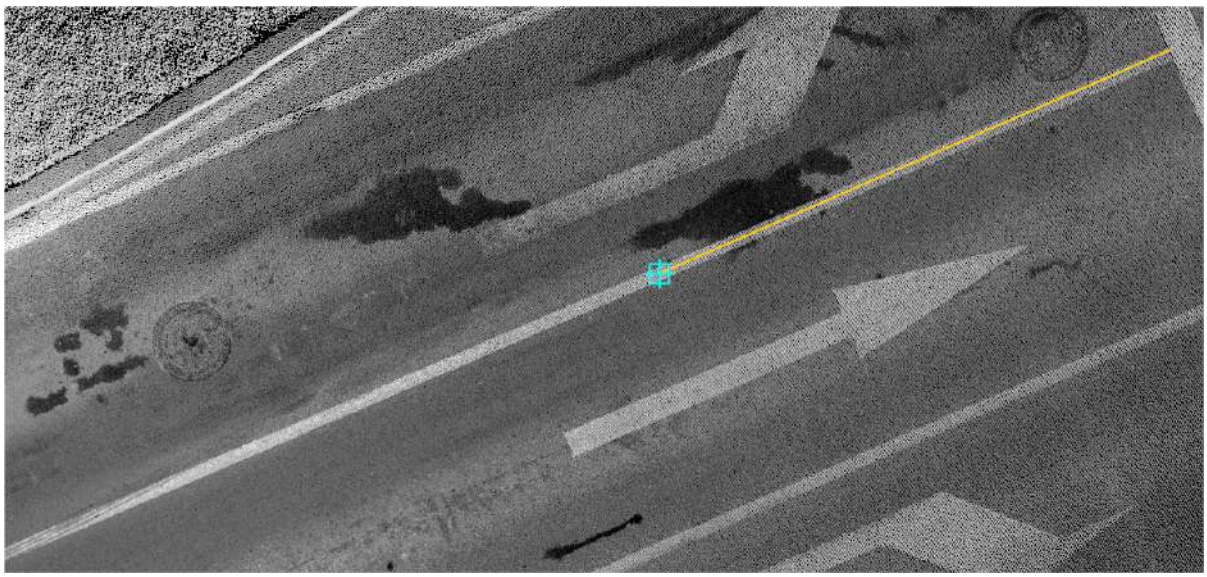
6.Use shortcut keys for **real-time shape adjustment** :

- he program is automatically **interrupted** due to the reasons such as drastic changes in intensity, low curb height, actual termination, actual interruption, etc.; if the automatic detection does not grow correctly, users can use the **Space** to manually **interrupt**.
- After **interruption**, users can **right click** to go back to adjust the shape.
- After **interruption**, users 3 and 4 can be repeated, followed by automatic growing.
- If the point cloud is missing in a large area or the intensity contrast is very low due to occlusion etc. and the program cannot detect it automatically, you can switch to manual mode with **C** and then draw by manually clicking on the key points, when the manual drawing is finished, you can double click to end the current mode or switch to automatic mode with **C**.

7.When the growth is completed and the shape is acceptable, press the Enter key (Enter, confirm key) to complete the growth of the solid line of the current lane.

Solid Line

- **Click the starting point and direction point**



Click the starting point and direction point

- **Growth results**

Solid line growth results

Dashed Lane Detection

Function Description: This tool enables the detection of dashed lanes using points. Use the left mouse button to select two points to obtain the starting point and direction point along the clicked direction, and start automatic lane line detection from the starting point.

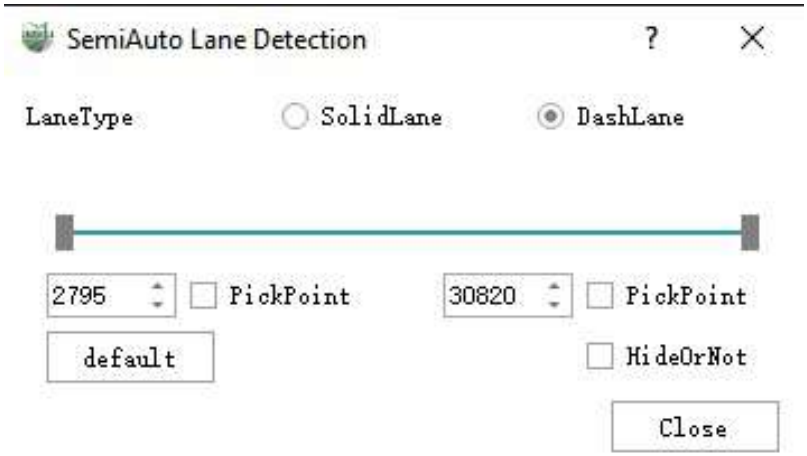
> Note: This function is only effective for areas where the lane line and road surface have high recognition in point cloud intensity.

Steps

1.Click **Detect Lane** button in the Road Shape panel.

2.The **Detection Settings dialog box** will appear.

Dashed Lane Dialog



Dashed Lane Dialog

Parameter Description:

- **LaneType**：Line type. Select dashed line.
- **Intensity Adjustment**：

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

3.3D window tips:

Shortcut name	Function
Space	Interrupt growth
Enter	Confirm growth result
C	Toggles between automatic/manual drawing mode
B	Toggles between batch or one-by-one plotting in manual mode
H	switch off/on the top left corner prompt

Current model	Current model
Right mouse button	Rewind node (only works after interruption, not during growth)

The lane dash detection function offers two modes, a semi-automatic mode, which automatically detects the lane by means of "Start + Length + Spacing", and a manual mode, which follows the manual drawing mode of the "Draw lane dash" function. Both modes can be switched between during the drawing process by means of C.

4.The initial drawing mode is "semi-automatic". **Click Left on the mouse** on a point on the lane line point cloud as the **start point**, and then **left mouse click again** to select another point on the lane line as the **growth direction**.

5.Automatic lane detection begins from **Click the starting point, direction point**.

> During the growth process, the center of the screen is automatically adjusted to the current growing point; at the same time, the point cloud can be viewed by **angle adjusting, dragging, and zooming** through **left mouse button, right mouse button, and scroll wheel**.

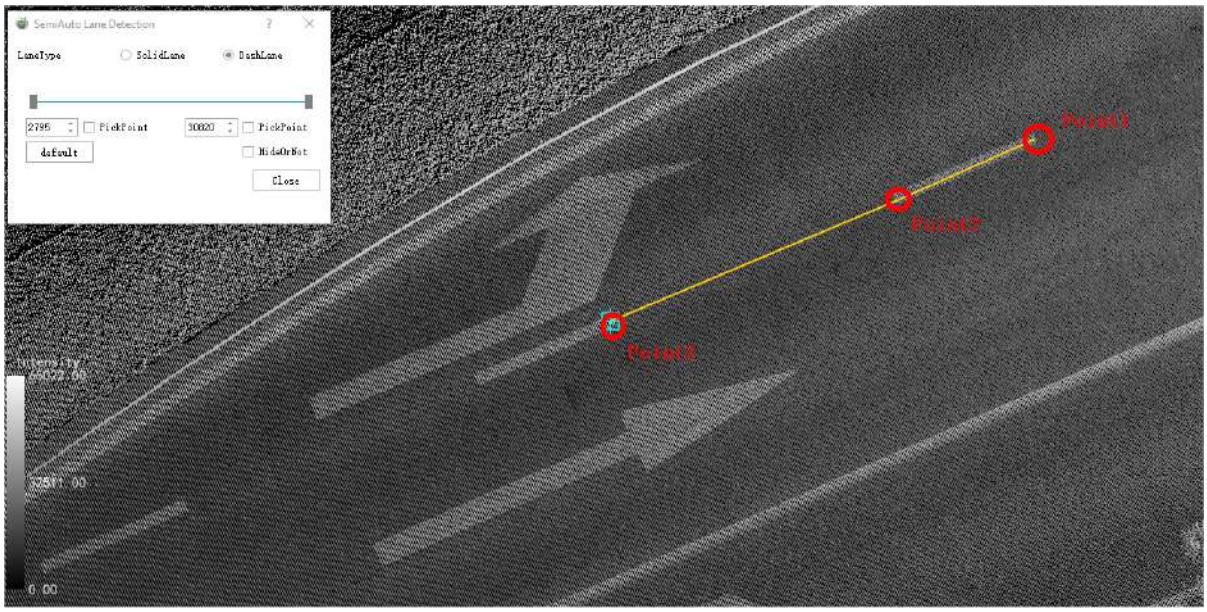
6.Use shortcut keys for **real-time shape adjustment** :

- The program is automatically **interrupted** due to drastic changes in intensity, actual termination, actual interruption, etc.; Users can use the **space** to manually **interrupt** if the automatic growth is not correct growth.
- After **interruption**, Users can use **right mouse button** to go back (go back a dashed line) to adjust the shape.
- After **interruption**, Users can repeat operations 3 and 4, followed by automatic growth.
- If the point cloud is missing in a large area or the intensity contrast is very low due to occlusion etc. and the program cannot detect it automatically, you can switch to manual mode with **C** and then draw it manually by batch drawing or section by section (you can switch with the shortcut key **B**, for details refer to Lane Dash Lines Drawing). When finished, you can double-click to end the current mode or use **C** to switch to automatic mode.

7.When the growth is completed and the shape is acceptable, press the **Enter** (Enter, confirm key) to complete the growth of the dotted line of the current lane, otherwise, the user can continue to **right-click** to go back and adjust the threshold again for growth.

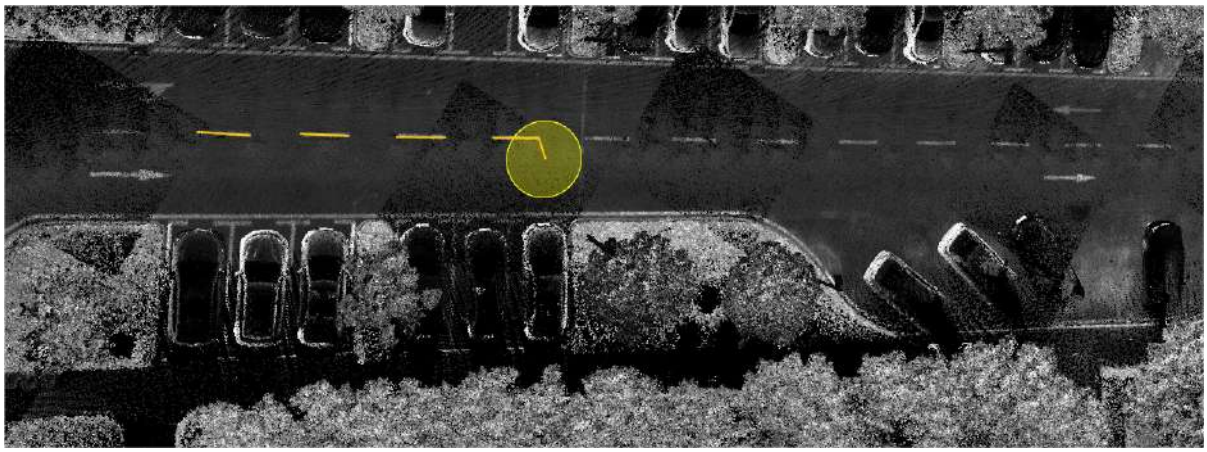
Rendering

- **Click the starting point and direction point**



Click the starting point and direction point/p>

- **Automatic or manual interruption during growth**



Dashed line growth results

- **Growth results**

Dashed line growth results

Detect Road

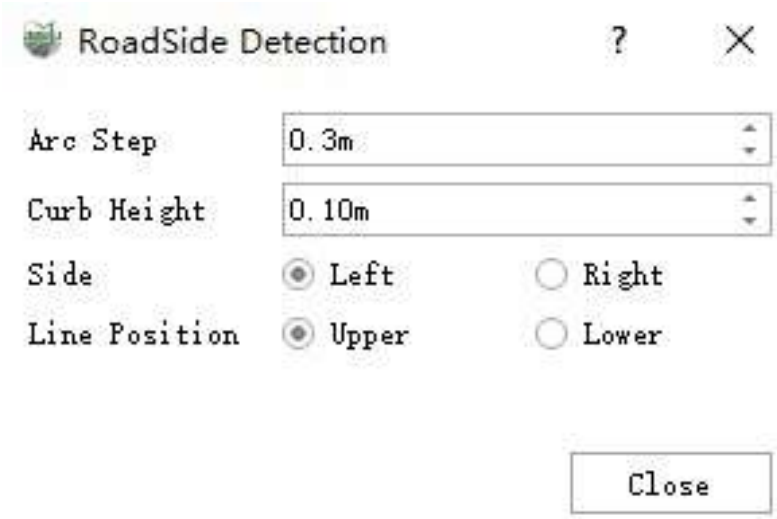
Function Description：This tool enables the detection of roadsides using points. Left click to select two points to obtain the starting point and direction point. Along the selected direction, automatic roadside detection will be carried out from the starting point.

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

Steps

1.Click the **Detect Road** button in the Road Shape panel.

2.A **Detection Settings dialog box** will appear:



Pop up Detection Settings dialog box

Parameter Description:

- **Arc Step**: The default is 0.3 meters, and the curb may have a larger arc. If the step length is set too large, the arc part will not fit the actual point cloud enough. It is recommended to be less than 0.3 meters, 0.1 meters above.
- **Curb Height**: The default is 0.1. If the peripheral curb is lower than the set value, it cannot be detected. The default value is recommended.
- **Side**: The default is the left side. It indicates which side of the straight line formed by the starting point and direction points of the point selection is located in the actual path point cloud, which is selected according to the actual situation.
- **Line Position**: The default is upper, which means that the final generated line is located on the upper or lower edge of the curb. The **upper line** is calculated by adding the height of the curb to the point Z value of the **lower line**. Therefore, the position of the upper line can be adjusted by **adjusting the height of the curb**.

3.3D window tips:

Shortcut name	Function

Space	Interrupt growth
Enter	Confirm growth result
C	Toggles between automatic/manual drawing mode
B	Toggles between batch or one-by-one plotting in manual mode
H	switch off/on the top left corner prompt
Current model	Current model
Right mouse button	Rewind node (only works after interruption, not during growth)

Two modes are available in the curb detection function, a semi-automatic mode, where automatic detection is carried out by means of the "start point + direction" mode, and a manual mode, which follows the manual drawing mode of the "draw curb line" function. Both modes can be switched between during the drawing process by means of C.

4.The initial drawing mode is "semi-automatic". Left click to select a point on the road surface point cloud as the starting point, and then left click again to select another point on the road surface as the direction point.

> Note: The clicked starting point and direction point must be clicked on the surrounding road surface. The distance between the clicked point and the actual curb point cloud is recommended to be more than 0.1 meters and within 0.5 meters.

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

> During the growth process, the center of the screen is automatically adjusted to the point currently growing; at the same time, the angle of the point cloud can be adjusted, dragged, and zoomed to view the point cloud through the left, right, and scroll wheels of the mouse.

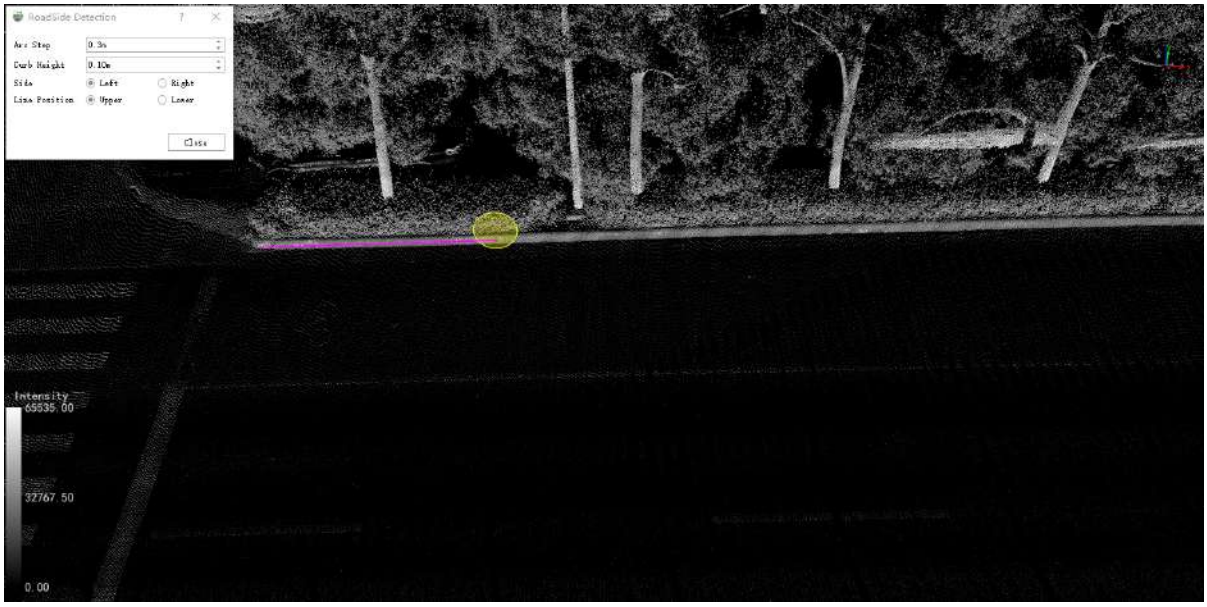
6.Use shortcut keys for **real-time road shape adjustment**:

- The program is automatically **interrupted** due to the reasons such as low curb height, actual termination, actual interruption, etc.; if the automatic growth does not grow correctly, the user can use the **Space** to manually **interrupt**.
- After **interruption**, the user can **right click** to go back to adjust the shape.
- After **interruption**, operations 3 and 4 can be repeated, followed by automatic growth.
- If the point cloud is missing in a large area due to occlusion, etc., and the program cannot detect it automatically, you can switch to manual mode with **C** and then draw by manually clicking on the key points, and when the manual drawing is finished, you can double-click to end the current mode or switch to automatic mode with **C**.

7.When the growth is completed and the shape is acceptable, press the **Enter**(Enter, confirm key) to complete the growth of the solid line of the current lane.

Straight Road Curb

- **Click the starting point and direction point**

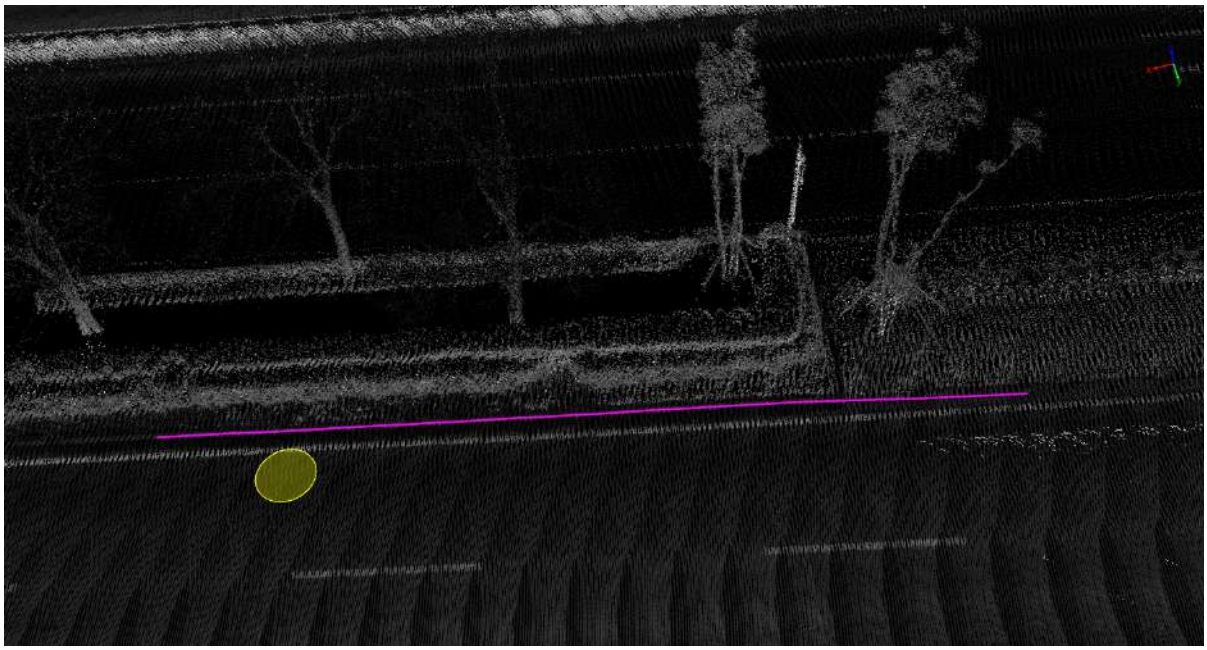


Click the starting point and direction point

- **Interrupt** and click again

Interrupt

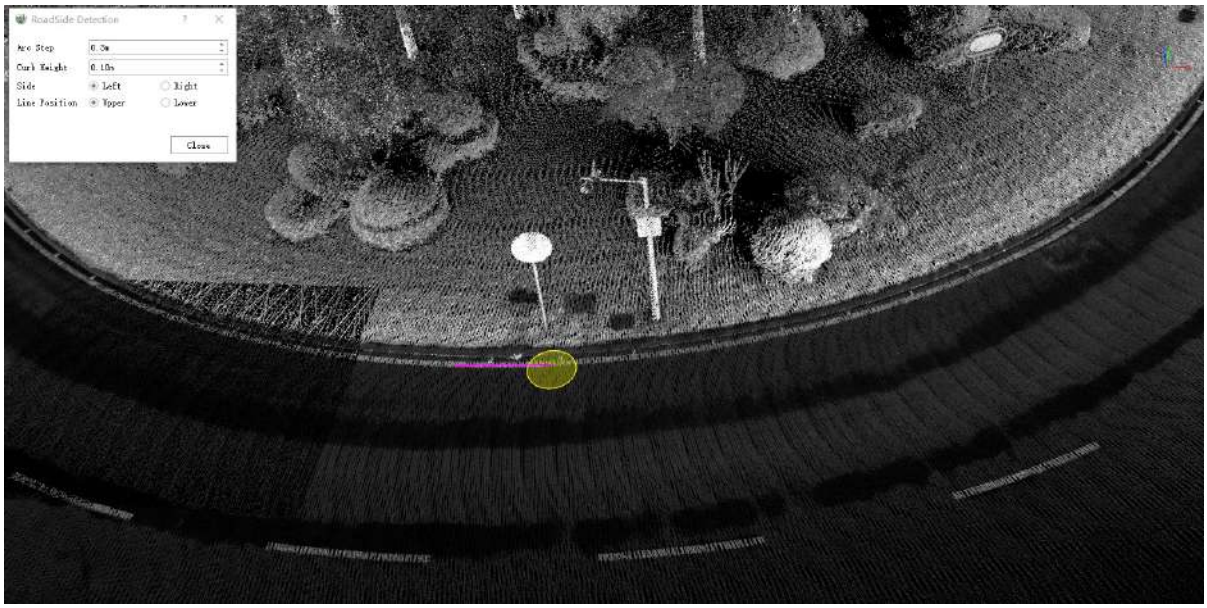
- **Growth result**



Curb growth result

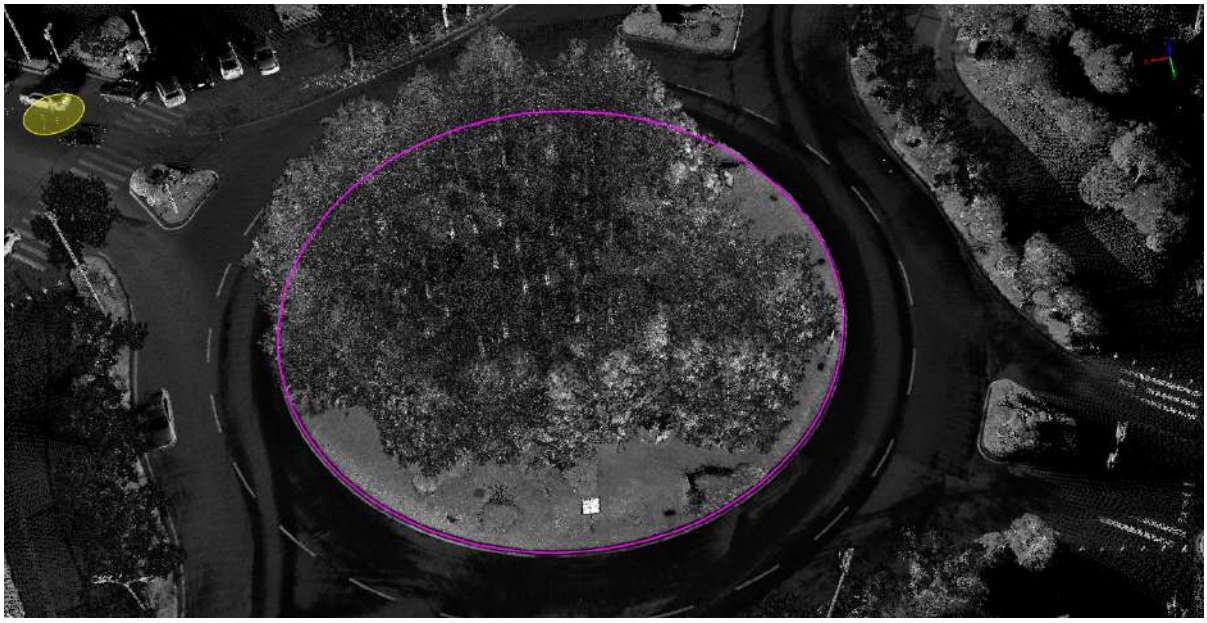
Closed Round Curb

- **Click the starting point and direction point**



Click the starting point and direction point

- **Growth result**



Round curb growth result

Road Surface

The road surface module is mainly used for the **vectorized manual** production of various ground signs such as various ground arrows, crosswalks, deceleration signs, and no-stop lines.

Includes:

- Guide arrow
  - Add Straight
  - Add Left
  - Add Right
  - Add Left/Right
  - Add Straight/Left
  - Add Straight/Right
  - Add Straight/Left/Right
  - Add Turn Round
  - Add Straight/Turn Round
  - Add Left/Turn Round
  - Add Left Merge
  - Add Right Merge
  - Add Right/Turn Round
  - Add Straight/Right/Turn Round
  - Add Left/Right/Turn Round
- Horizontal marking
  - Add CrossWalk
  - Add Longitudinal Speed Reduction
- Pavement graphic markings
  - Add CrossWalk Warning
  - Add Speed Reduction
  - Add No-Parking
  - Add BicycleLane
  - Add No U-Turn
  - Add Font
- Template Matching
  - TemplateMatchingSidebarParametersDetail
  - TemplateMatchingCustomTables
  - TemplateMatchingAutoMatch
  - TemplateMatchingManual
  - TemplateMatchingDatabaseFile
  - New Template

Add Straight

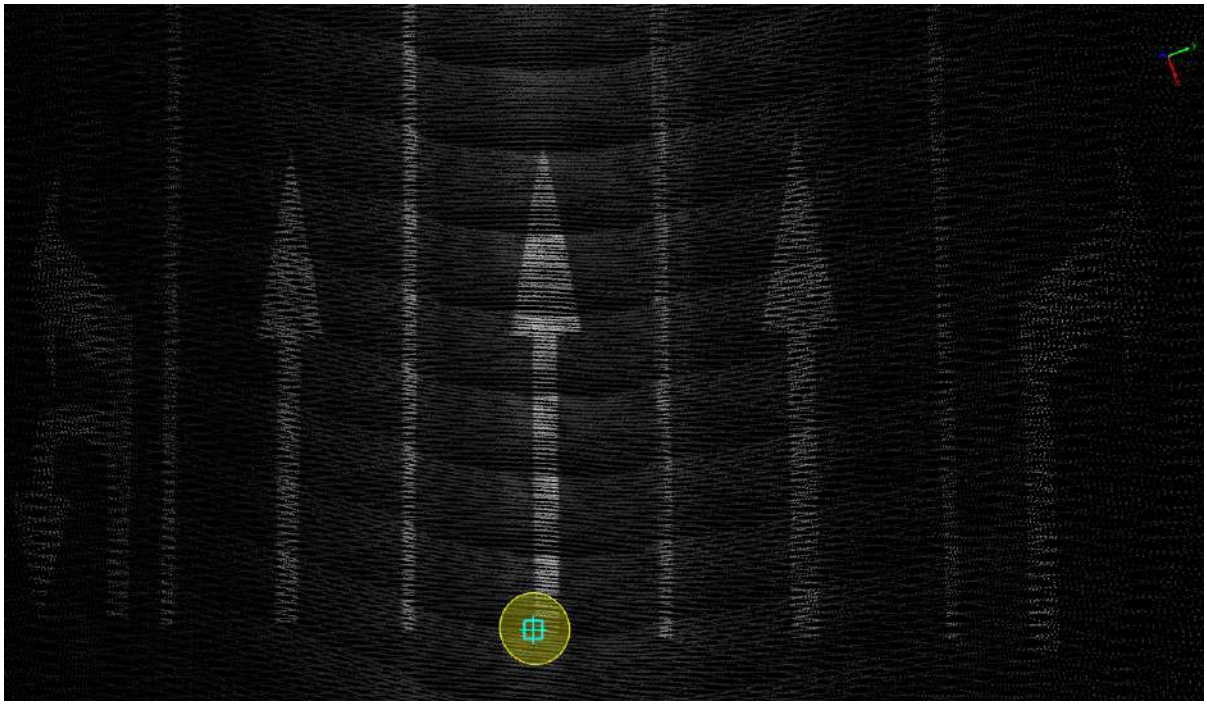
Function Description：The straight arrow means that the current lane can only go straight. This tool is used to manually select the point cloud to vectorize the straight arrow.

Steps

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

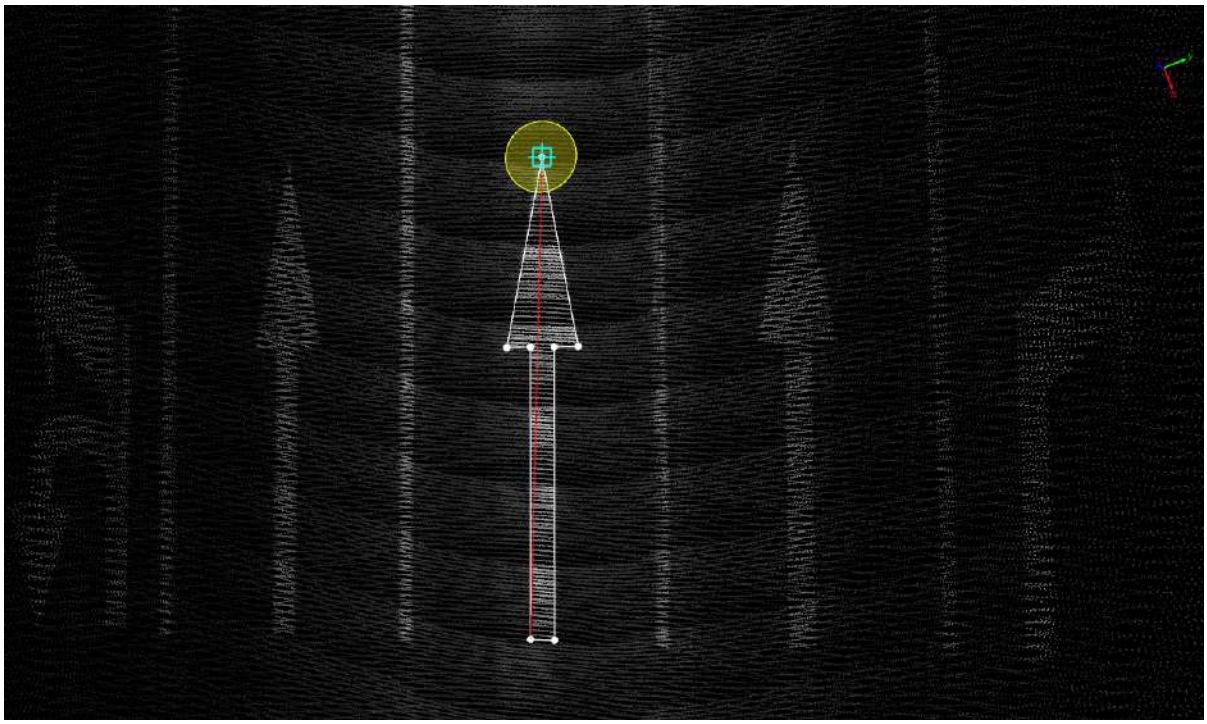
> - The key points in the arrow icons need to be aligned with the actual point cloud.
> - The red point is the first key point and the green point is the second key point.

2.**Left mouse click** to select the lower center of the straight line arrow point cloud with the following details and the **first key point** in the icon.



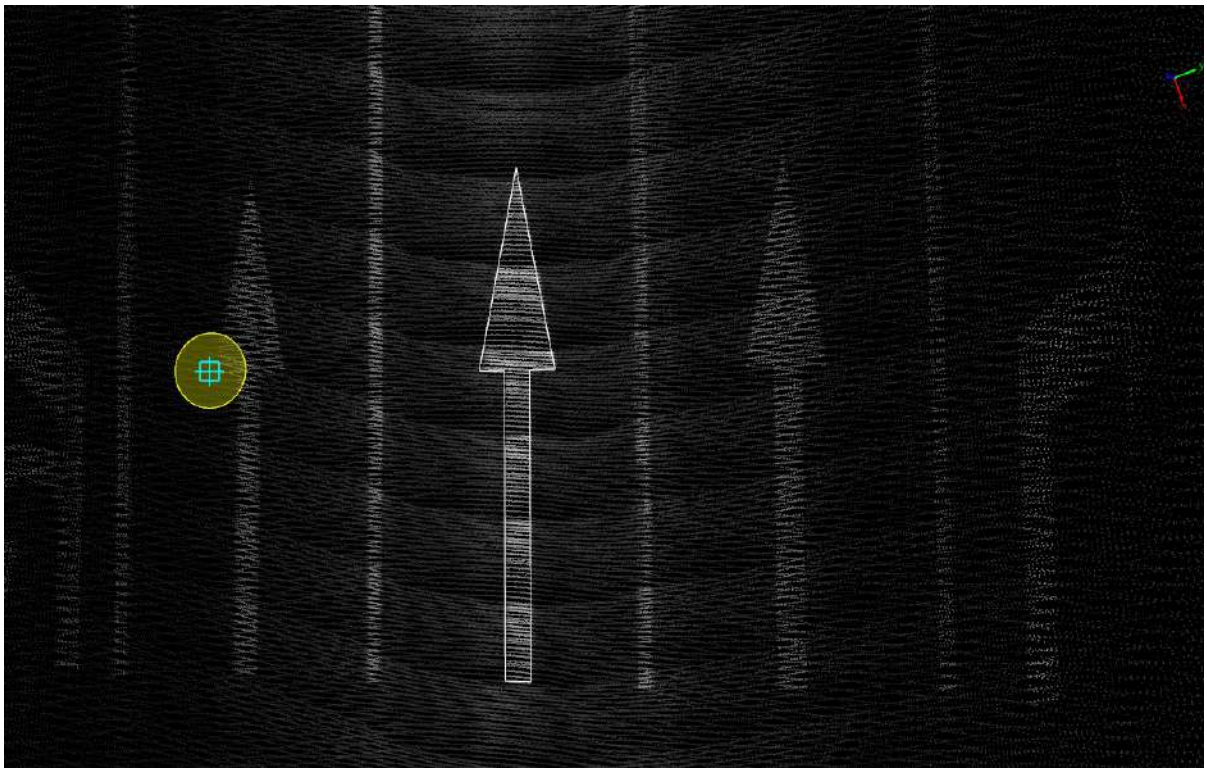
First key point

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



Second key point

Results Show



Straight arrow drawing

Add Left

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

Steps

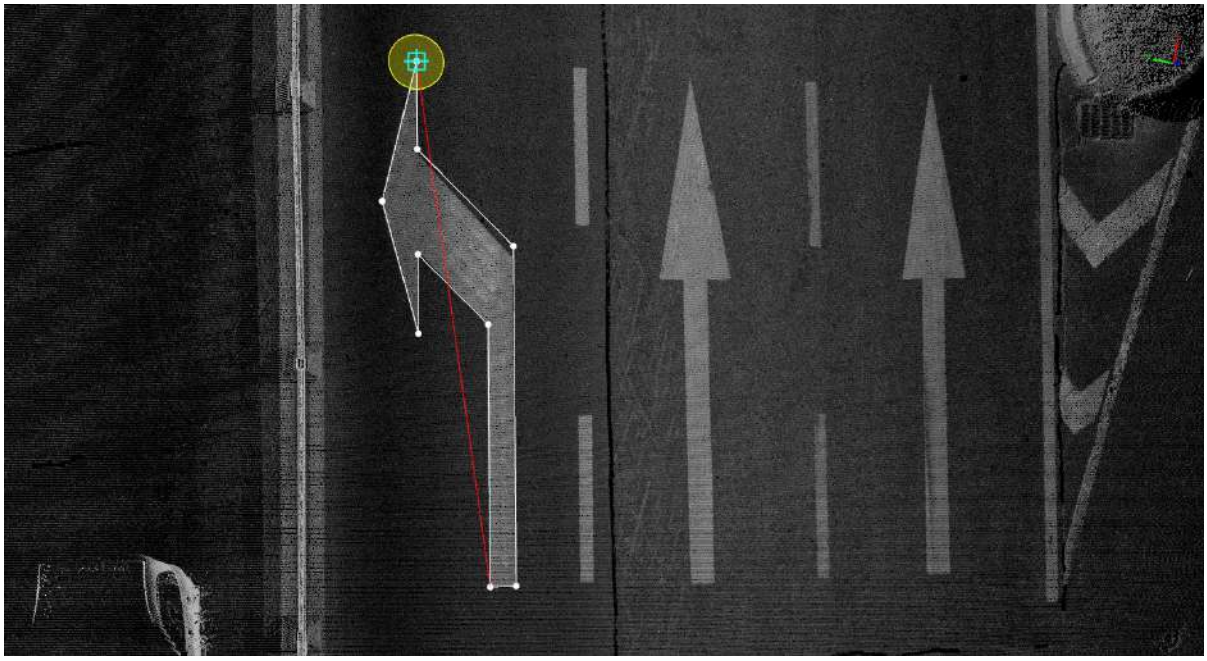
1.Click the **Add Left** button in the Road Surface panel.

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

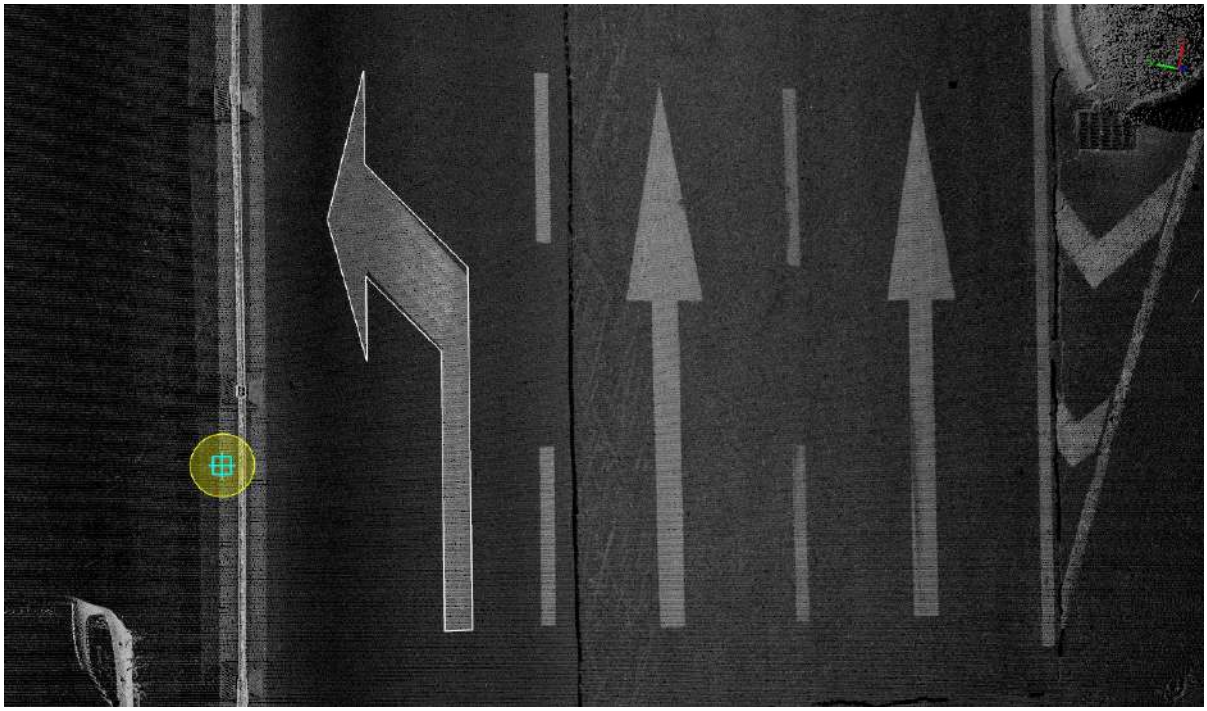
2.**Left mouse click to select the key point** of the left arrow point cloud, the details are as follows, **the first key point** in the icon.

First key point

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



Second key point



Turn left arrow drawing

Add Right

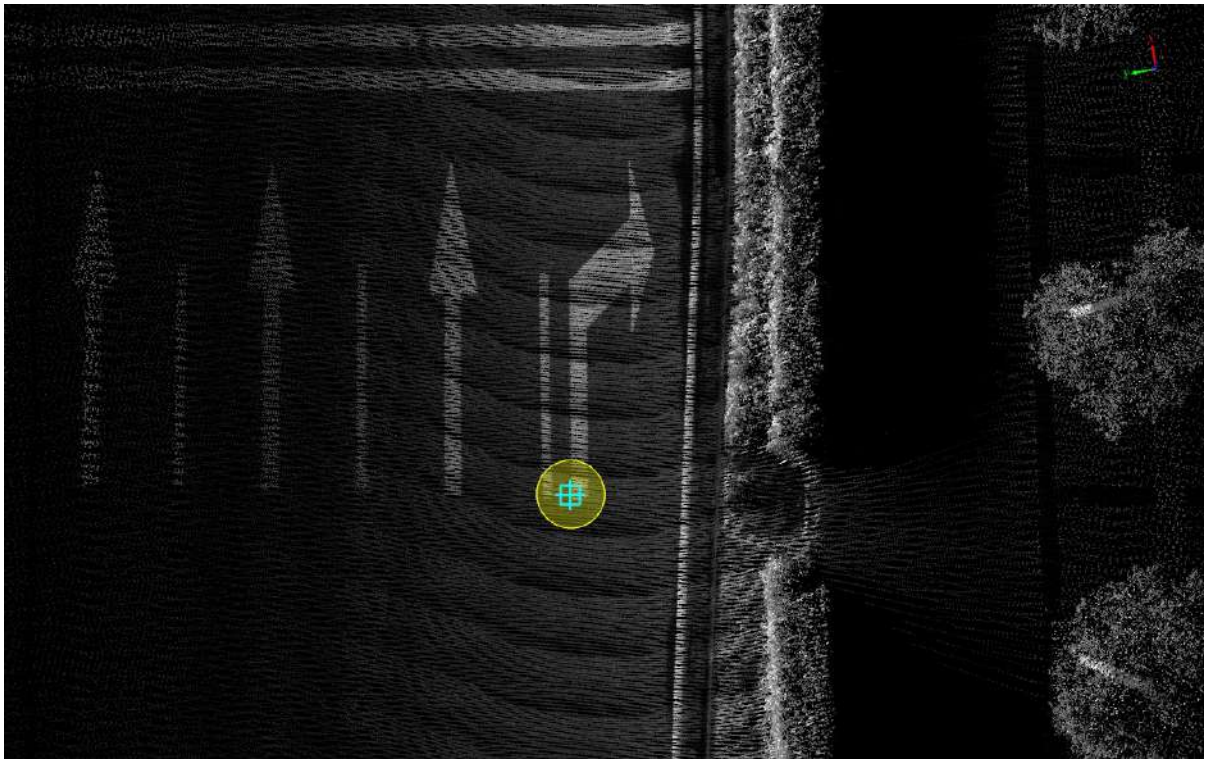
Function Description：The turn right arrow indicates that the current lane can only be turned right, this tool is used to manually tap the point cloud for vectorization of the right turn arrow.

Steps

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

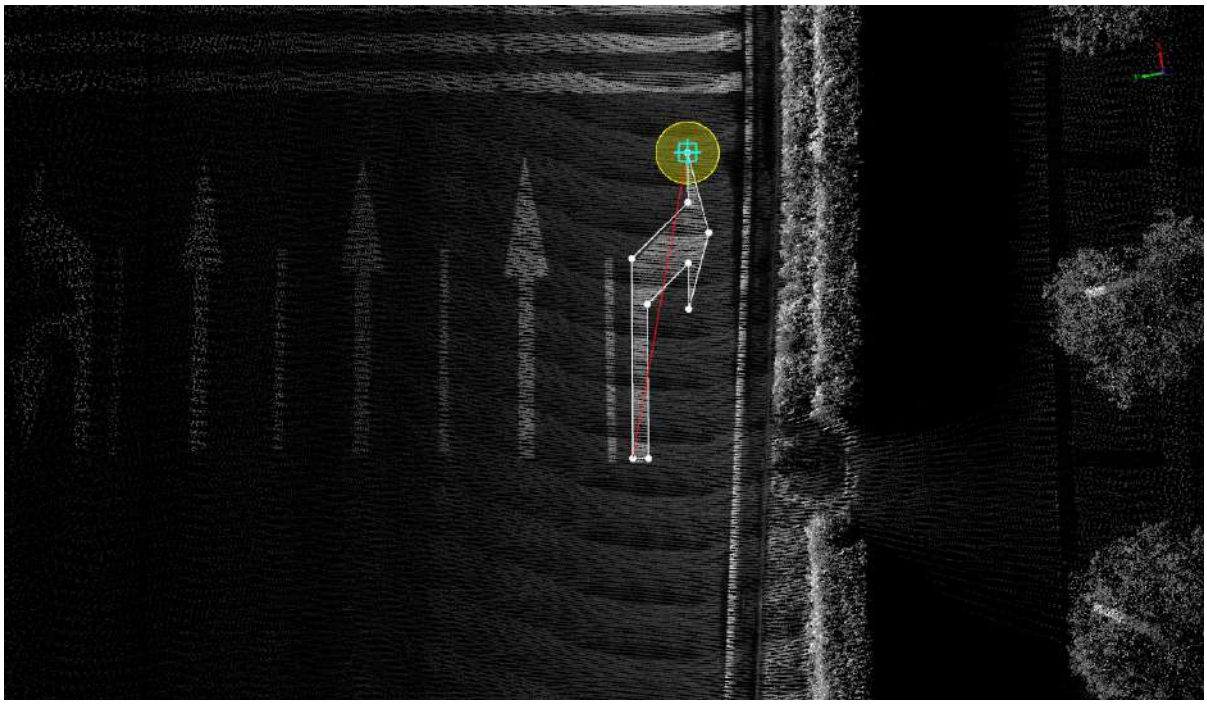
> - The key points in the arrow icons need to be aligned with the actual point cloud.
> - The red point is the first key point and the green point is the second key point.

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

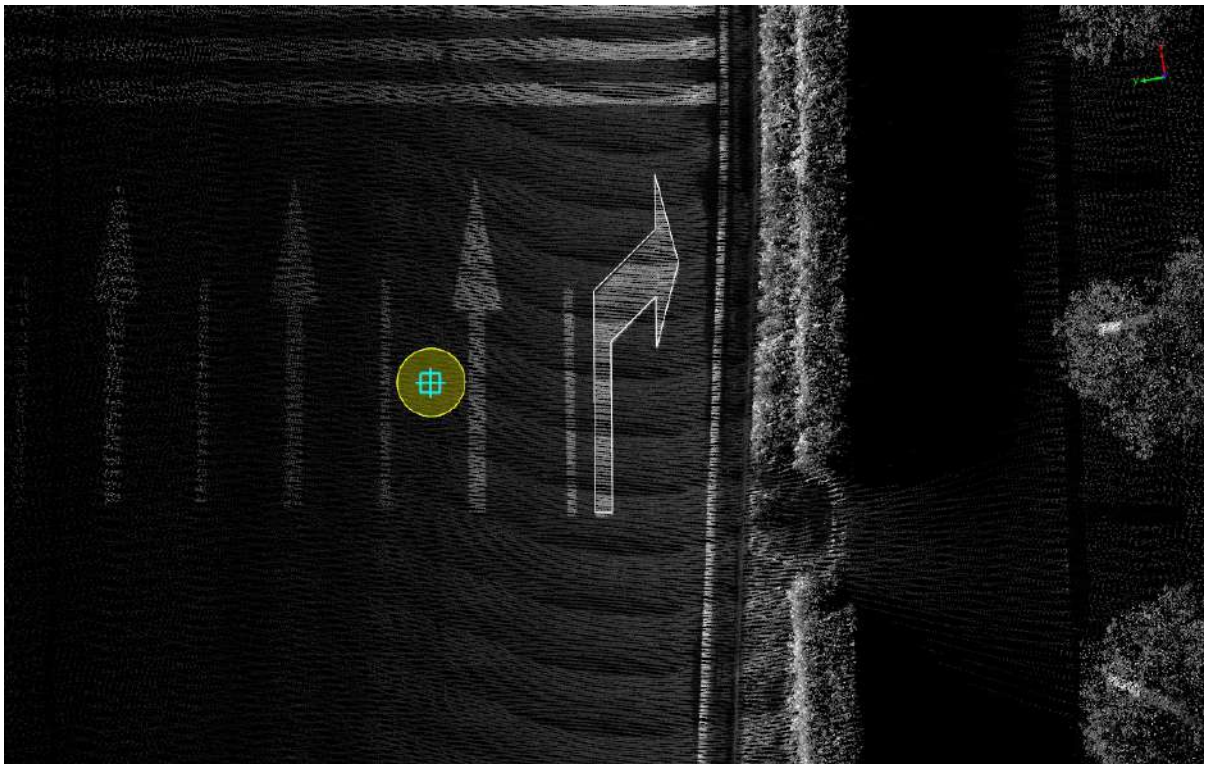


First key point

3.Move the mouse to see how the **template** fits into the **actual point cloud**, **left mouse button click** to select the **second key point** to complete the turn right arrow production, details are as follows, icon in **second key point**.



Second key point



Turn right arrow drawing

Add Left/Right

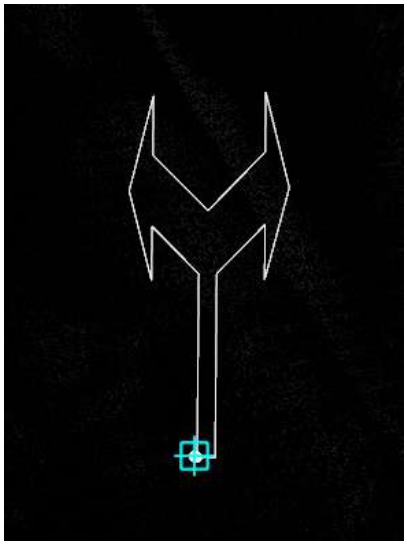
Function Description：The left/right arrow indicates that the current lane can only turn left or right. This tool is used to manually click on the point cloud to vectorize the left/right arrow.

Steps

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

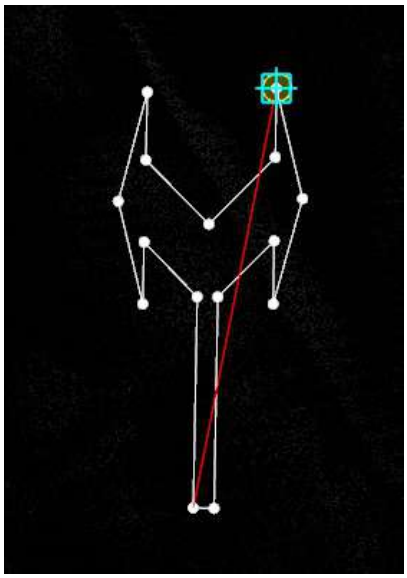
> - The key points in the arrow icons need to be aligned with the actual point cloud.
> - The red point is the first key point and the green point is the second key point.

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

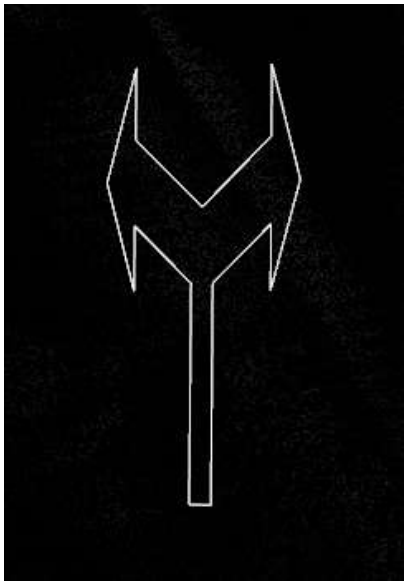


First key point

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



Second key point



Turn Left/turn right arrow drawing

Add Straight/Left

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

Steps

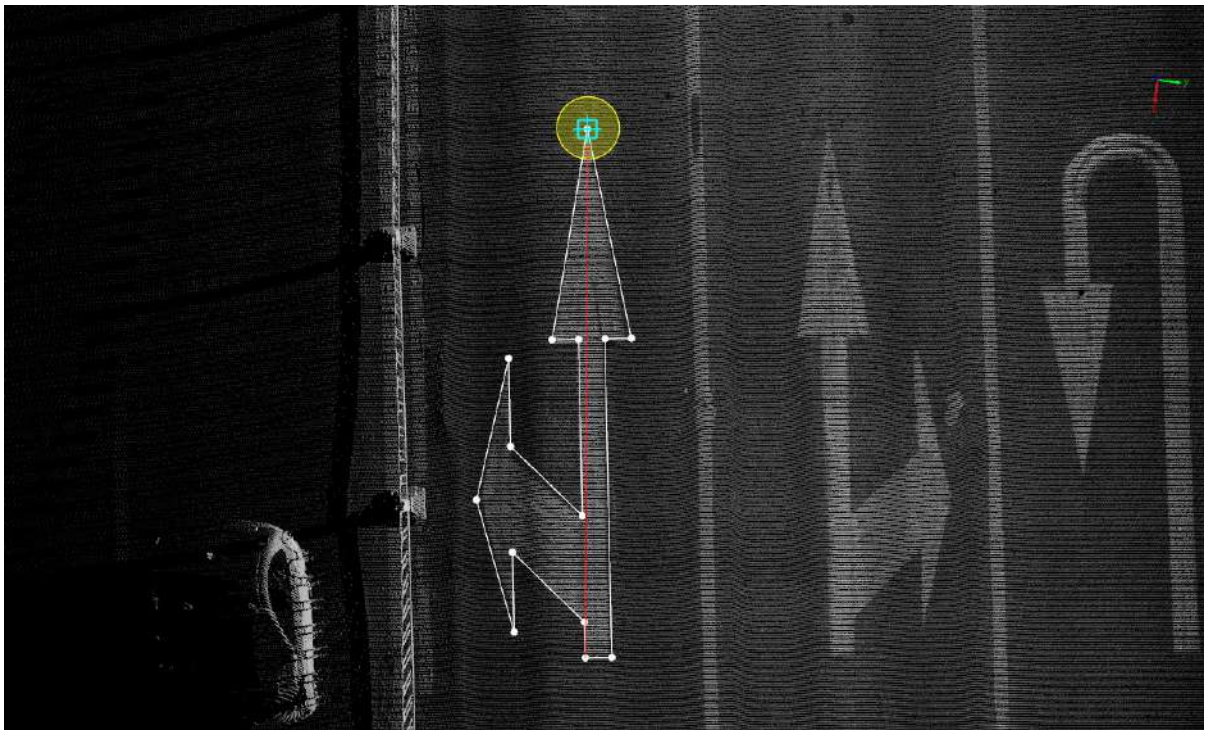
1.Click the **Add Straight/Left** button in the Road Surface panel.

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

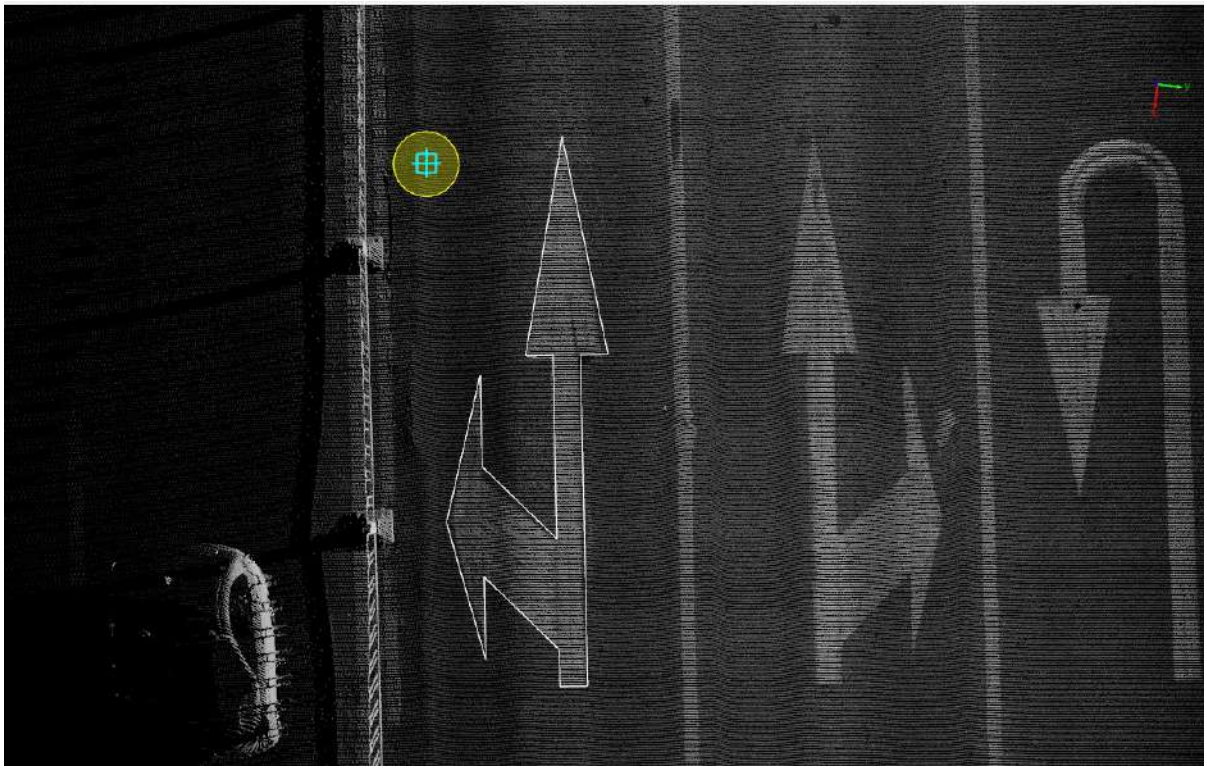
2.**Left mouse click** to select the straight/turn left arrow point cloud key point, the details are as follows, the icon in the **first key point**.

First key point

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



Second key point



Straight/turn left arrow drawing

Add Straight/Right

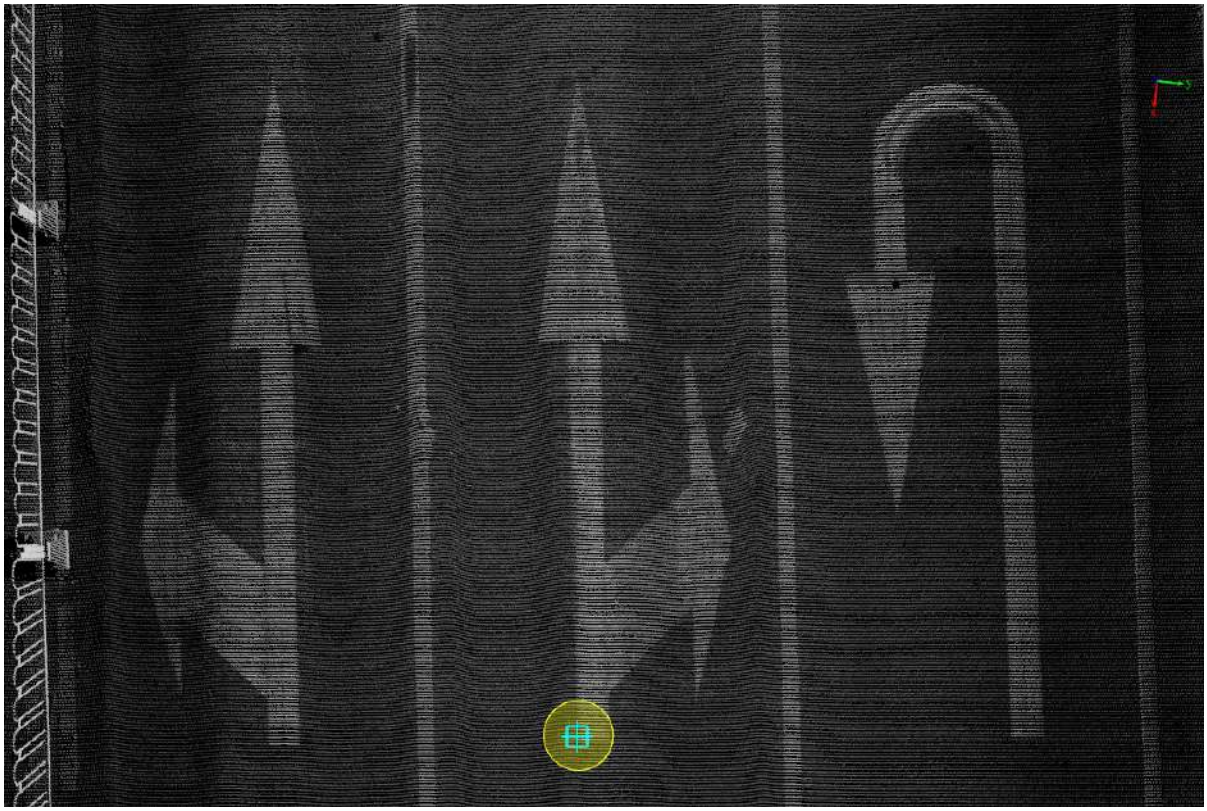
Function Description：The straight/turn right arrow indicates that the current lane can turn right or go straight. This tool is used to manually click on the point cloud to vectorize the straight/right arrow.

Steps

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

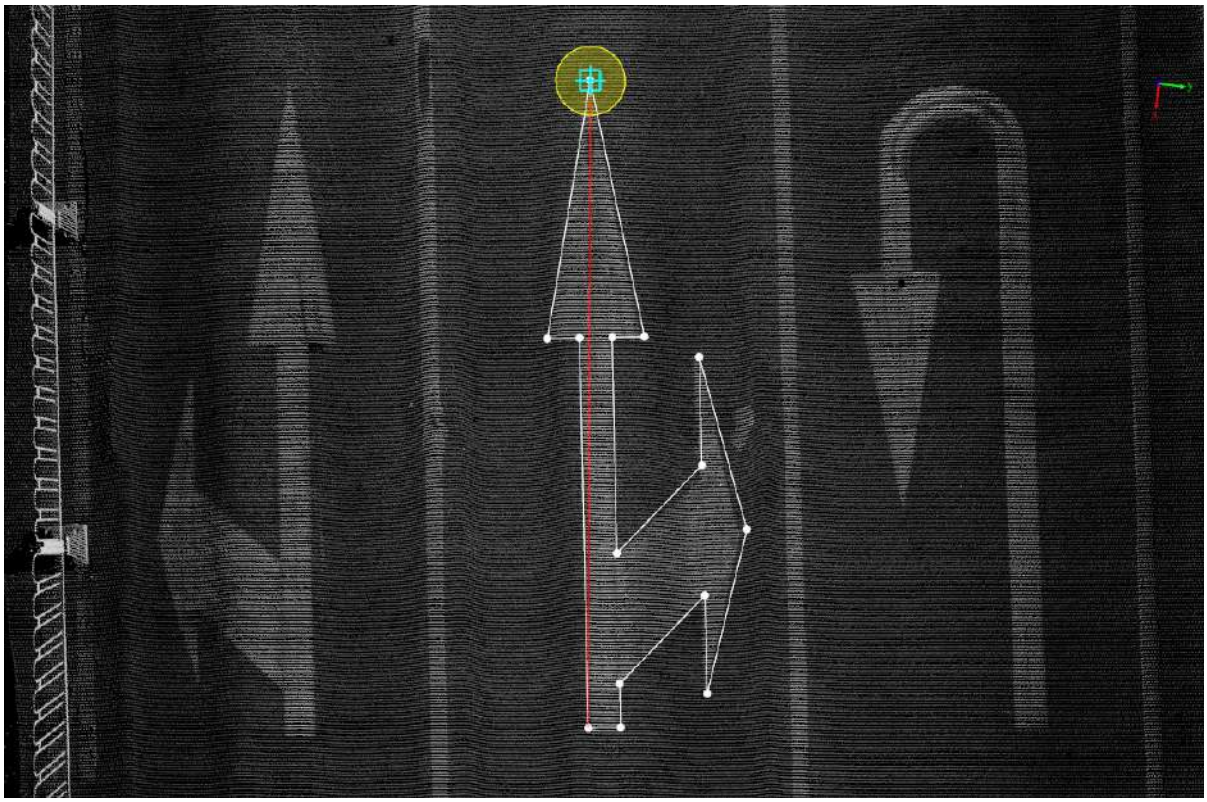
> - The key points in the arrow icons need to be aligned with the actual point cloud.
> - The red point is the first key point and the green point is the second key point.

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

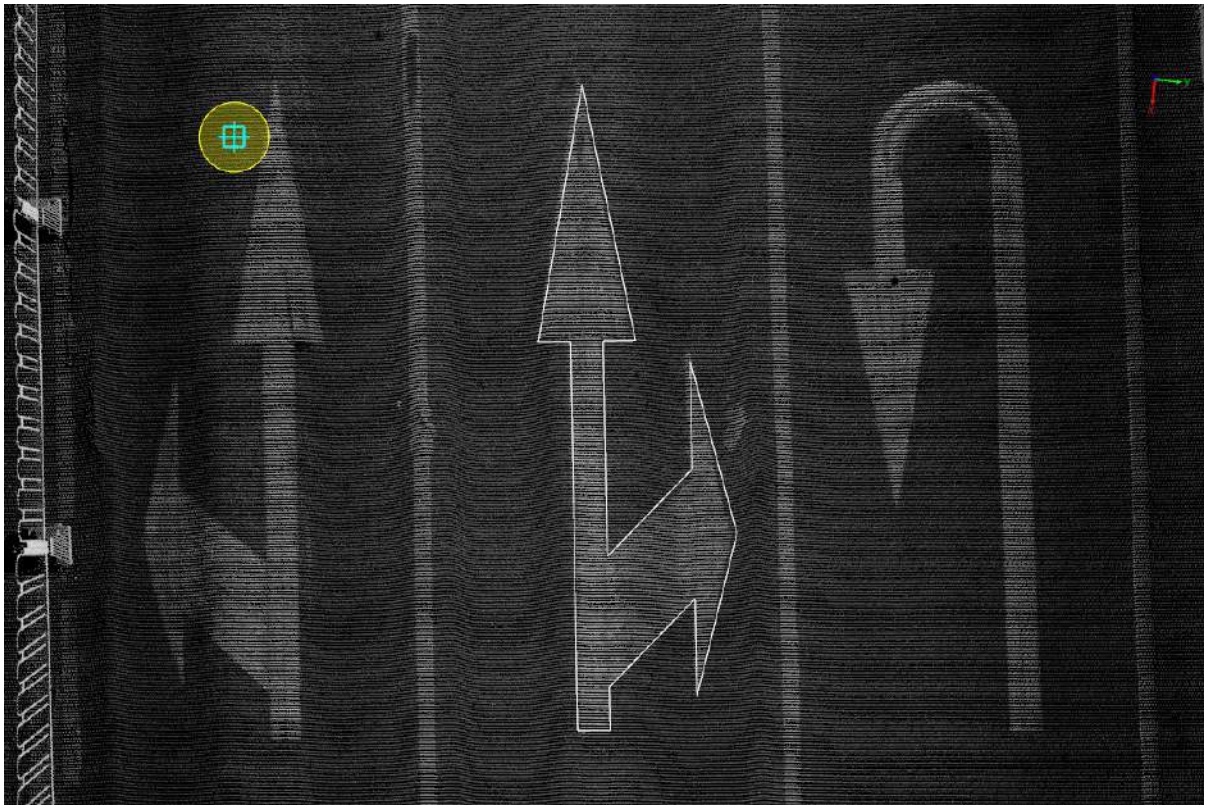


First key point

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



Second key point



Straight/turn right arrow drawing

Add Straight/Left/Right

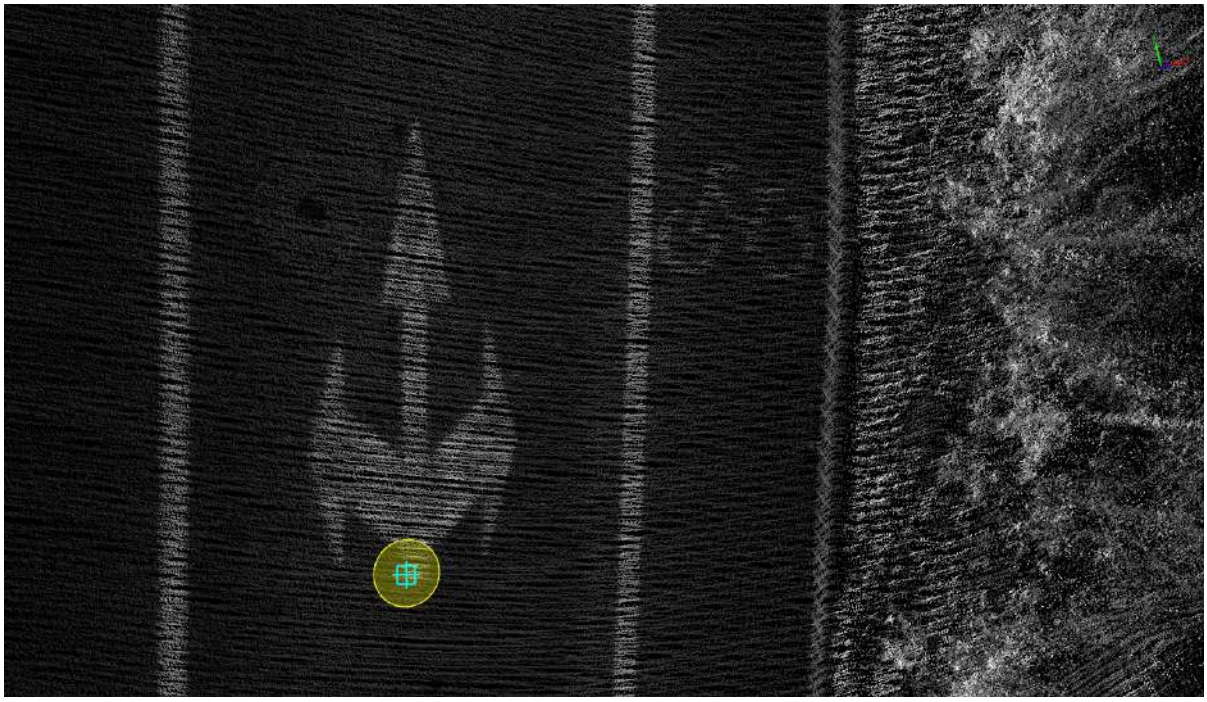
Function Description：The straight/turn left/turn right arrows indicate that the current lane can go straight, turn left, or turn right. This tool is used to manually click on the point cloud for vectorization of straight/turn left/turn right arrows.

Steps

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

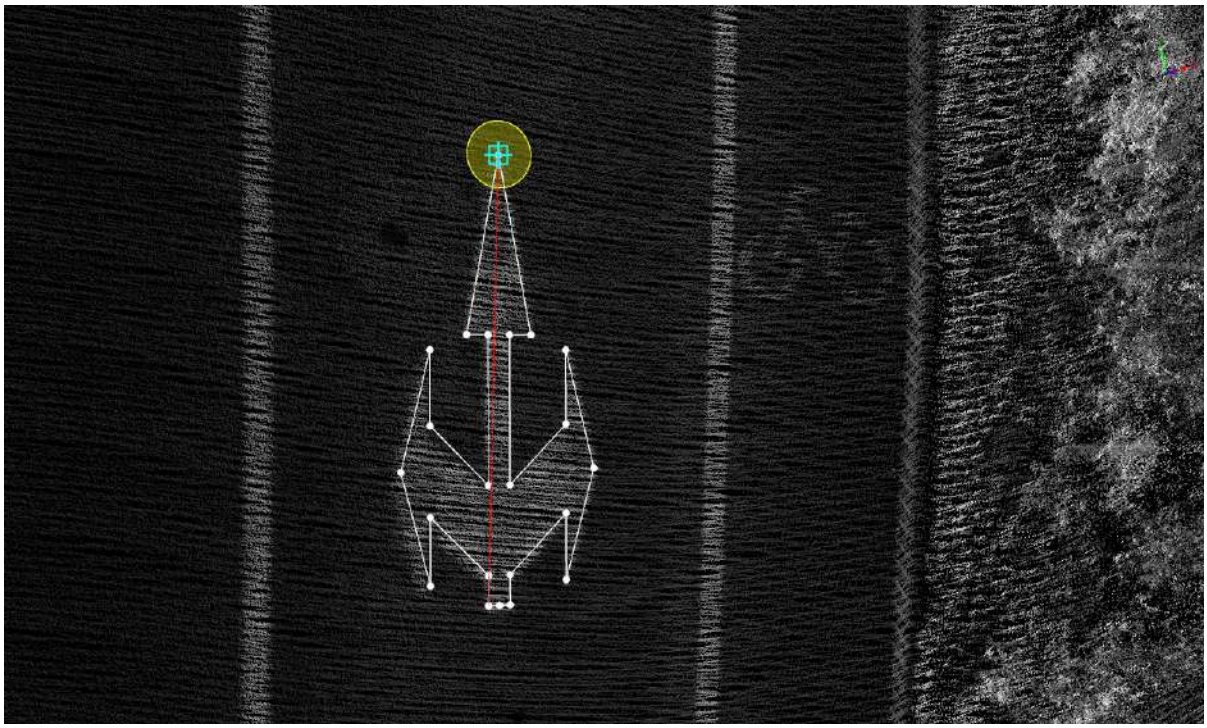
> - The key points in the arrow icons need to be aligned with the actual point cloud.
> - The red point is the first key point and the green point is the second key point.

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

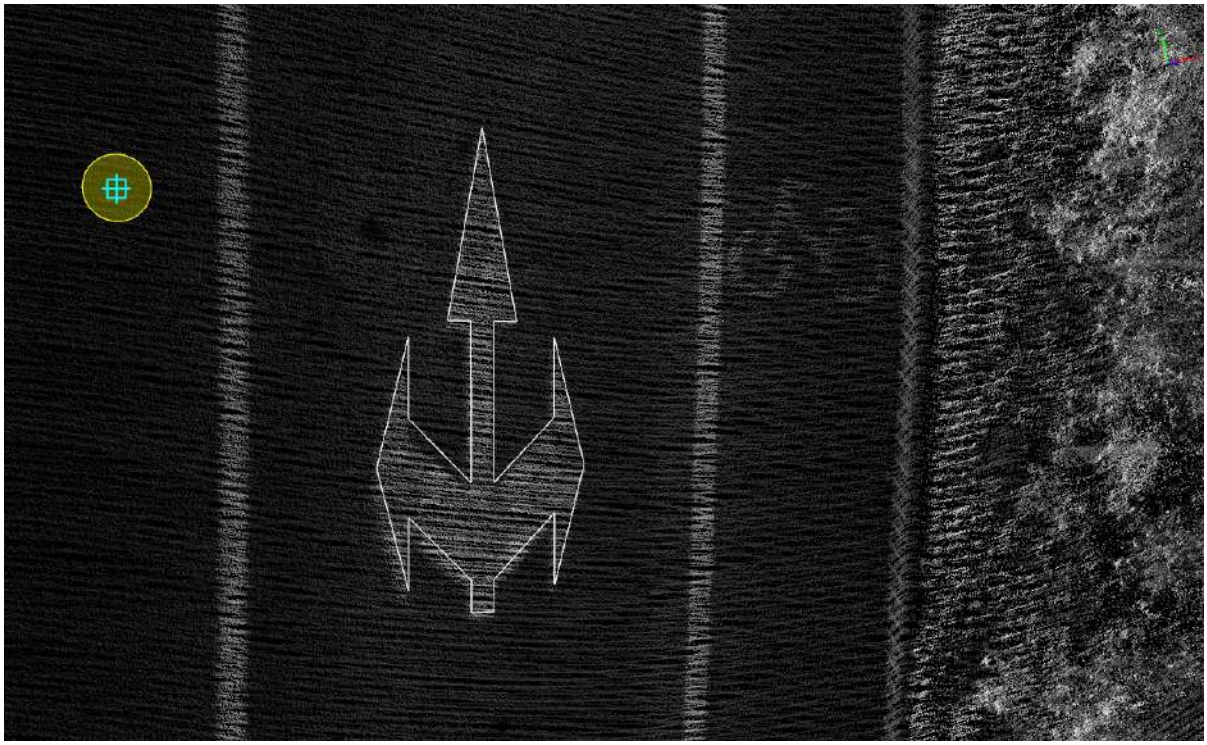


First key point

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



Second key point



Straight/turn left/turn right arrow drawing

Add Turn Round

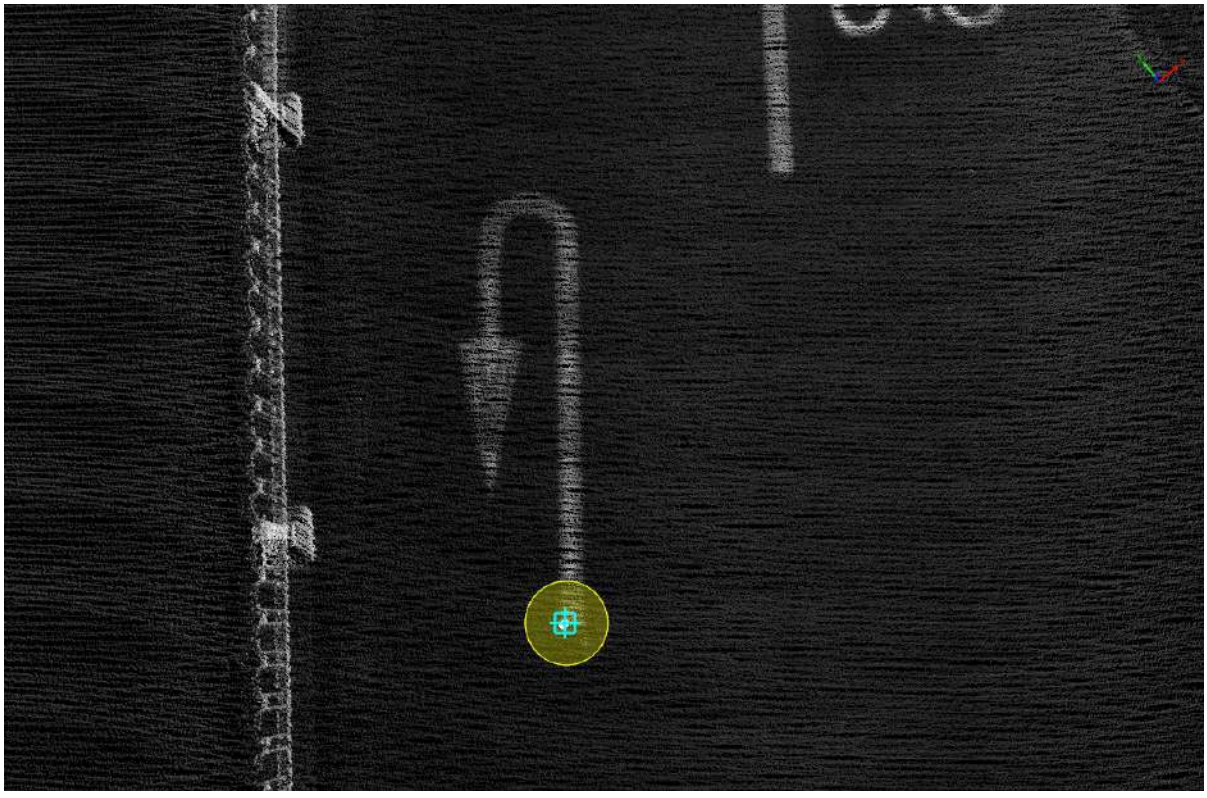
Function Description：U-turn arrow indicates that the current lane will be a U-turn. This tool is used to manually click on the point cloud to vectorize the U-turn arrow.

Steps

1.Click the **Add Turn Round** button in the Road Surface panel.

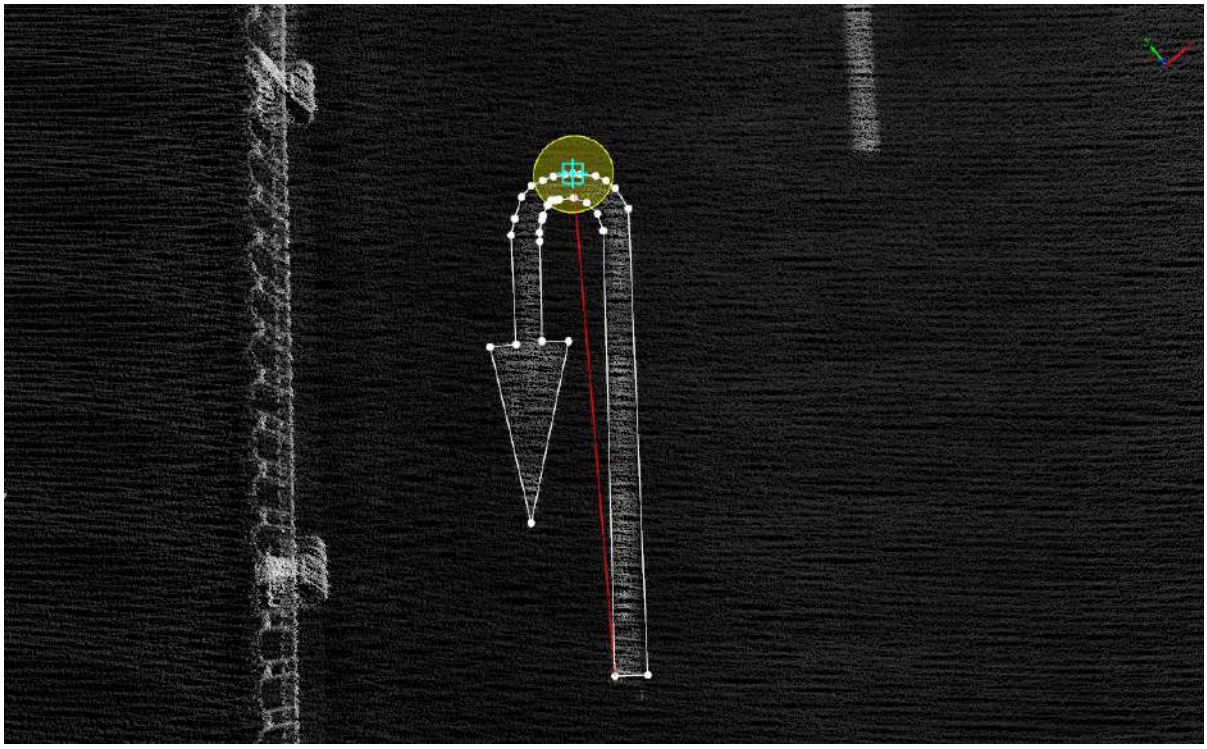
> - The key points in the arrow icons need to be aligned with the actual point cloud.
> - The red point is the first key point and the green point is the second key point.

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

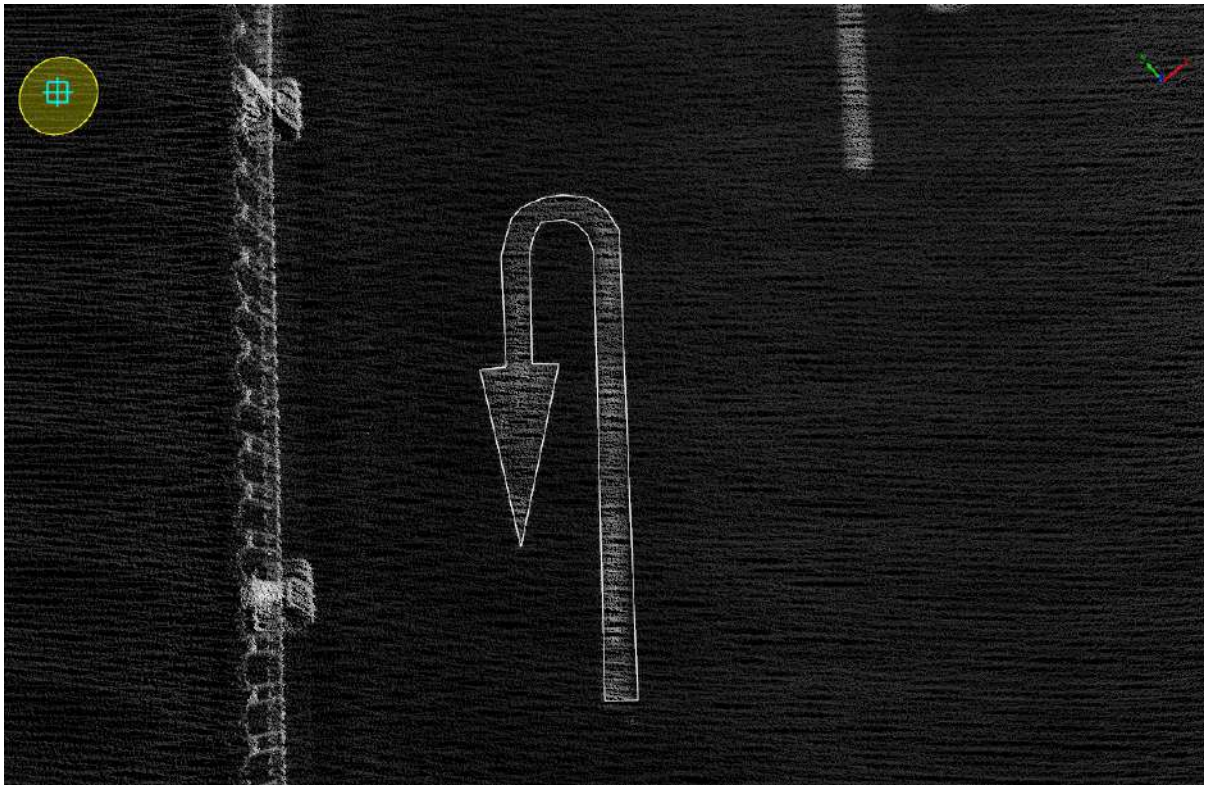


First key point

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



Second key point



U-turn arrow drawing

Add Straight/Turn Round

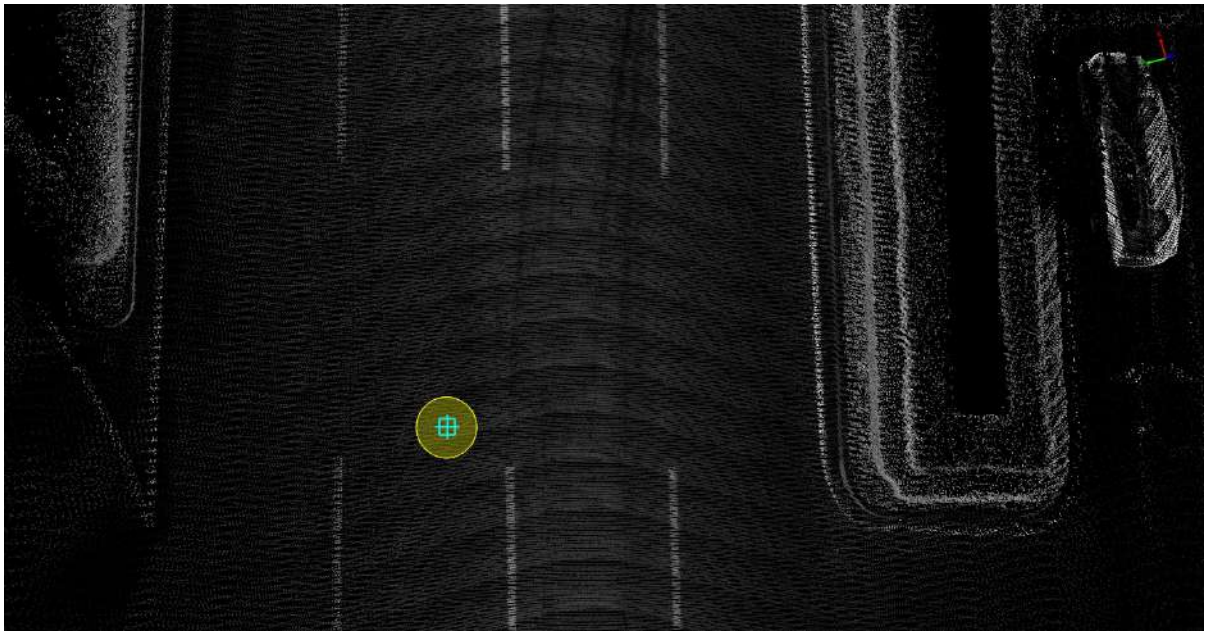
Function Description：Straight/U-turn arrow indicates that the current lane can be U-turn or go straight. This function is used to manually click on the point cloud to vectorize the straight/U-turn arrow.

Steps

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

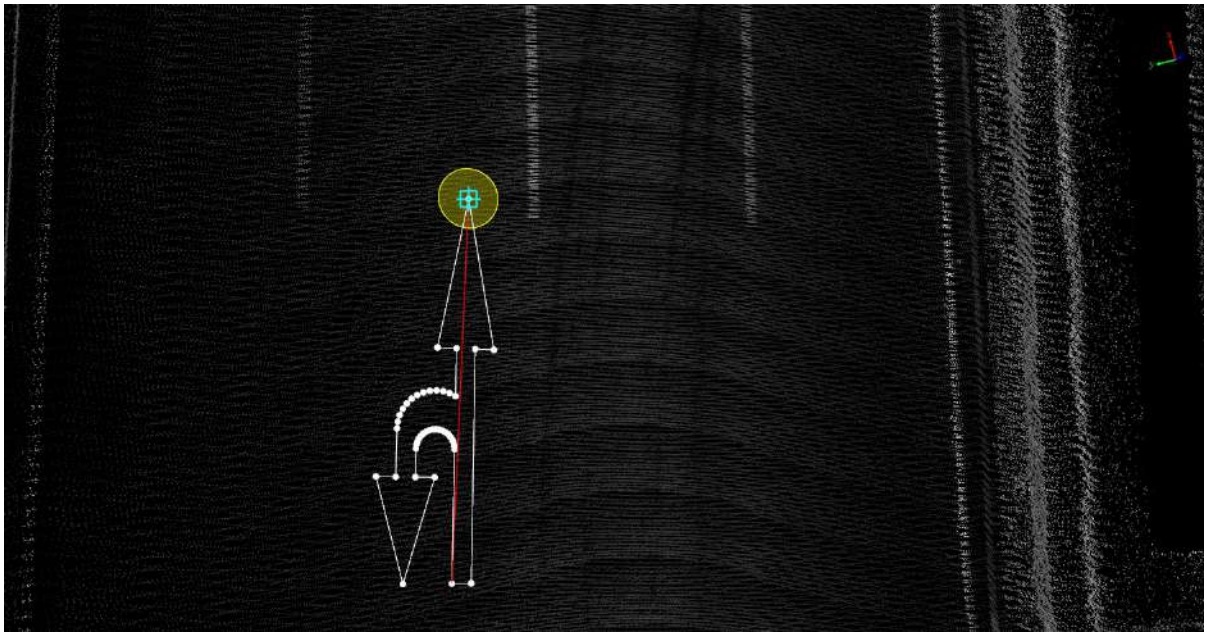
> - The key points in the arrow icons need to be aligned with the actual point cloud.
> - The red point is the first key point and the green point is the second key point.

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

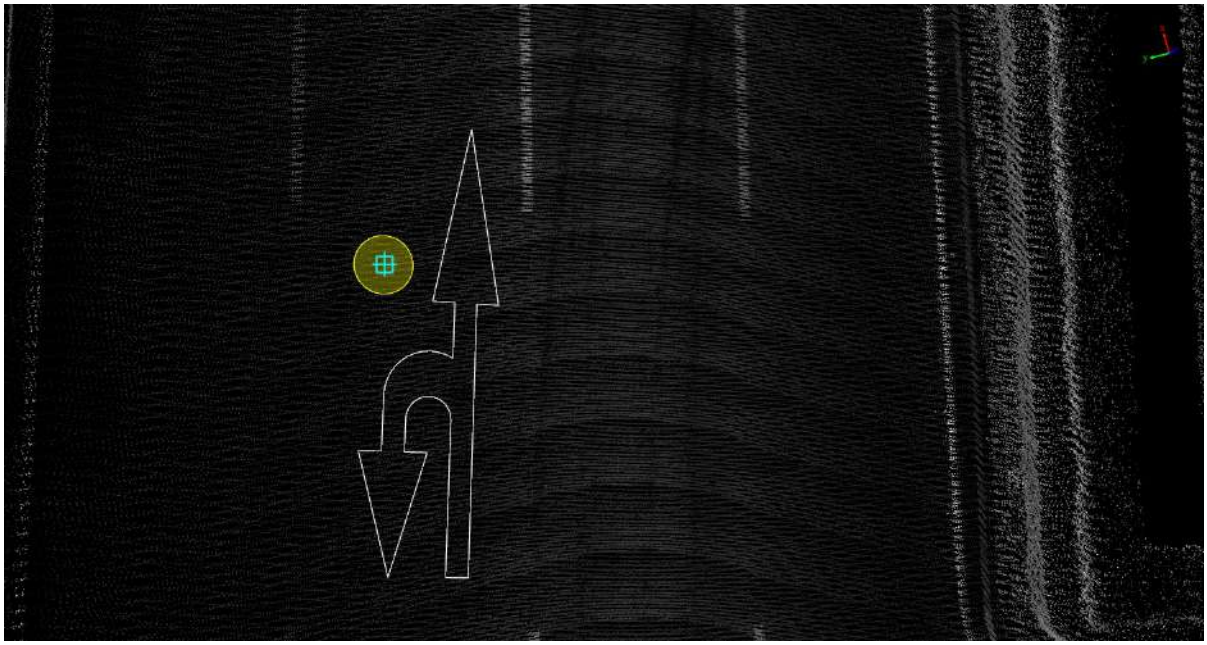


First key point

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



Second key point



Straight/U-Turn arrow drawing

Add Left/Turn Round

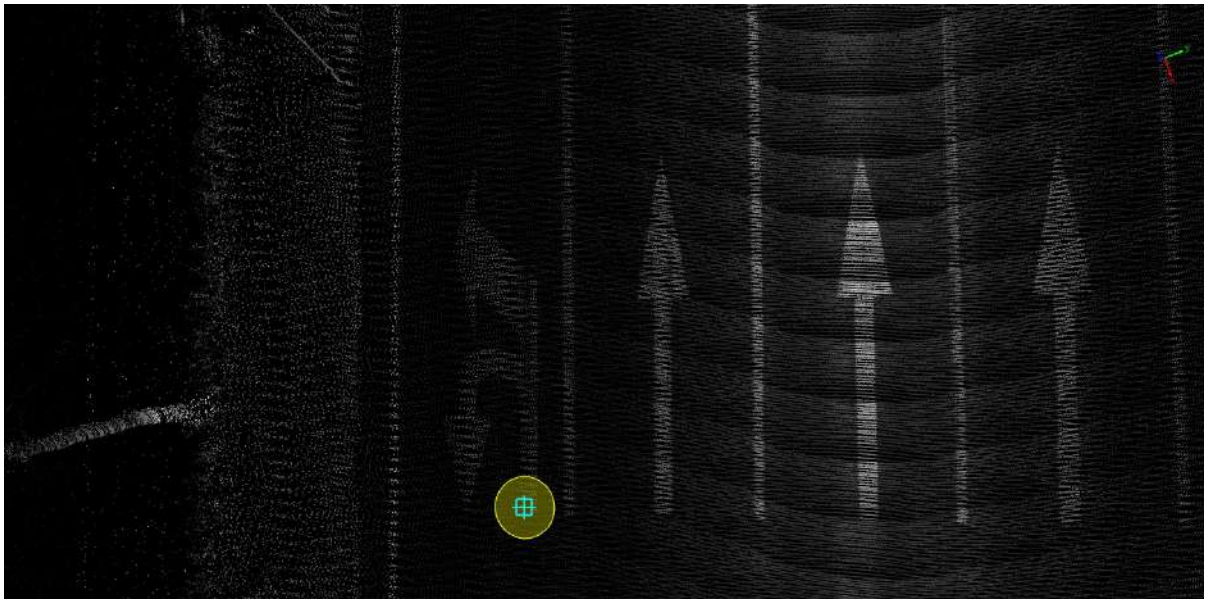
Function Description：Turn left/U-turn arrows indicate that the current lane can turn left or turn around, this tool is used to manually click on the point cloud for vectorization of turn left/U-turn arrows.

Steps

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

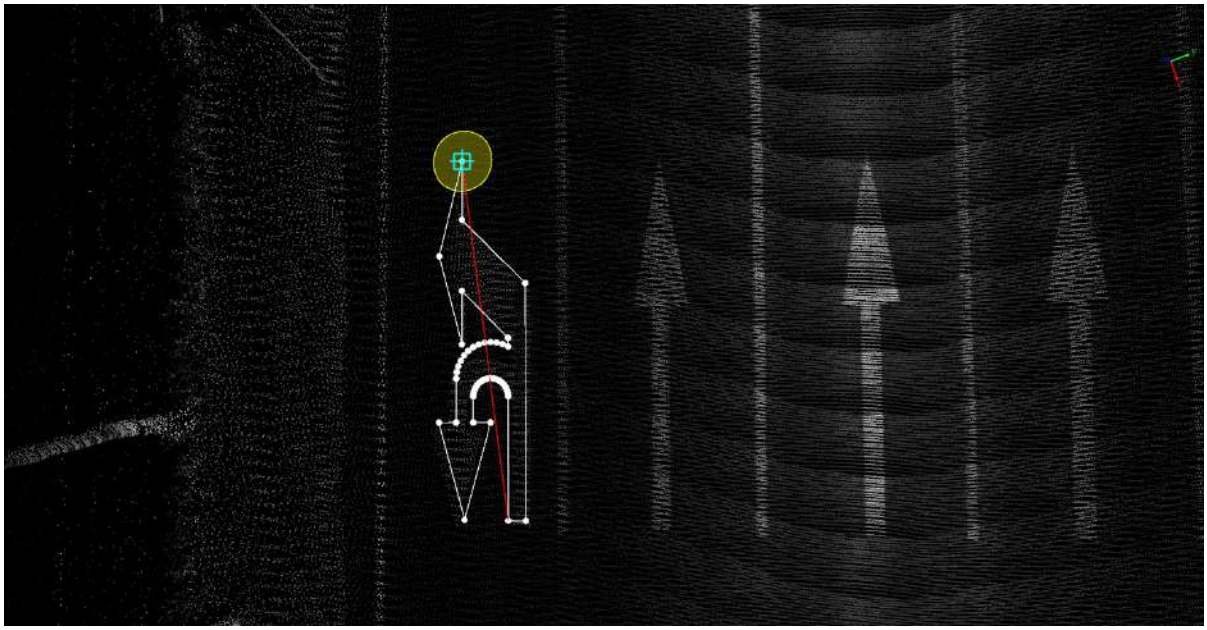
> - The key points in the arrow icons need to be aligned with the actual point cloud point cloud.
> - The red point is the first key point and the green point is the second key point.

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

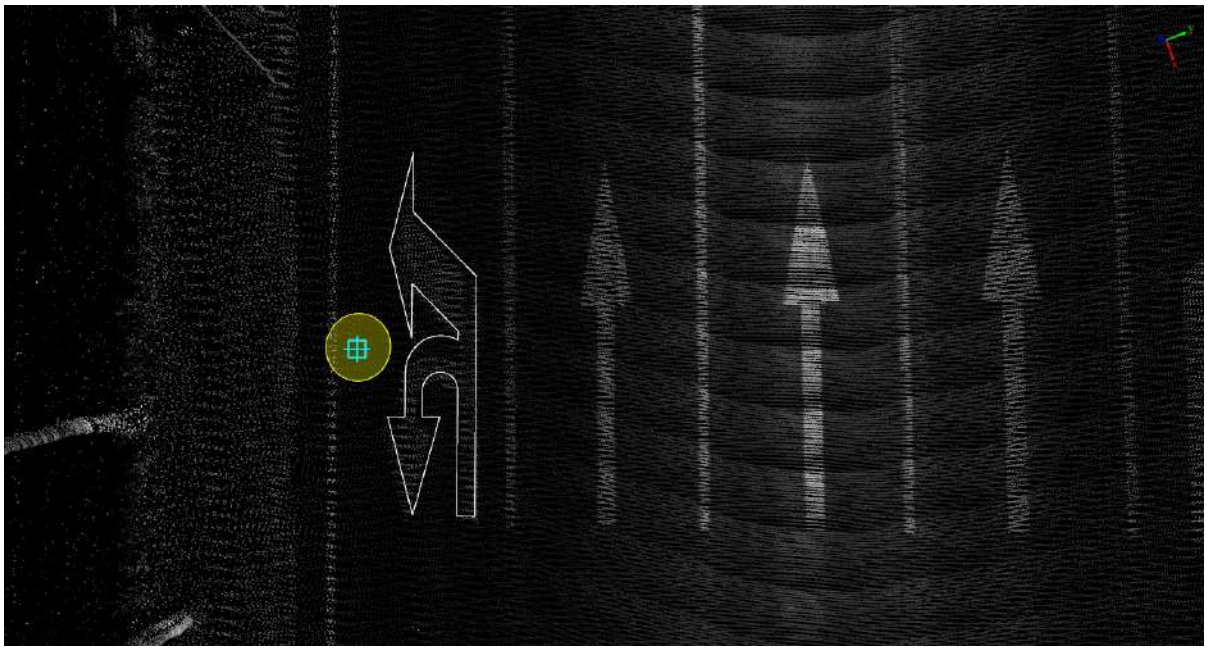


First key point

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



Second key point



Turn left/U-turn arrow drawing

Add Left Merge

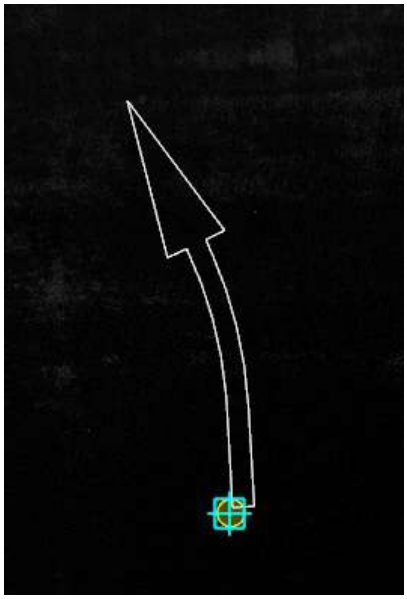
Function Description：The left merge arrow indicates that the current lane merges to the left. This tool is used to manually select the point cloud to vectorize the left merge arrow.

Steps

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

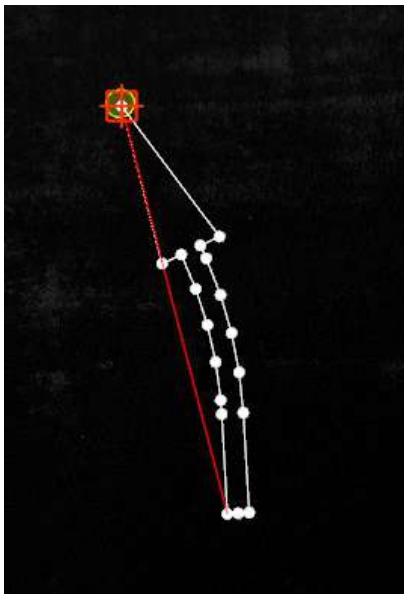
> - The key points in the arrow icons need to be aligned with the actual point cloud point cloud.
> - The red point is the first key point and the green point is the second key point.

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

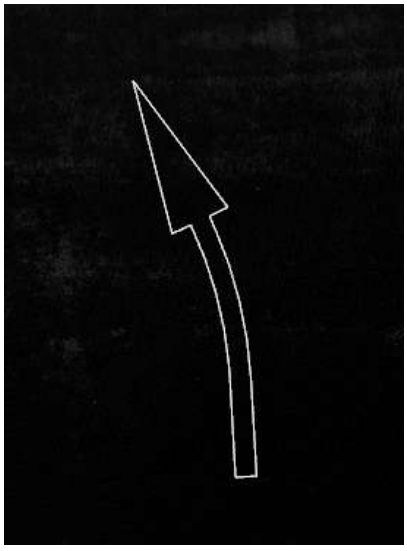


First key point

3.Move the mouse to check the fit of the **template** and the **actual point cloud**, left click **the second key point** to complete the left merge arrow. The details are as follows, **the second key point** in the icon.



Second key point



Left merge arrow drawing

Add Right Merge

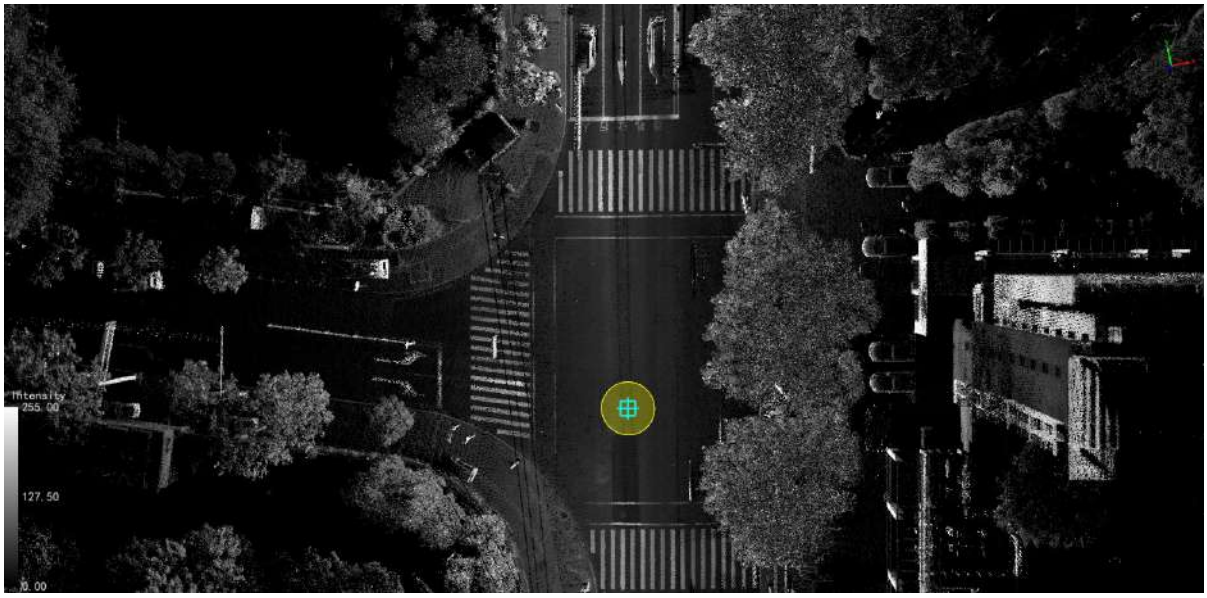
Function Description：The right merge arrow indicates that the current lane is merging to the right, this tool is used to manually click on the point cloud to vectorize the right merge arrow.

Steps

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

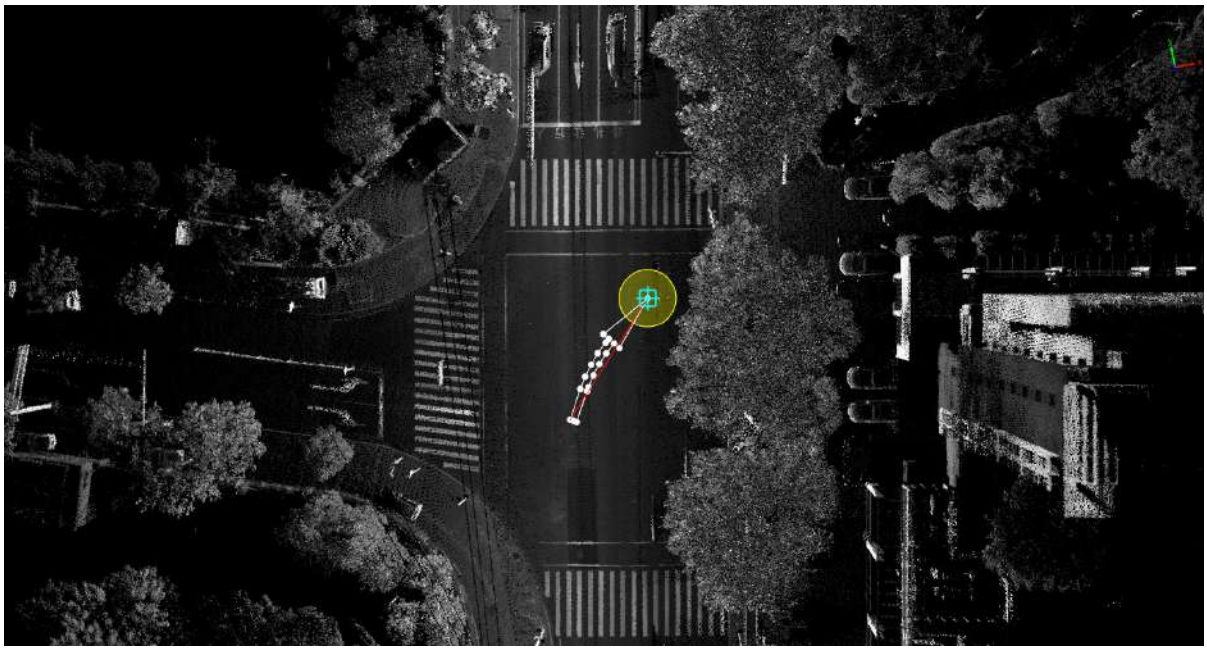
> - The key points in the arrow icons need to be aligned with the actual point cloud.
> - The red point is the first key point and the green point is the second key point.

2.**Left mouse click** to select right merge arrow point cloud key point, details below, icon in **first key point**.

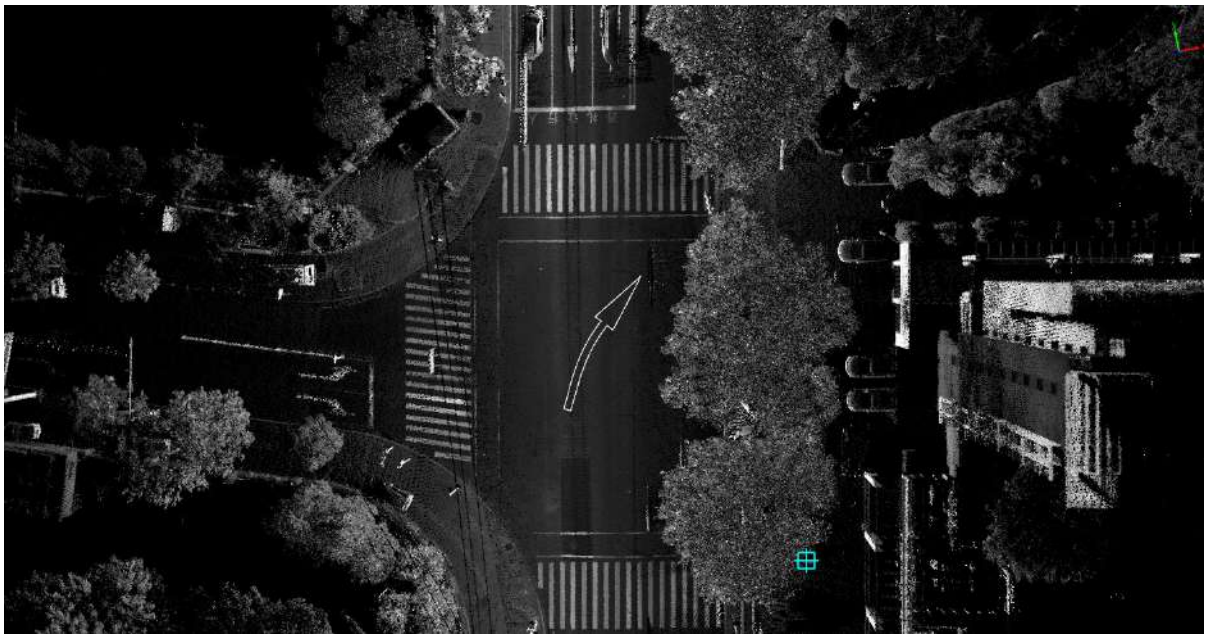


First key point

3.Move the mouse to see how the **template** fits into the **actual point cloud**, **left mouse click** to select the **second key point** to complete the creation of the merging arrow to the right, details are as follows, icon in **second key point**.



Second key point



Left merge arrow drawing

Add Right/Turn Round

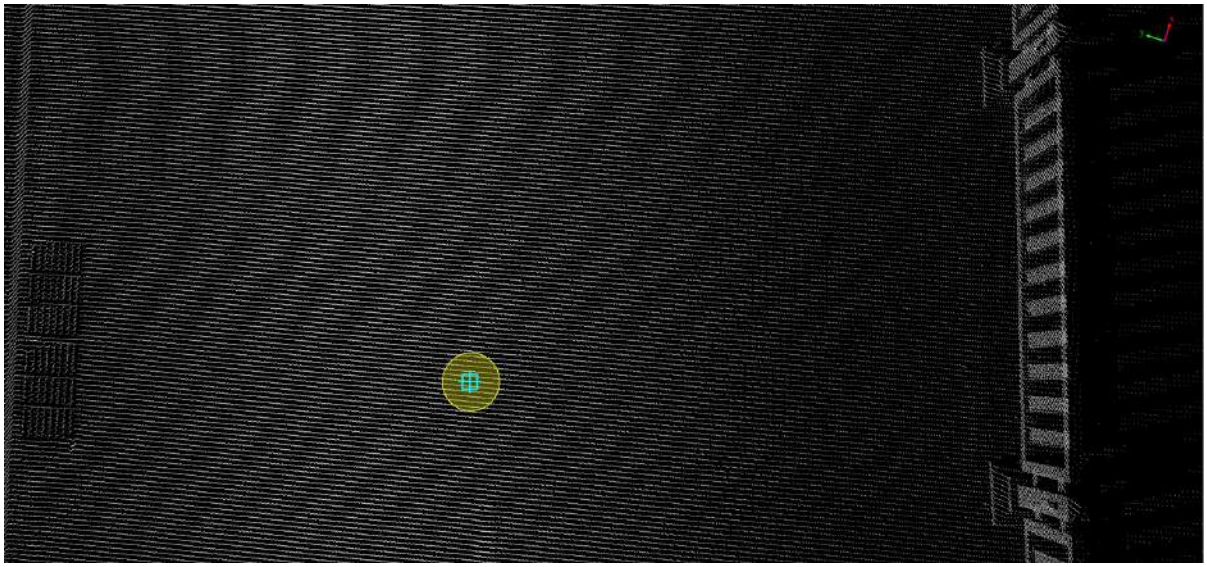
Function Description：The turn right/U-turn arrow indicates that the current lane allows a turn right or U-turn. This tool is used to manually click on the point cloud to vectorize the turn right /U-turn arrows.

Steps

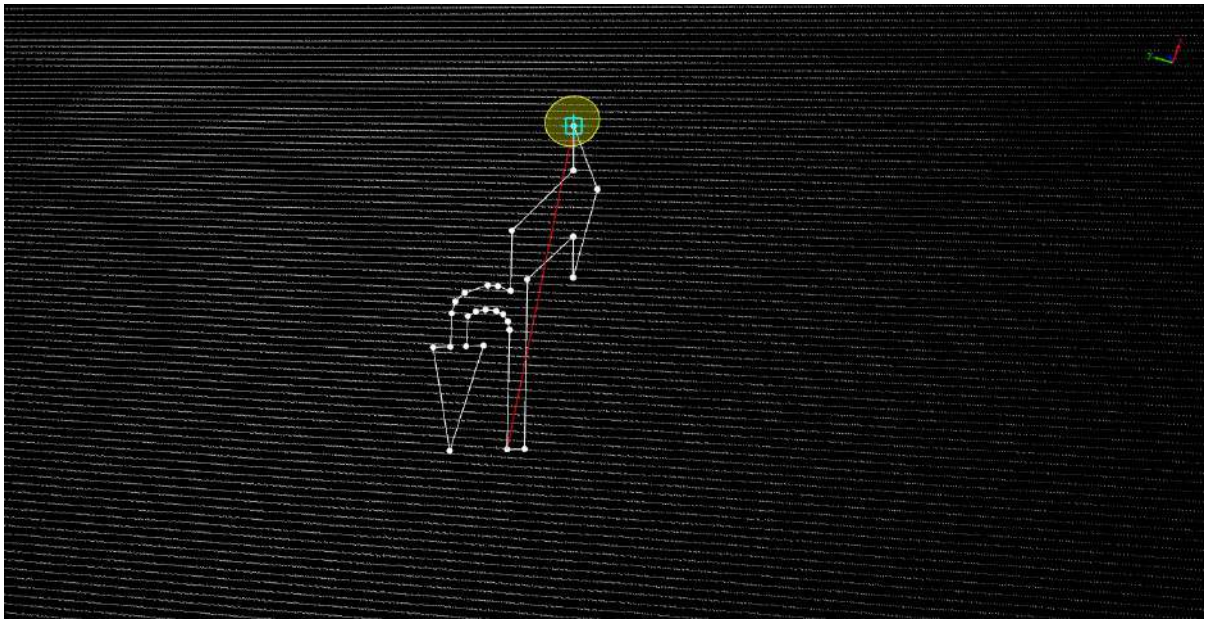
1.Click the **Add Right/Turn Round** button in the Road Surface panel.

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

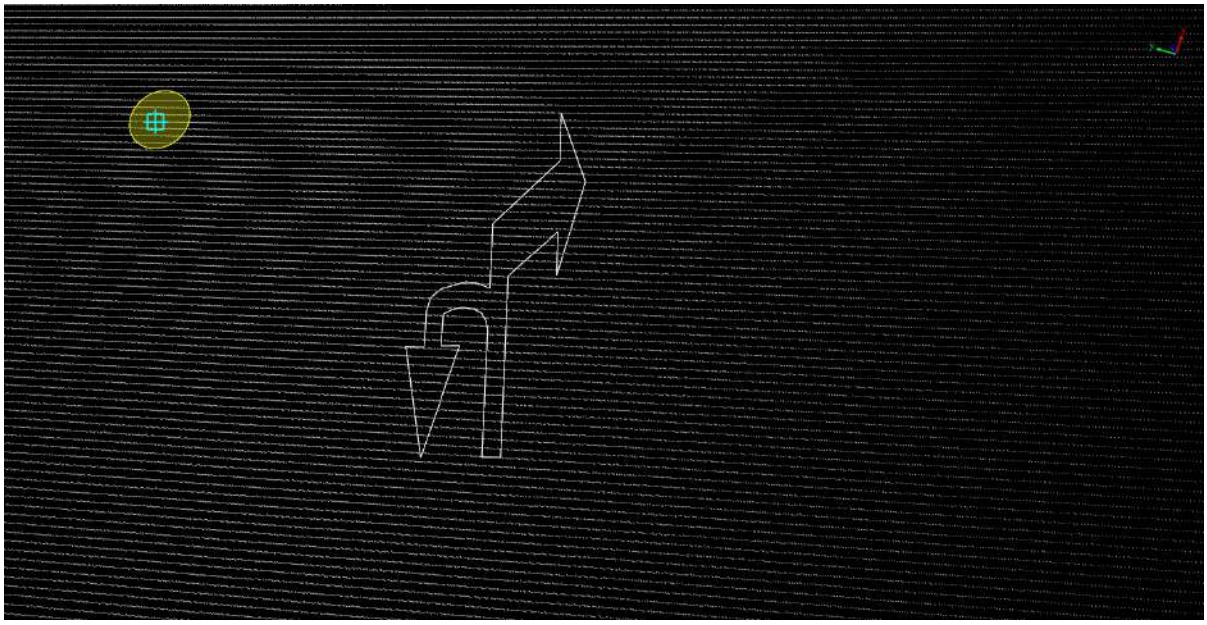
2.**Left mouse click** to select the point cloud key point, the details are as follows.



First key point



Second key point



Turn right /U-turn arrow drawing

Add Straight/Right/Turn Round

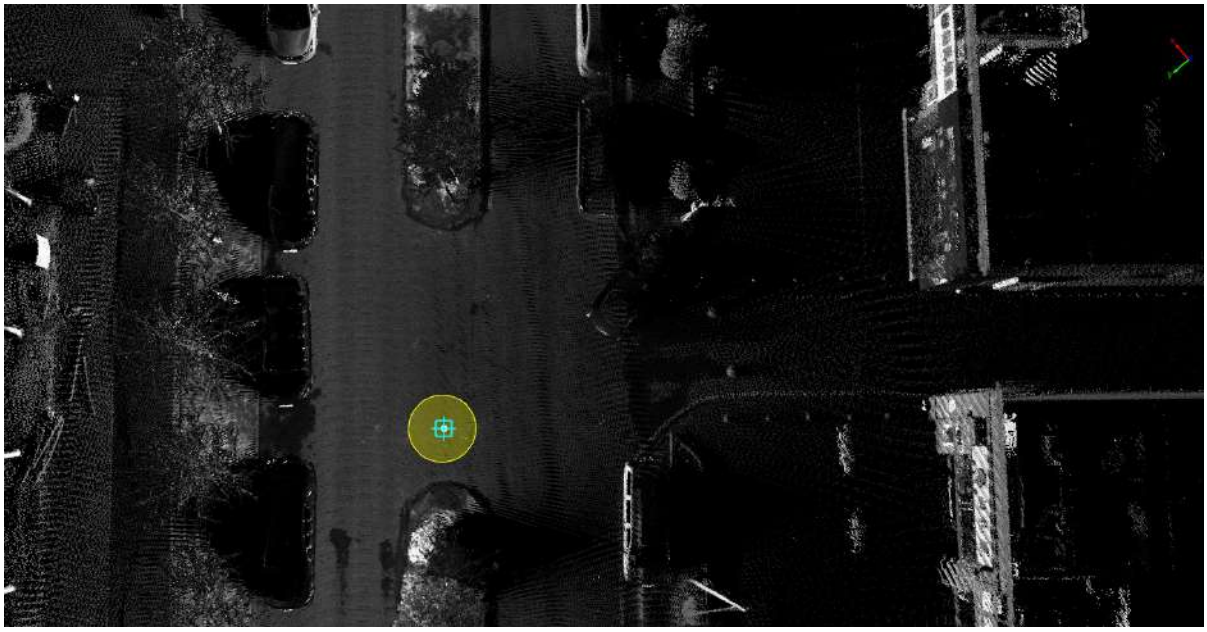
Function Description：Straight/turn right/U-turn arrows indicate that the current lane allows going straight, right-turn and U-turn. This tool is used to manually click on the point cloud to vectorize the straight/turn right/U-turn arrows.

Steps

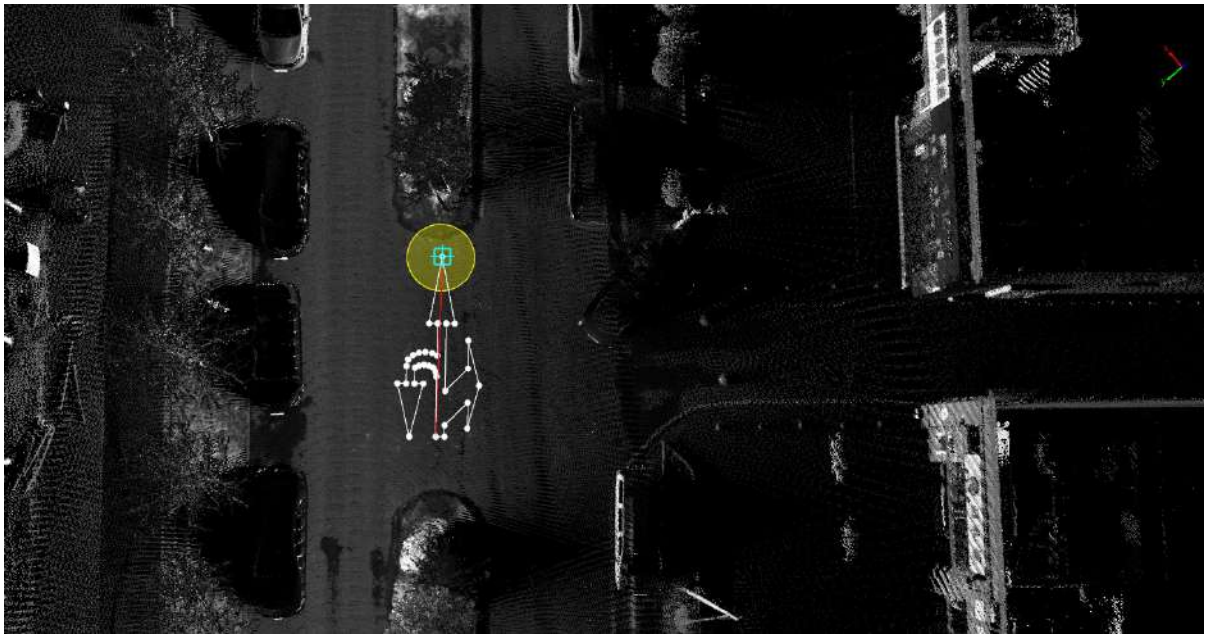
1.Click the **Add Straight/Right/Turn Round** button in the Road Surface panel.

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

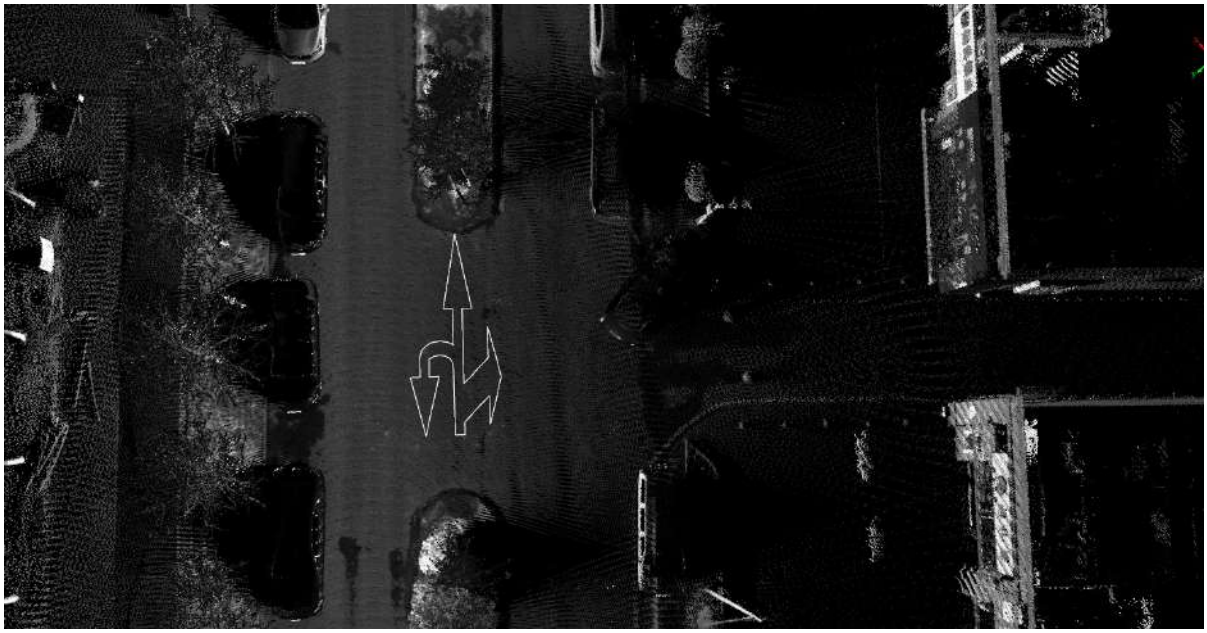
2.**Left mouse click** to select the point cloud key points, the details are as follows, the icon of **first key point**.



First key point



Second key point



Straight/turn right/U-turn arrow drawing

Add Left/Right/Turn Round

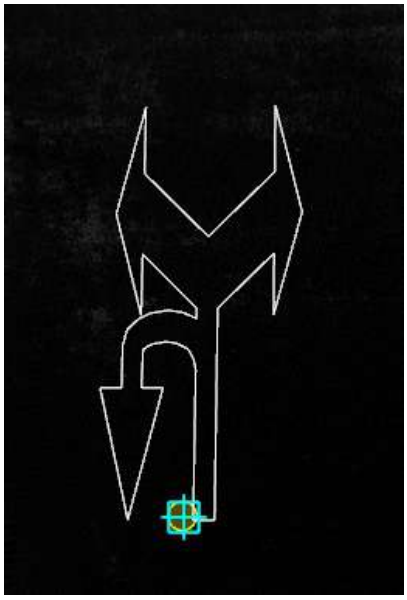
Function Description：Turn left/turn right/U-turn arrows indicate that the current lane can turn left, right, or turn around. This tool is used to manually click on the point cloud for vectorization of turn left/turn right/U-turn arrows.

Steps

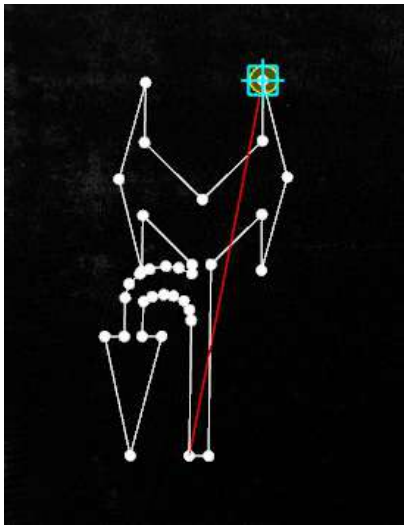
1.Click the **Add Left/Right/Turn Round** button in the Road Surface panel.

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

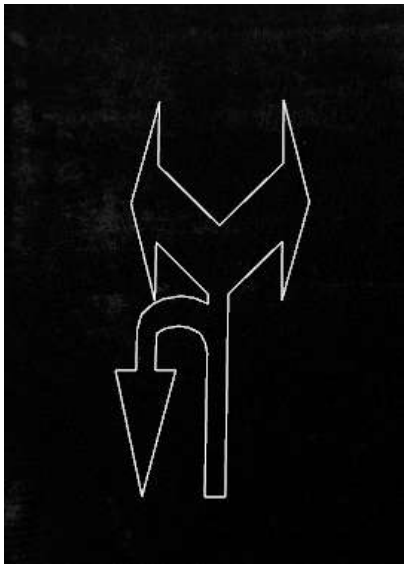
2.**Left mouse click** to select the point cloud key points, the details are as follows, the icon of **first key point**.



First key point



Second key point



Turn left/turn right/U-turn arrow drawing

Add CrossWalk

Function Description：This tool enables the outlining of crosswalks. Use three key points, manually click the point cloud, and make the vectorization of the crosswalk.

Steps

1.In the drop-down icon in the Road Surface panel, click the **Add CrossWalk** button.

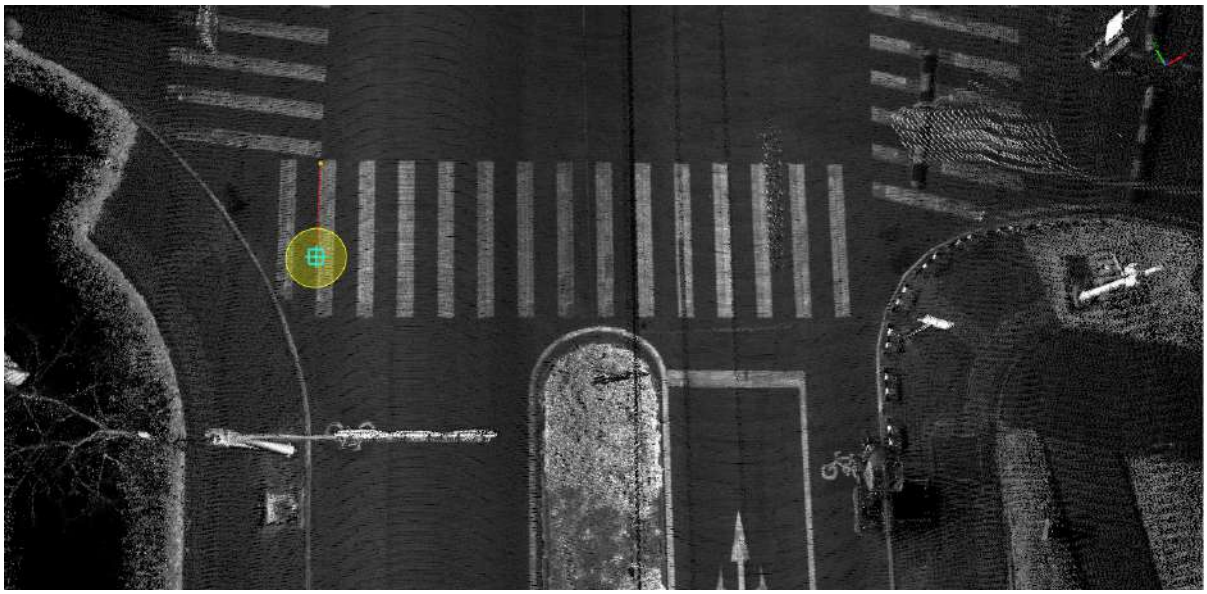
> - The **yellow point** in the figure below is the key point, and the key point needs to be aligned with the actual point cloud.
> - **Yellow point** position is the **position of the left click**.

2.3D window tips.：

Shortcut name	Role
C	Switch between standard/custom drawing modes
H	Turn off/on top left corner alert
Current model	Current model

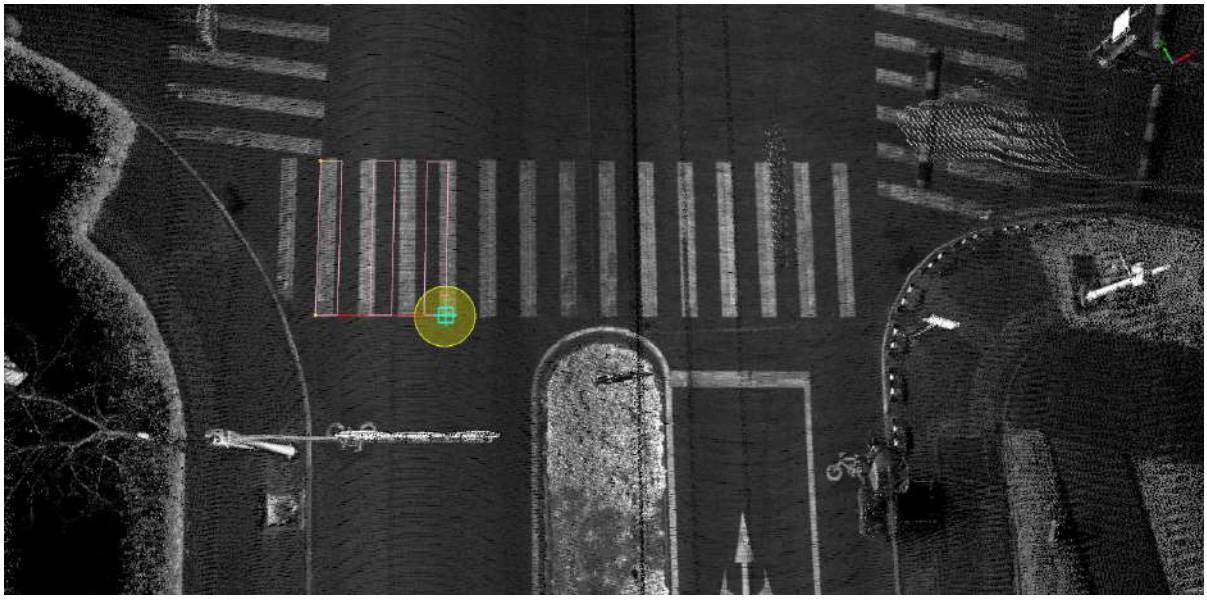
3.Standard model：

（1）**Left mouse click** to select the **first key point** of the crosswalk point cloud, the details are as follows, the **yellow point** position is the **first key point** in the icon.



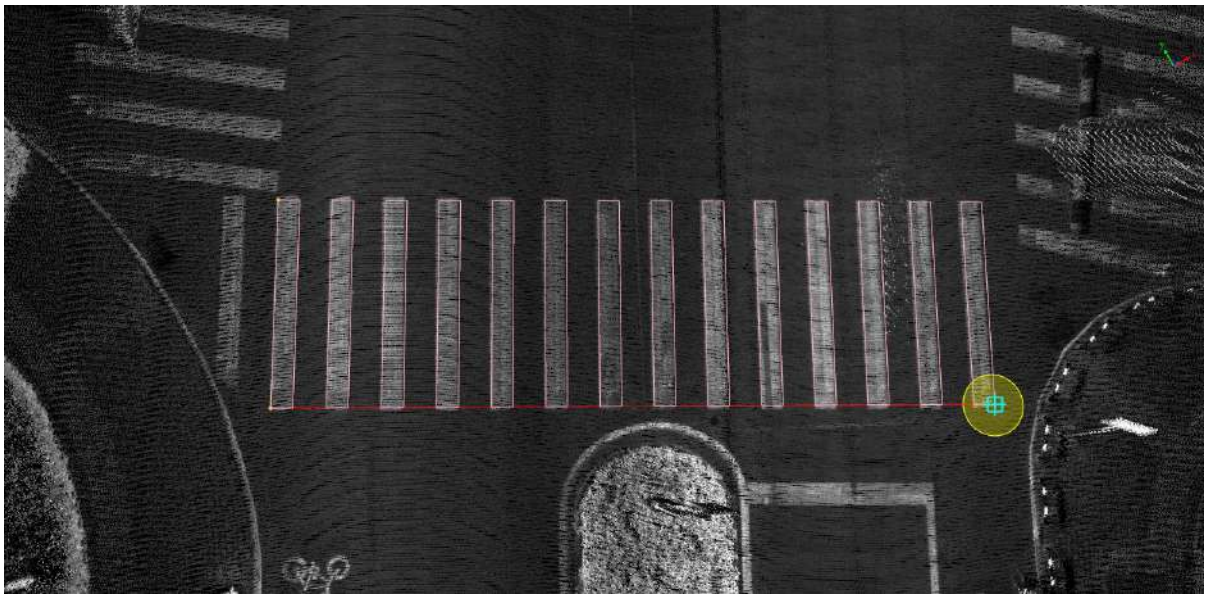
First key point

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

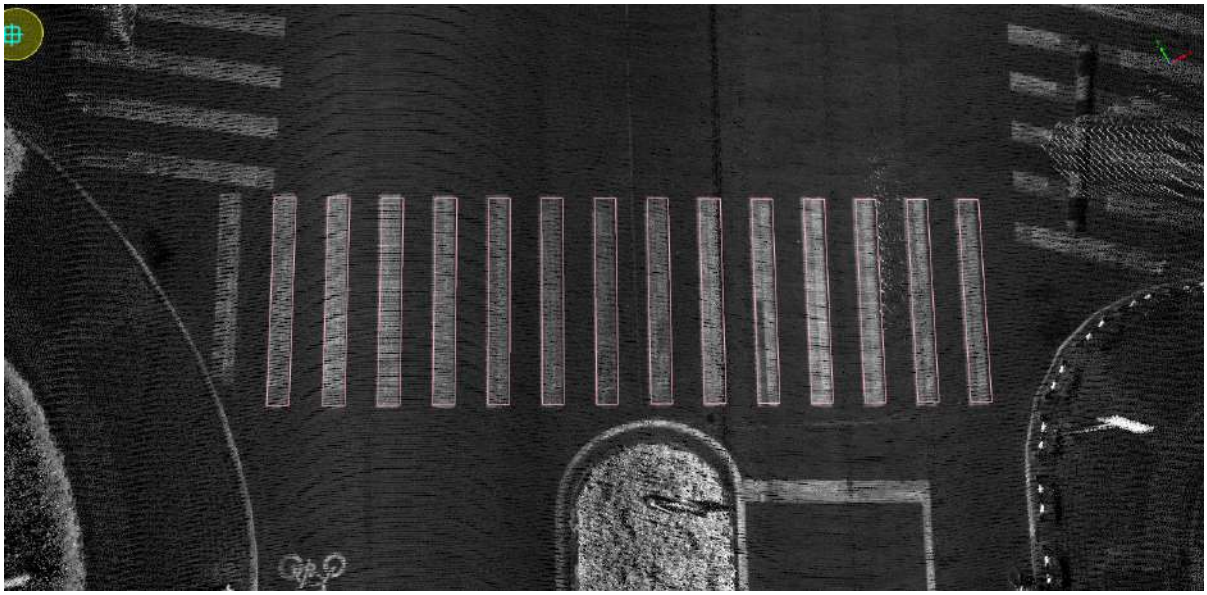


Second key point

（3）Move the mouse to check the fit of the **template** and the **actual point cloud**, Left click **the third key point**, and determine the **crosswalk length**, The details are as follows, the **yellow point** is the **third key point in the icon**.



Third key point

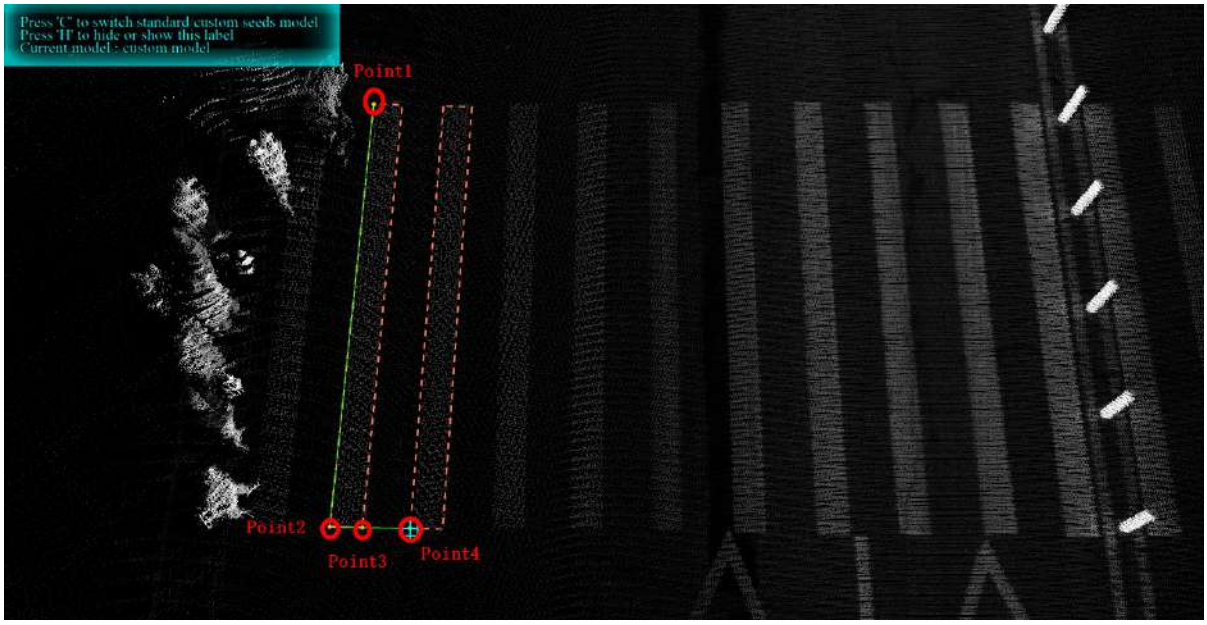


Crosswalk drawing

4.Customised models：

（1）Use the shortcut key **C** to switch the mode to custom model, the current model is displayed as custom model.

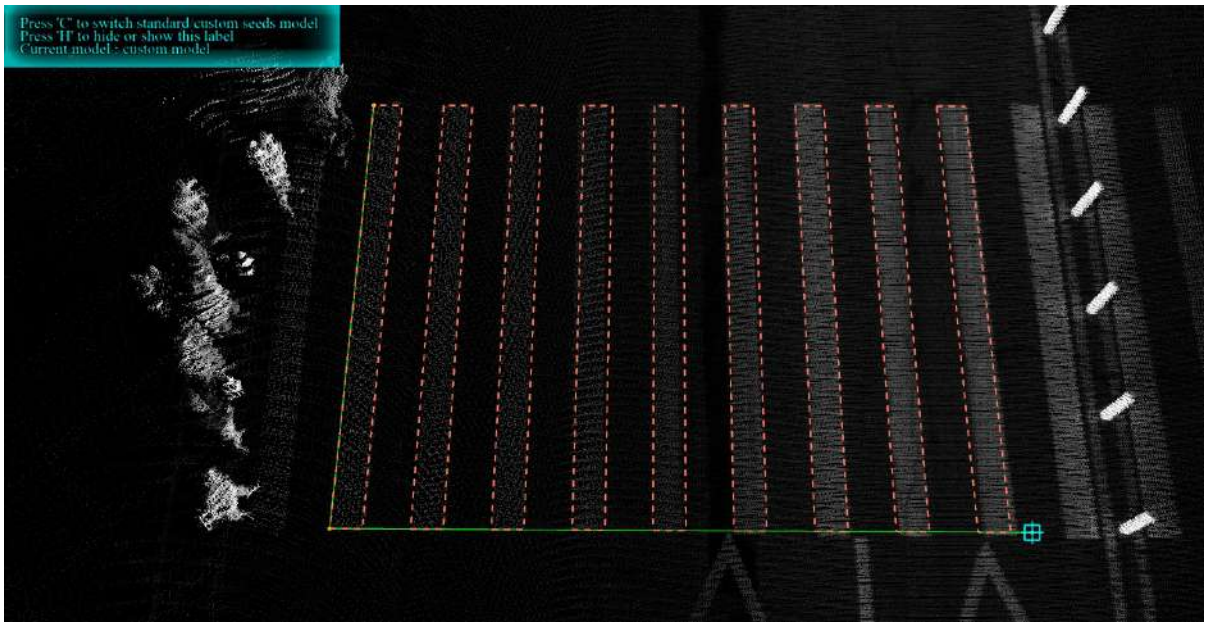
（2）Click with the left mouse button on each of the four key points shown below



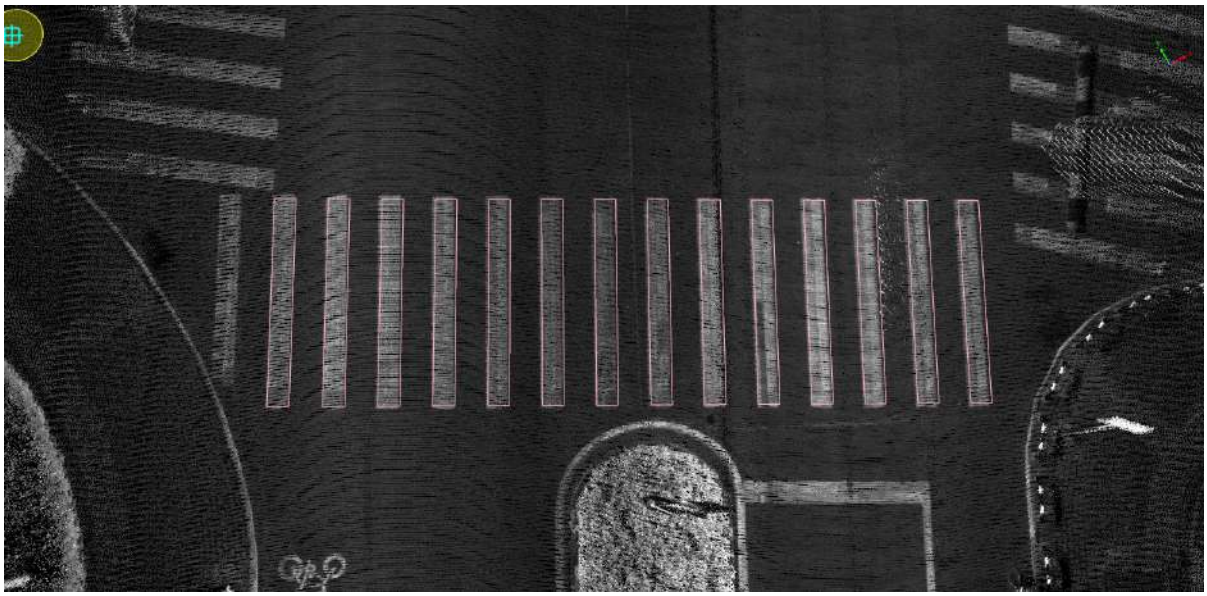
Custom Mode Key Points

> Notes：
>
> - For the first two points, click on the long side of the outside of the pedestrian crossing
> - for the third point, click on the intersection of the pedestrian crossing in the same line as the second point, the first three points define the shape of a pedestrian crossing
> - For the fourth point, click on the corner point where the adjacent pedestrian crossing is in line with points 2 and 3 in the direction parallel to the pedestrian crossing, thus determining the spacing of the pedestrian crossing

（3）Finally, drag the mouse to pull out the live vector and double-click to end the drawing when it fits with the pedestrian crossing point cloud



Custom mode real time vector



Pedestrian crossing drawing

Add Longitudinal Speed Reduction

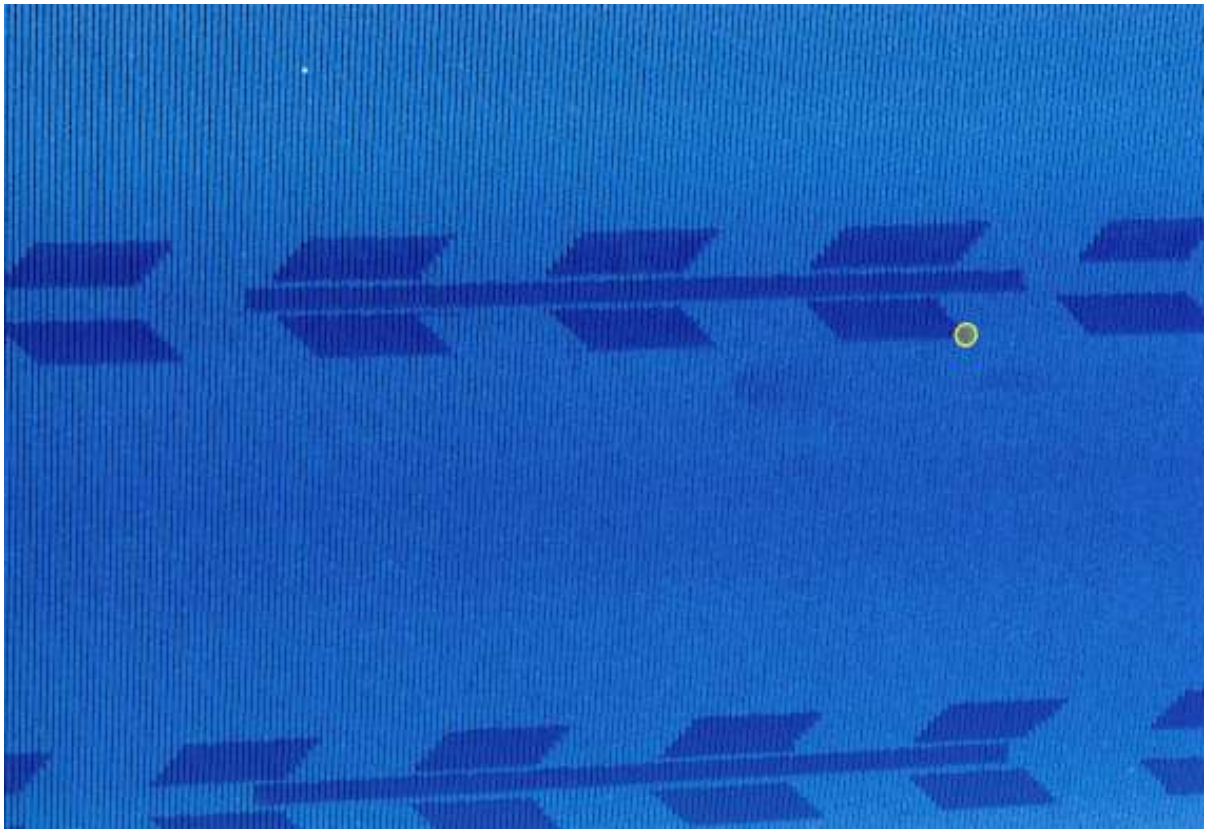
Function Description：This tool can be used to outline Vertical Deceleration Markings. For three key points, manual point selection point cloud for vectorization of vertical deceleration markings.

Steps

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

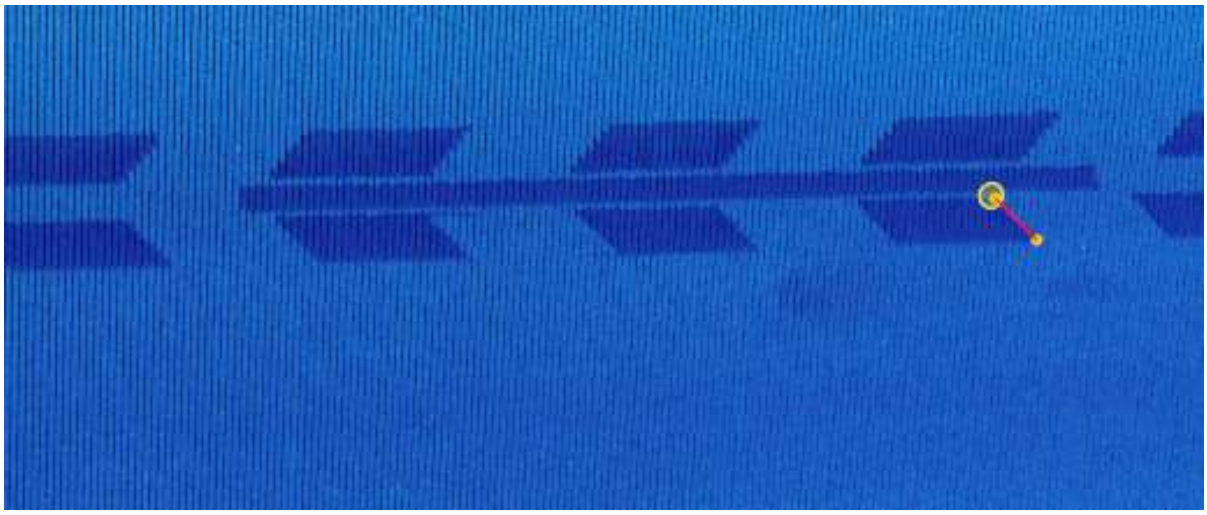
> - The **yellow point** in the figure below is the key point, which needs to be aligned with the actual point cloud.
> - **Yellow point** position is **left mouse click** position button click.

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



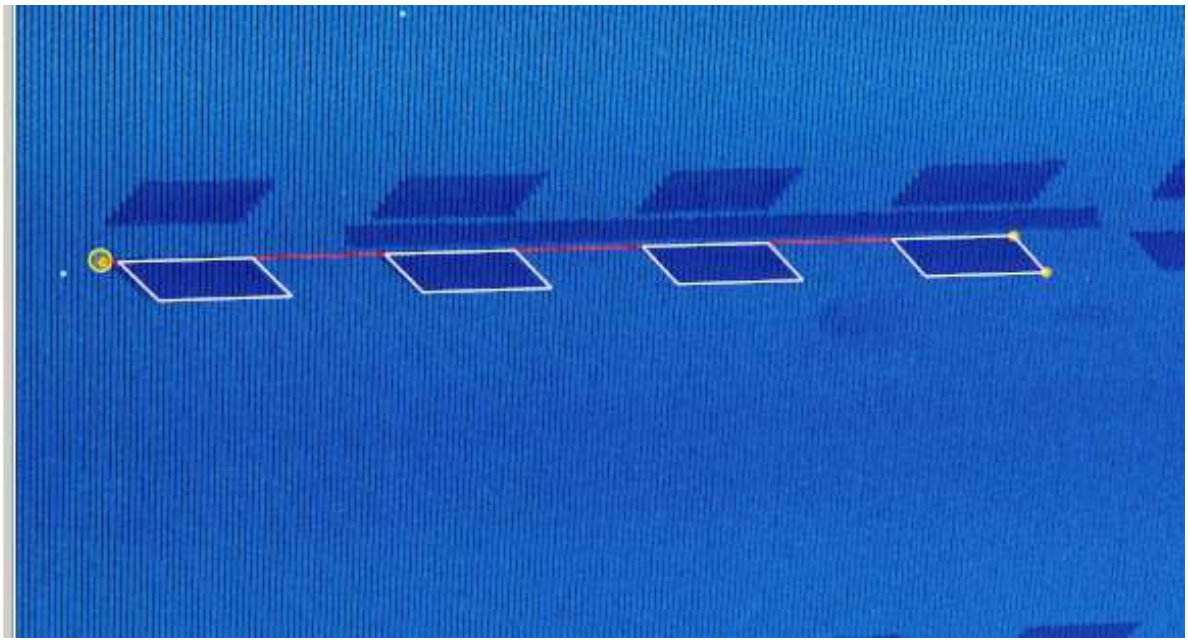
First key point

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

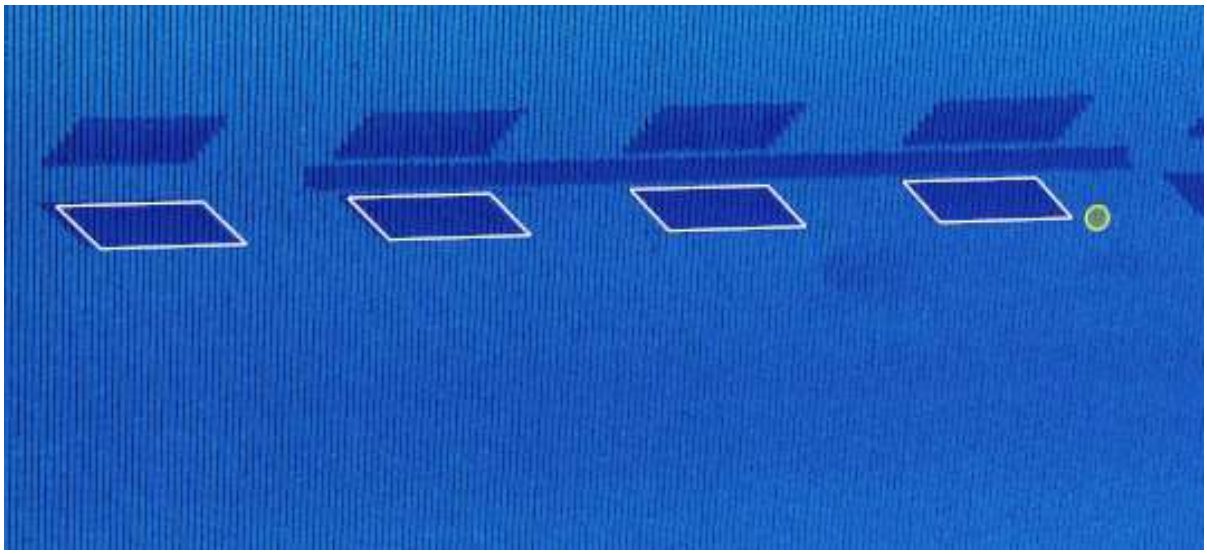


Second key point

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



Third key point



Vertical deceleration markings drawing

Add Speed Reduction

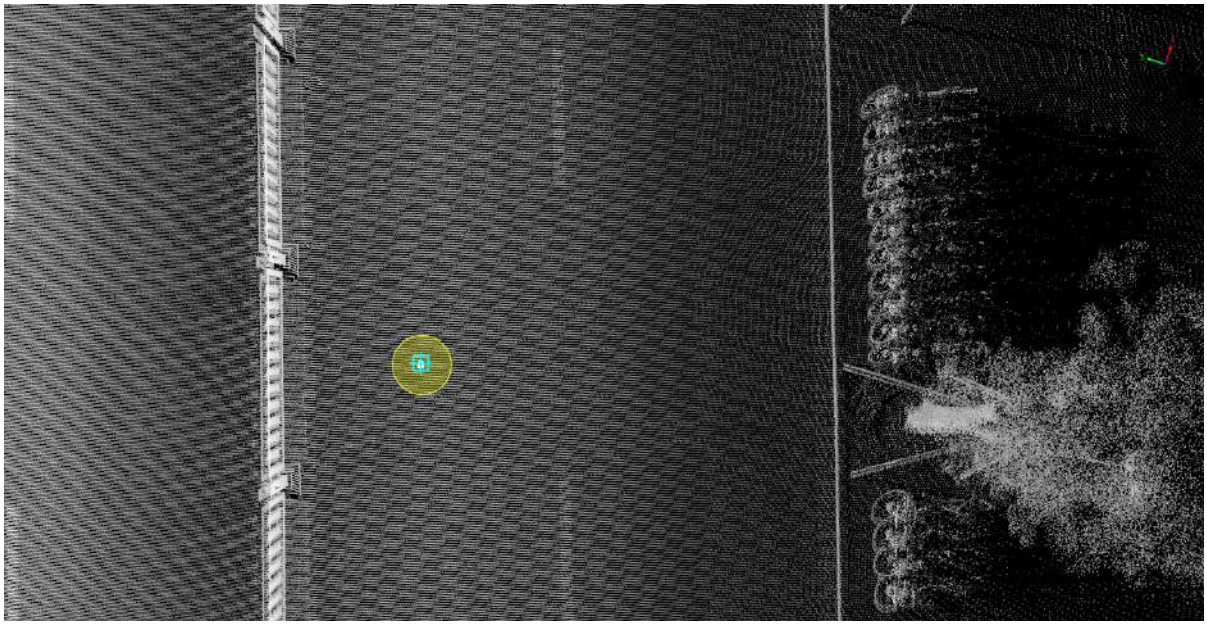
Function Description：This tool can be used to use two key points, manually click on the point cloud, and make the vectorized production of the reduce speed mark.

Steps

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

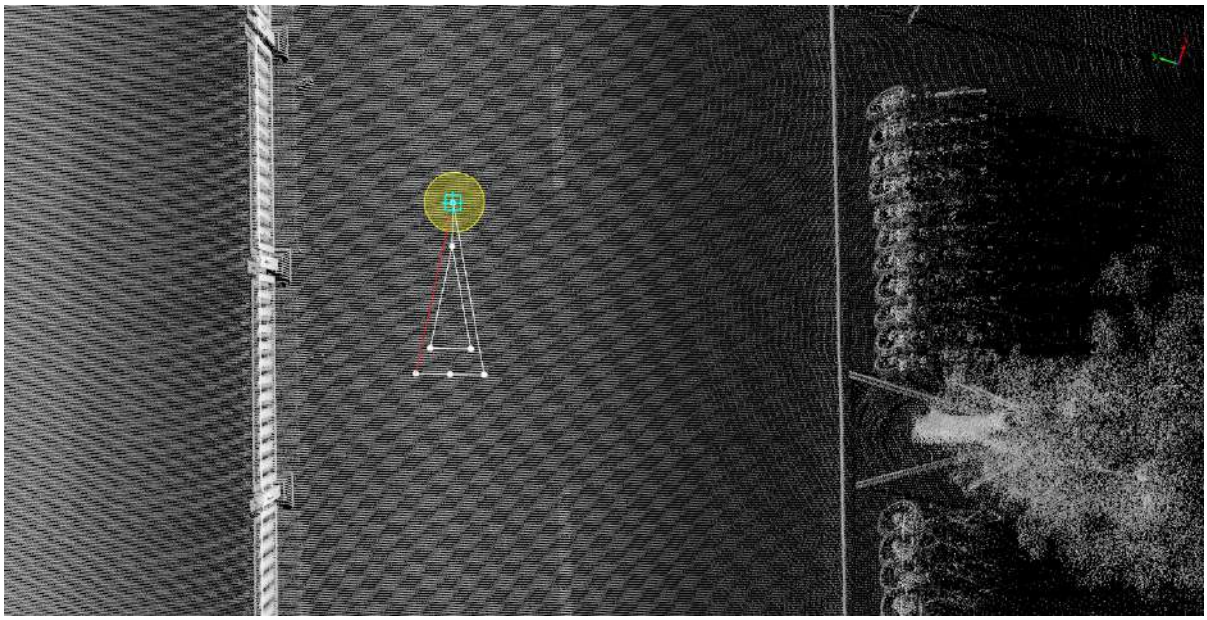
> - The key points in the arrow icons need to be aligned with the actual point cloud.
> - The red point is the first key point and the green point is the second key point.

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

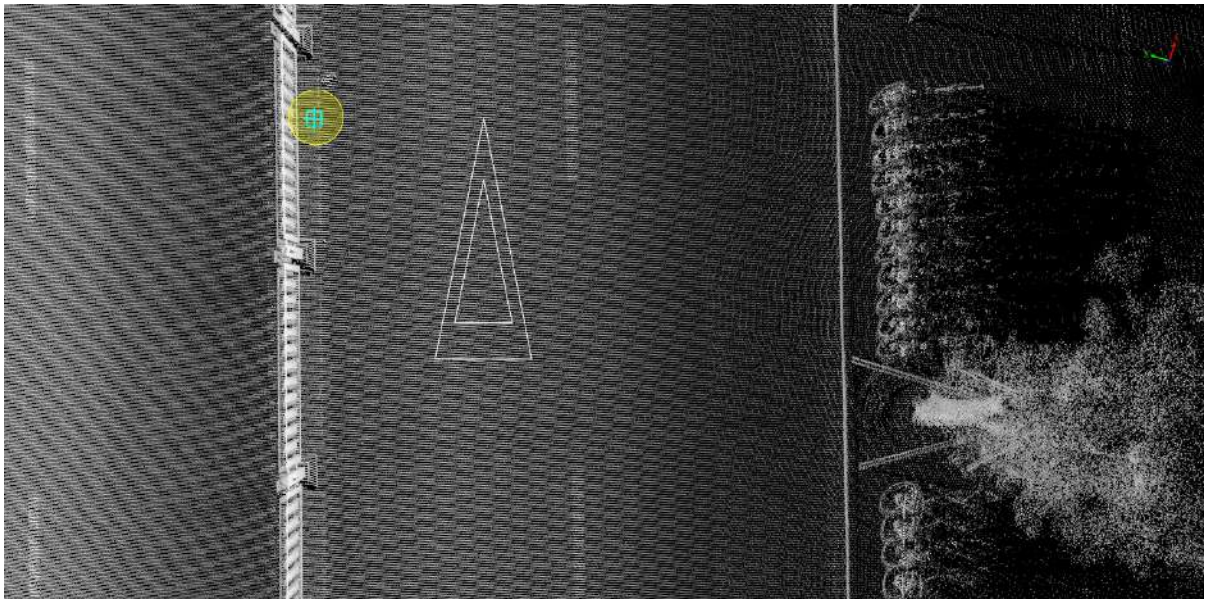


First key point

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



Second key point



Reduce speed mark drawing

Add CrossWalk Warning

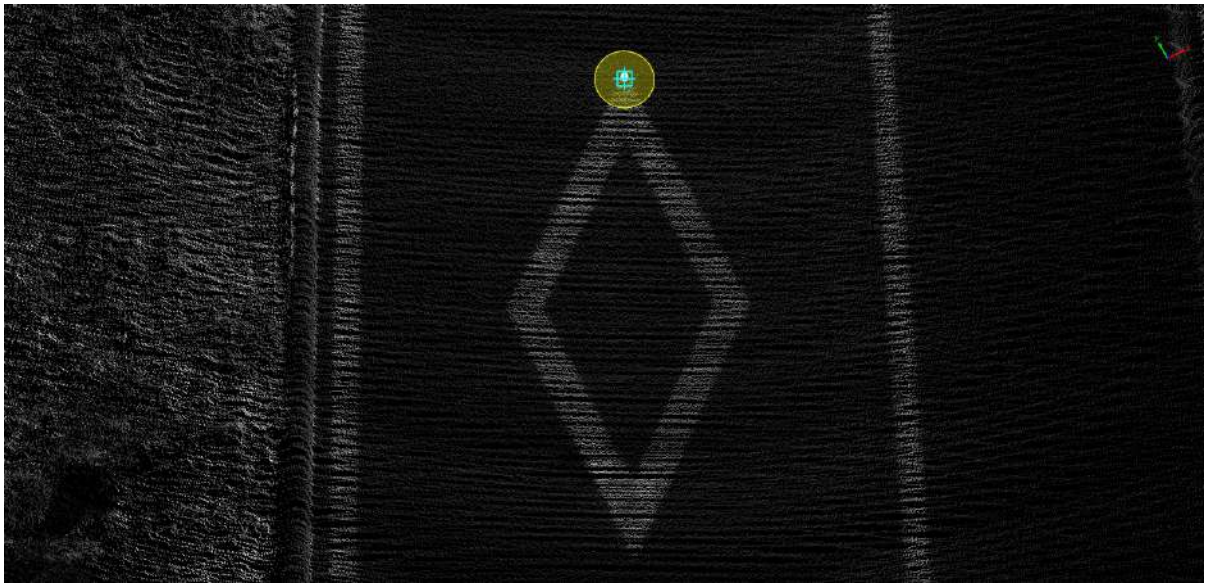
Function Description：The crosswalk warning sign is used to predict the crosswalk appearing ahead. This tool can be applied using two key points. Manually click on the point cloud to vectorize the crosswalk warning.

Steps

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

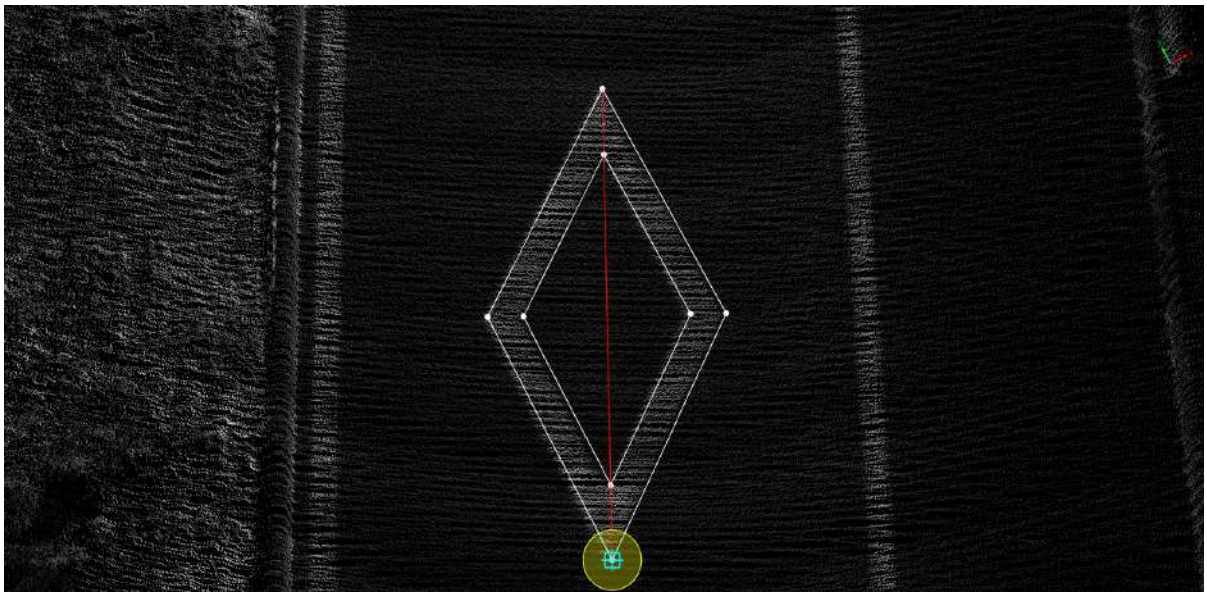
> - The key points in the arrow icons need to be aligned with the actual point cloud.
> - The red point is the first key point and the green point is the second key point.

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

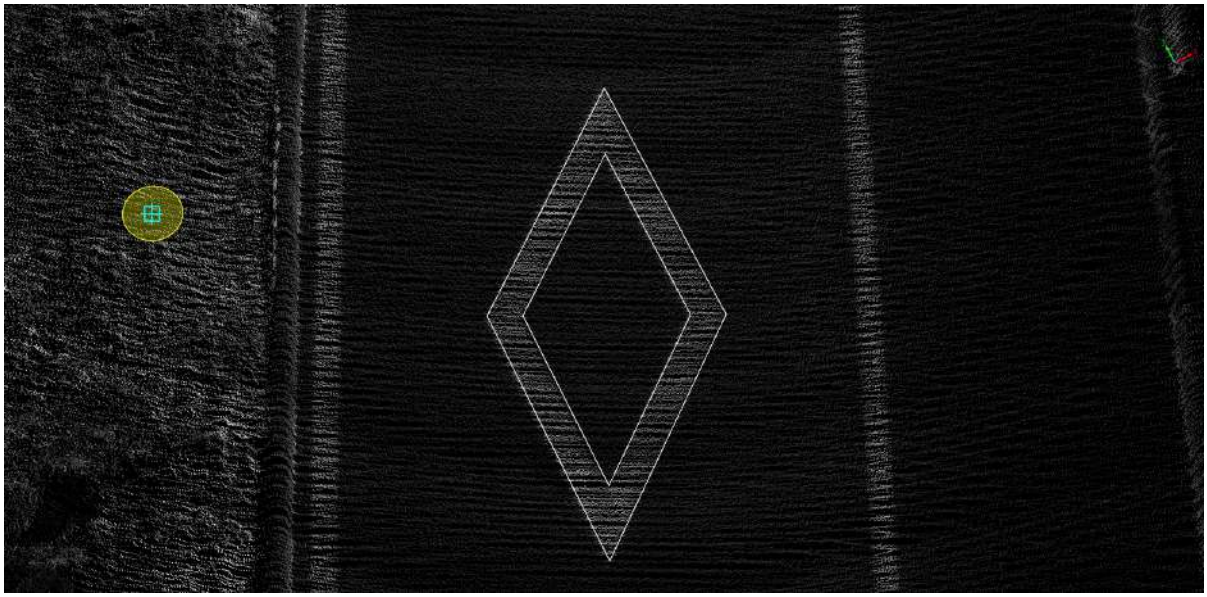


First key point

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



Second key point



Crosswalk warning drawing

Add No U-Turn

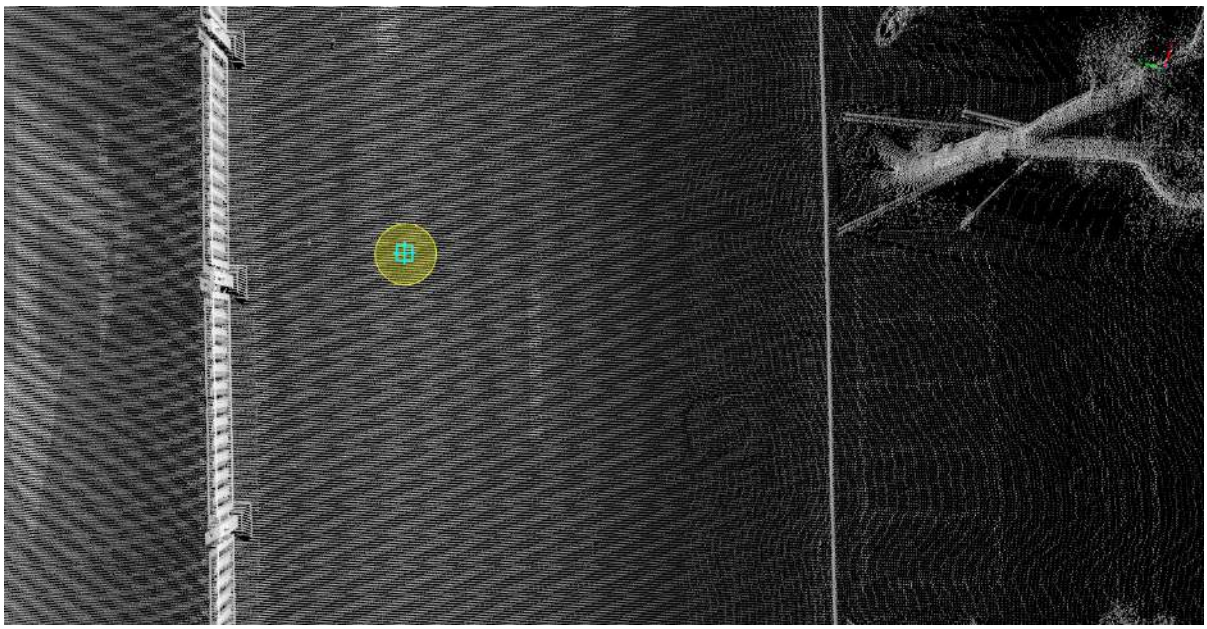
Function Description：The no U-turn marker indicates that the current lane does not allow U-Turns. This tool is used to manually click on the point cloud to create the vectorized production of the no U-turn marker.

Steps

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

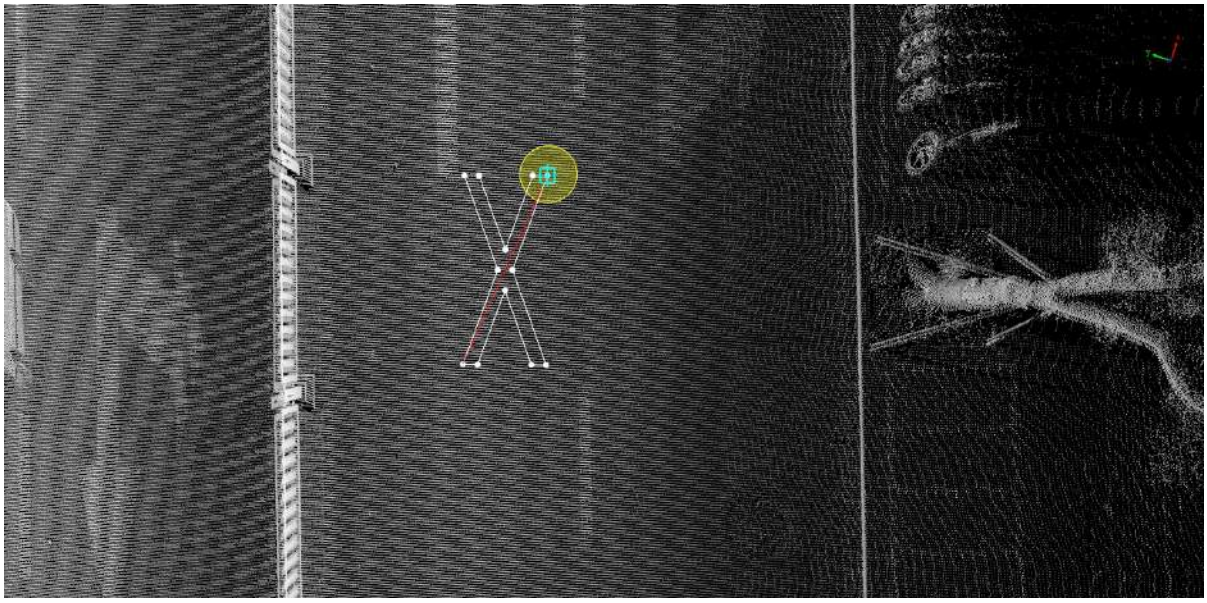
> - The **red points** in the arrow icons are key points, and the user needs to align the key points with the actual point cloud.
> - **Red point** position is **left mouse button click** position.

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

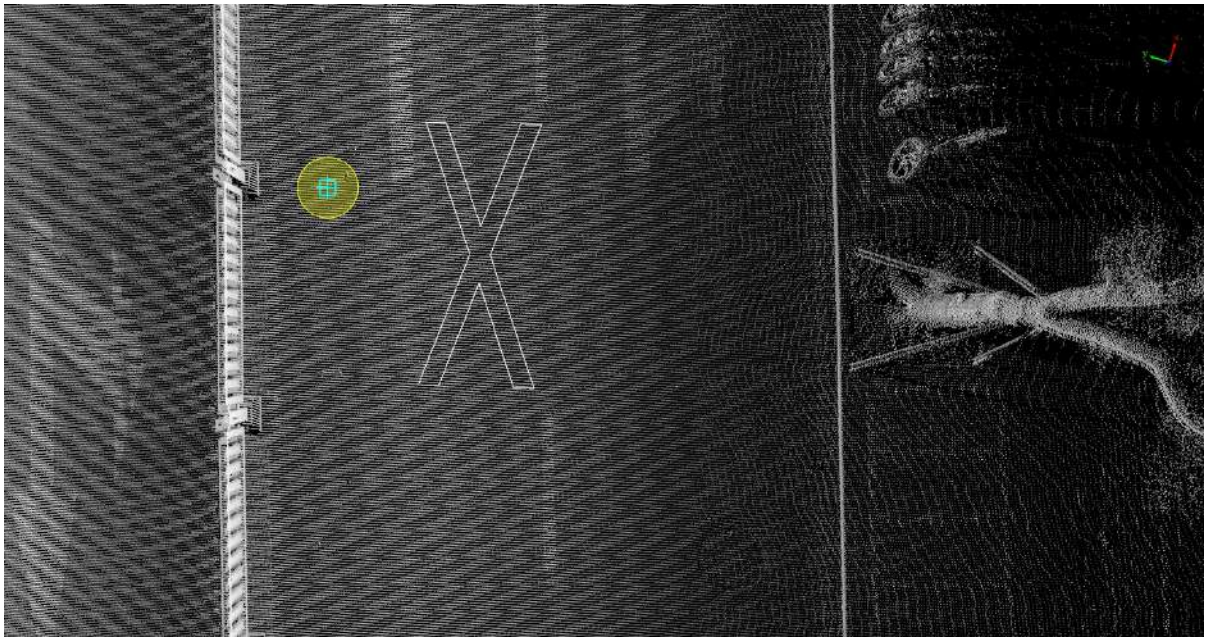


First key point

3.Move the mouse to see how the **template** fits into the **actual point cloud**, **left mouse click** to select the **second key point** to complete the production, details are as follows, **yellow point** is the **second key point** in the icon.



Second key point



No U-Turn marker drawing

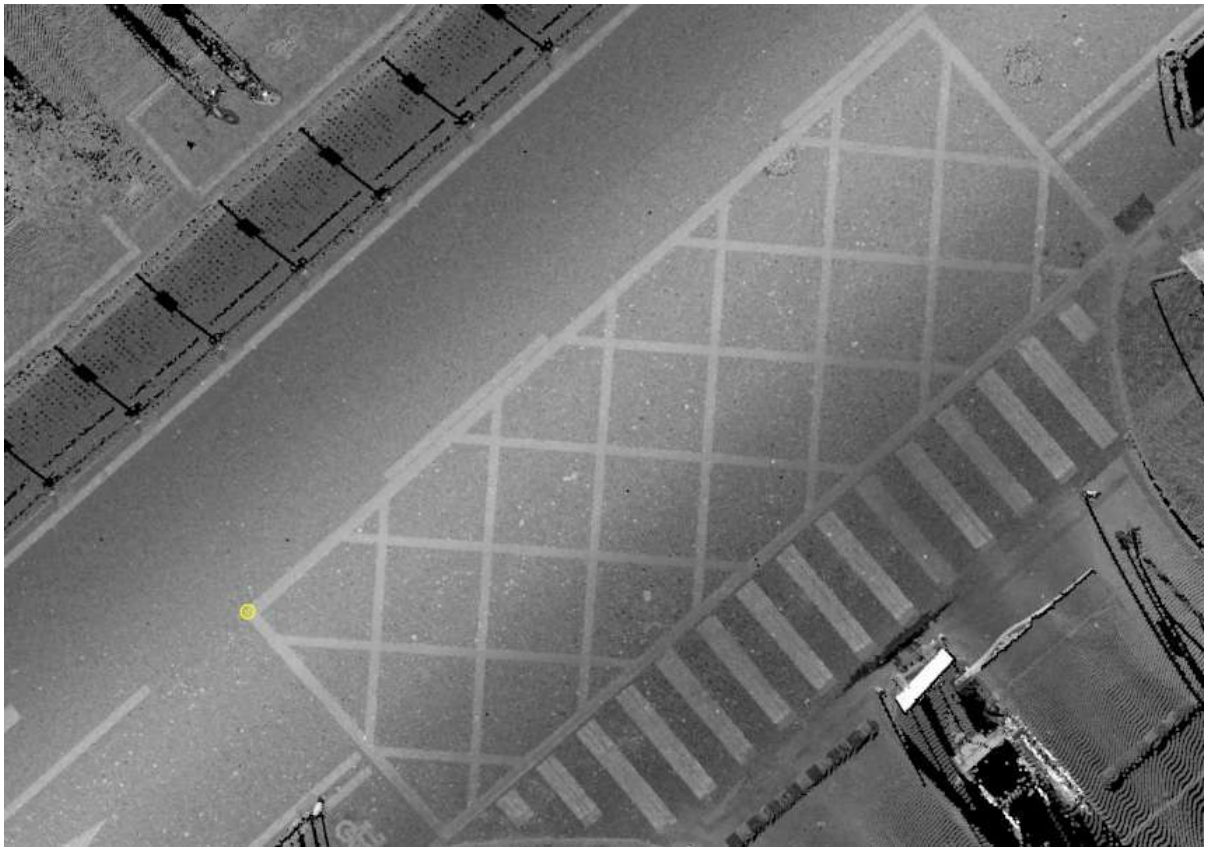
Add No-Parking

Function Description：This tool can be applied using three key points to determine the size of the outer rectangle, and internal auxiliary three key points to determine the interior. Manually click on the point cloud to vectorize the no-stop line.

Steps

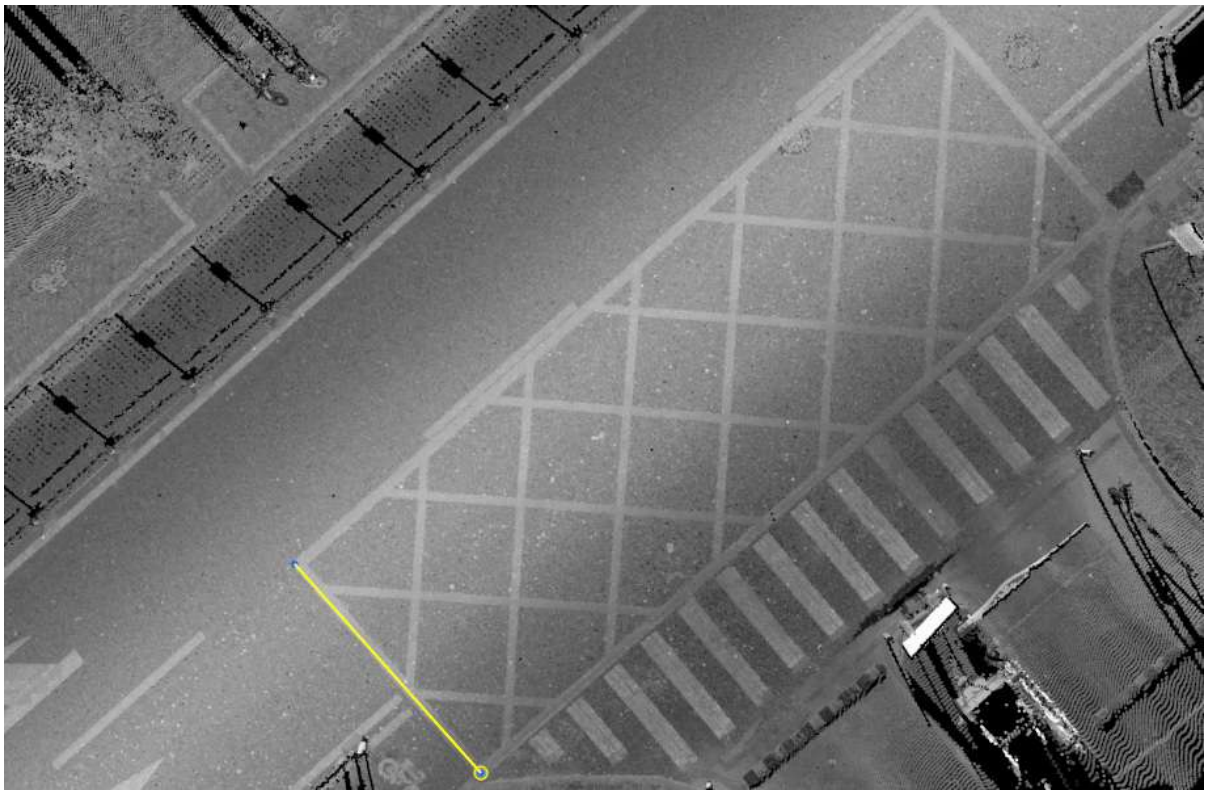
1.In the Road Surface panel, click the drop-down icon on the right, and select the **Add No-Parking** button.

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



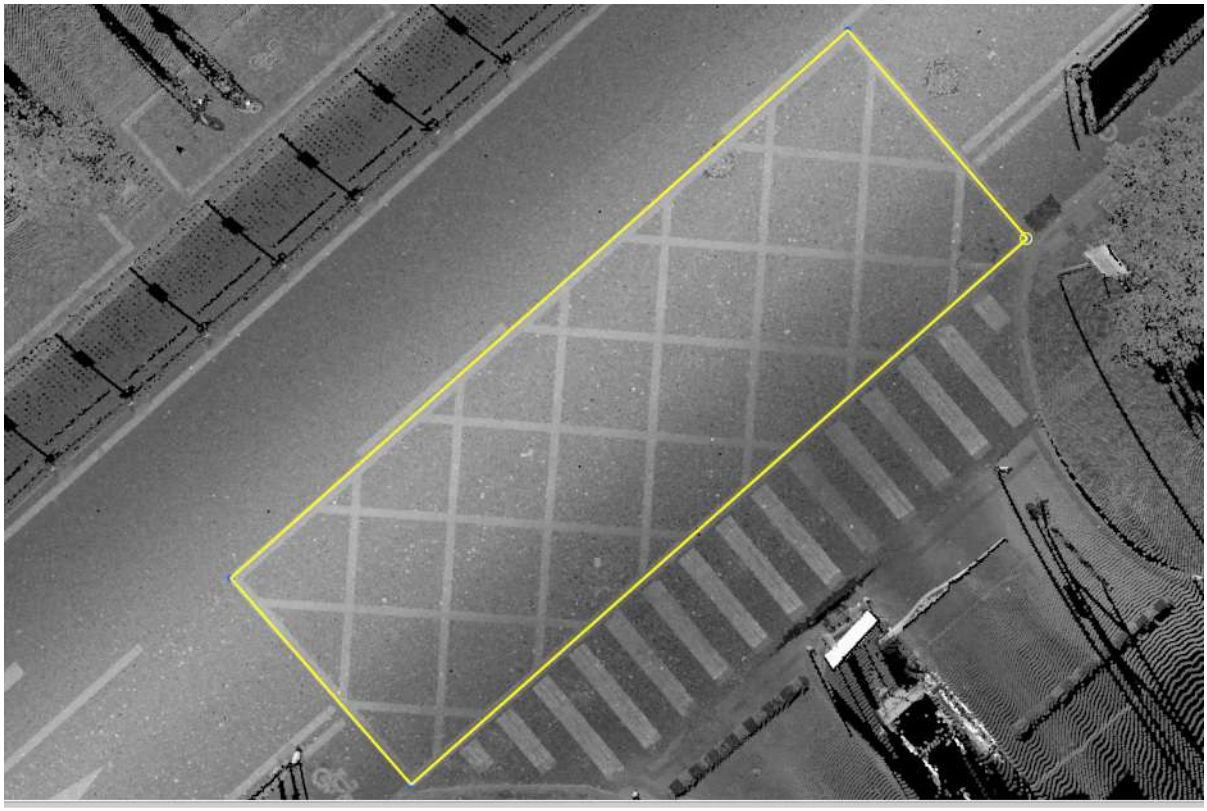
First key point

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



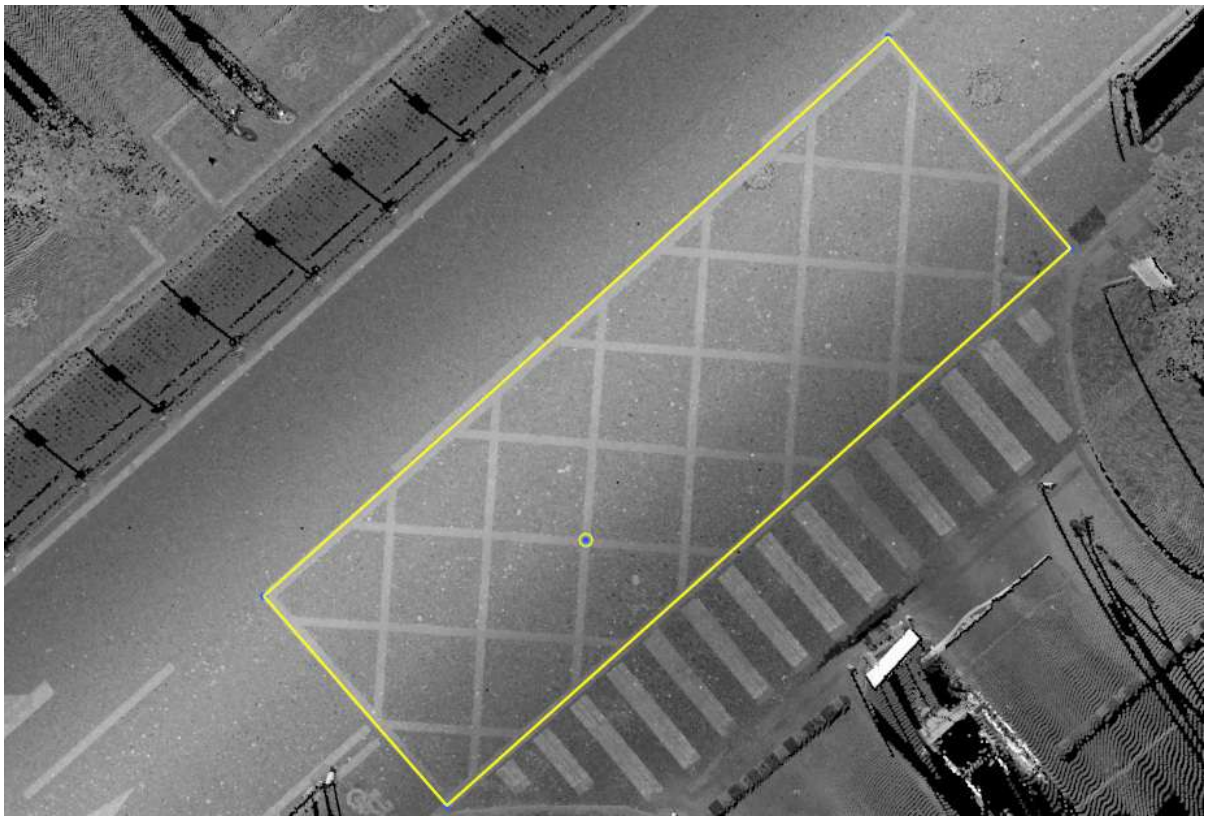
Second key point

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



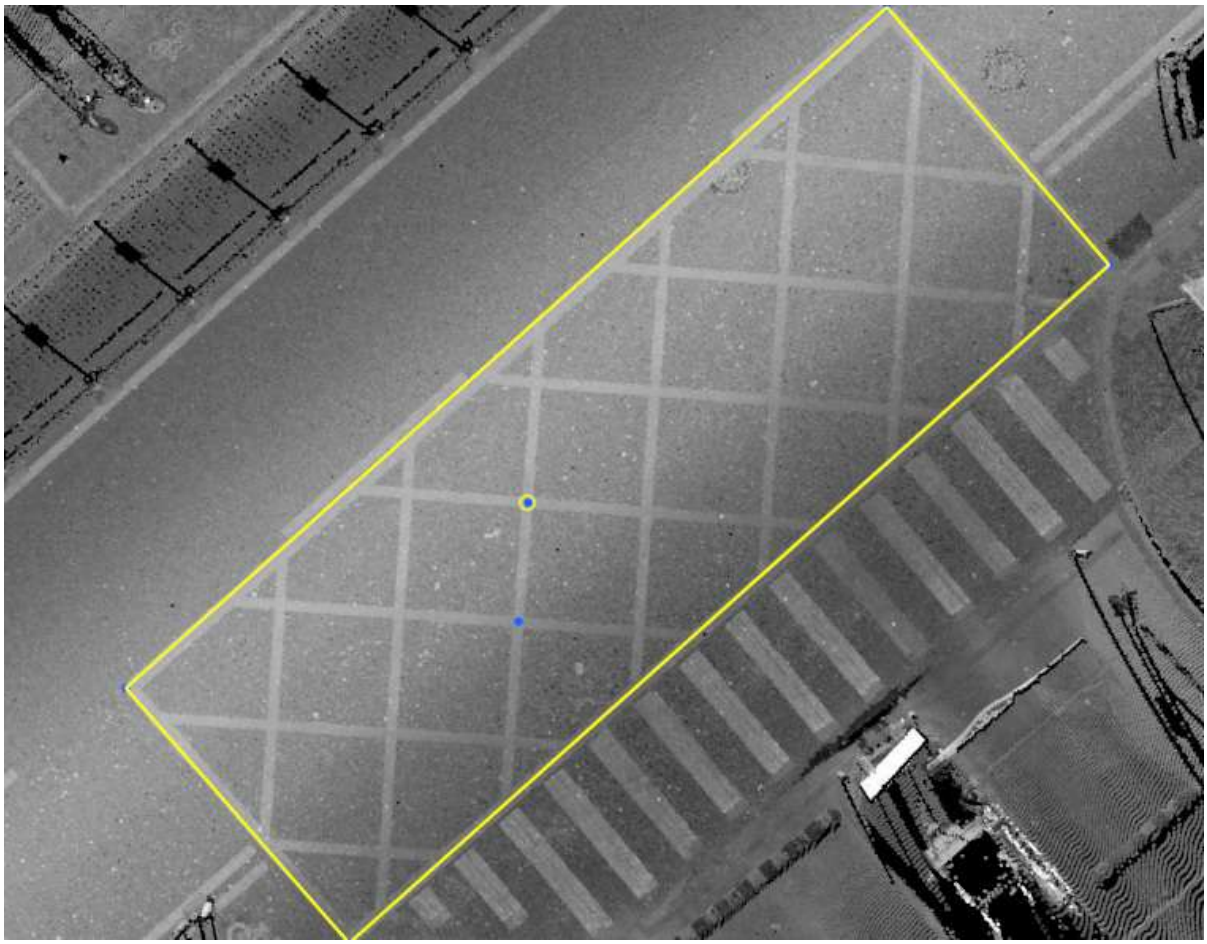
Third key point

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



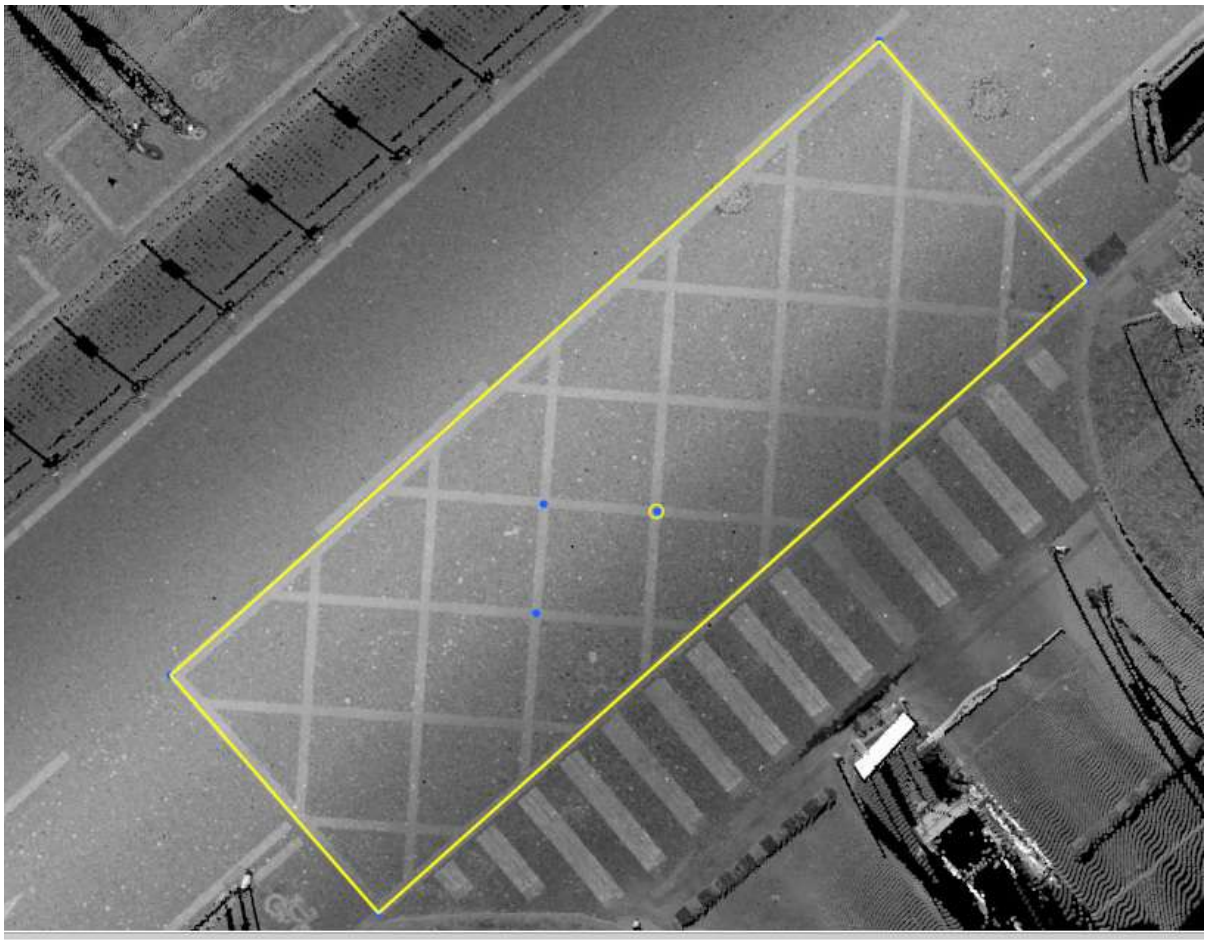
Fourth key point

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

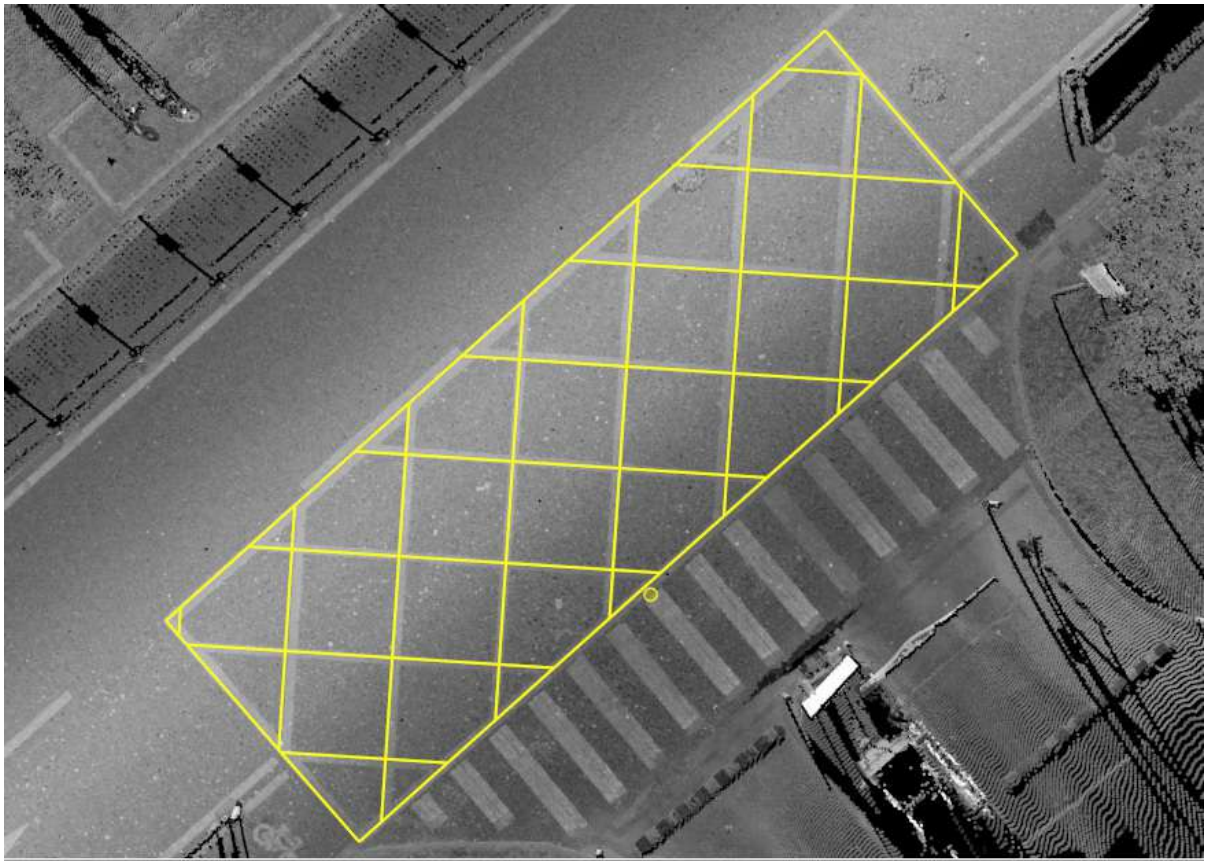


Fifth key point

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



Sixth key point



No-ParkingArea Drawing

Add BicycleLane

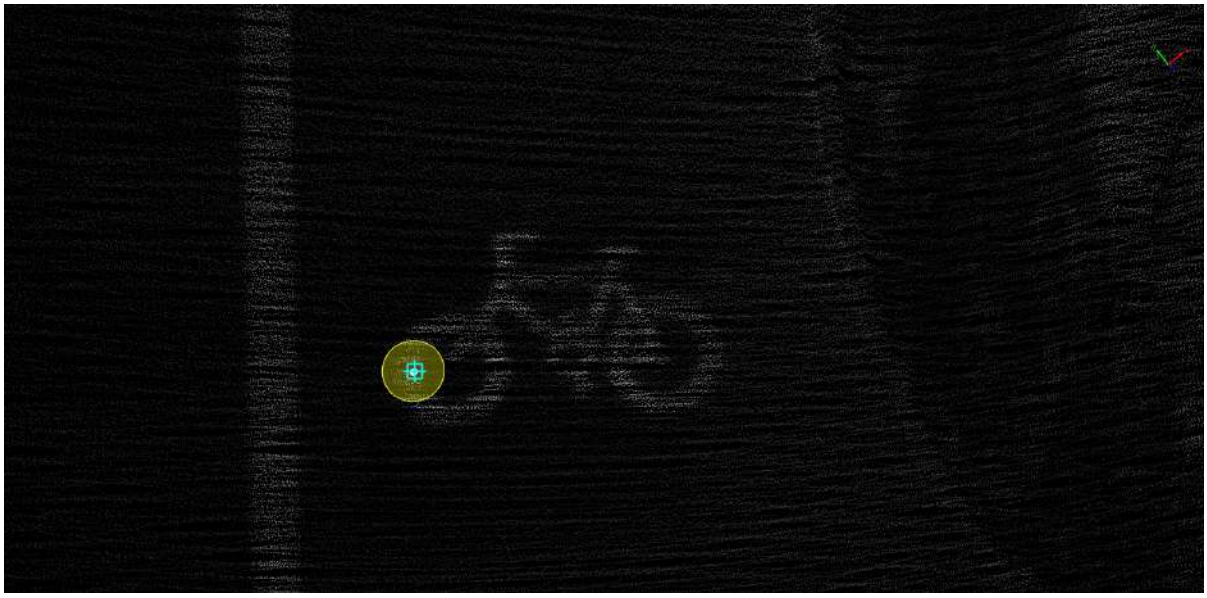
Function Description：The bike lane marker indicates that the current lane is a non-motor vehicle lane. This tool is used to manually click on the point cloud to create vectorized bike lane markers.

Steps

1.Click the **Add BicycleLane** button in the Road Surface panel.

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

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



First key point

3.Move the mouse to check the fit of the **template** and the **actual point cloud**, **left click the second key point** to complete the production, the details are as follows, **Yellow point** is the **second key point** in the icon.

Second key point

Bike lane marker drawing

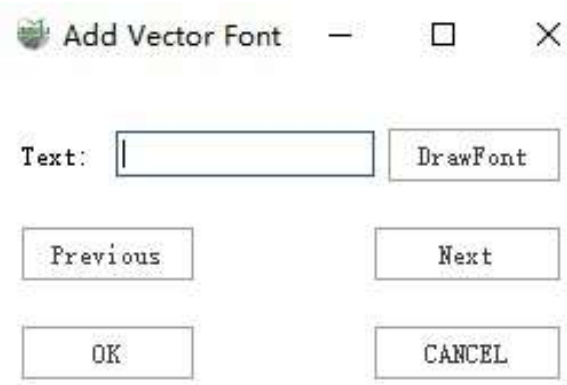
Add Text

Function Description: This function is used to create vectorized text for ground vector text and English characters.

> From the first character to the confirmation, it is one group of operations. The vector result of one group of operations is one object, such as "Bus Lane," which consists of four characters and is one group of operations, resulting in one object.

Steps

1.Click the **Add Text** button to open the Add Vector Text dialog box.



Dialog Box

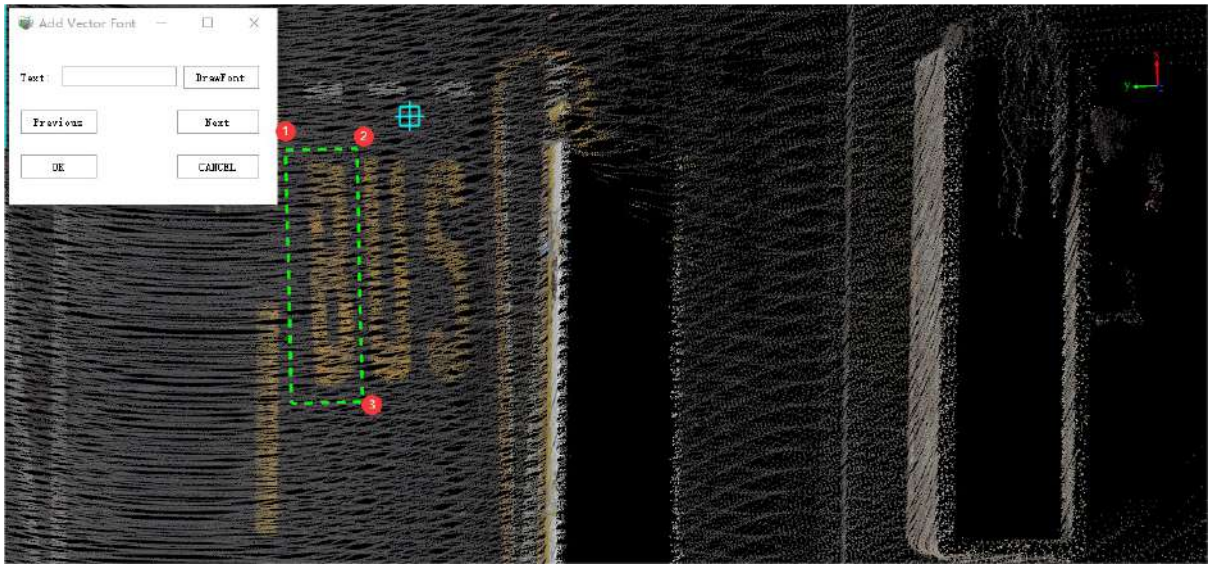
Parameter Description

- **Text**: The text to be entered, which can be a single letter, a single Chinese character, or a single digit.
- **Draw Text**: Display the entered text within the rectangular area of the point cloud.
- **Previous**: Use the previous vector text as the current processing unit for rotation, translation, scaling, and other operations.

  > From the first character to the confirmation, it is one group of operations. "Previous" and "Next" refer to the current operation.

- **Next**: Save the current vector result and start the next operation.
- **OK**: End the current group of operations and save the vector results.
- **Cancel**: End the current group of operations and discard the vector results.

2.**Left-click** three points to determine the position of the current text. The direction of the box represents the initial direction of the text.



Text Position

3.Enter the text in the dialog box, for example, "Bus," and click the **Draw Text** button to display the actual vector on the point cloud.

Actual Display of Vector

4.If you need to move a vector, simply place the mouse cursor over any dashed edge of the bounding rectangle. When the dashed edge turns green, it indicates selection. Then, drag the mouse to move the vector object to the desired location.

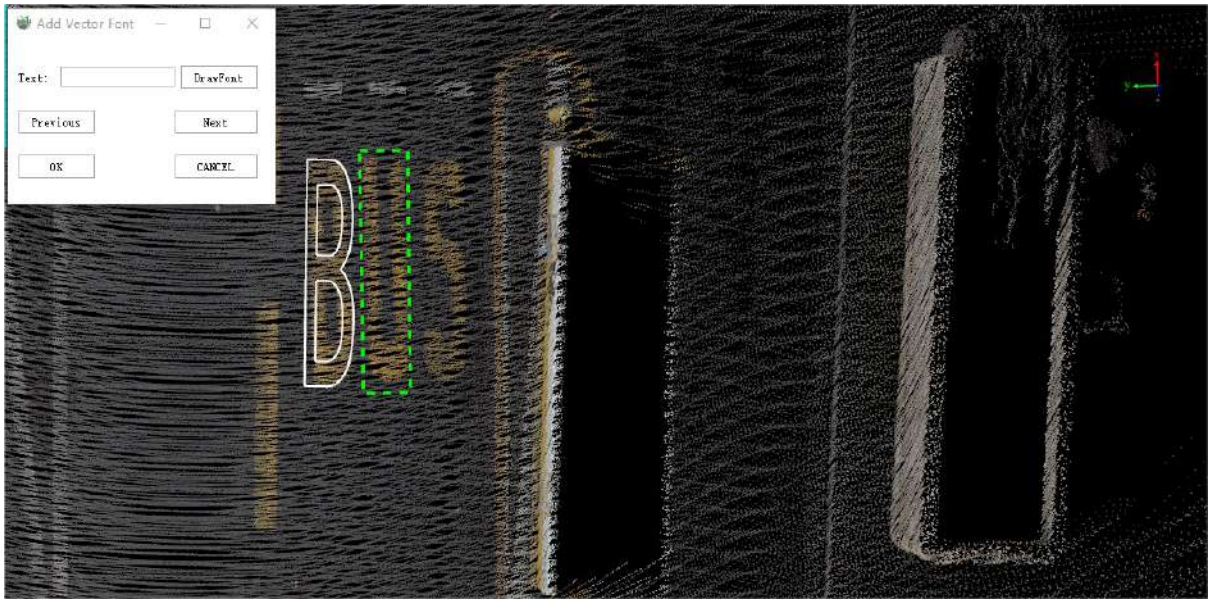
For uniform scaling or flipping operations on vector objects, position the mouse cursor over any of the four corner points of the bounding rectangle. When they turn green, it signifies selection. Next, drag the mouse to perform uniform scaling, and dragging it in the opposite direction can achieve flipping.

To perform non-uniform scaling on vector objects, place the mouse cursor over the key points on the four edges of the bounding rectangle, and then drag the mouse. The scaling will occur on the side corresponding to the selected edge.

If you wish to rotate a vector object, position the mouse cursor over the rotation point of the bounding rectangle and move the mouse cursor as needed.

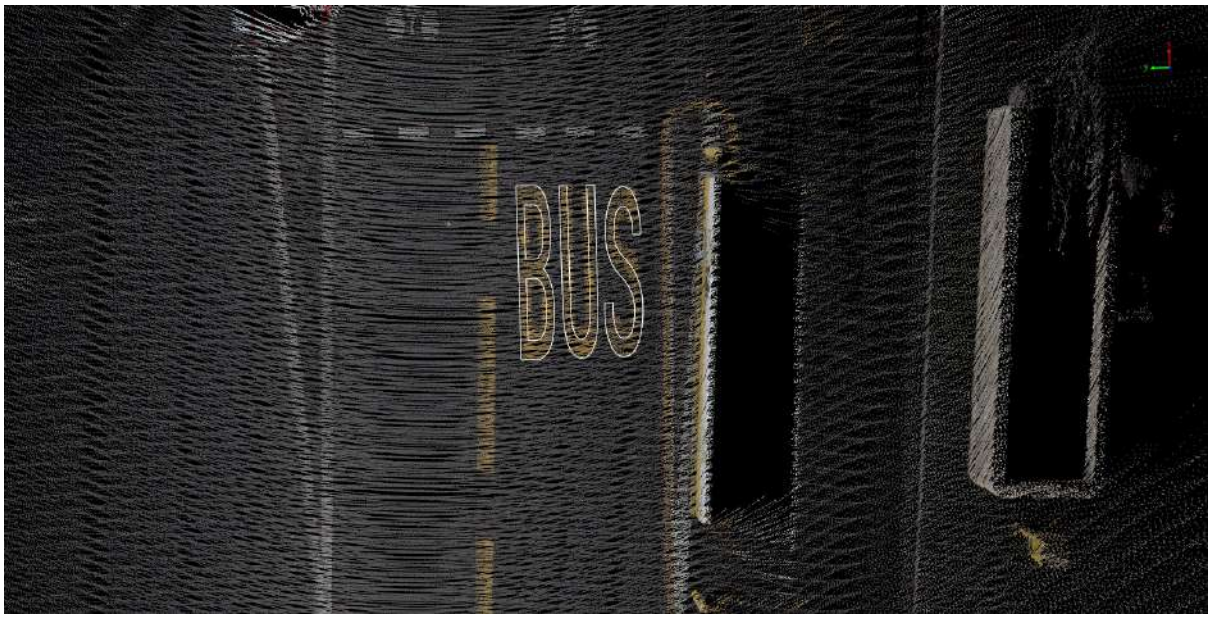
Adjustment

5.Click **Next** to proceed to the next text in the current object.



Create Next

6.Follow steps 2 to 4 to create text until the last character of the current object. Click **OK** to automatically save the current object.



Complete Creation

Template matching

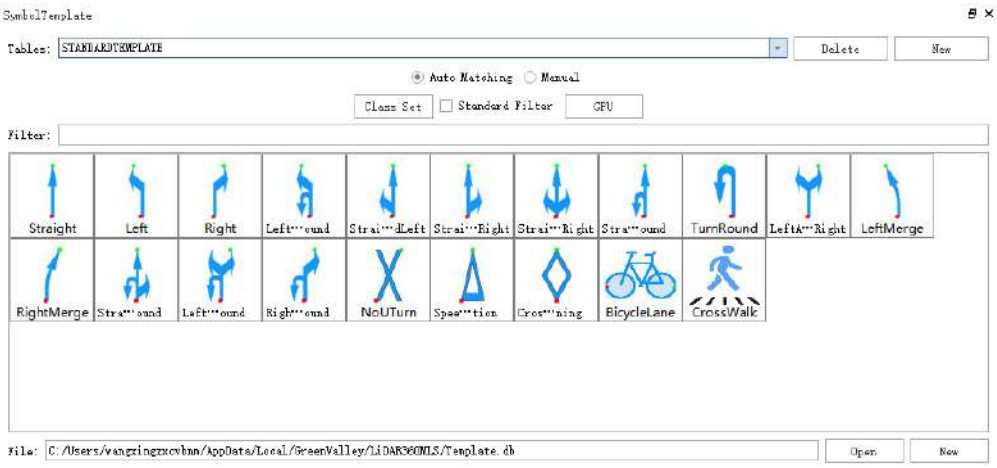
Function Description:This tool can be used to apply a symbol from the SymbolTemplate.
1.Automatic matching: 1.built-in arrow template of Chinese national standard size, as well as ground signs such as crosswalk warning, deceleration sign, and no U-turn of Chinese national standard size, three keys can be used click the box to select the point cloud to perform automatic template matching, and directly get the vectorized result obtained by fitting the point cloud.

2.Manual addition: For different forms of ground marks, manual drawing function can be performed, and the point cloud can be manually described and saved as a custom template. The custom template can be manually added or used for automatic matching.

Steps

1.Click **Template Matching** in the Road Surface panel button.

2.The **Template Matching** sidebar should pop on the right side of the software, as shown in the figure below, for button meanings see Detailed explanation of template matching sidebar parameters.



Sidebar preview

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

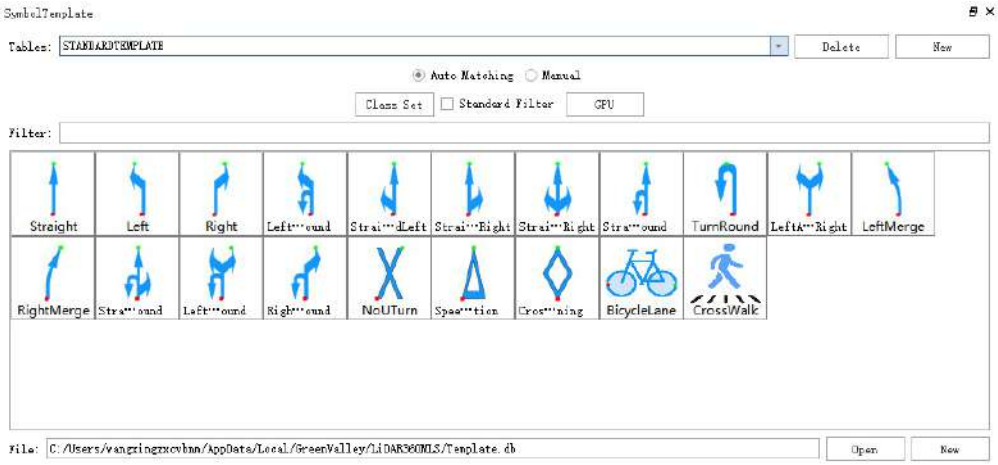
4.**Auto Matching**: Use the currently activated form and use the three-key point method to perform automatic template matching, see Template Automatic Matching.

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

Template Matching Para

Function Description: Describes the template matching sidebar parameters in detail.

Sidebar



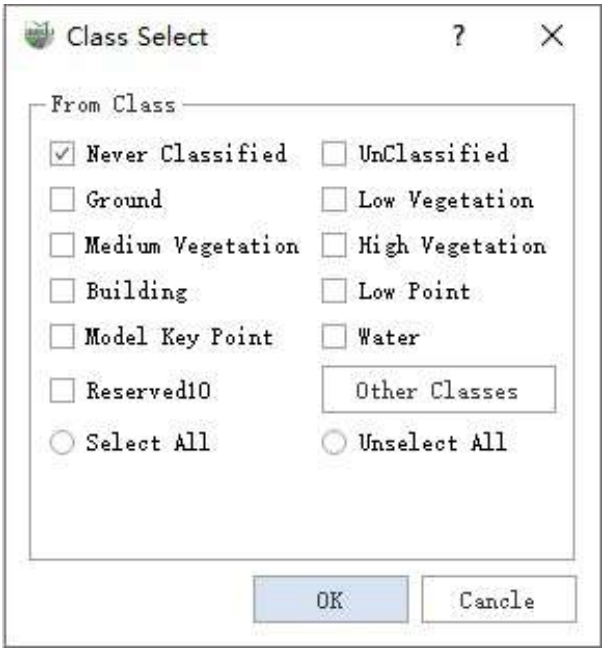
Sidebar

Parameter Description

- **Tables**: The SymbolTemplate module uses a database to store various template tables, as well as user-defined tables.
  - Custom tables: Users can add and delete new templates by themselves, see Template Matching Custom Tables.
  - The name of the built-in template table is STANDARDTEMPLATE, which cannot be deleted and does not support adding custom templates.

> Built-in tables cannot be deleted or modified.

- **Delete, New**: Used to create and delete custom tables, see Template Matching Custom Tables.
- **Filter**: Enter the template name, perform a quick search, and the preview window below will display real-time linkage.
- Auto Matching: Use the currently activated table and use the three-key point method to perform automatic template matching, see Template Automatic Matching.
- **Manual creation**: Use the currently activated table to make manual vector creation, see Template Manual Production.
- **Class Select**: Set the point cloud type used for detection, all are selected by default. If the point cloud data has been classified, only the category of the ground is selected, which can speed up the calculation efficiency and reduce the influence of other categories of noise.



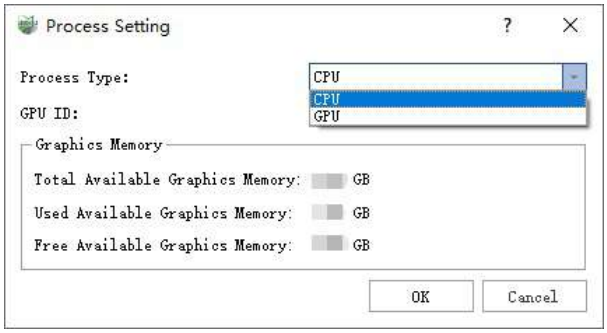
Class select

- **Standard Filter**: Unchecked by default, if checked, the recognition results will be filtered according to the Chinese national standard size, and the results that are too small or too large will be filtered out.

  > The length of ground signs on domestic urban roads is not less than 3m, and the length of ground signs on domestic expressways is not less than 6m. If standard filter is enabled, results with a length of less than 3m will be filtered out.

- **Mode: CPU, Mode: GPU**: Set the operation mode, users can choose CPU or GPU settings, running in GPU mode can improve the operation efficiency to a certain extent.

  > GPU mode requires computer hardware and software environment support, if the conditions are not met, even if checked, it will still run in CPU mode.



Processing settings

> The three functions of class select, standard filter, and CPU/GPU are only available when **the current table is the built-in standard table of the software and it is in automatic matching mode**.

- **File**: The currently opened database file. For details, see TemplateMatchingDataBase.
- **Open, New**: Open other database files and create new database files. For details, see TemplateMatchingDataBase.
- **Strength adjustment**: It is used to manually select the maximum and minimum values to stretch the strength in places where the strength is not easy to distinguish. Available only when the current table is a custom table.

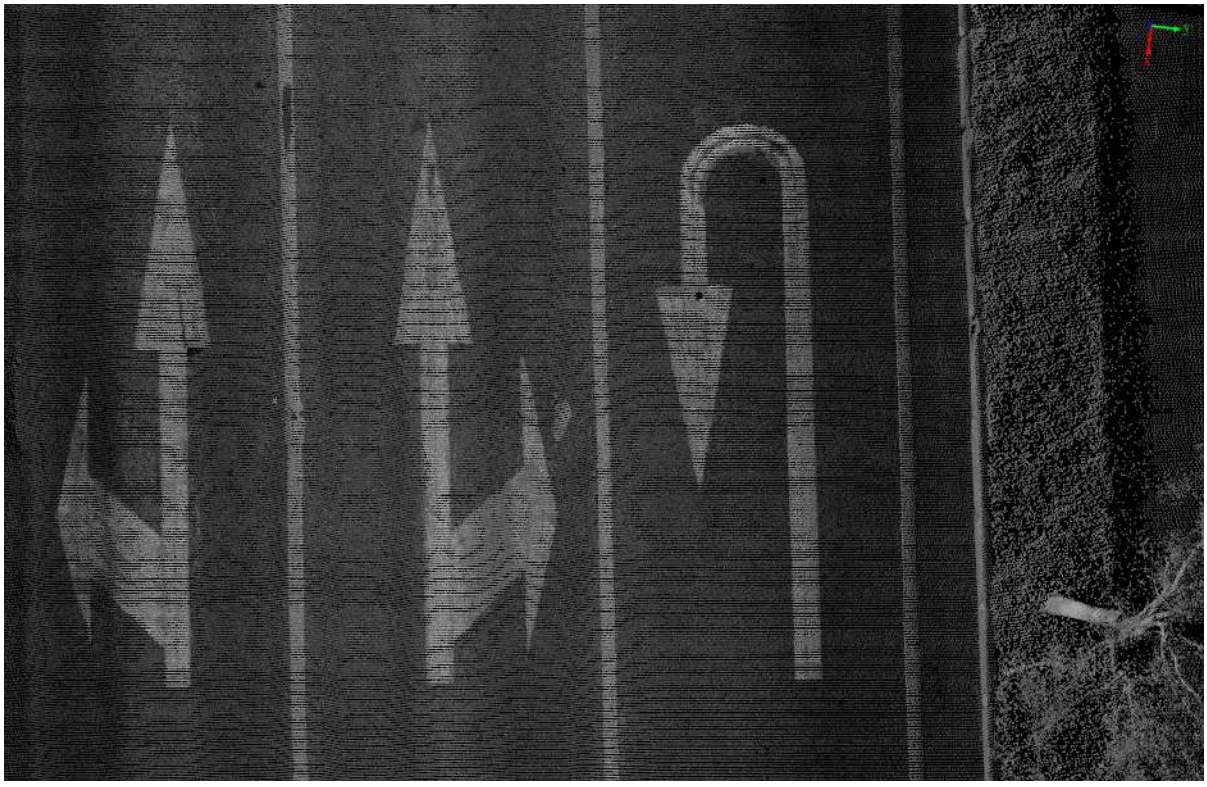
Dialog

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

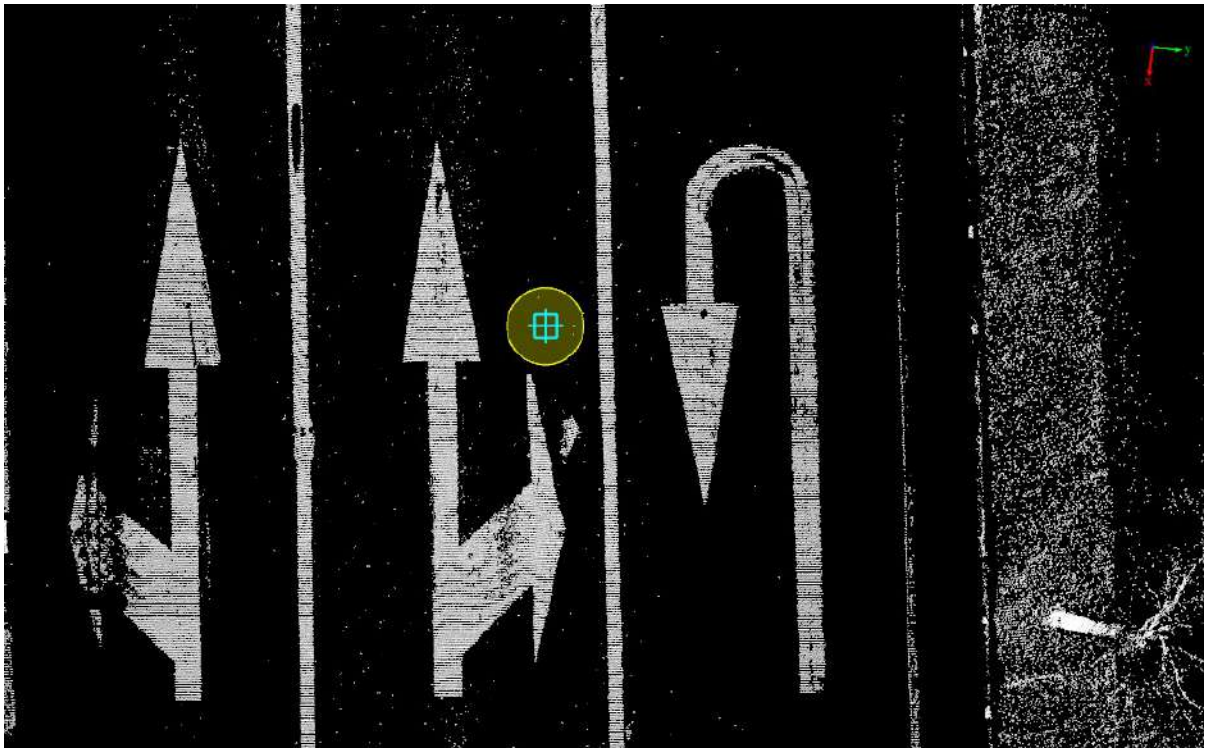
  > Scroll bar, value box, click box, the three are linked together, the user can choose one of the operation methods to adjust the strength, and the point cloud display will also be linked in real time (stretch the strength of the overall point cloud).

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

Intensity adjustment effect



Original point cloud screenshot

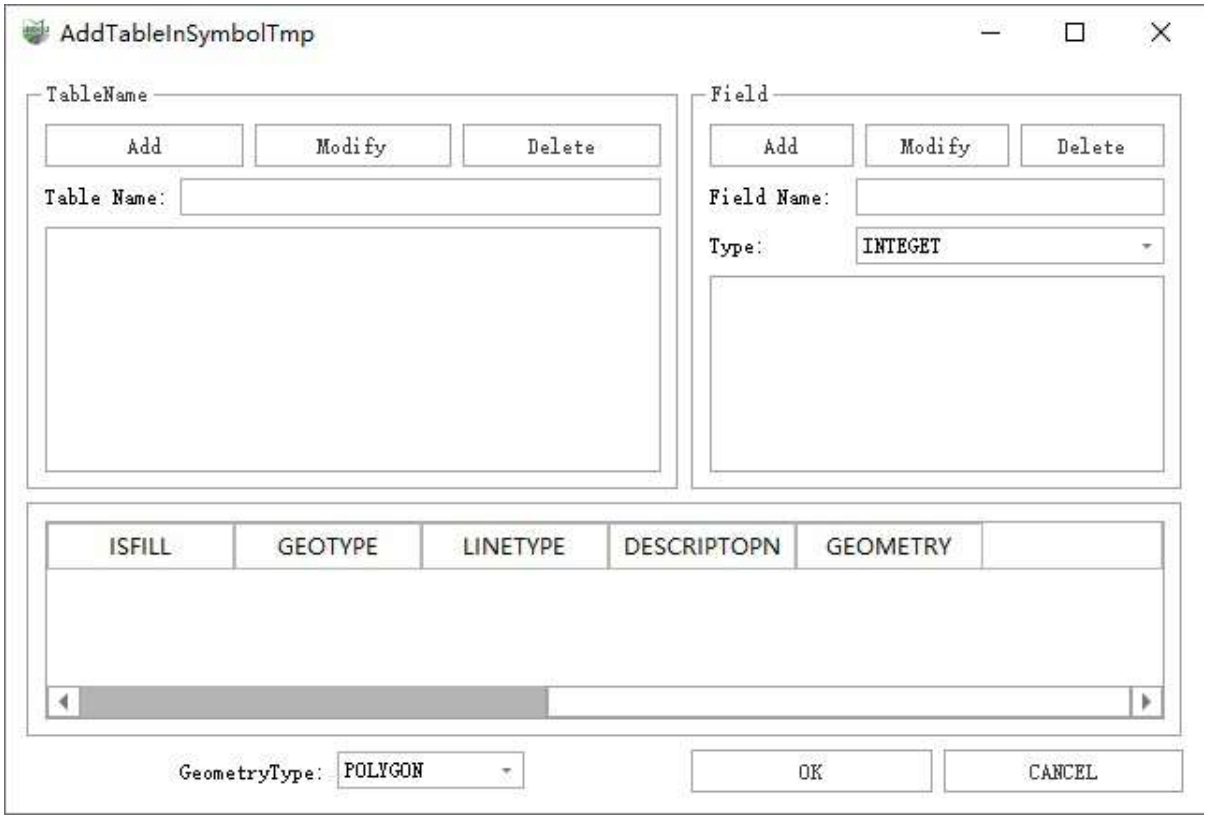


The minimum value is adjusted to 14, the maximum value is adjusted to 18, and the box is not selected to hide

Template Matching Custom Tables

Function Description: This tool can be used to create a custom table, create a custom template.

Dialog



Dialog

Parameter Description

- **Table Name**
  - **Table Name**: Enter the table name.
  - **Add**: Confirm to add table.
  - **Modify**: Modify the added table name and field.
  - **Delete**: Delete the selected added table.
- **Field**: Add and delete fields to the selected table.

  > Each new table has built-in fields, which cannot be deleted or changed.

  - **Field Name**: the newly added field name.
  - **Type**: field type.
    - REAL: decimal.
    - INTEGET: integer.
    - TEXT: text.
  - **Add**: Confirm to add field.
  - **Modify**: Modify the added field name and type.
  - **Delete**: Delete the selected added field.

- **Table Preview**: Real-time preview of fields in the table.
- **GeometryType**: The geometry type stored in the current table.
  - POINT: Point (not commonly used).
  - LINE: Line (not commonly used).
  - POLYGON: Single contour (commonly used).
  - MULTIPOLYGON: Multi-contour, Chinese, English and other texts or other ground signs with multiple contours, use this type (commonly used).
  - TEXT: Text type (only for backup).

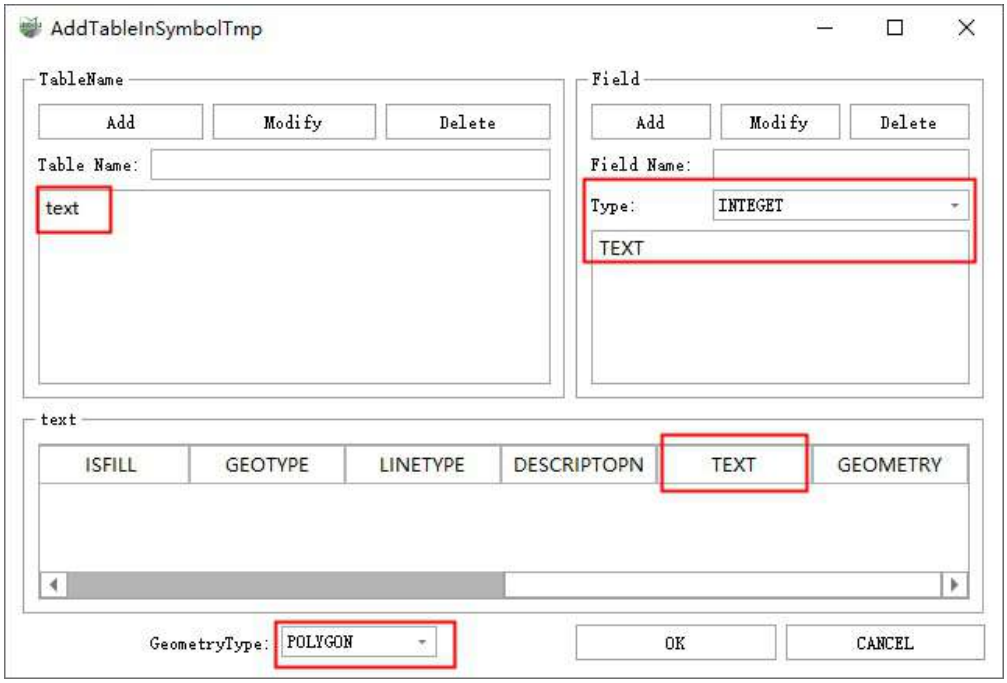
Steps

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

> Built-in table, cannot be deleted.

2.Create a new table:

- Enter the table name (required) and click the Add button.
- Add a new field (optional), enter the field name, select the type, click the Add button.
- Set the table type (required).
- Click the OK button.



New table

Add results

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

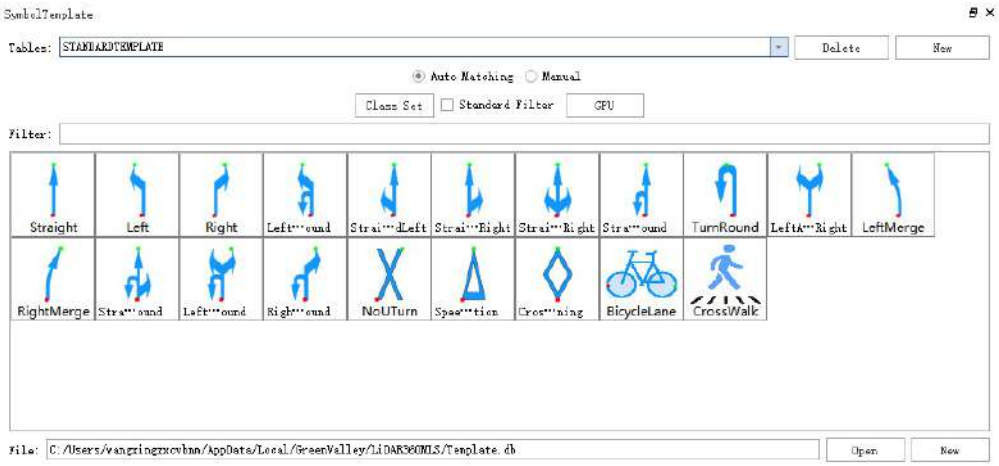
> Built-in tables, cannot add new templates.

Template Automatic Matching

Function Description: This tool can be used to select the target point cloud in a box, and automatically detect and vectorize the ground marks displayed in the template file.

> The built-in templates cannot be added, deleted, or modified. The built-in templates can be selected at one time. Multiple templates can be detected at the same time, and multiple results can be generated at the same time, such as generating arrows and deceleration signs at the same time.
>
> Custom template file, can be added, deleted and modified, custom template **only supports** one frame selection, a single template is detected, **only** generates a single result.

Parameters dialog



Parameter Description:

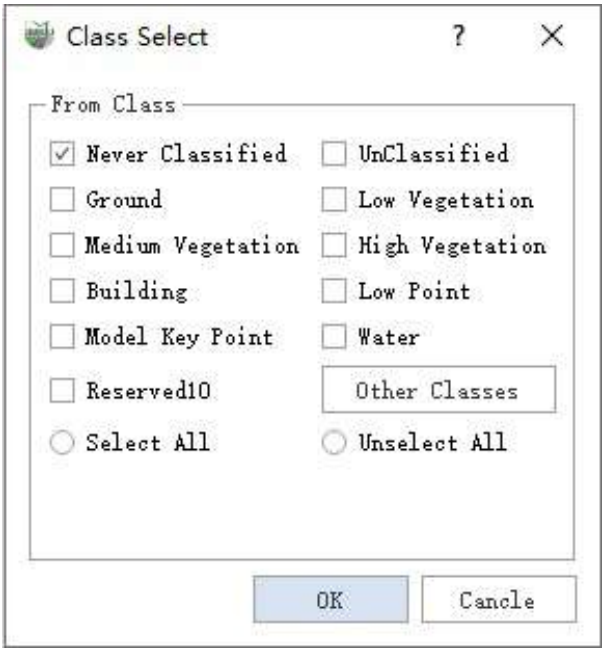
- **Tables**: Used to select the template table to be used, the main table below displays the content of the currently selected template table in the form of graphics + name.

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

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

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

  - **Other Classes**: Used to display and select more classes: used to display and select more classes.
  - **Select All**: Check all classes.
  - **Uncheck All**: Uncheck all classes.
  - **OK**: Complete the classes settings.
  - **Cancel**: Cancel classes settings.



- **Standard Filter**: Appears if and only if the currently used template table is the built-in template table and the mode is automatic matching, simple filtering is performed using the size of the ground signs of urban roads and expressways in China, and the filter is too large or too small result.

> If the user is not familiar with the national standard size of ground signs, or the size of the current road ground signs is not standard, or the current road ground signs have various shapes, **not recommended** to check this option.

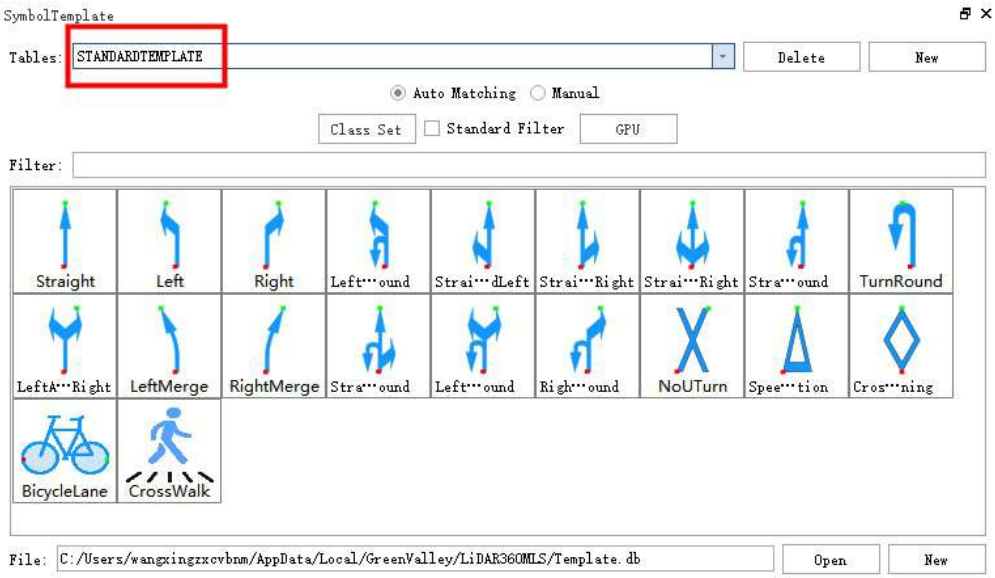
- **Mode: CPU, Mode: GPU**: Appears if and only if the currently used template table is the built-in template table and the mode is auto match, and is used to select the operation mode of the function.

> The default is **CPU** mode. When the computer has a **GPU** environment, the user can check this option, which can slightly improve the efficiency of a single detection.

Built-in Standard Template Automatic Matching Steps

The built-in standard template can be selected at one time to detect multiple targets.

1.Select the built-in standard template table.

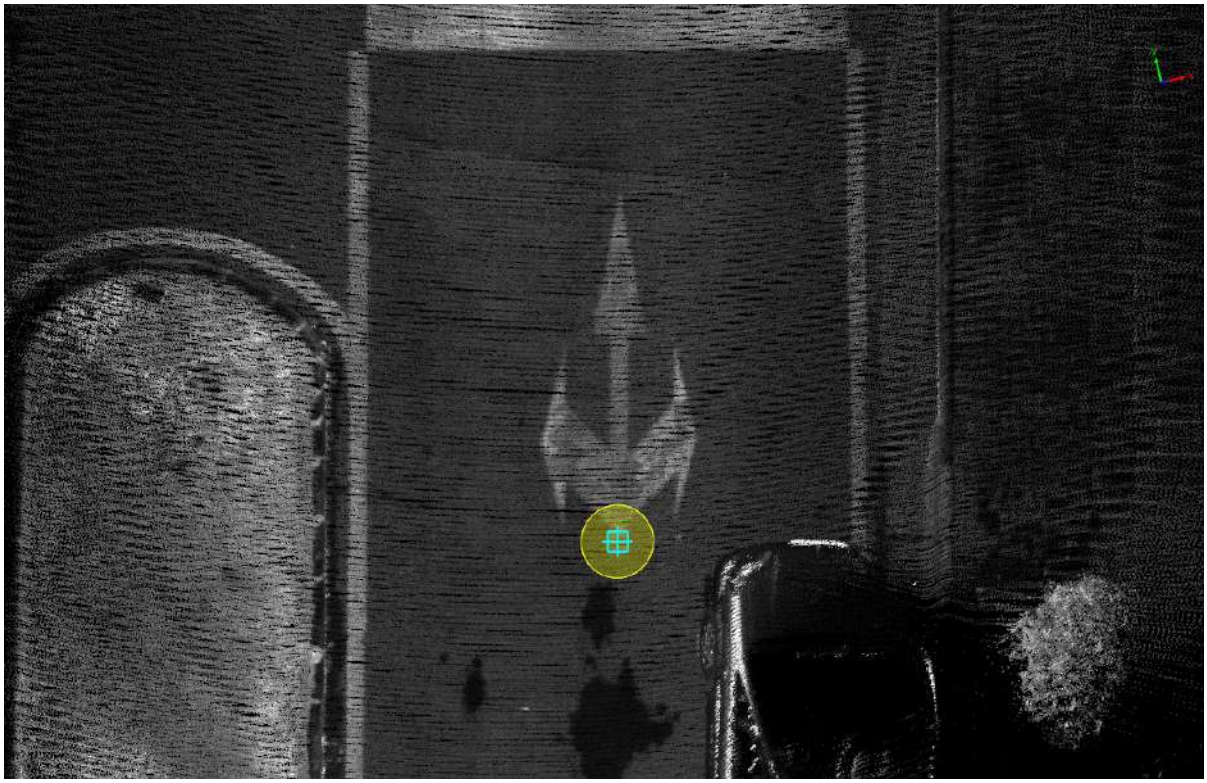


Select table

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

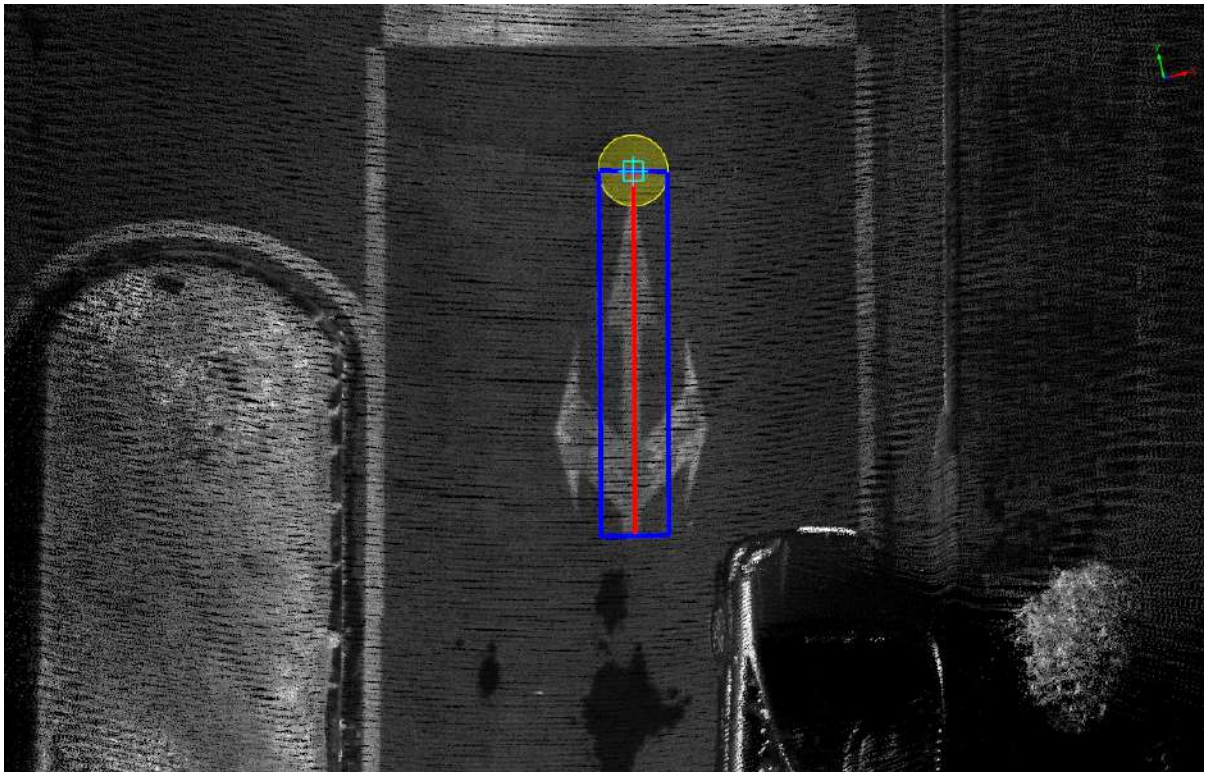
(1)If the point cloud data has classification information, it is strongly recommended that in the class selection, set the input class to the class of the ground point.

(2) If the point cloud data has no classification information, or use all classes as input classes, must select the first key point and the second key point on the ground, and the third key point can be arbitrarily selected according to the size of the pull box.



Key point drawing

3.Left mouse button, click the second key point.

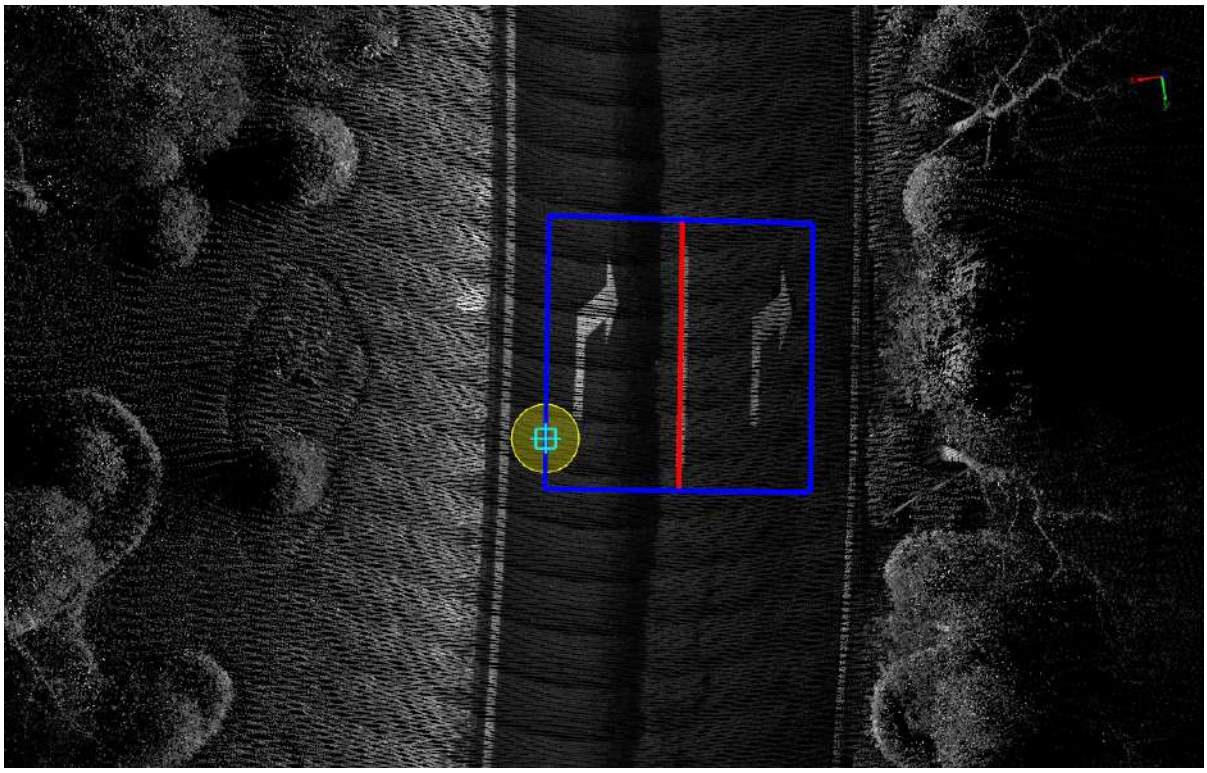


Key point drawing

4.Left mouse button, click the third key point, and then automatically start detection.

> In order to ensure the detection efficiency and detection accuracy, the size of the pull box is **recommended** slightly larger than the ground mark.

Recommended size of standard template table pull box

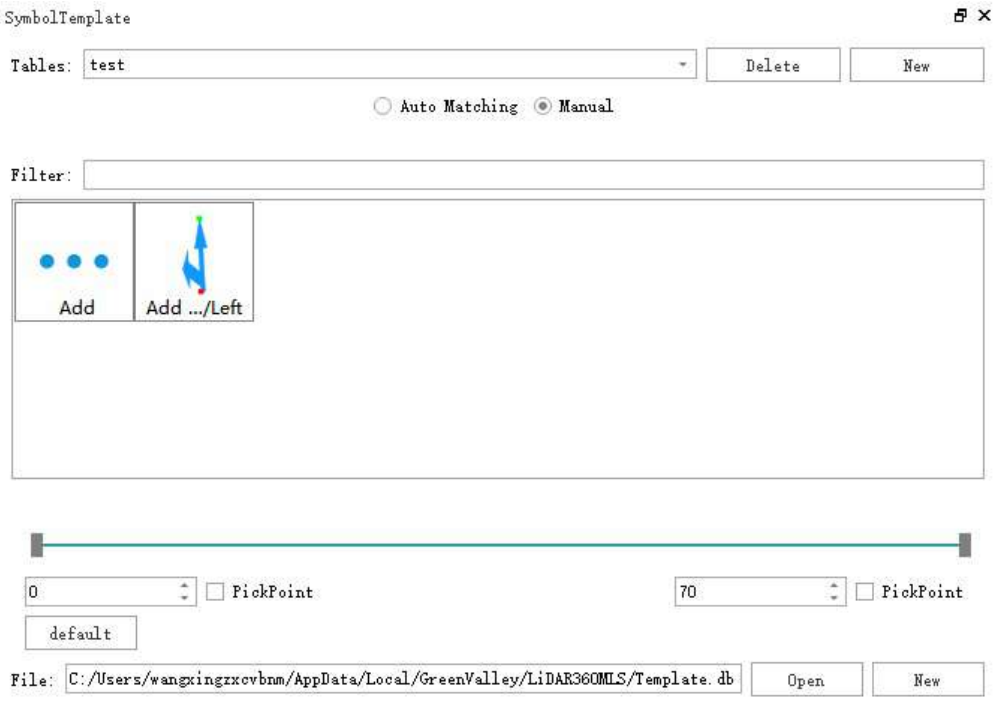


Detection results

Custom Template Automatic Matching Steps

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

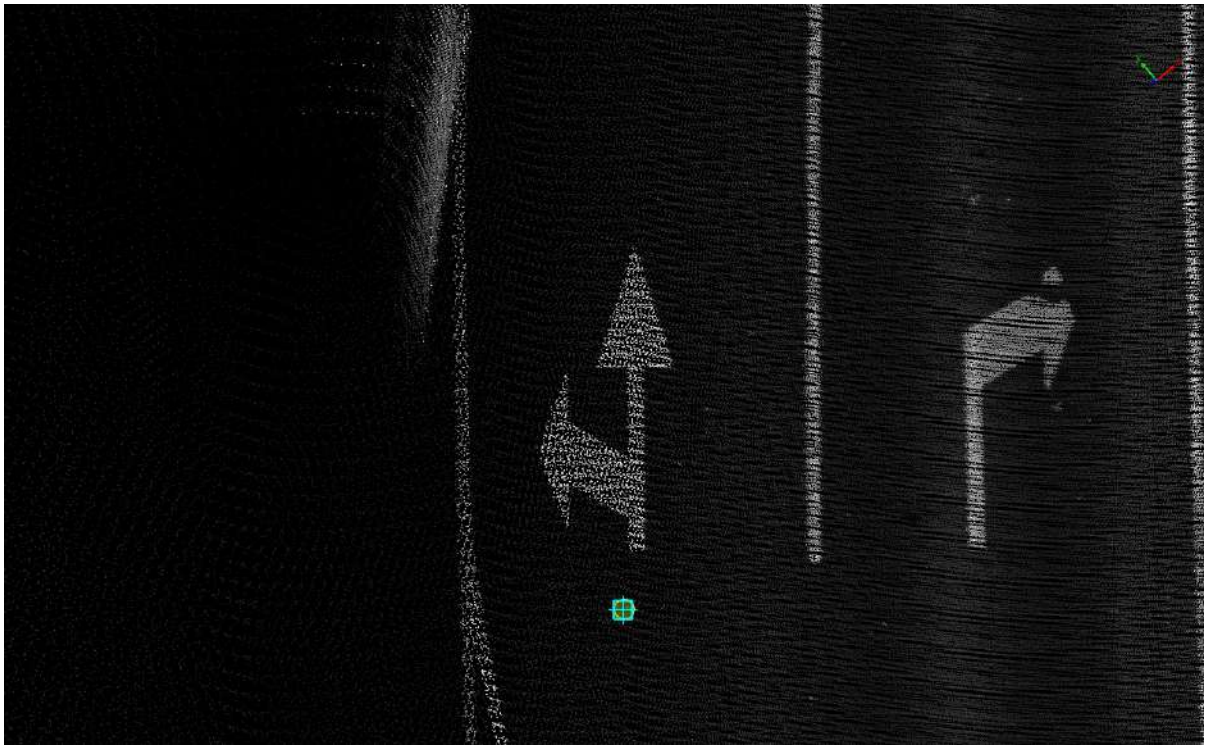
1.Select custom template table.



Select table

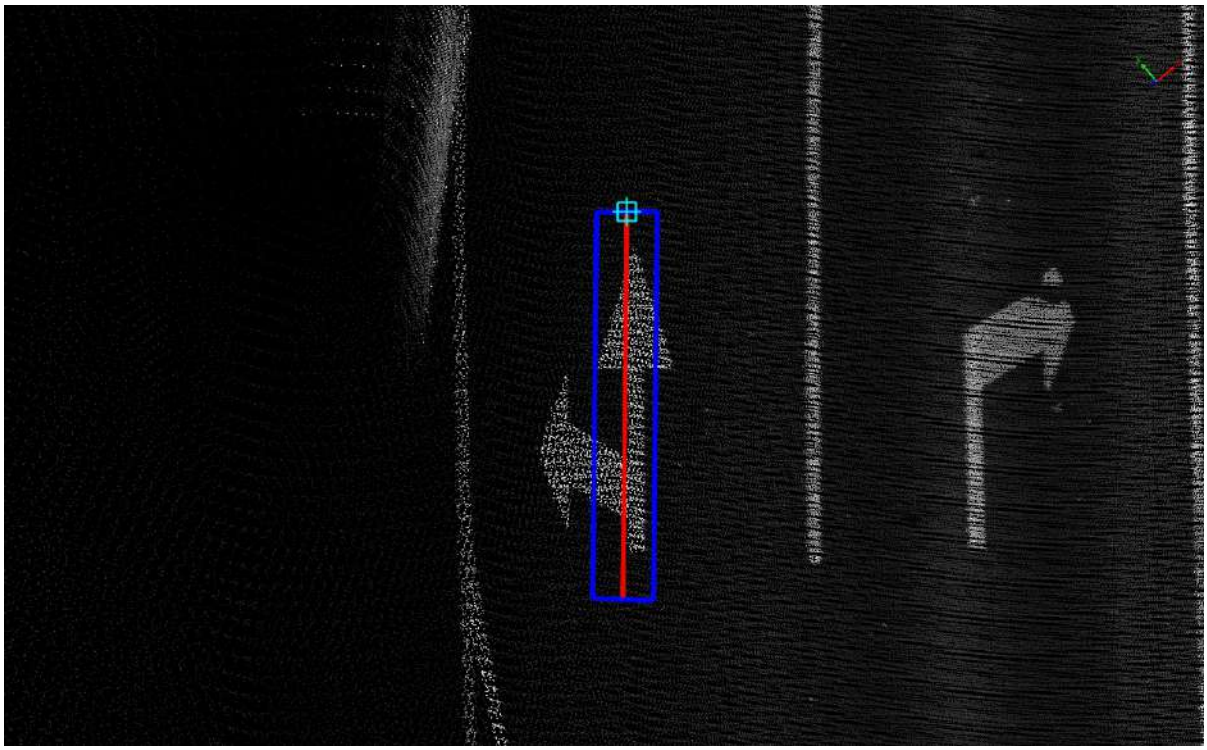
2.Left mouse button, click on the first key point of the target area.

> If the point cloud data has classification information, it is strongly recommended that in the class selection, set the input class to the class of the ground point.
>
> If the point cloud data has no classification information, or use all classes as input classes, **must** select the first key point and the second key point on the ground, and the third key point can be arbitrarily selected according to the size of the pull box.



Key point drawing

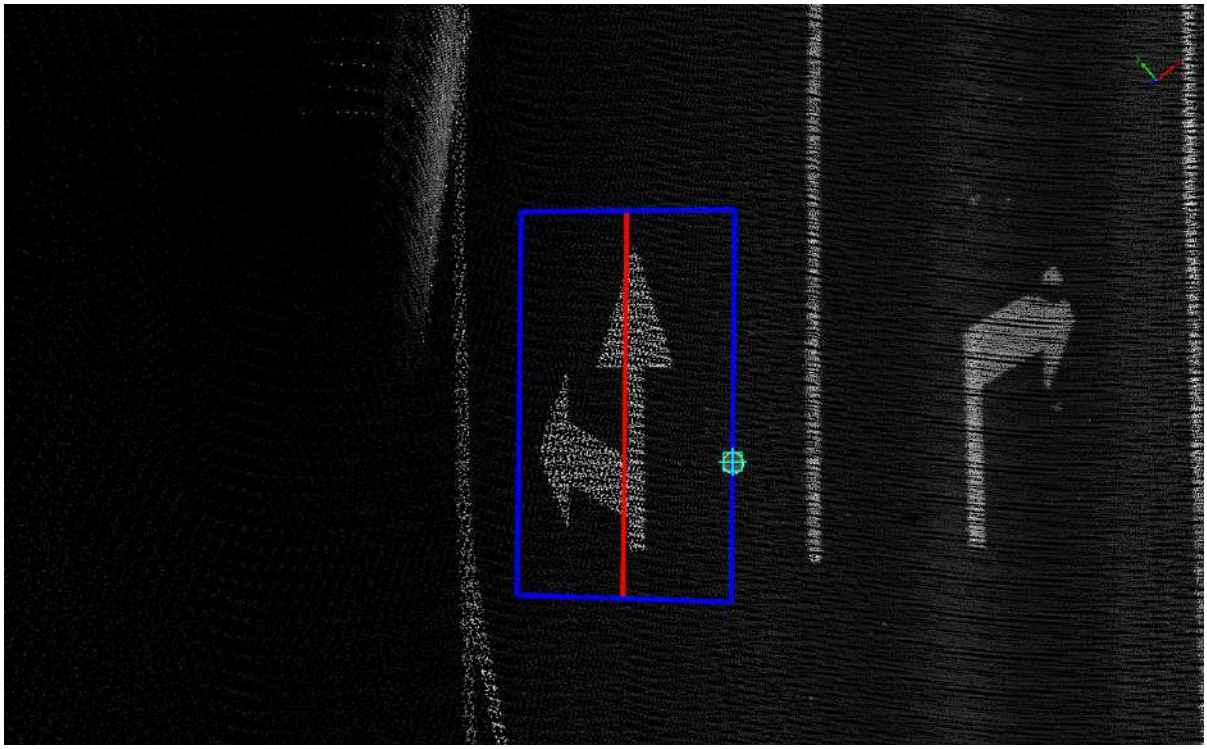
3.Left mouse button, click the second key point.



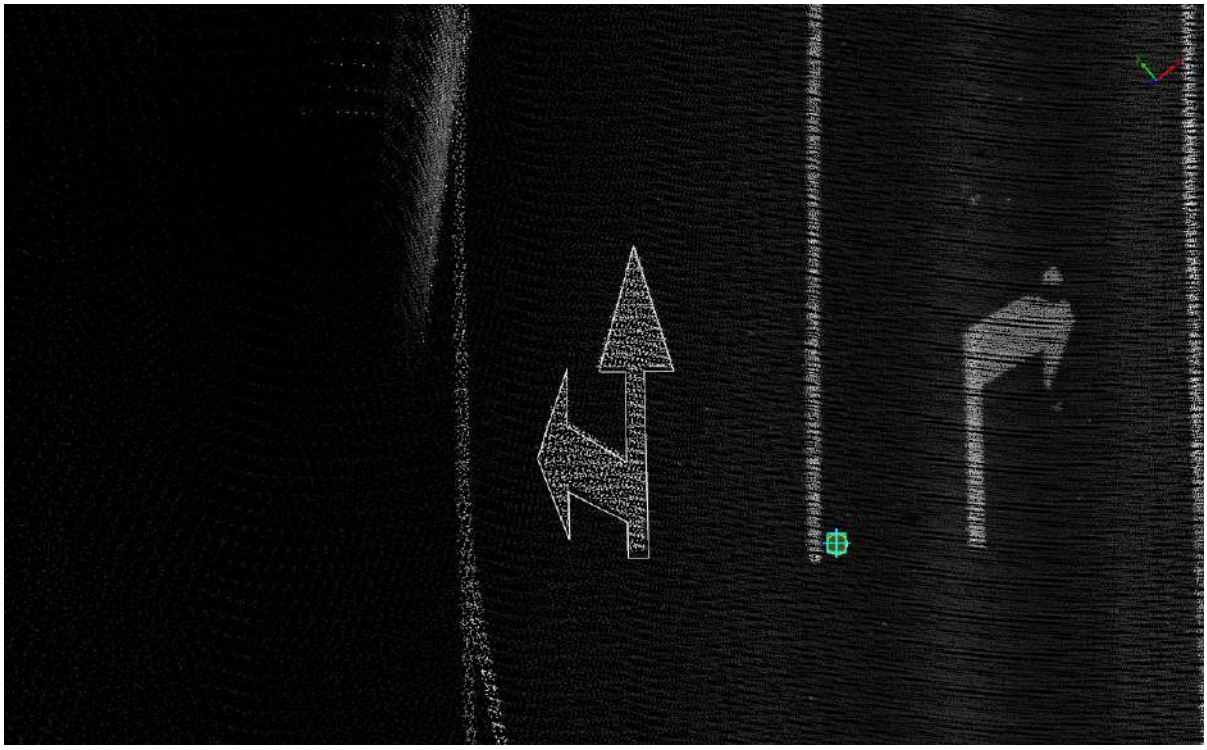
Key point drawing

4.Left mouse button, click the third key point, and then automatically start detection.

> In order to ensure the detection efficiency and detection accuracy, the size of the pull box is **recommended** slightly larger than the ground mark.



Recommended size of standard template table pull box



Detection results

Template Matching Manual

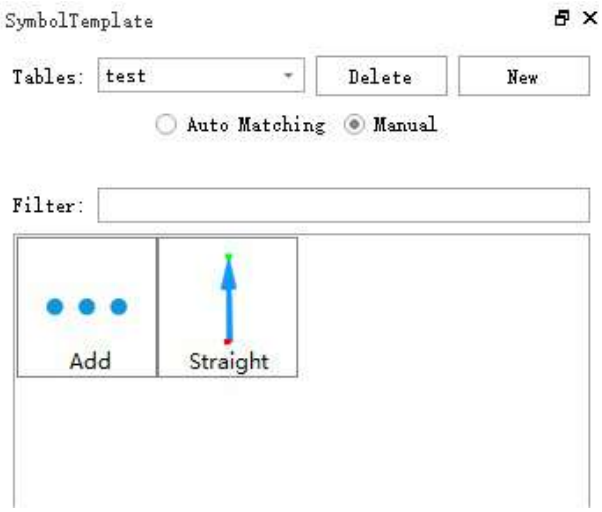
Function Description: This tool can be used to make a new template, or manually add vector results from an existing template.

New template creation

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

Template based, manually drawn

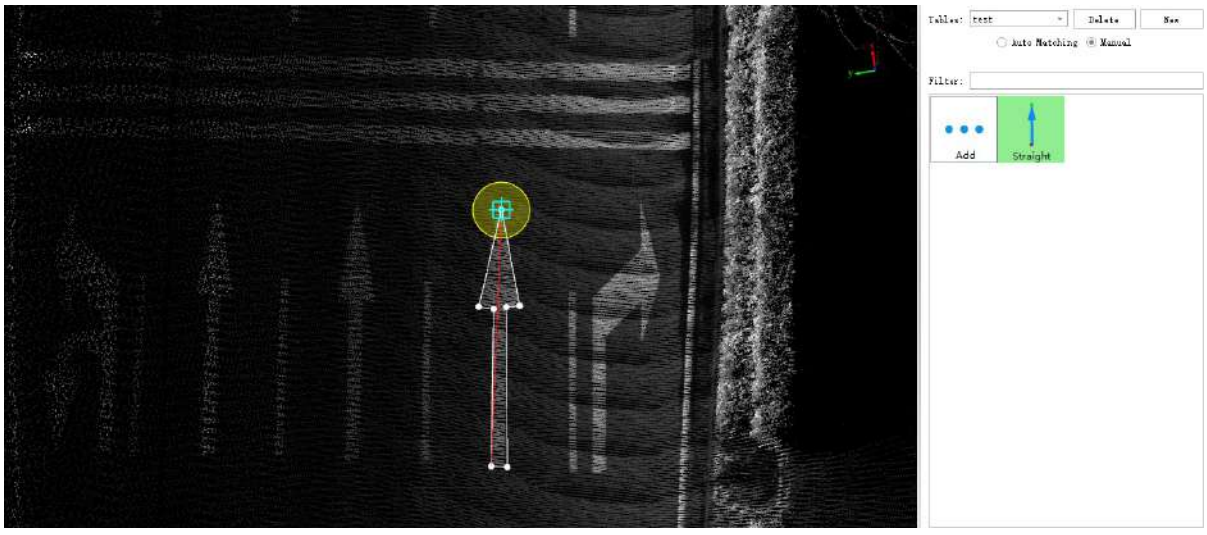
1.Left mouse button in the preview window, select the template to be added manually.



Table

> In the preview icon of the template, the red point is the initial point, and the green point is the direction point.

2.Manual addition, operation method, refer to StraightArrow.



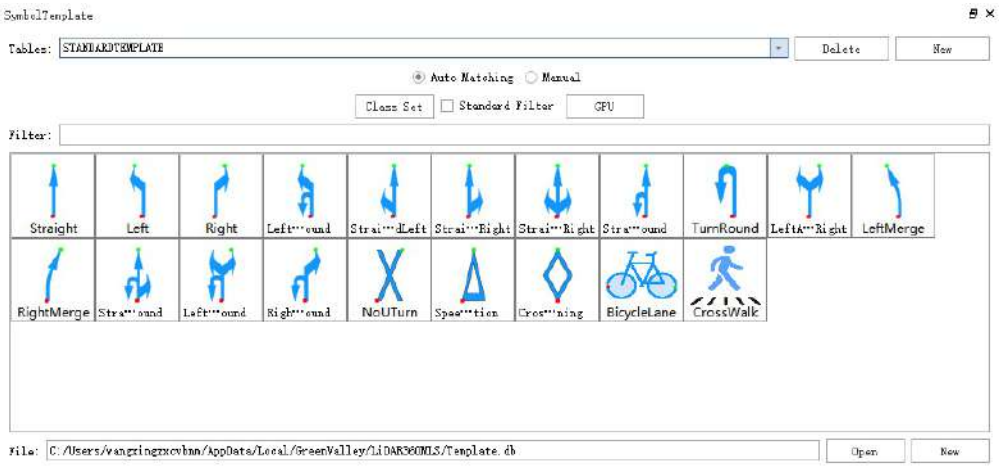
Rendering

Template matching database file

Function Description: This tool can be used to create and open database files.

> Open built-in database file by default.

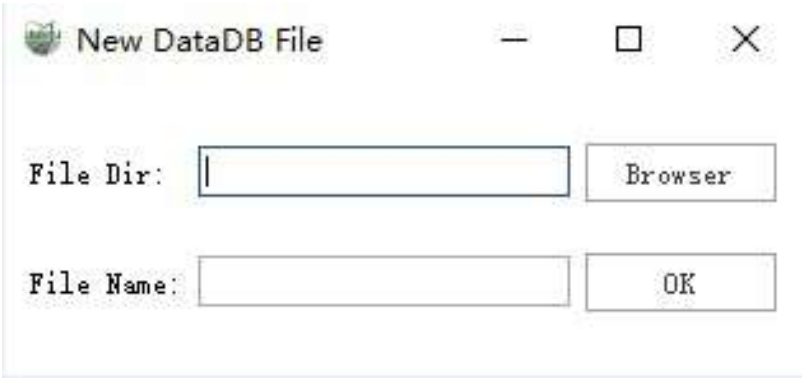
Steps



Shortcut hints

1.Click the **Open** button to open the **Select File dialog box** and select the database file.

2.Click the **New** button to open the **New DataDB File dialog box**.



3.Click the Browser button corresponding to the folder path to set the location of the new database file.

4.Enter a database file name.

5.Click the **OK** button to complete the new creation.

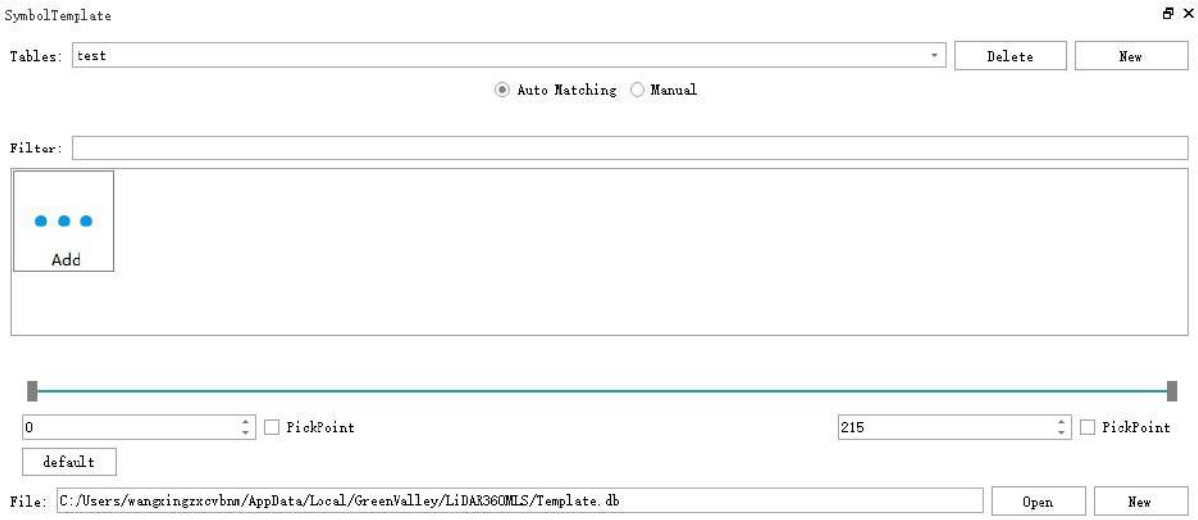
Template Matching New template

Function Description: In the custom table, this tool can be used to create a new custom template.

> New templates can only be created in custom table.

Steps

1.Select a custom table.

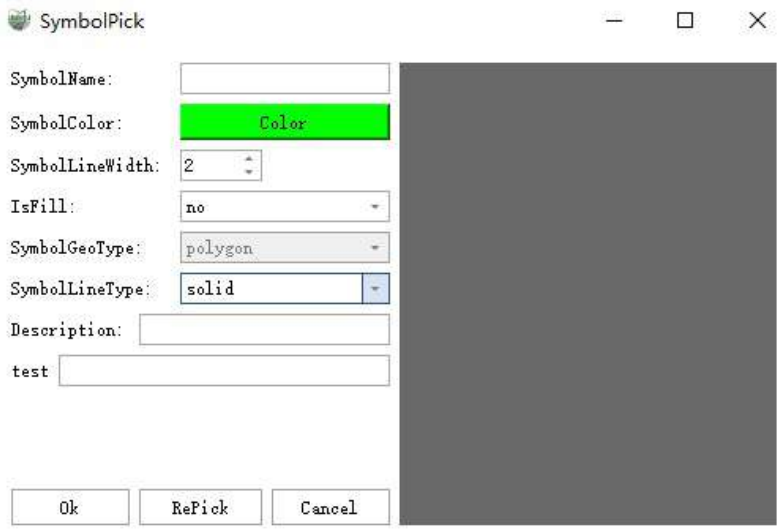


Custom form

2.Click the **Manual creation** checkbox.

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

4.The **SymbolPick** dialog box should pop up.



Dialog

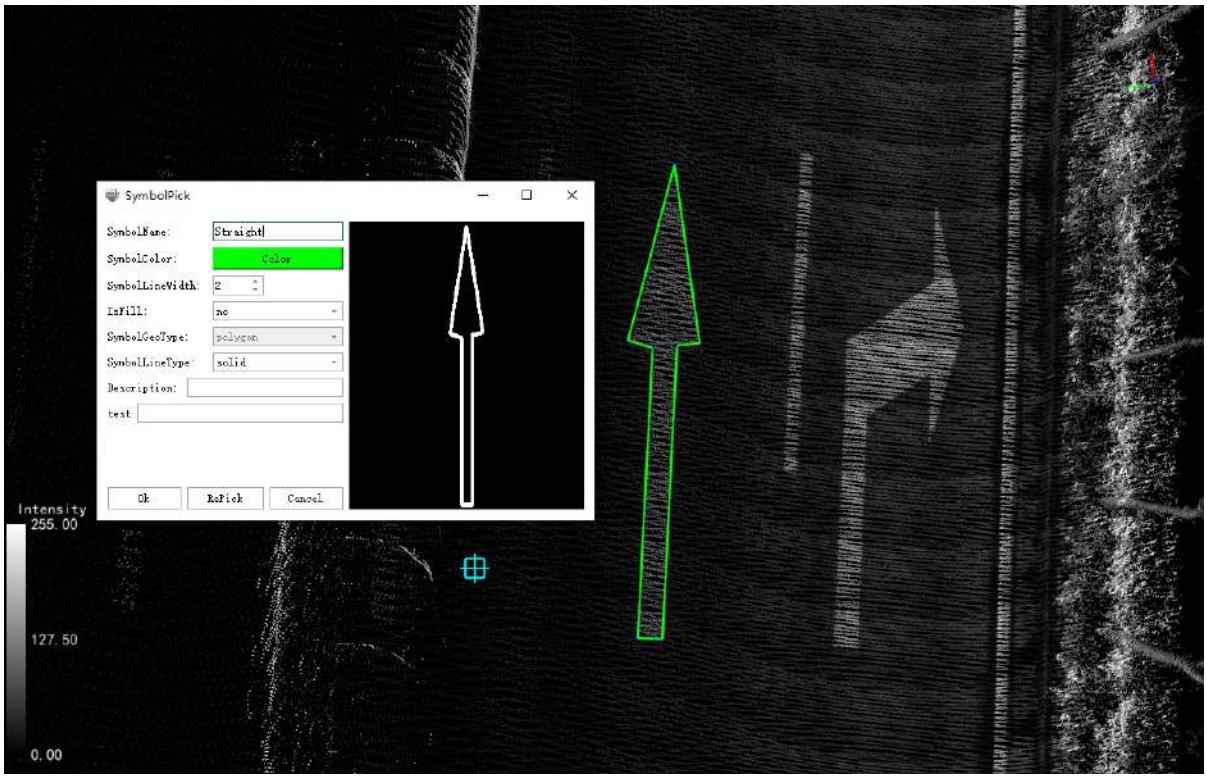
Parameter Description

- **SymbolName**: Template name.

- **SymbolColor**: Set the color of the template.
- **SymbolLineWidth**: Set the template line width.
- **Isfill**: Whether to fill the template.
- **SymbolGeoType**: Set the geometry type of the template.
- **Description**: Text description.
- **Real-time preview**: Real-time preview of the template clicked by the mouse.

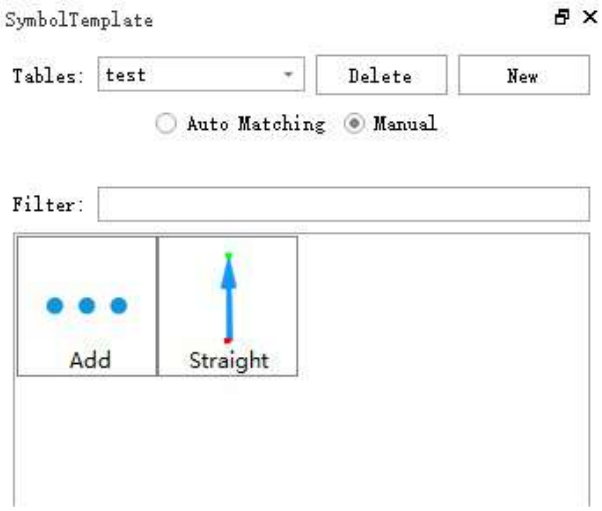
5.Click the left mouse button to select the inner and outer contours of the target point cloud consecutively.

> It is necessary to ensure that the head and tail are connected, that is, the last point coincides with the first point.



New template rendering and real-time preview

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



New template result

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

Extraction by DeepLearning

- Road Marking
- Road Vector
- Merge To Lane

Road Marking

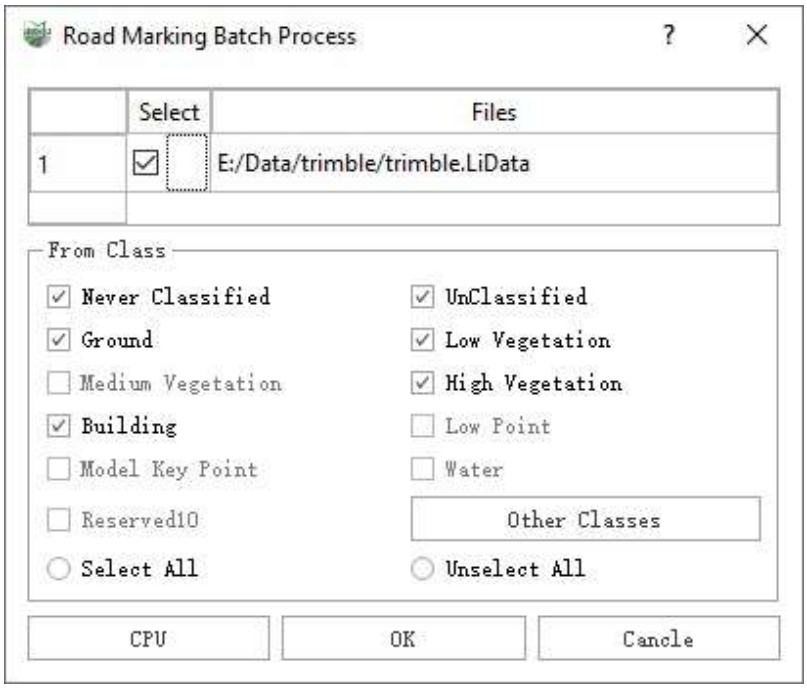
Function Description: This tool enables automatic, deep learning-assisted ground identification extraction for a large amount of point cloud data.

> **Strongly recommended**: Before using ground identification batch processing, perform point cloud classification, and only select the category where the ground point is located as the input class, which can effectively improve processing efficiency and detection result accuracy.

Steps

1.Click **Road Marking** button.

2.The **Road Marking Batch Process** box should pop up.



Road marking batch process

Parameter Description

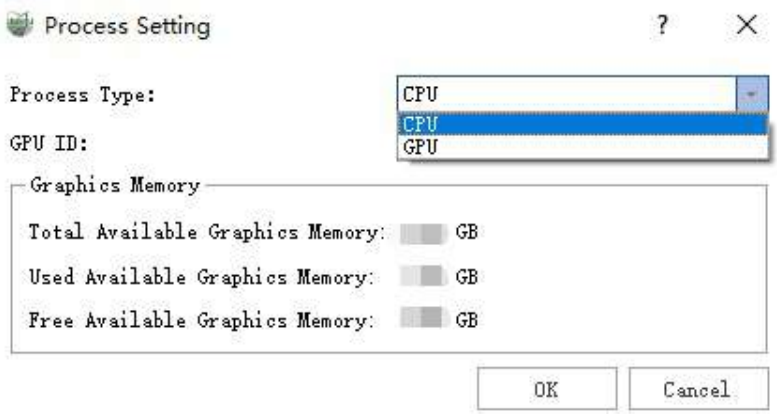
- **Select**: Check/uncheck the point cloud data to be processed.
- **From Class**: Select the class point that needs to be calculated and users can select multiple.
  - **Other Classes**: Click to show other classes.
  - **Select All**: Click to select all classes.
  - **Unselect All**: Click to cancel all selections.

> If the current point cloud data is not classified, only the "Create point, unclassified" option is available;
>
> **Strongly recommended**: Before using ground identification batch processing, perform point cloud classification, and only select the class where the ground point is located as the input class, which can effectively improve processing efficiency and detection result accuracy.

- **Mode: CPU, Mode: GPU**: Set the operation mode and users can choose CPU or GPU settings, running in

GPU mode can improve the operation efficiency to a certain extent.

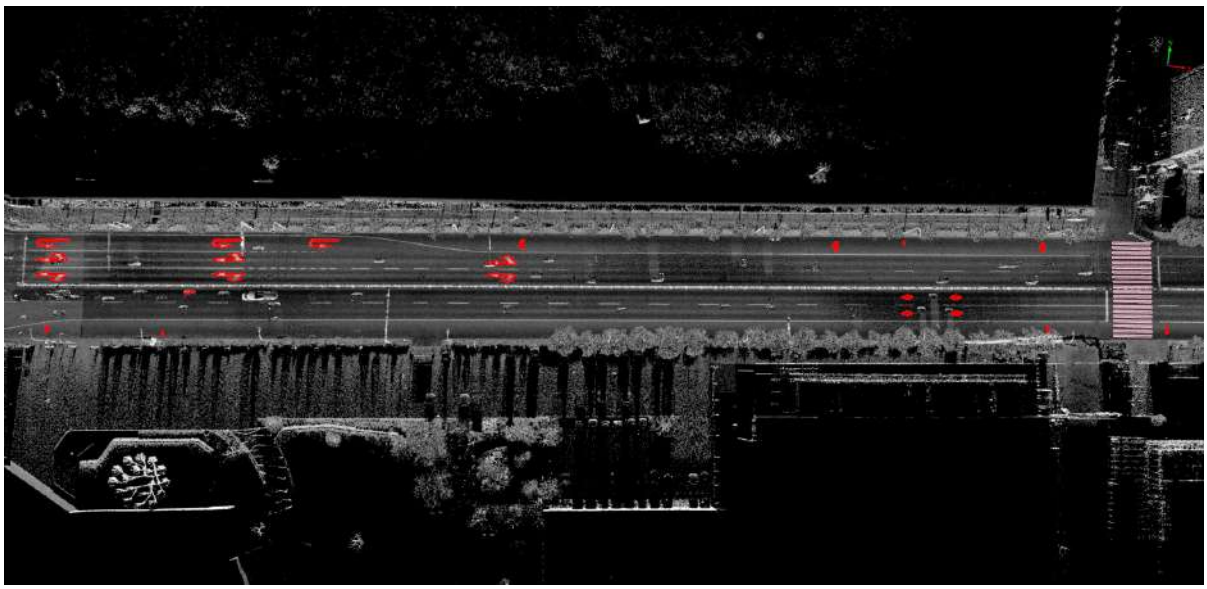
> GPU mode requires computer hardware and software environment support, if the conditions are not met, even if checked, it will still run in CPU mode.



Processing settings

- **OK**: After setting the corresponding parameters, click the **OK** button to start automatic detection.
- **Cancel**: Cancel and exit the function.

3.Click **OK** to enable automatic identification of road surface identifiers and vectorize them automatically. Crosswalk signs will be stored in the software's built-in crosswalk layer, and other signs will be stored in the built-in road surface signs.



Processing settings

Road Vector

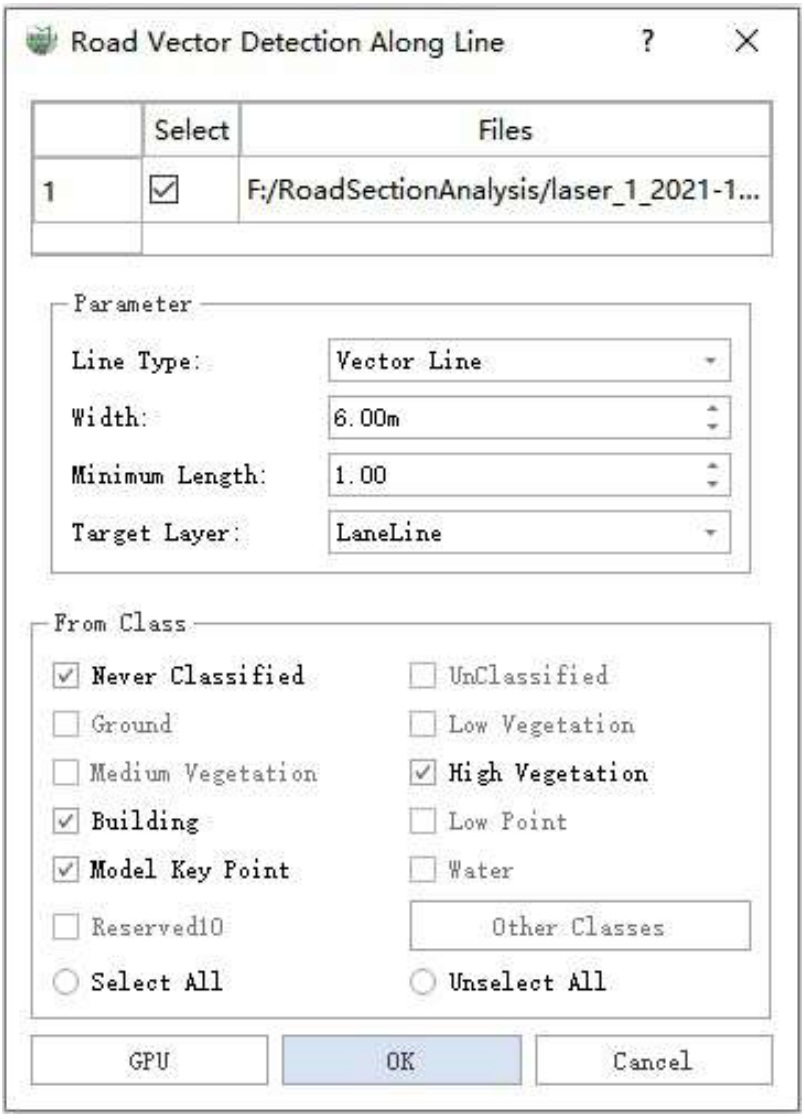
Function Description: Based on the deep learning solution for road vector detection, the choice of the detection area is along the route.
Road vectors include lane lines (all are solid line segments, which need to be combined with the Merge into Lane Line function to generate complete lane solid lines, lane dashed lines), ground arrows, crosswalks, crosswalk warnings, yield signs, and other ground-related vectors.

> **Strongly Recommended**: Before using this function, perform point cloud classification, and **only select** the category where the ground points are located as the input category, which can effectively improve processing efficiency and detection accuracy.

Steps

1.Click the **Road Vector** button in the Map Elements panel.

2.The **Parameter Setting Dialog Box** pops up.



Road vector detection along the route

Parameter Description

- **Select**：Check/uncheck the point cloud data that needs to be processed.
- **Parameters**：
  - **Line Type**: This function needs to determine a **baseline**, this parameter is used to set the origin of the baseline.
    - **Vector Line**: Select an existing vector line as the baseline.
    - **Custom Line**: Manually create a vector line as the baseline.
    - **Trajectory**: Select a section of trajectory as the baseline, **recommended use**.

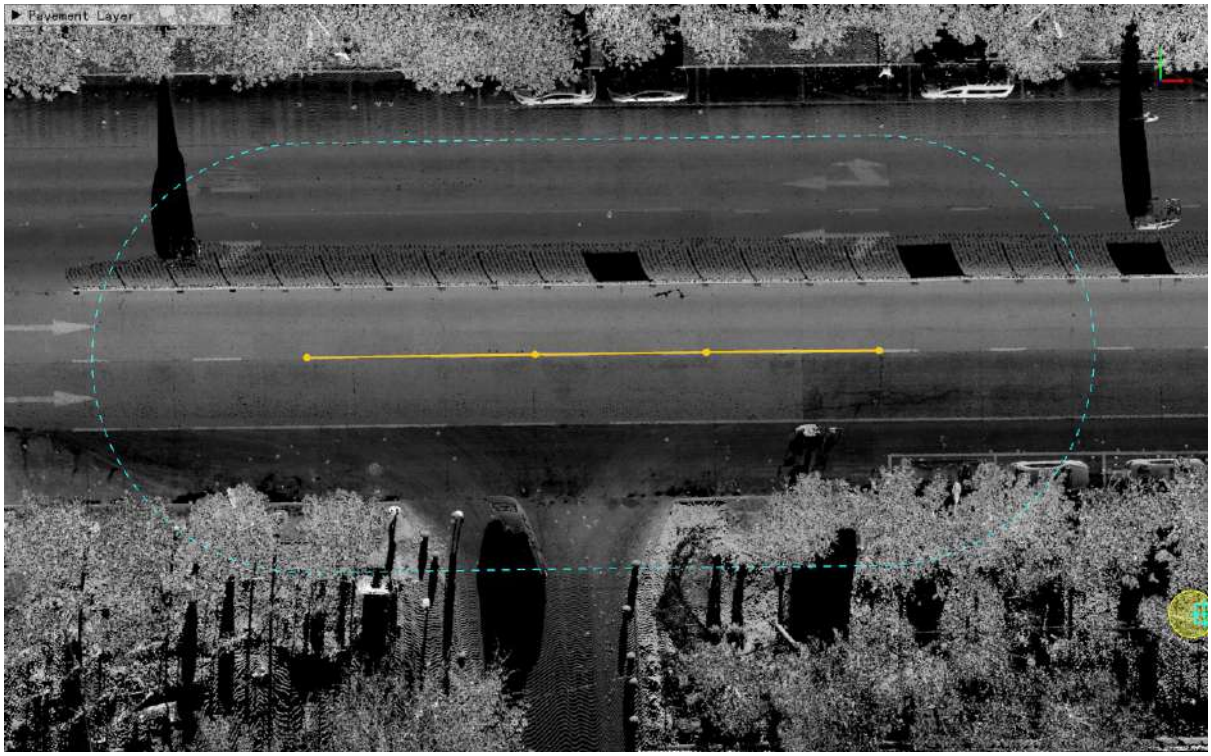
    > The Trajectory option is only available when there is trajectory data in the current project.
  - **Width**: Used in conjunction with the Line Type, forming a polygon area with the baseline as the centerline and the set width to the left and right, and performing road vector detection within this area.
  - **Minimum Length**: The detected too-short lines (lane lines) are often incorrectly extracted. Lane lines shorter than the minimum length will be filtered out
  - **Target Layer**: The layer where the lane line result is to be stored.
- **From Class**: Set the point cloud category used for detection.

  > he current point cloud data is not classified, only the "Create point, Unclassified" option is available
  >
  > **Strongly Recommended**: Before using this function, perform point cloud classification, and only select the category where the ground points are located as the input category, which can effectively improve processing efficiency and detection accuracy.
- **Mode: CPU, Mode, GPU**: Set the operation mode, you can choose CPU or GPU settings, running in GPU mode can improve operation efficiency

  > GPU mode requires computer hardware and software environment support, if the conditions are not met, even if selected, it will still run in CPU mode.
- **OK**: After setting the corresponding parameters, click the OK button to start the automatic detection
- **Cancel**: Cancel and exit the function

3.Baseline setting and polygon area generation

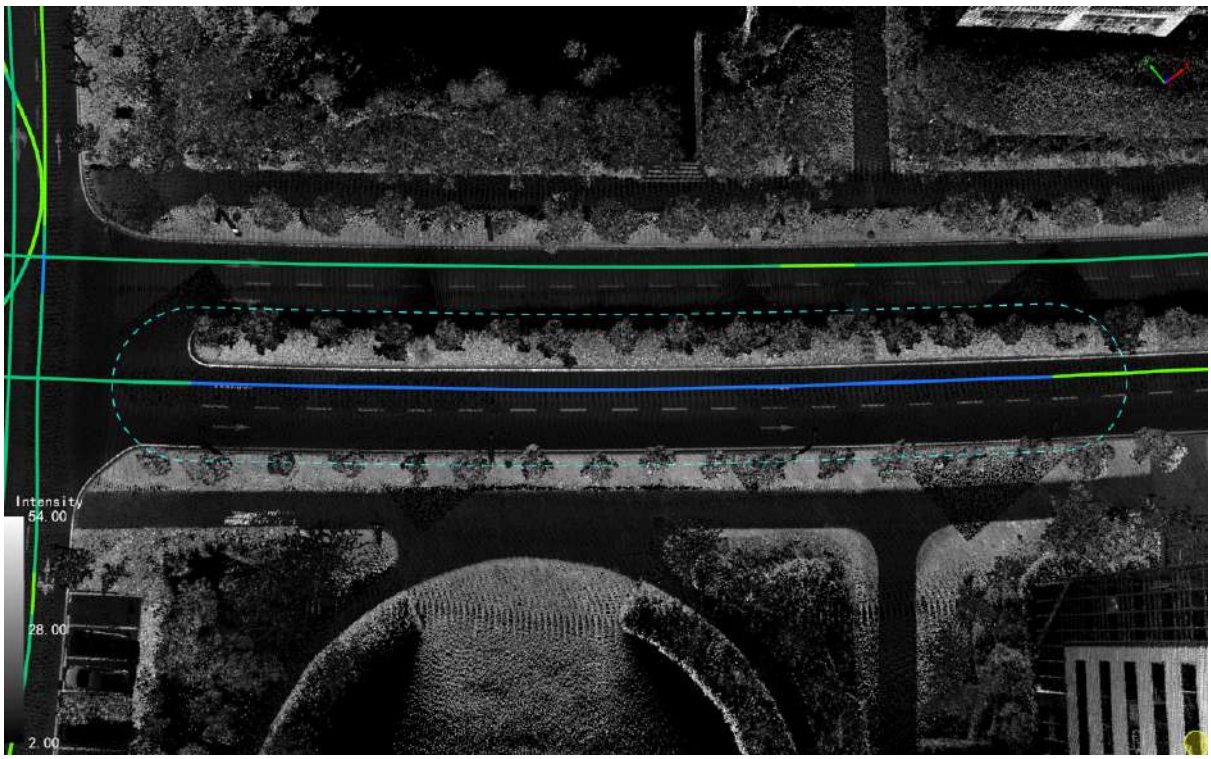
- **Vector Line Mode**: Left-click with the mouse to select an existing vector line, automatically generate a polygon area based on the width, and adjust the polygon area by adjusting the width.
- **Custom Line Mode**: Left-click with the mouse in sequence to add a vector line, double-click with the mouse to end the vector line making, automatically generate a polygon area based on the width, and adjust the polygon area by adjusting the width.



Vector Line Mode and Custom Line Mode Polygon Area

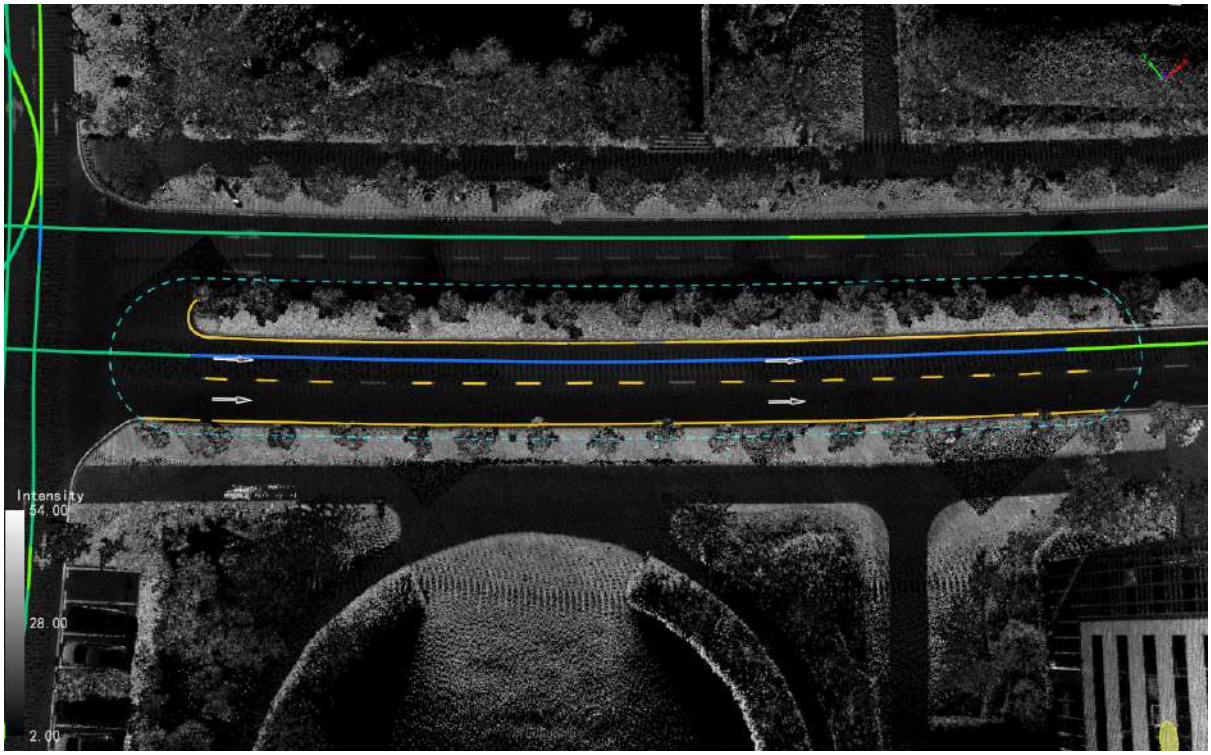
- **Trajectory Mode, recommended use**

First, left-click with the mouse to select the starting point of the trajectory segment, then left-click with the mouse to select the endpoint of the trajectory segment. Automatically generate a polygon area based on the width, and adjust the polygon area by adjusting the width.



Trajectory Mode Polygon Area

4.Click **OK** to start automatically recognizing road surface symbols and automatically vectorizing them. The crosswalk symbols will be stored in the built-in crosswalk layer in the software, the lane lines will be stored in the layer set by the **Target Layer**, and other symbols will be stored in the built-in road surface symbols.



Result Display

In the resulting figure, all lane lines are solid lines and need to be combined with the Merge into Lane Line function to generate complete lane solid lines and lane dashed lines.

Merge To Lane

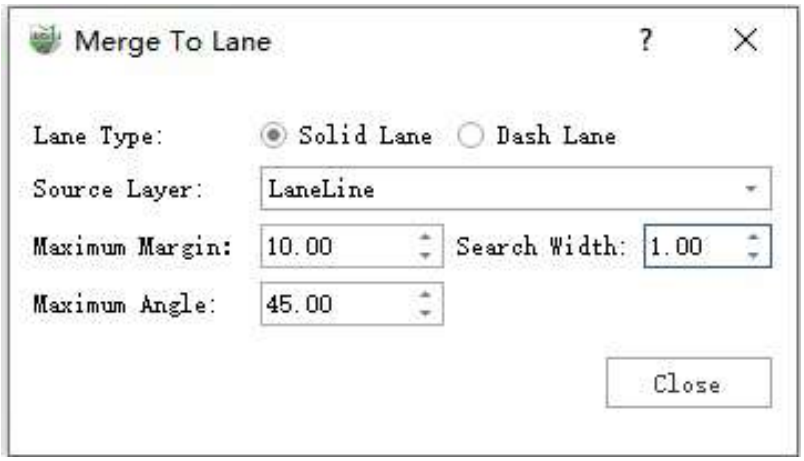
Function Description: This function is mainly used to merge similar and sequentially connected line vectors into lane solid lines or lane dashed lines.

> Strongly advised that before using this feature, please check and correct the geometry of line vectors to ensure correct merging, and that there is no error in the geometry of the merged lane lines.

Steps

1.In the Road Vector Detection panel, click the **Merge To Lane** button.

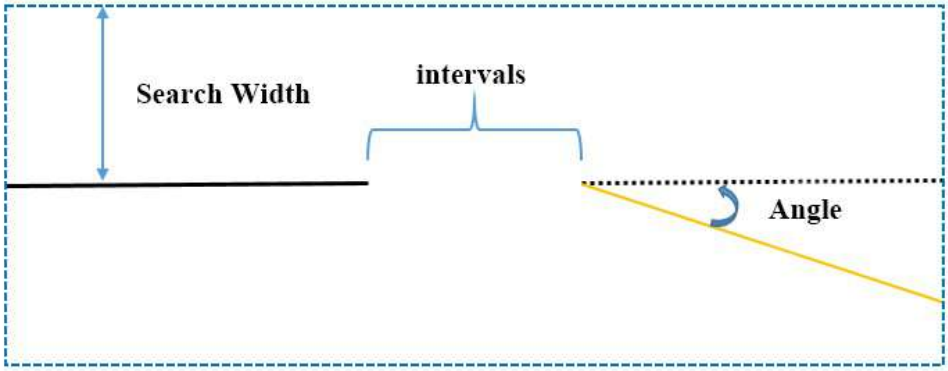
2.The **Merge To Lane dialog box** pops up:



Dialog Box

Parameter Description:

- **Lane Type**: Set the type of lane lines to be generated
- **Source Layer**: The source of the lines to be merged. Only the line vectors in the source layer can be selected and merged.
- **Maximum Margin**: The maximum gap for sequential connection. Lines with a gap larger than this value will not be merged, refer to the example diagram.
- **Search Width**: Search for the next line within a certain width on the left and right along the forward direction of the line, refer to the example diagram.
- **Maximum Angle**: The maximum angle for sequential connection. Lines with an angle larger than this value will not be merged, refer to the example diagram.



Schematic diagram, the black solid line is the first line; the yellow solid line is the second line

3.Window tips：

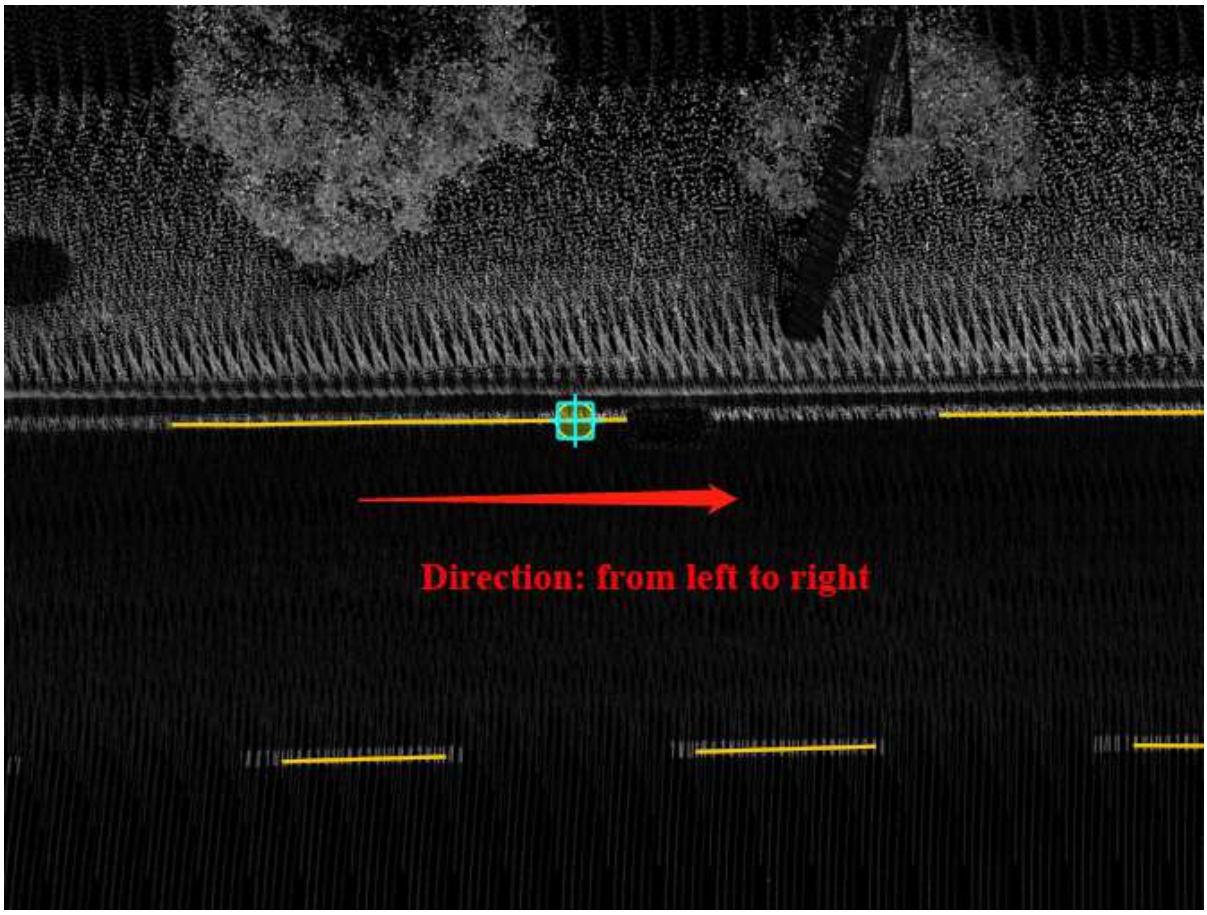
Shortcut Key Name	Function
Space	Interrupt merging
Enter	End merging
C	Switch between automatic/manual mode
H	Close/Open the top left corner prompt
Current model	Current drawing mode
Right mouse button	Retract a single node (only effective after the interruption, invalid during growth)

This function provides two modes, one is automatic mode, which automatically merges forward by selecting a vector line. The other is manual mode, where line nodes are added by clicking the left mouse button. The two modes can be switched by pressing C during the drawing process.

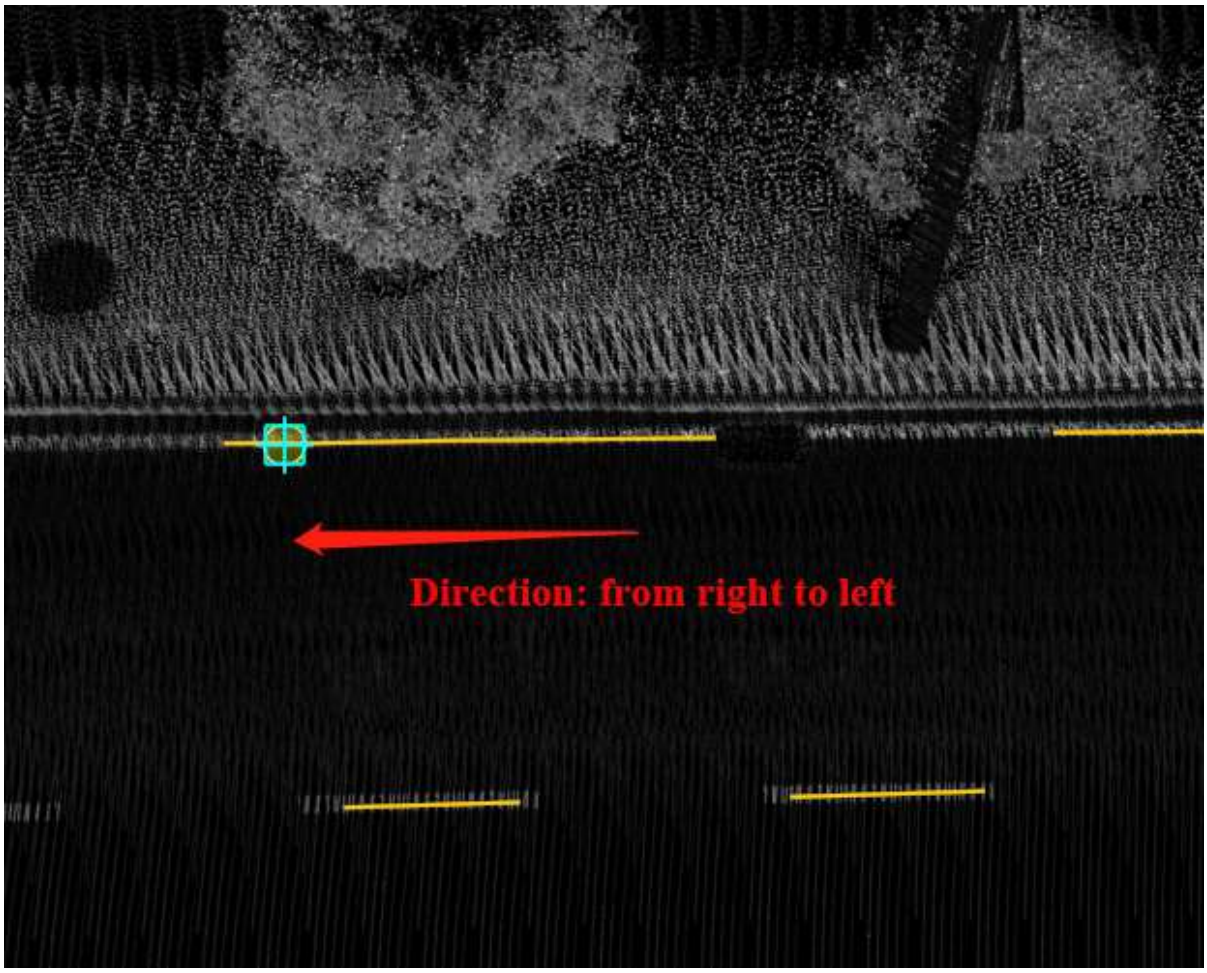
Solid Line Merge

Choose a solid line as the lane line type.

1.Select the first line. Note that when selecting the first line, the mouse selection position determines the direction of the line. The principle of selection is that the mouse selection position is the tail end of the first line.

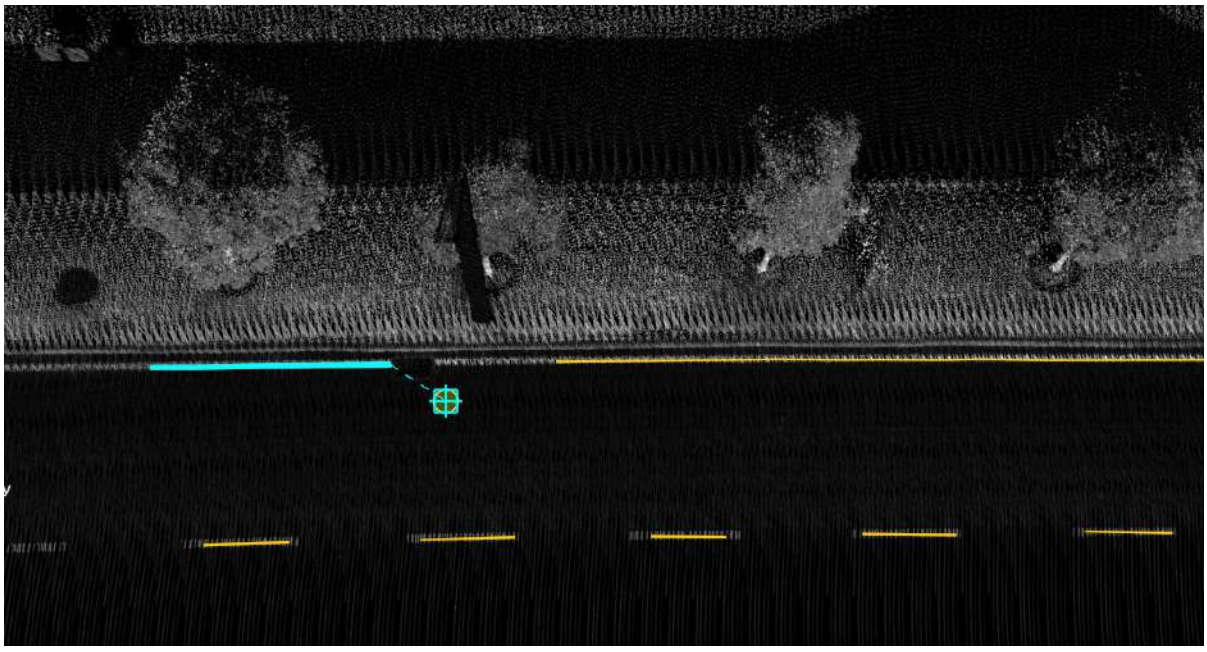


Schematic diagram, if the mouse selects the right side of the line, then the right side is the tail end



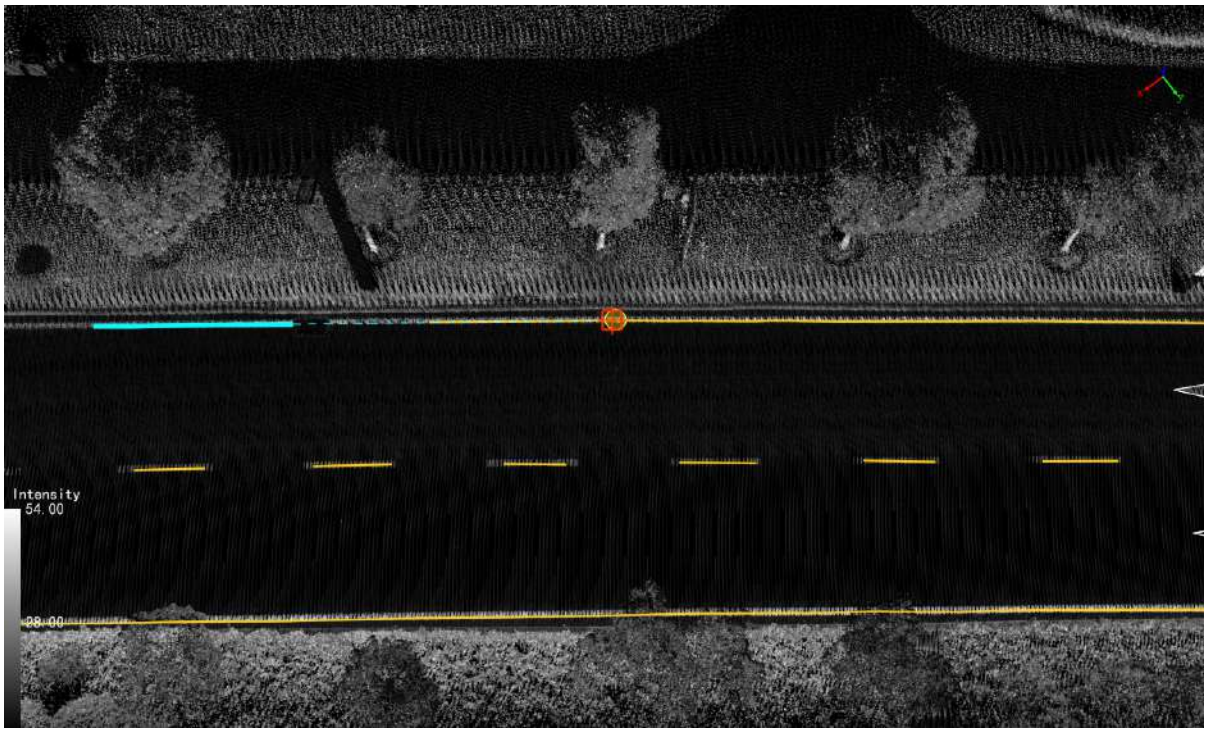
Schematic diagram, if the mouse selects the left side of the line, then the left side is the tail end

2.Select the first line to determine the direction of the merged line, click with the left mouse button, and select the first line according to the principle above.



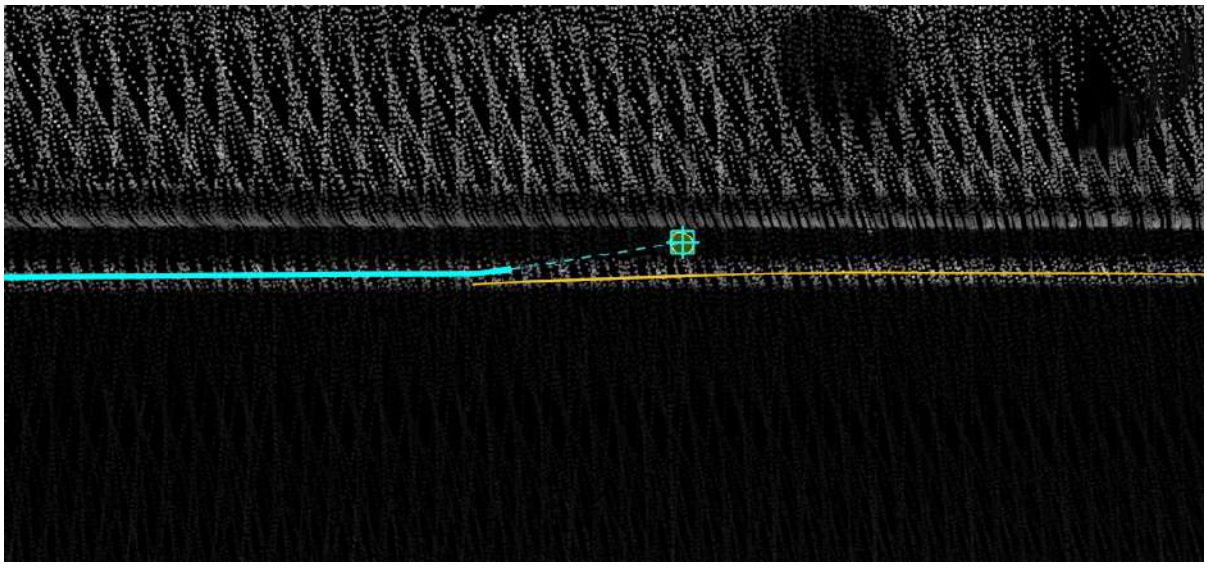
Select the first line

3.Select the second line. You can start automatic forward merging by clicking on **any position** of the second line with the left mouse button until the conditions are not met and the auto-merge is automatically terminated.

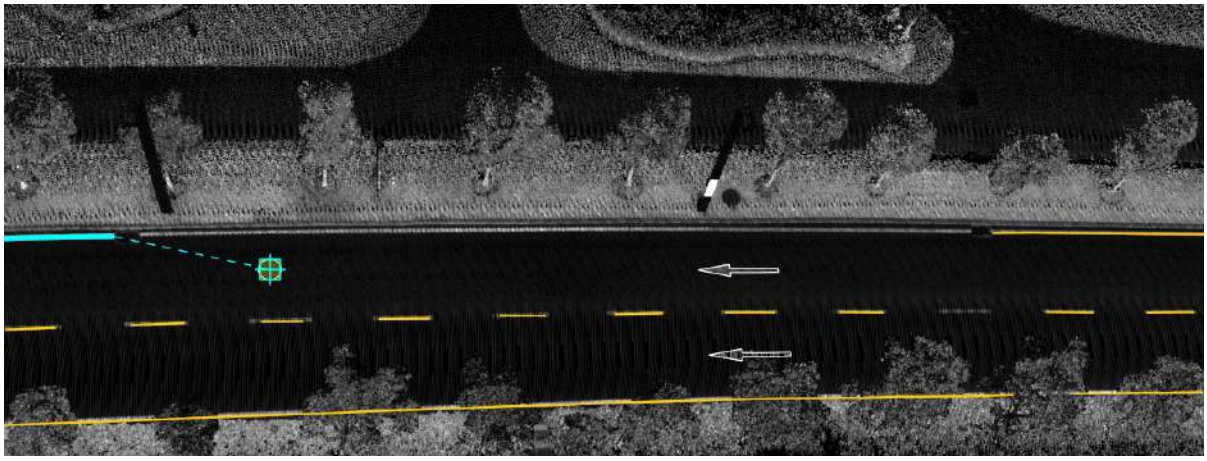


Select the second line

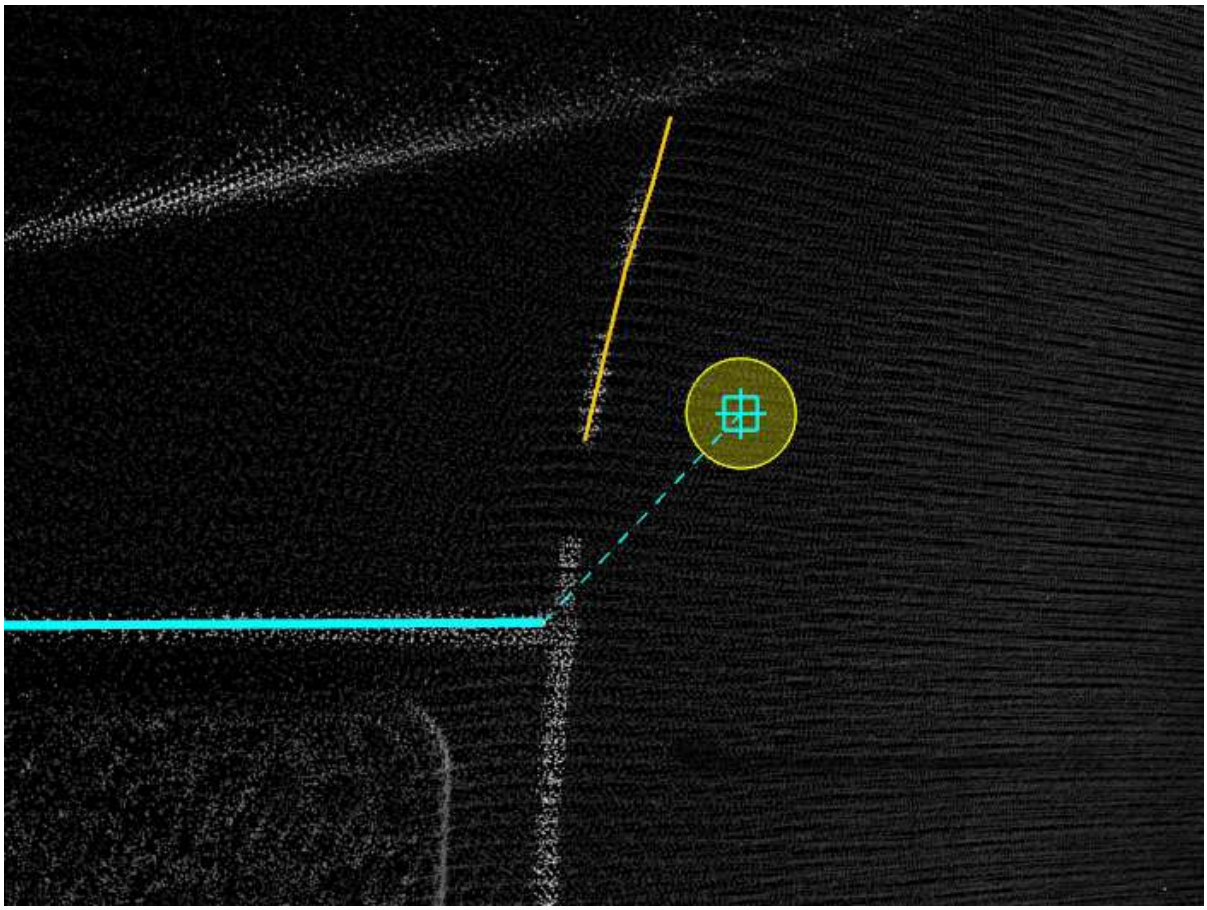
4.Interruption Conditions



Interruption condition: Backfolding occurs



Interruption condition: Maximum gap exceeded



Interruption condition: Maximum angle exceeded

5.After the interruption, detect again:

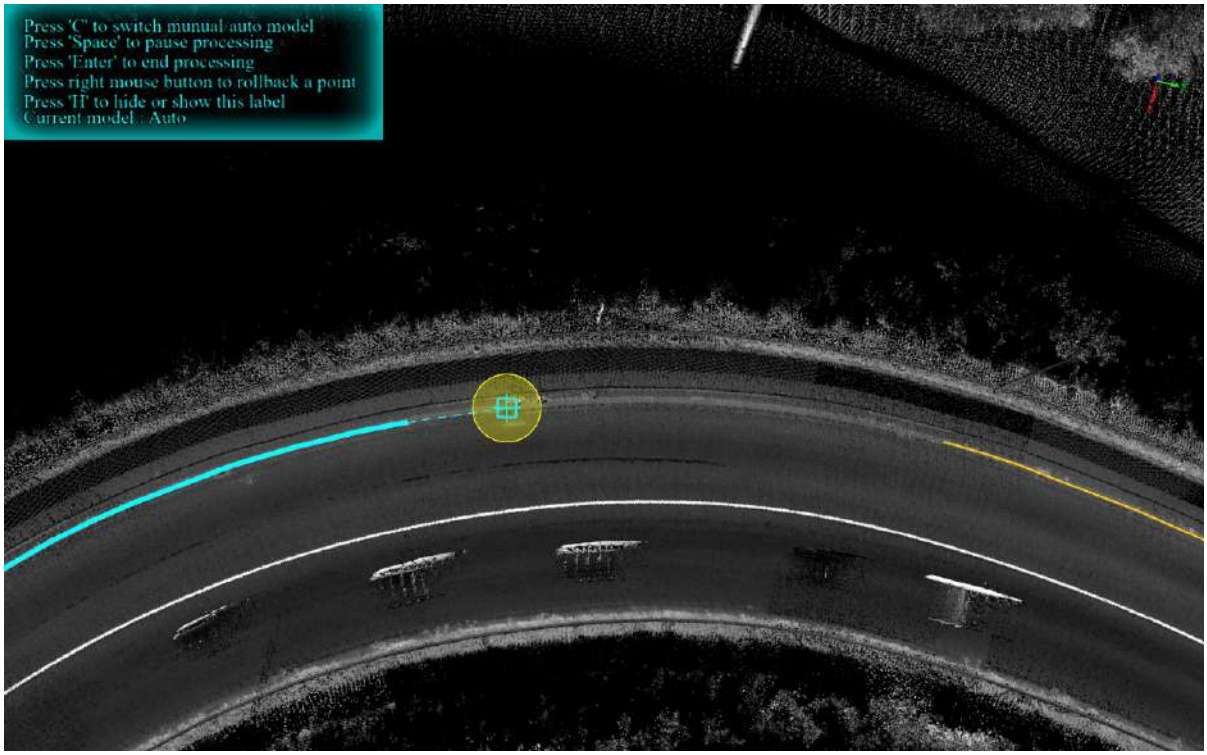
- Automatic mode: After the interruption is generated in the above situation, the forward direction of the line is known at this time, so just repeat step 2, and re-select the next line to start the automatic merge again.

> If the reason for the interruption is back folding, to prevent the final line from having back folding, it is strongly recommended to withdraw the function first, adjust the line geometry, and eliminate the back folding.

- Manual mode: After the interruption is generated in the above situation, if the next line is too far from the current line, or the angle is too large, and the **direct connection will not conform to the actual scene**, you

need to manually add points. Press the C key to switch to manual mode, and add points one by one with the mouse.

> Switch back to automatic mode: 1. Double-click to end manual mode, and automatically switch to automatic mode; 2. Press the C key to switch.
>
> After the interruption, you can use the right mouse button to retract, to adjust the shape.



Scenes that need to manually add points

6.After the merge (or drawing) is completed and the shape is acceptable, press the Enter key (Enter, confirmation key) in automatic mode to complete the merge of the current lane solid line.

Dashed Line Merge

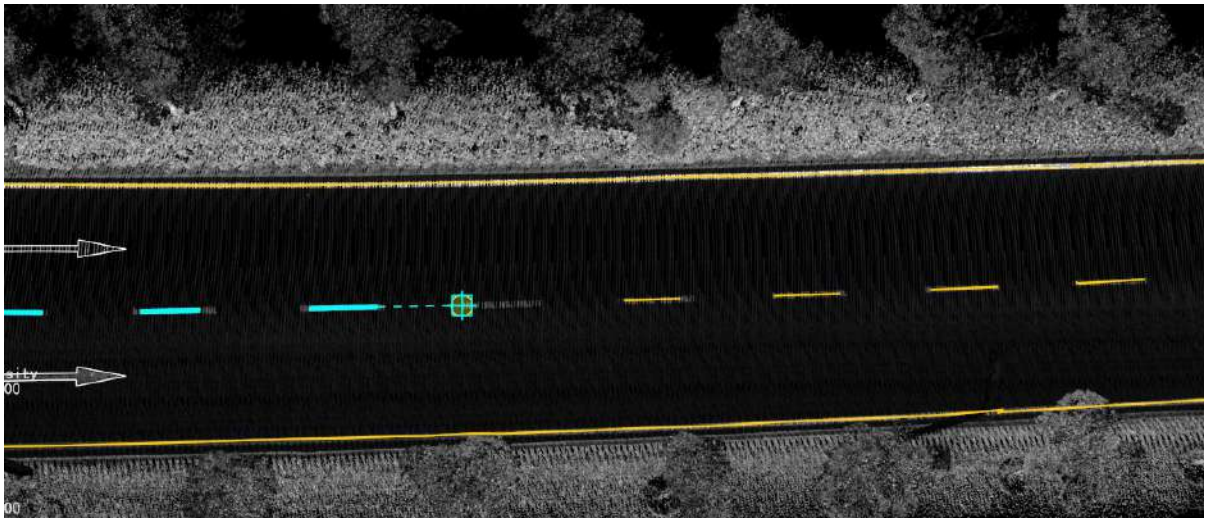
Choose the dashed line as the lane line type.

1.Select the first vector line, refer to solid line merge.

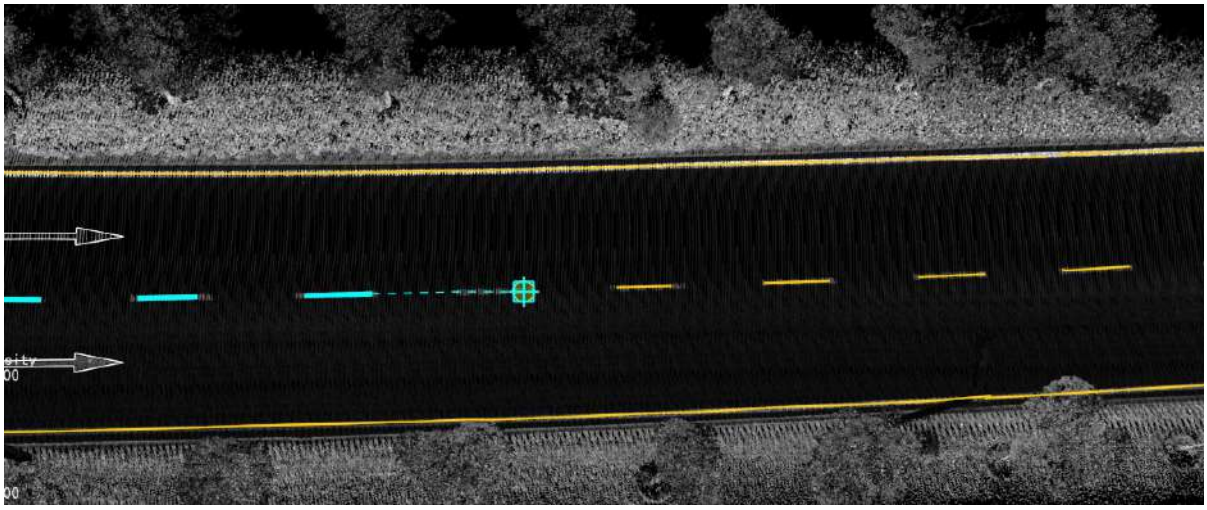
2.Select the second vector line, refer to solid line merge.

3.Interruption conditions, refer to solid line merge.

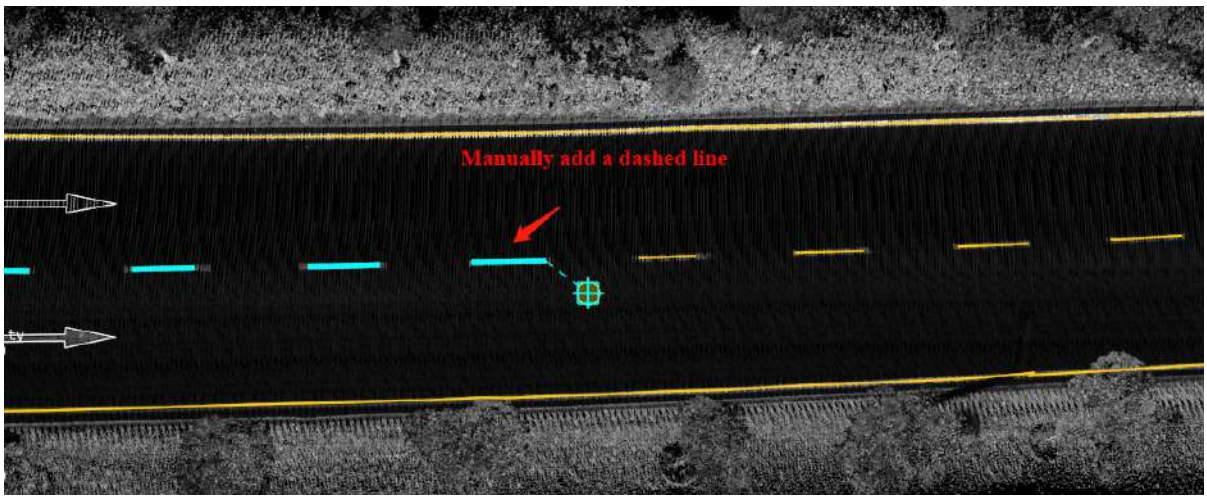
4.In the case of interruption, if you switch to manual mode, unlike the solid line merge which adds points one by one, the dashed line merge needs to add two points at the same time to add a dashed line segment.



Left-click with the mouse, manually add the starting point of the dashed line segment



Left-click with the mouse, manually add the endpoint of the dashed line segment



Manual adding effect

5.Detect again after the interruption, refer to the solid line merge.

6.After the merge, refer to the solid line merge.

Facility

Road Facilities

Road Facility Detection

Linear Facility Extraction

Rail Way Detection

Pole Seed Editor

Feature Objects

Road Facilities

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

It includes the following functions：

- Detect Pole
- Detect PowerLine
- Add TrafficSign
- Add Parking Spot
- Add Rain/Strainer
- Add Manhole

Detect Pole

Distinguished by **operating mode** and is divided into four modes:

- Single Point mode
- Two Point mode
- Polygon mode
- Grip Ground mode

Single Point Mode

Function Description: The Single Point Mode for pole detection. To apply, manually click on the top of the pole as the key point, and automatically generate the rod by setting the search radius.

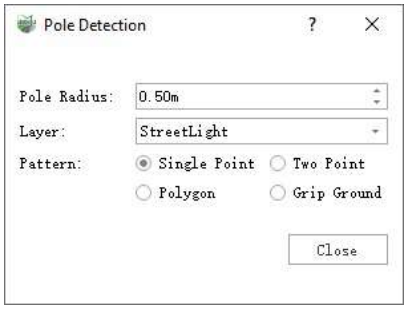
> The generated rod is **precisely perpendicular** to the ground, while the real rod is **not necessarily perpendicular** to the ground due to the **instrument scanning unilateral, tilt itself, thick at the bottom and thin at the top**, etc.

Steps

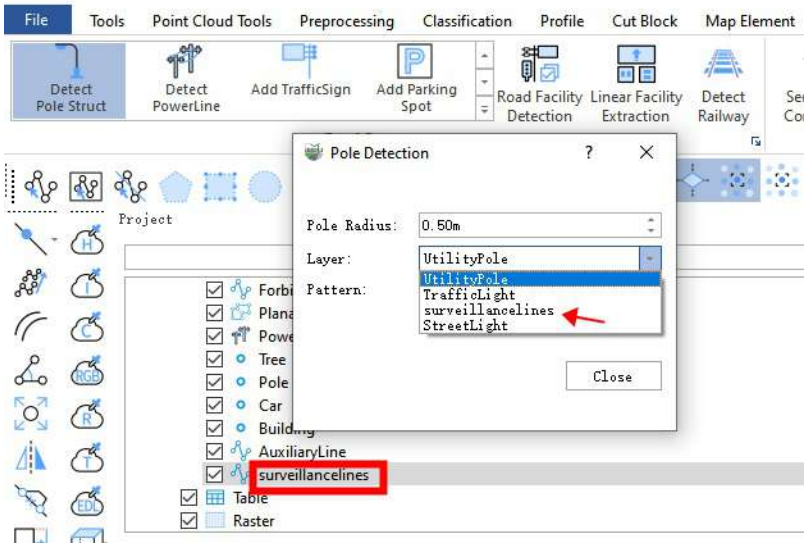
1.In the Road Facilities panel, click on the drop-down logo on the right and select **Detect Pole** button.

2.**Pole Detection** setting dialog box should pop up.

- Single point mode.



Single Point Mode Dialog



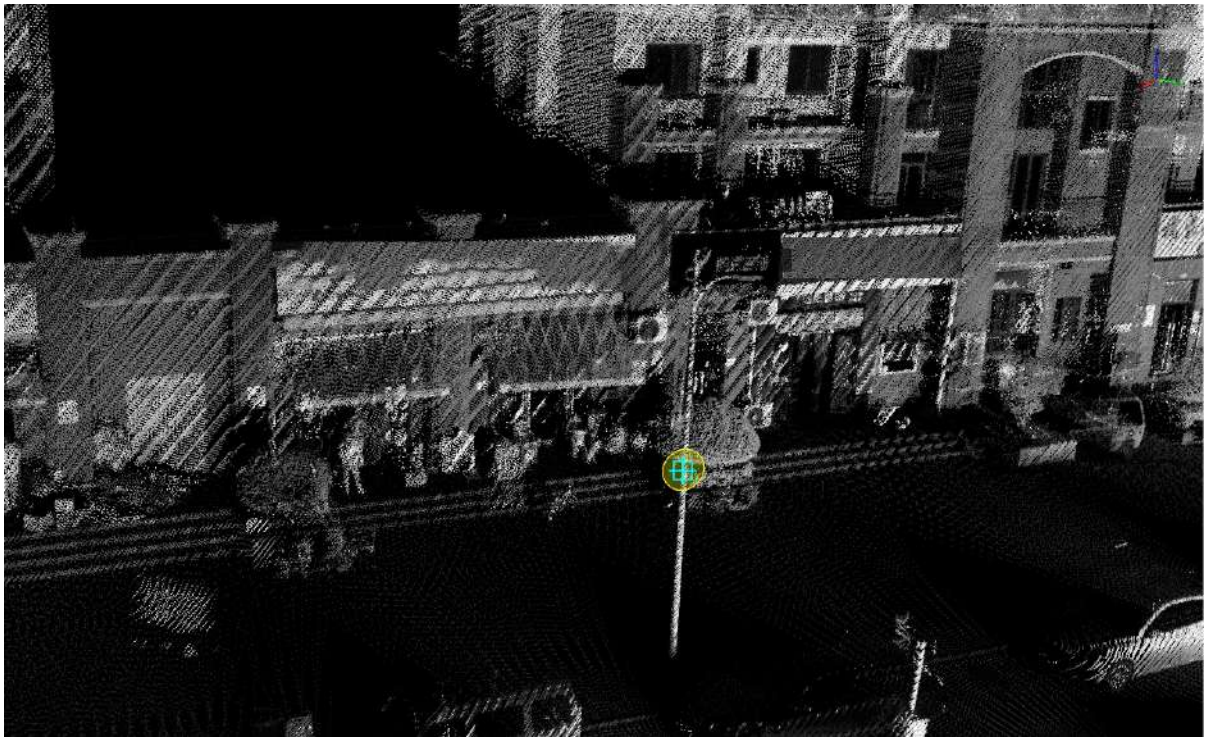
Single Point Mode Dialog Custom Line Layer

Parameter description:

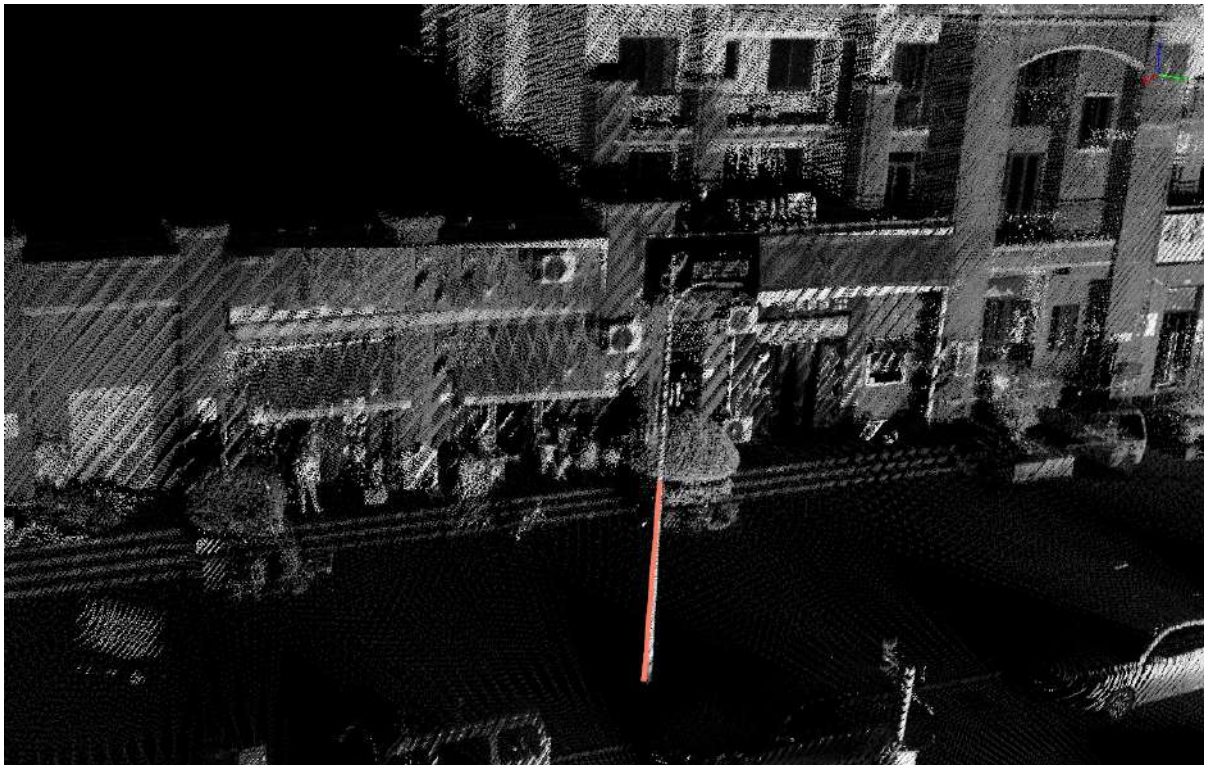
- **Rod Radius**: The default is 0.5 meters, which means that the self-clicked **key point** is the reference point, and the radius of the selected box **rod is the radius, and the cylinder** is **vertically downward**.
- **Layer**: The default is street light, the options are: **streetlights, utility poles**, and also custom layers to add a line layer, indicating the layer where the vectorization results are deposited.

- **Mode**: The default is **single point mode**, which means the operation mode of **clicking on key points**.

3.**Click on the top** of the pole **with the left mouse button to automatically generate a single pole**. The details are shown in the figure. The yellow dot **is the key point** clicked by the mouse. It is recommended to click on the top position of the rod.



Click on the key point



Single point mode renderings

Two Point Mode

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

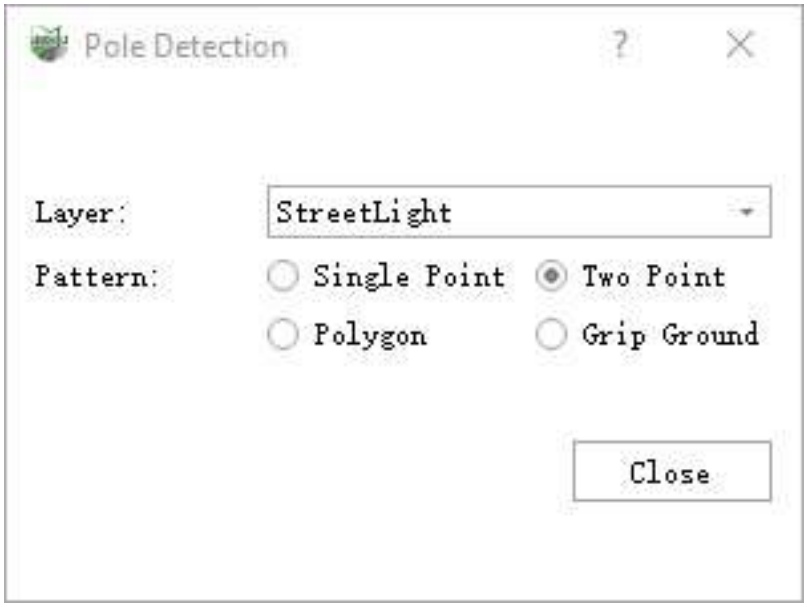
> The generated bar is **exactly in line with the two points clicked**.

Steps

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

2.The **Pole Detection** setting dialog box should pop up.

- Two Point mode.



Two-point modal dialog box

Parameter description:

- **Layer**: Default is street light, and the options are: **street light, utility pole**, Users can also customize the layer to add a line layer, indicating the layer where the vectorization results are deposited.
- **Mode**: The default is **Two Point**, which means the operation mode of **clicking on key points**.

3.**Clicktop of the rod with the left mouse button**, the details are as shown in the figure, and **yellow point** is the key point selected by mouse.

Click on the key point at the top of the pole

4.**Click the bottom of the rod with left mouse button**, the details are shown in the figure, **yellow point** is the key point of the mouse click.

Two-point mode renderings

> Note: The two-point mode with the profile drawing effect will be better and the efficiency will be faster.

Polygon Mode

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

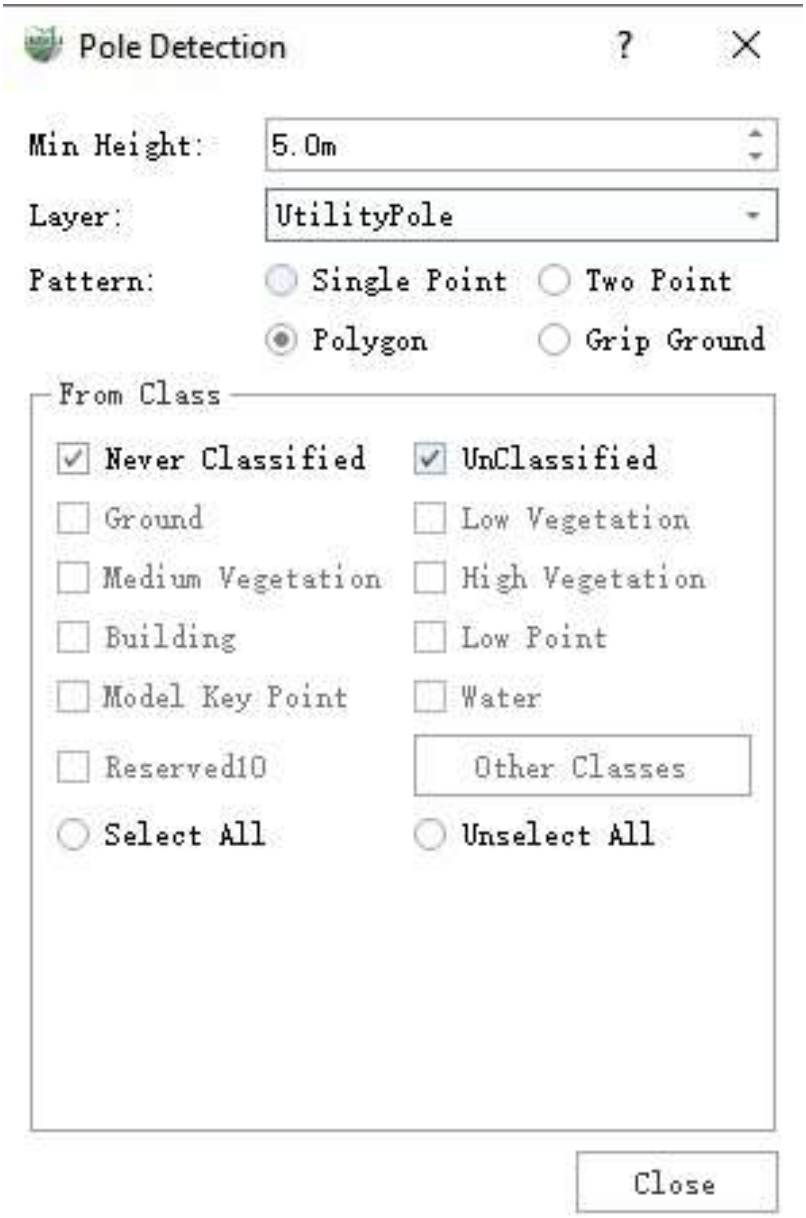
> The generated rod is **precisely perpendicular** to the ground, while the real rod is **not necessarily perpendicular** to the ground due to the **instrument scanning unilateral, tilt itself, thick at the bottom and thin at the top**, etc.

Steps

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

2.The **Pole Detection** setting dialog box should pop up:

- Polygon mode.



Polygon Mode Dialog box

Parameter description:

- **Min Height**: The default is 5.0 meters, pole-shaped objects **below** set height **will not** be extracted, at the same time, if the height is set too low , **it is easy to cause the shape of trees similar to the rod mistakenly extracted, recommended 5.0 meters**.
- **Layer**: The default is street light, the options are: **Street light, telephone pole**, and also custom layers to add a line layer, indicating the layer where the vectorization results are deposited.
- **Mode**: Default is **Single Point Mode**, which indicates the operation mode of **Tap Key Point**.
- **Source category**: Set the input category for category filtering, if the data has been classified, you can select only the point cloud category where the rod is located to improve detection accuracy.

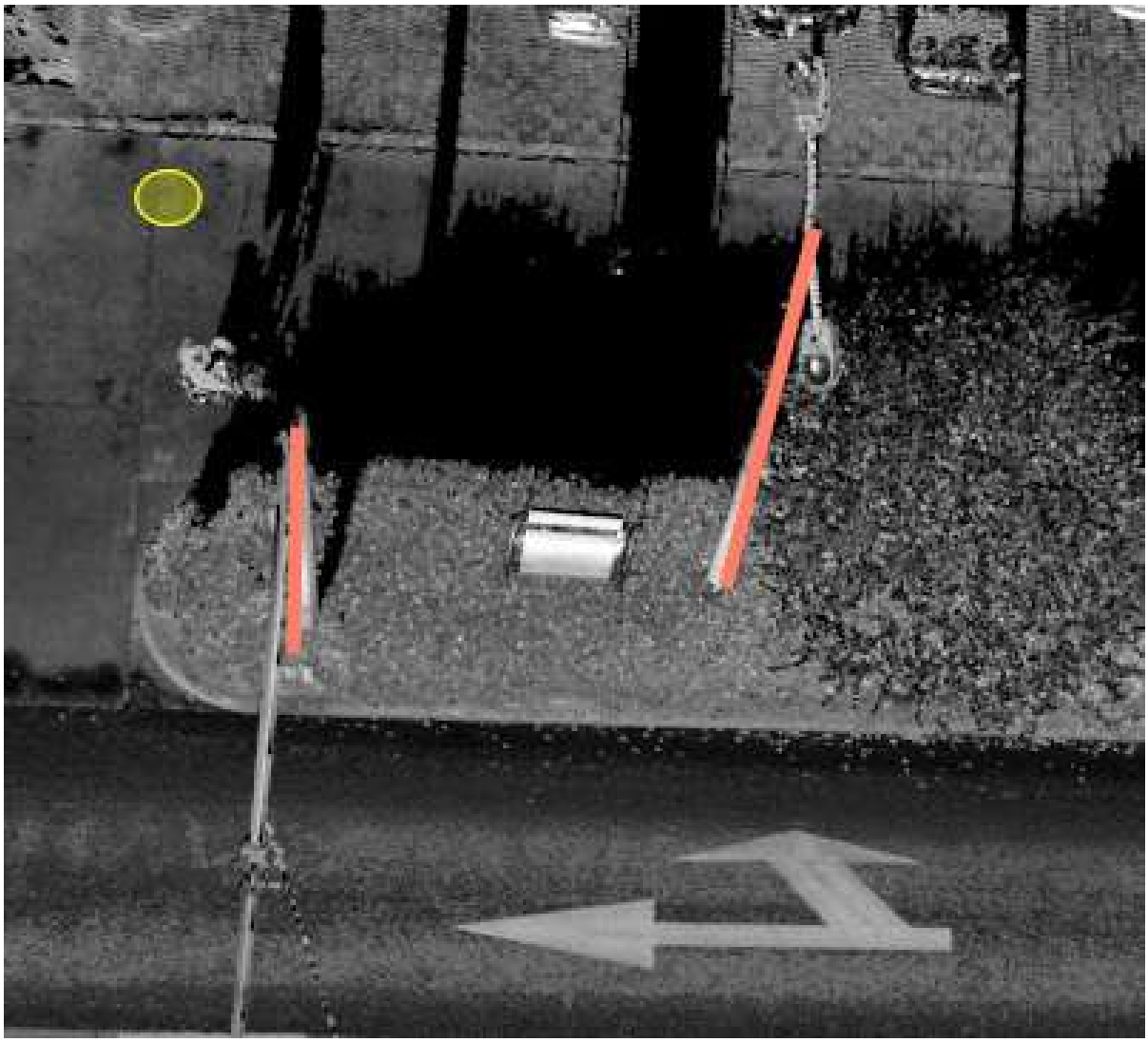
3.**Click the left mouse button in order to select the polygon area where the rod is located, double click the left mouse button to end** polygon area box selection, and automatically start **multiple rod detection**, the details are as shown in the figure, and the **yellow point** is the **key point**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

> **When the box is selected, try to narrow down unnecessary areas to speed up the extraction process**.

Click on the key point

4.Wait for the **detection completion pop-up window** or **detection failure pop-up window** pop-up, that is, complete **polygon mode rod automatic extraction**.

> When the boxed polygon area, **no** or **not detected** any rod, the **Detection Failure** pop-up window will appear, and the **Detection Complete** pop-up window will appear for the rest.



Detection completed

Polygon mode effect

Grip Ground Mode

Function Description: The grip point mode of pole detection. To apply, manually click on any point on the pole, and then click on the ground point, according to the set height of the rod, the rod vectorization result is generated.

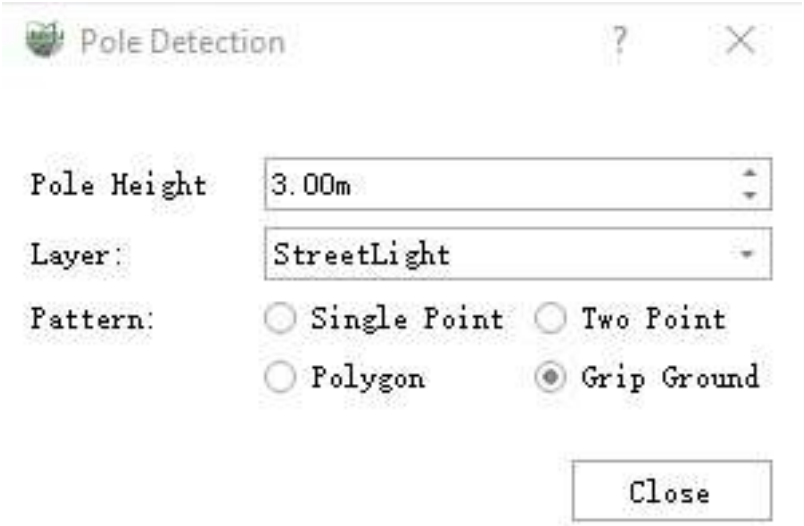
> The generated pole **rises from the clicked ground point, with a precisely set height**.

Steps

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

2.The **Pole Detection** setting dialog box should pop up.

- Grip mode



Grip mode

Parameter description:

- **Pole Height**: The height of the generated pole. In grip mode, the height is set, and the subsequent pole heights are generated according to that height.
- **Layer**: The default is street light, the options are: **streetlights, telephone poles**, and also custom layers to add a line layer, indicating the layer where the vectorization results are deposited.
- **Mode**: The default is **single point mode**, which means the operation mode of **Clicking on key points**.

3.**Click any point on the pole with left mouse button**, the details are as shown in the figure, the **yellow point** is the **key point** of mouse click.

Click on the key point of the lever

4.Click **Surrounding ground points with left mouse button**, the details are shown in the figure, **Yellow point** is the key point of the mouse click.After clicking, a three-dimensional vector line will be generated according to the height of the pole set in advance.

Click on the ground point

Renderings

Detect PowerLine

Function Description: Multiple power lines are clicked at once, and they grow automatically at the same time to generate power line vectors.

> Mainly used for simultaneous growth of multiple parallel power lines.

Steps

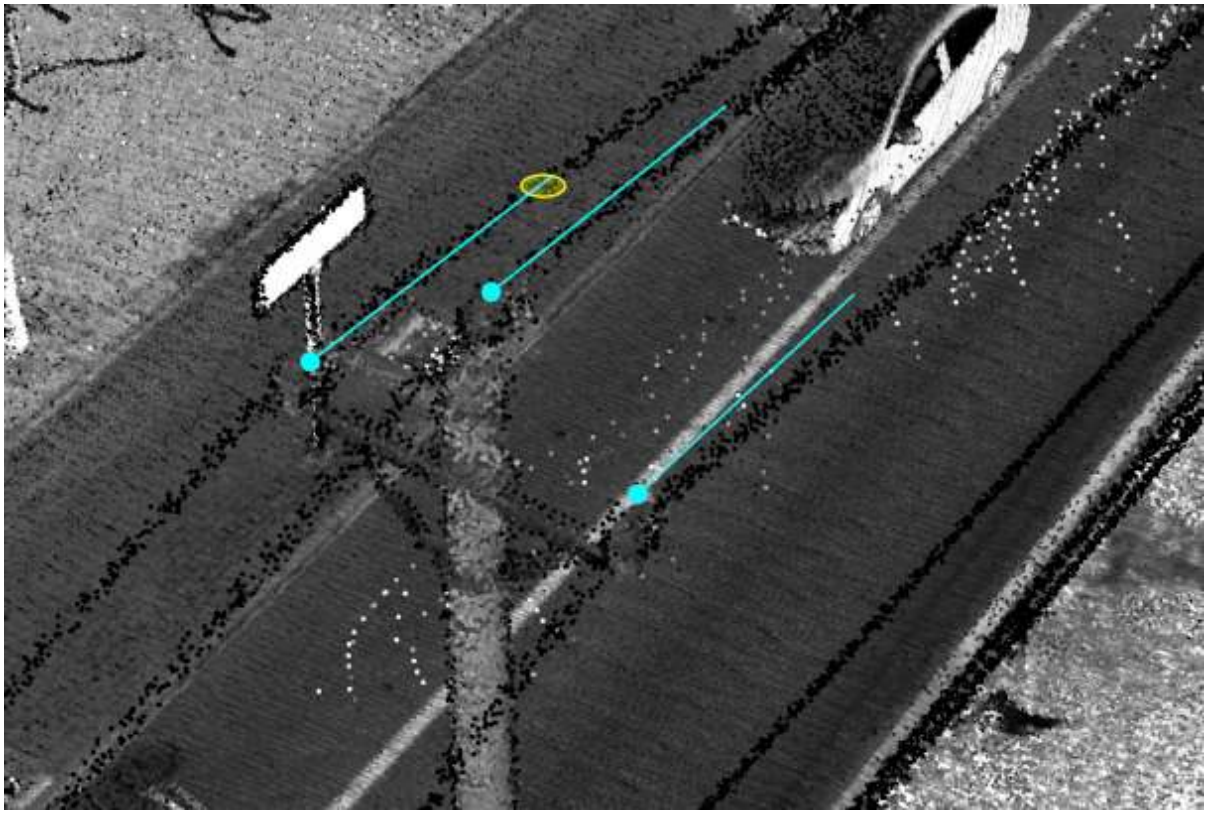
1.In the Road Facilities panel click on the drop-down logo on the right and select **Detect PowerLine** button.

2.Click **The starting point of the growth of each power line with left mouse button**, the details are shown in the figure, double-click at the starting point of the last power line to end the click.

Click on the key point

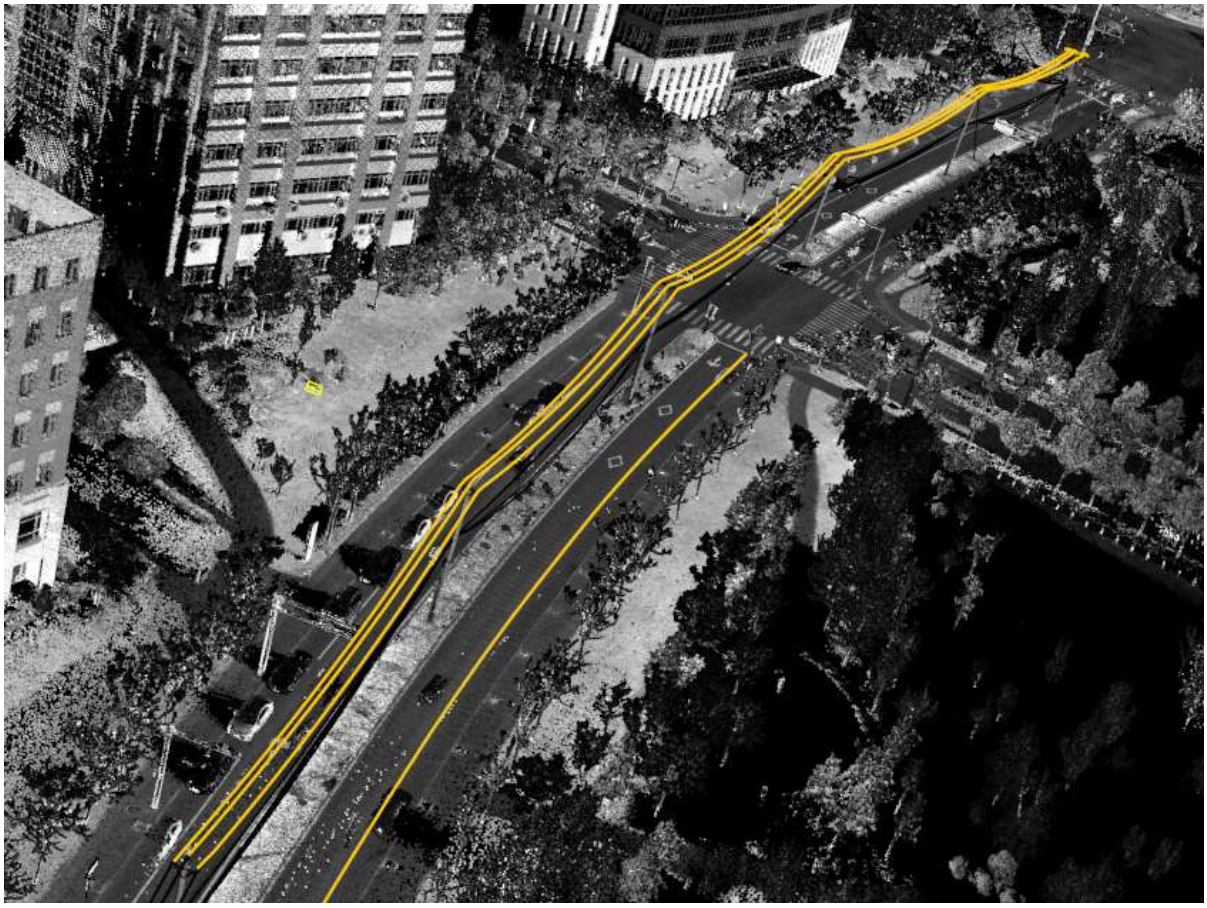
3.Click the mouse to get the growth direction of the power line.

> Just click on the direction of the first power line, and the directions of the other power lines will be automatically parallel.



Click on the growth direction

4.After clicking on the growth direction, it will automatically start to grow



Click on the growth direction

Add TrafficSign

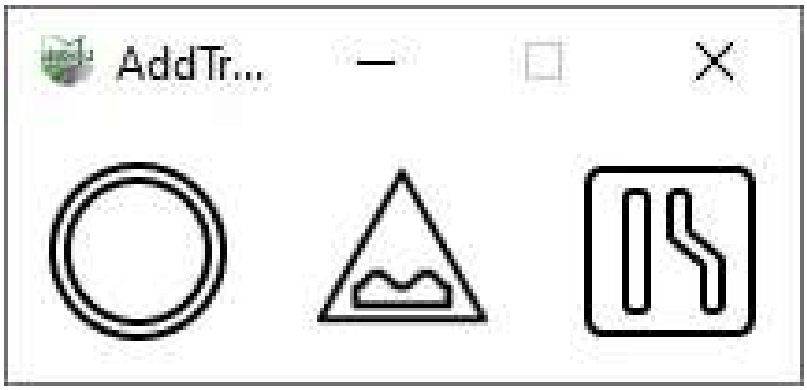
Function Description: This tool can be used to manually add traffic signs.

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

Steps

1.In the Road Facilities panel, click on the drop-down sign on the right and select **Add TrafficSign** button.

2.The **Settings dialog box** should pop up. Select the shape the user wants to draw:



Dialog box

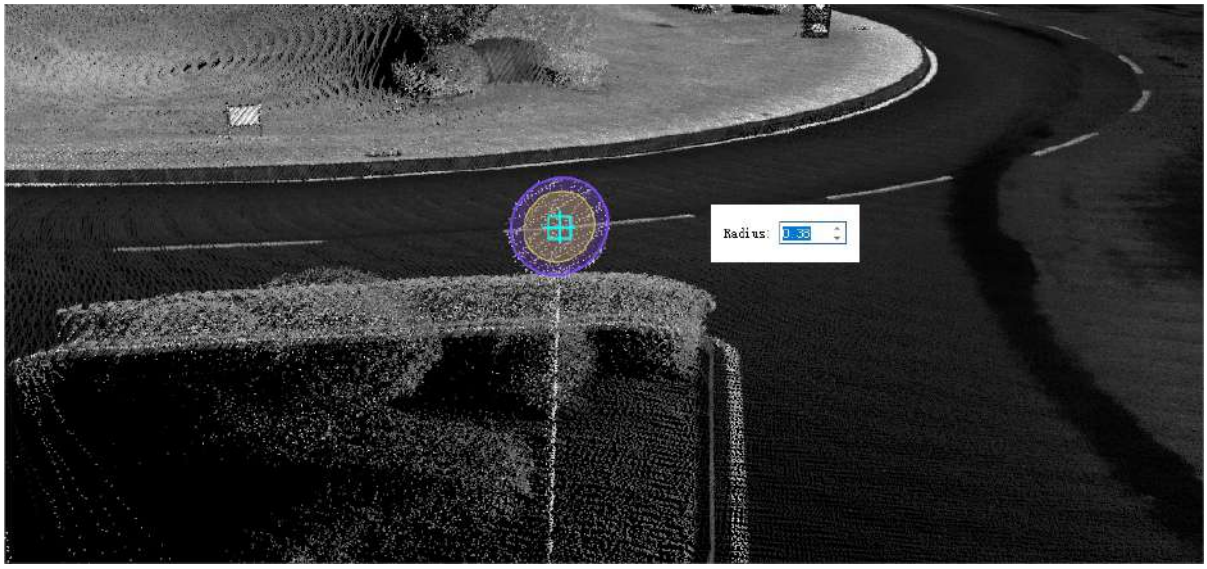
Parameter description:

- In order: round signs, triangular signs, rectangular signs.

3.Circle and triangle sign drawing: After selecting the shape to be drawn, the graphics to be drawn will be displayed on the mouse. Place the mouse on the center of the sign point cloud, use the shortcut keys to rotate, use the right mouse button to bring up the dialog box to adjust the size, **Double-click** to determine the current sign.

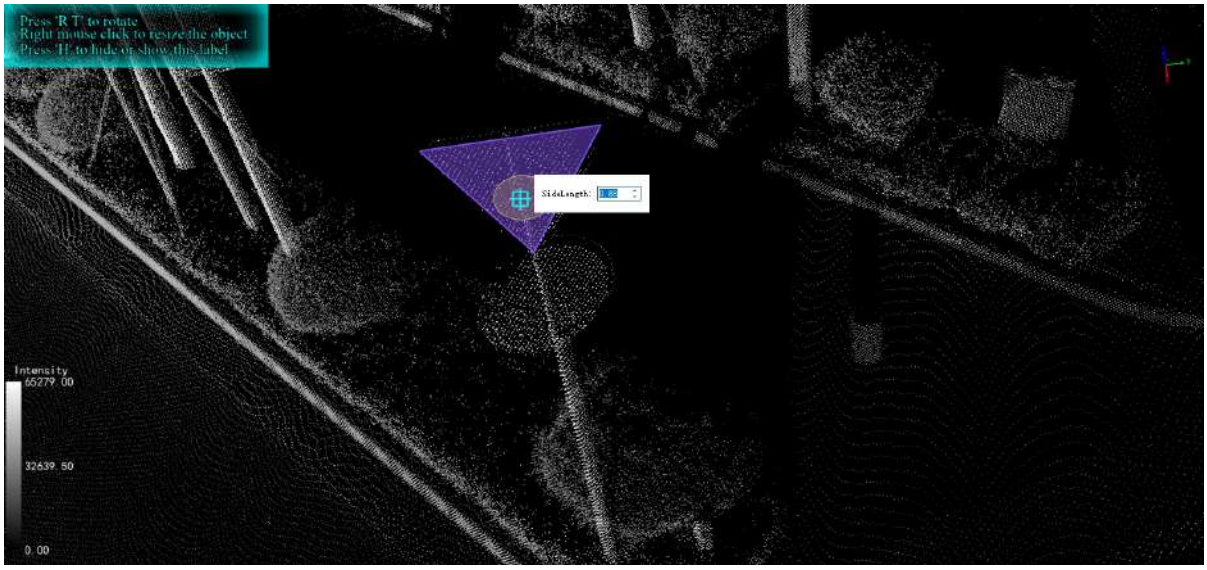
Shortcut key name	Function
R/r	Rotate counterclockwise
T/t	Rotate clockwise
Right mouse button	Change the size of the vector template

Round sign



Round

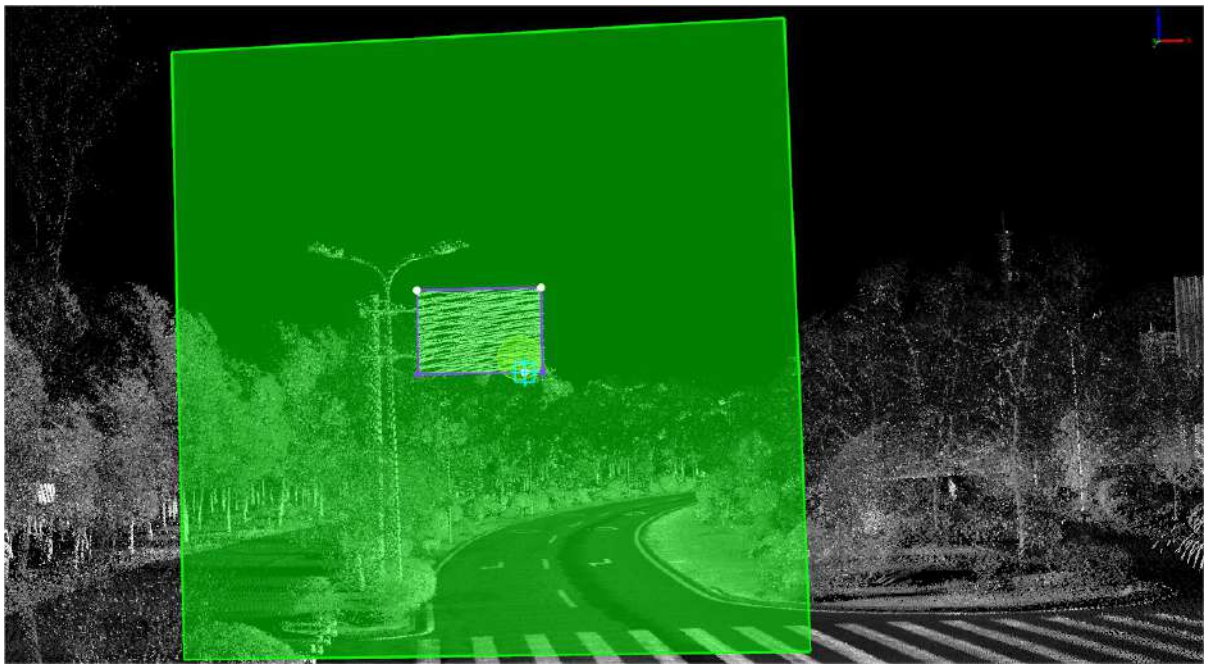
Triangle sign



Triangle

4.Rectangular sign drawing: Turn on the point cloud fitting function, click with the left mouse button on the point cloud plane of the road sign to generate a fitting plane, and continue to use the left mouse button to draw a rectangle with three points on the plane to draw a rectangle.

Rectangular sign



Rectangle

Add Parking Spot

Function Description: Manual generation of parking space vectors in two modes with and without common edges.

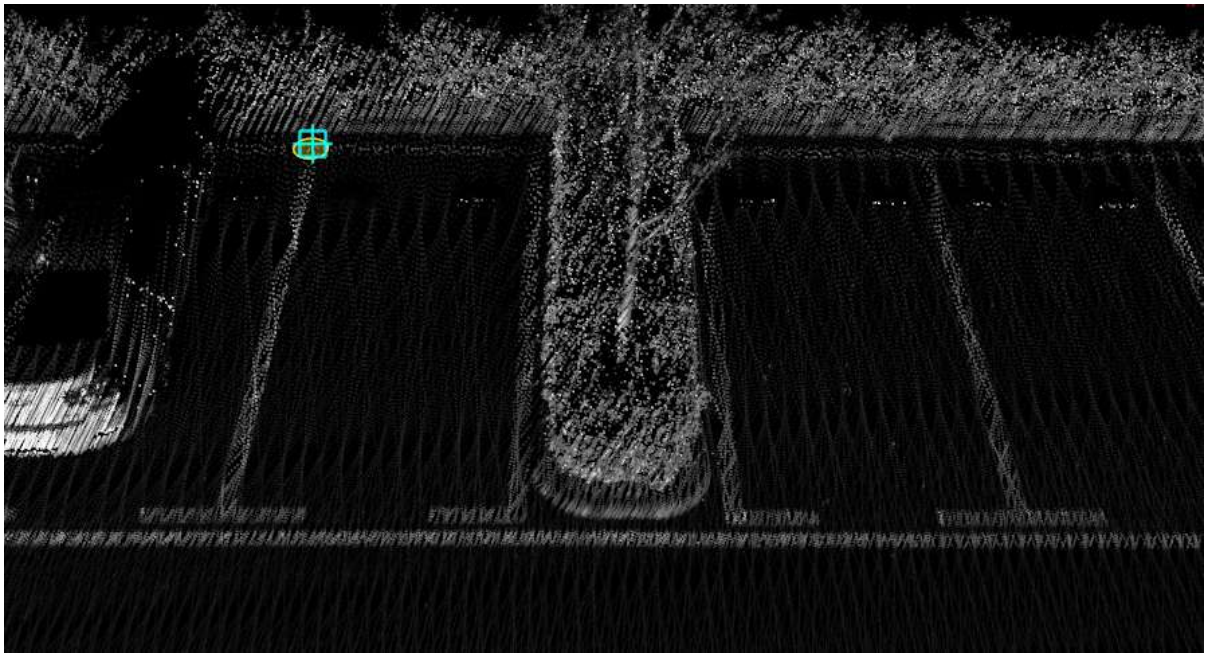
> For parking spaces with a communal edge, use a 4-point sub-pattern; for parking spaces without a communal edge, use a 5-point sub-pattern.

Steps

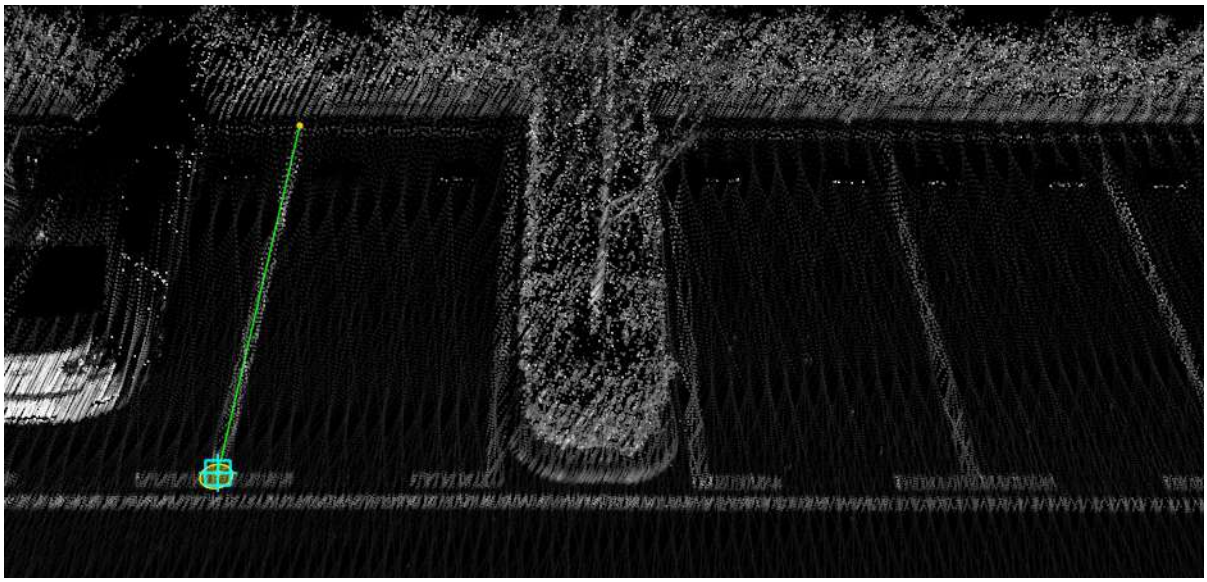
1.In the Road Facilities panel click on the drop-down sign on the right and select **Add Parking Spot** button.

No communal side parking:

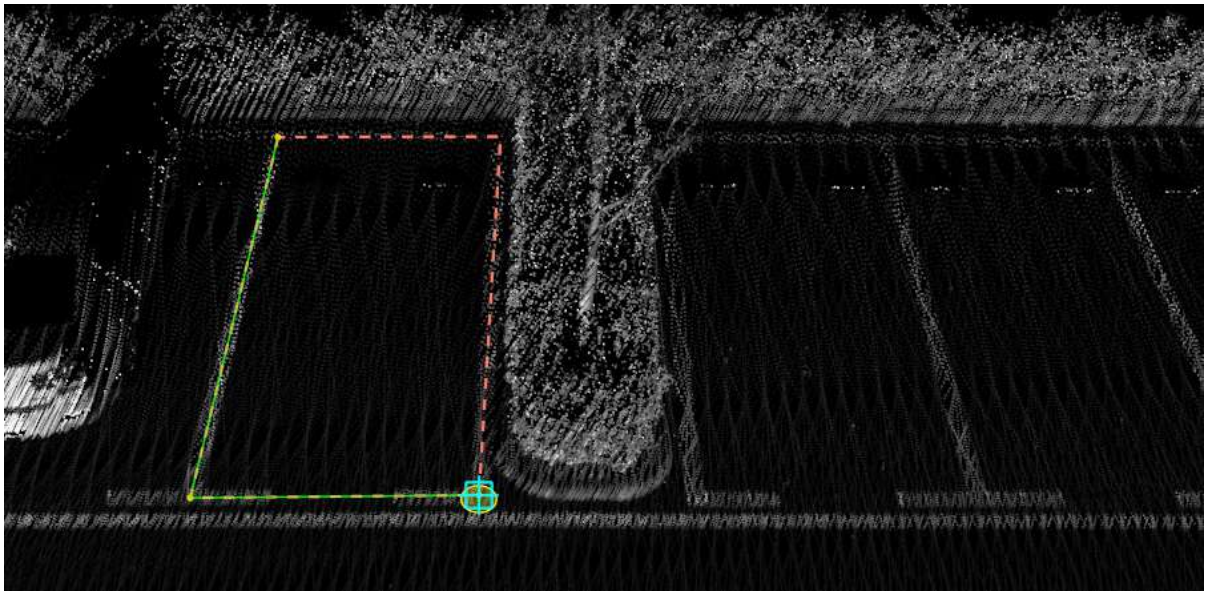
2.The function is activated and the default is 5-point mode, i.e. no common edges by default. **left mouse button** to tap the three corner points of a single parking space in turn for determining the size of the parking space



Key points

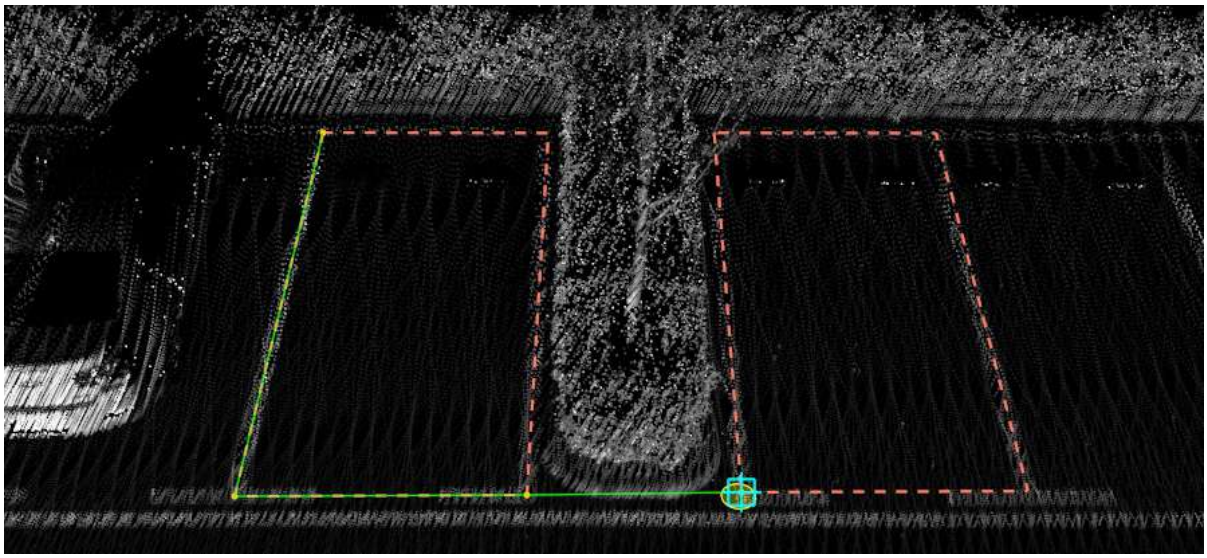


Key points



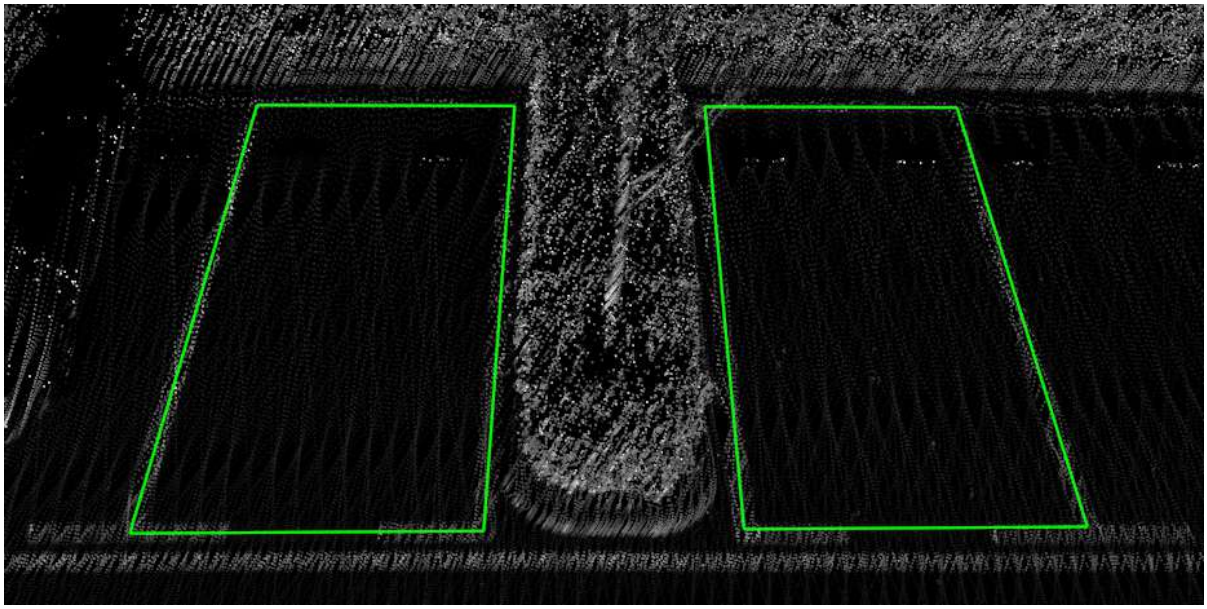
Key points

3.Move the mouse to see the second parking space move with the mouse, **left mouse button** tap the corner of the next parking space to determine the distance between the two parking spaces, special attention needs to be paid to the location of the tap



Key points

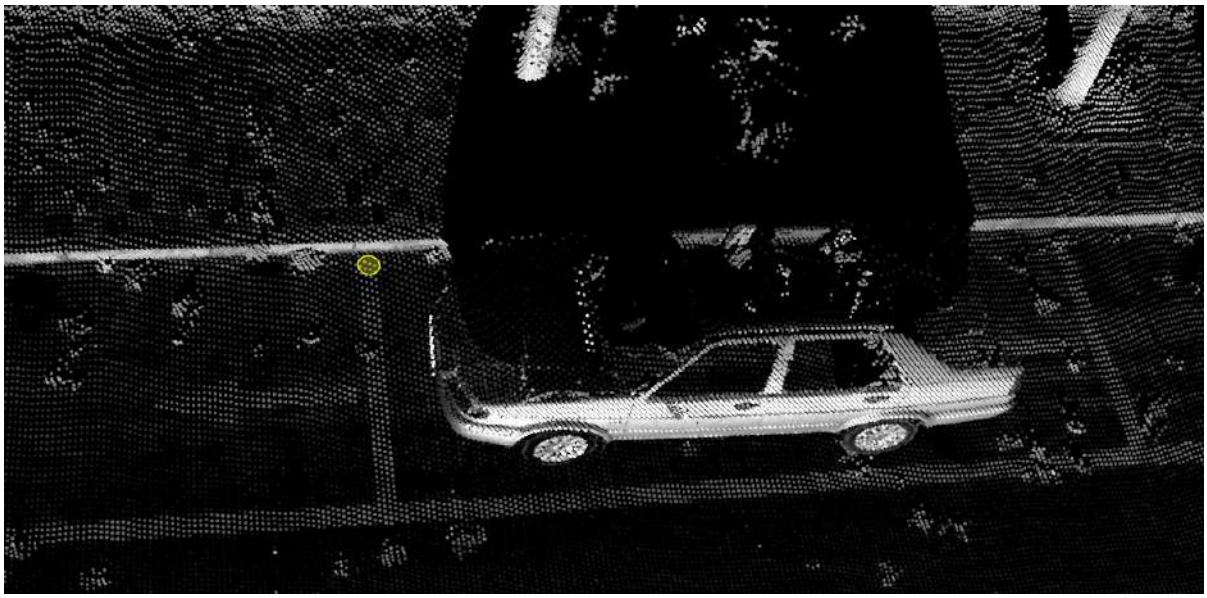
4.Move the mouse to automatically update the number of parking spaces in real time according to the size of the individual parking spaces and the spacing between them **double click** the fifth key point to complete the production



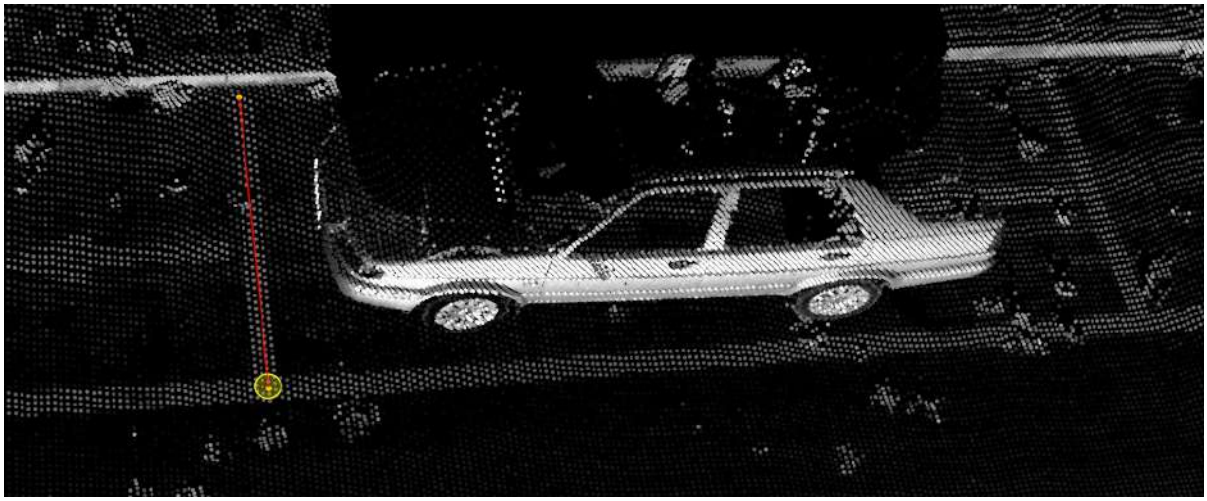
Key points

Communal side parking available

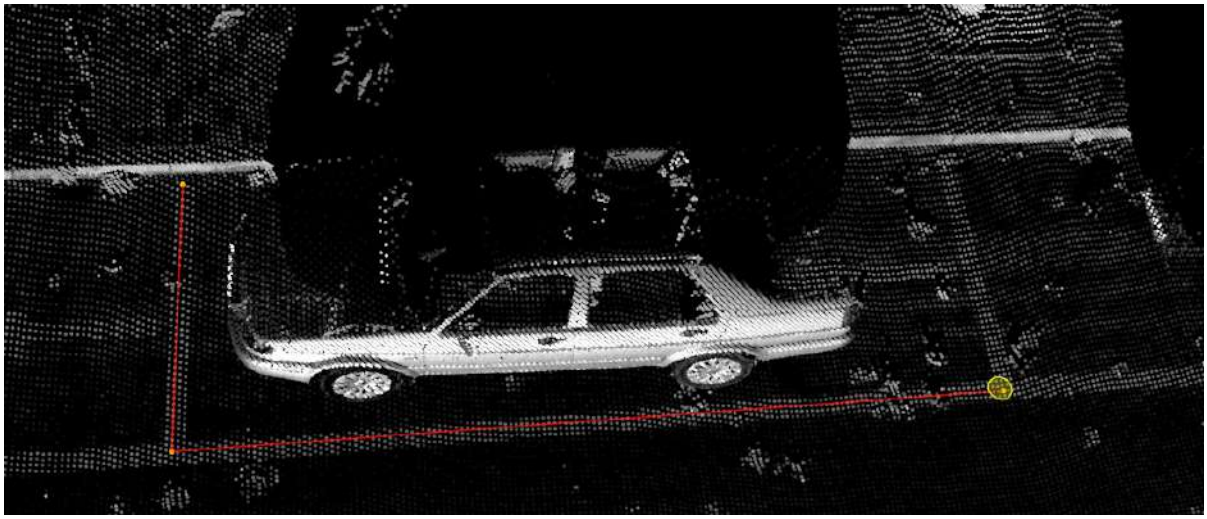
2.**Left mouse button** to select each of the three corner points of a single parking space



Key points

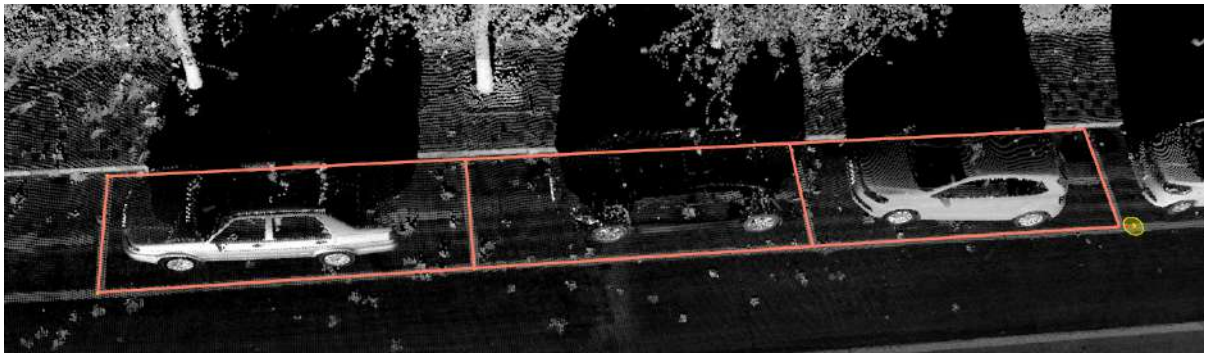


Key points



Key points

3.After three clicks to identify a single parking space, move the mouse and the number of spaces will be automatically updated in real time according to the length of the second key point and the fourth key point, **double click** the fourth key point to complete the production



Renderings

Add Rain/Strainer

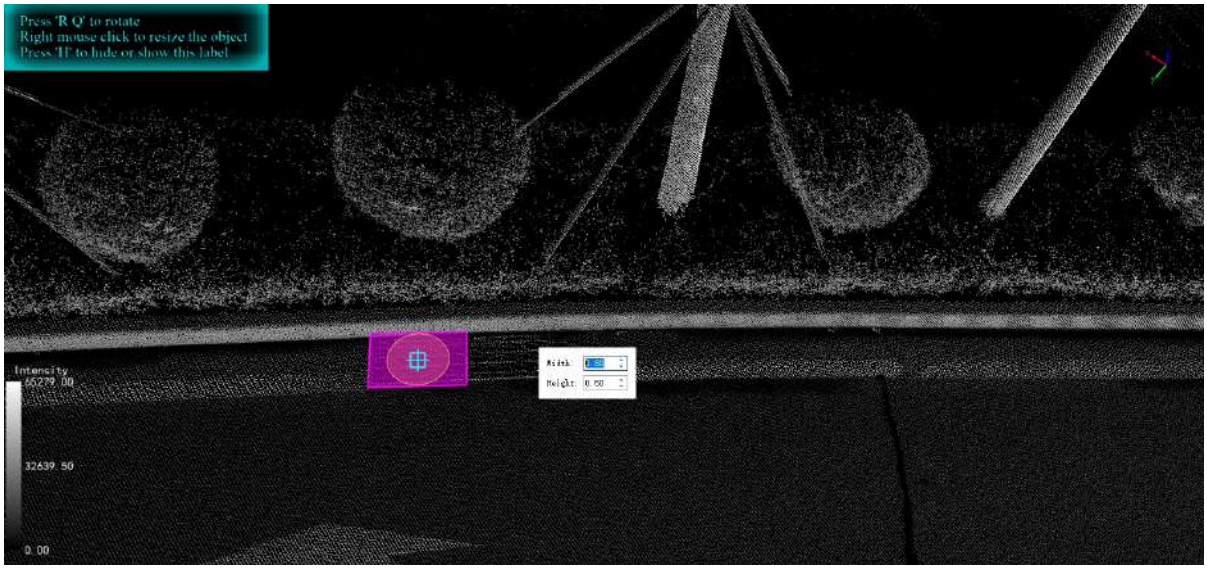
Function Description: This tool can be used to manually add a drain grate.

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

Steps

1.In the Road Facilities panel click on the drop-down sign on the right and select **Add Rain/Strainer** button.

2.The graphics to be drawn will be displayed on the mouse in real time. Place the mouse at the center of the point cloud, use the shortcut key to rotate, use the right mouse button to bring up the dialog box to adjust the size, and **Double-click** to determine the current vector.



Renderings

> Note: Shortcuts:

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

Add Manhole

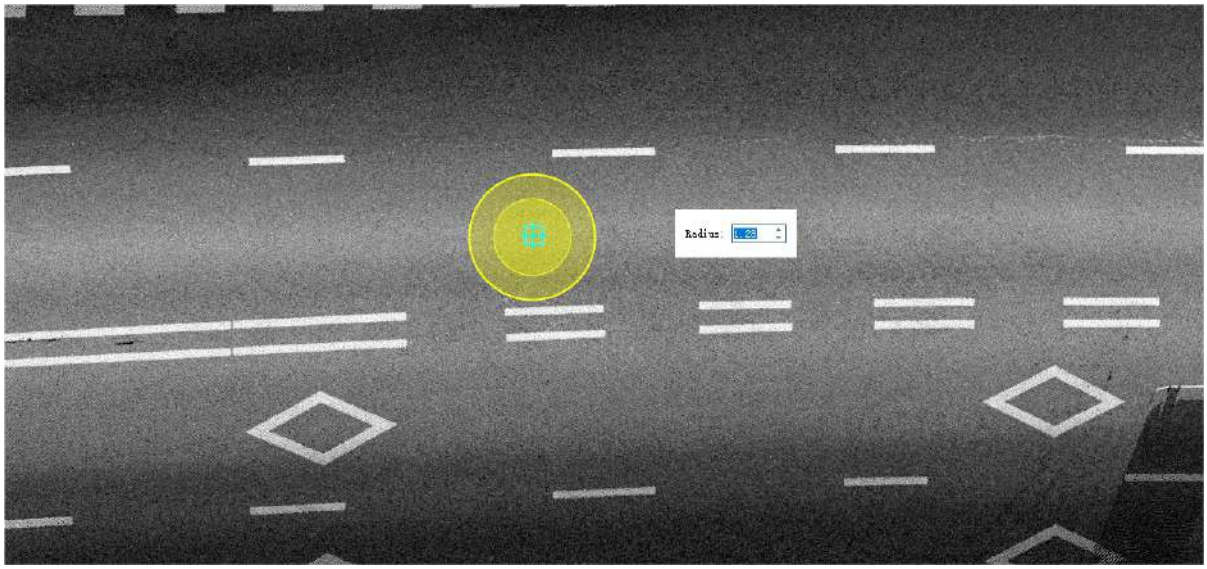
Function Description: This tool enables manually add manhole covers.

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

Steps

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

2.The graphics to be drawn will be displayed on the mouse in real time. Place the mouse at the center of the point cloud, use the shortcut key to rotate, use the right mouse button to bring up the dialog box to adjust the size, **Double-click** To determine the current vector.



Renderings

> Note: Shortcuts:Right mouse click to resize the object

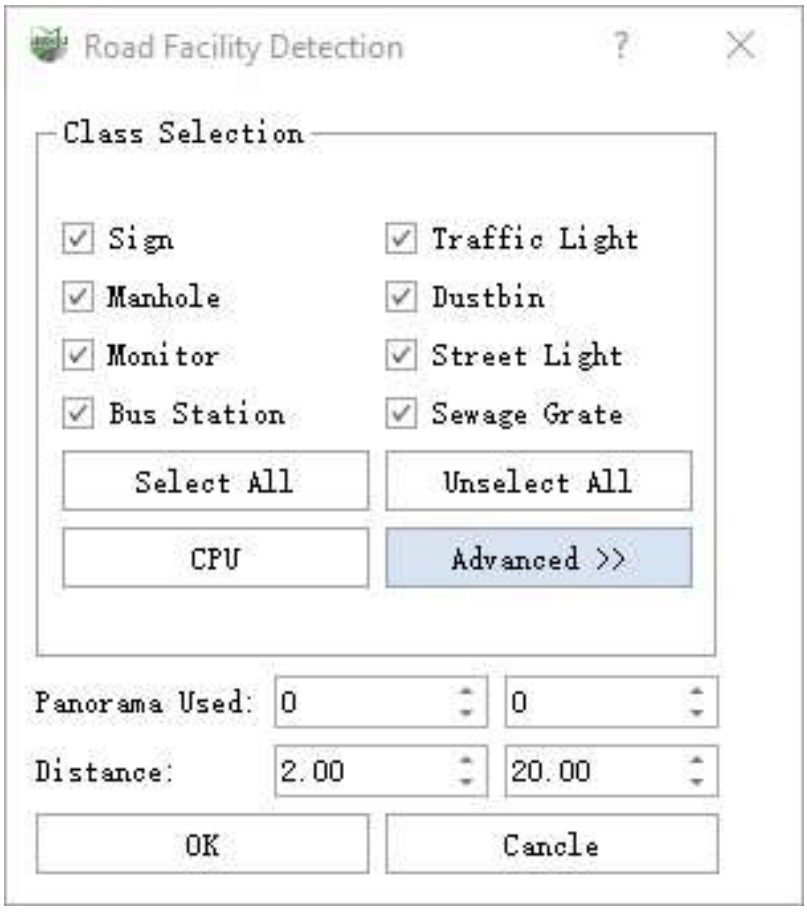
Batch processing of road facility recognition based on panoramic images

Function Description: This tool enables the use of panoramic images to assist in the automatic identification of road facilities. Among them, **traffic signs** reflect the results with plane-like elements (plane-like vectors), and **The rest** reflect the results with point elements (point-like vectors).

> Note: This function is only valid for engineering data with panoramic images.

Steps

1.Click **Road Facility Detection** button and the parameter dialog box should pop up.



Parameter dialog box

Parameter description:

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

  > The ticking options are proportional to the time-consuming, that is, the more the user checks, the longer the entire processing process will take. Please tick them according to the actual situation.

- **Select all/cancel all selections**: Click the button to check all supported categories/uncheck all.

- **Pattern**:**CPU**、**Pattern**:**GPU**:For details, please refer to Classify by Deeplearning to set the operation mode.

  > GPU mode requires computer hardware and software environment support, if the conditions are not met, even if the checkbox is checked, it will still run in CPU mode.

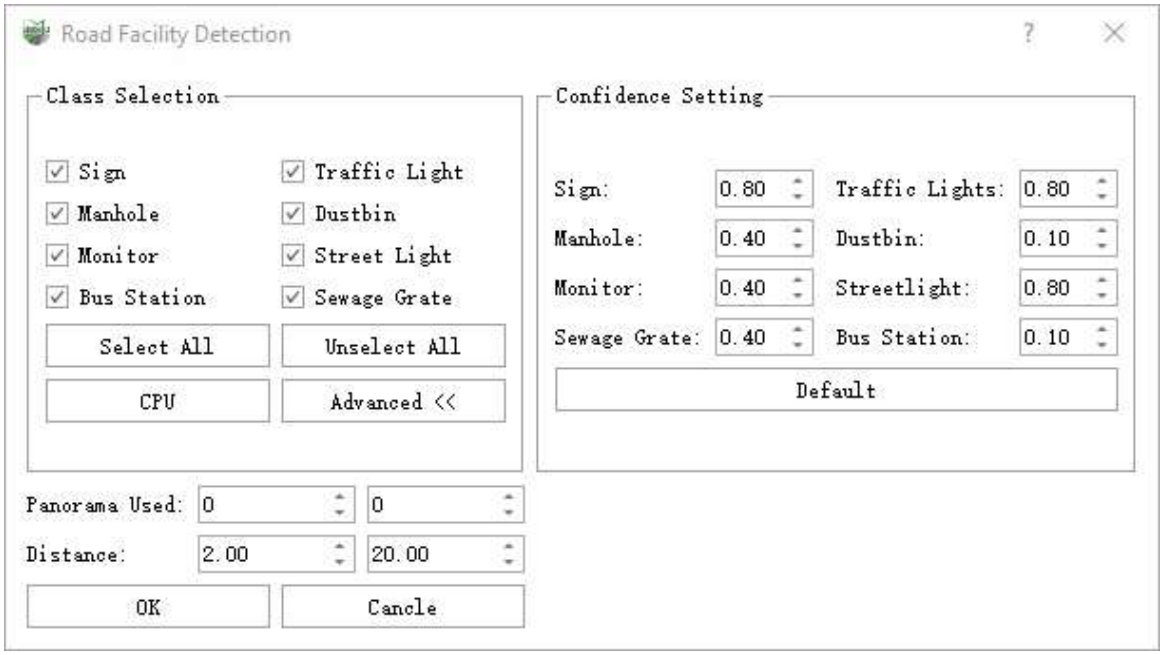
- **Use image**: Used to determine the picture number used in this processing, the default starts from 0, and the ending with the last picture in the project.

  > Can be used in conjunction with the selection frame, fill in the start and end numbers of the image where the area of interest is located, and the precise start and end image numbers will greatly reduce useless calculations.

- **Distance**: The distance between the target and the center of the camera, less than the shortest distance or greater than the maximum distance, will not be extracted, **the default minimum distance is 2.0 meters, and the maximum distance is 20.0 meters**.

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

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

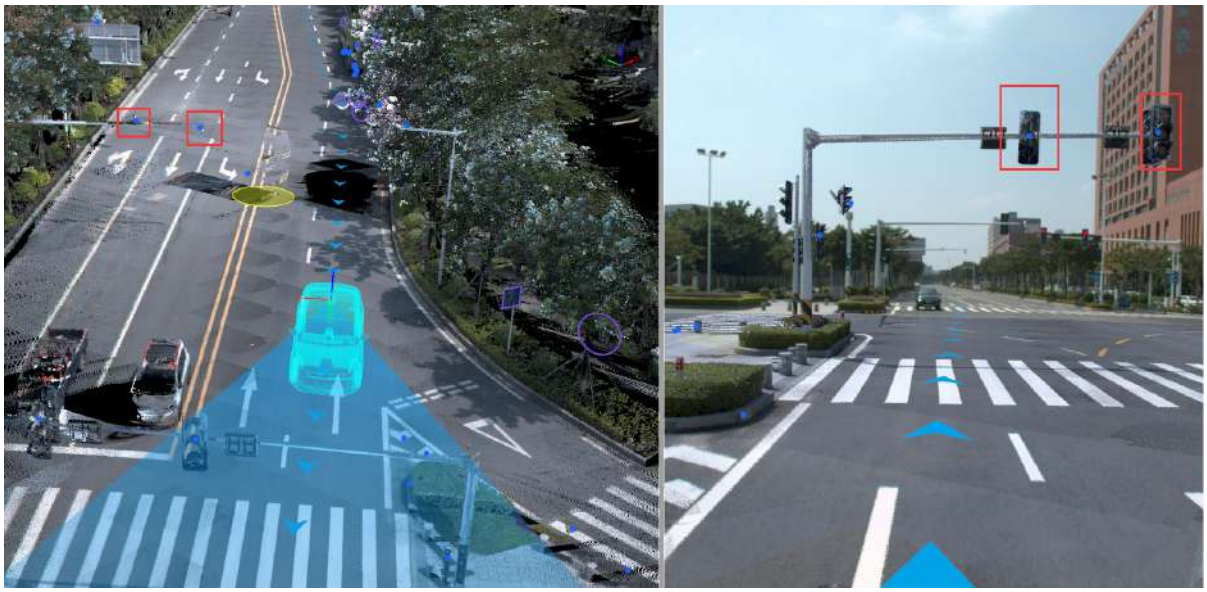


More dialogs

  - **Confidence Setting**: For the categories supported by the current function, set the corresponding confidence level separately. And the targets below the set confidence level will be ignored.
  - **Default**: This button is used to set the confidence level of all categories to the default value.

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

2.Click OK for automatic detection.



Detection results

Linear Facility Extraction

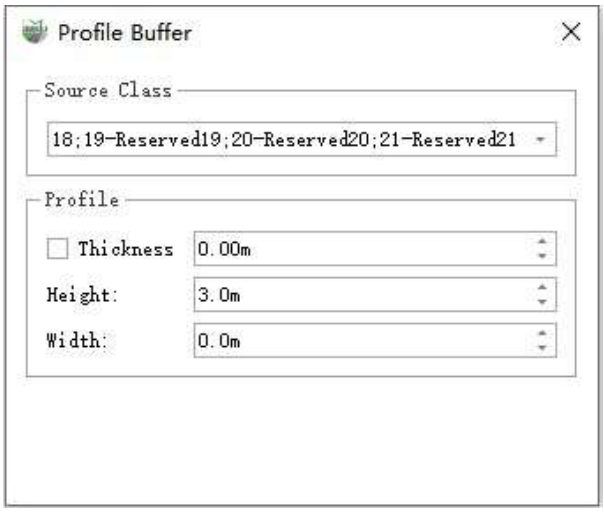
Function Description: This tool can be used to create and edit a template. Strip object template extraction is divided into template selection and editing, and banded object extraction. The following will be described in detail.

Description

- **Template selection and editing**: Including template selection, template point cloud, key point editing and template saving.
- **Strip object extraction**: Draw a path based on the selected template, and then perform detection and extraction use manual or semi- automatic methods.

Template editing and selection operations

- **Template View**: The template view is located on the left side, used to display the template point cloud and key points for the current application.
- **Template Selection**:When the function is activated, the program automatically loads the template file into the template selection box. Users choose a template that matches the detected object (paying attention to the template point cloud orientation). After setting up the template, the software automatically copies the template file to the user data folder (AppData), and subsequently reads template data from this folder.
- **Edit**: Enable and disable template editing. When template editing is enabled, the detection function button becomes unavailable. It is restored after closing template editing.
- **Create Template**: The interaction logic is the same as the profile. Draw a template point cloud with an appropriate thickness perpendicular to the direction of the strip-shaped object. The drawing result will be displayed in the left template view.
- **Profile Settings**: Used to set the point cloud category and size within the profile, achieving precise control over the template point cloud.
    - **Class**: Set the category points for generating the template.
    - **Thickness**: Display the length in the vertical direction of the profile. When checked, it indicates that the thickness is locked.
    - **Height**: Profile height, with the line as the starting height, and equal portions above and below set height.
    - **Width**: Draw the horizontal distance between two points.



> Pay special attention to the adjustment of 'Height,' for example, with railings. When ground points are not within the profile range, adjusting the 'Height' ensures that ground points fall within the profile, ensuring effective extraction.

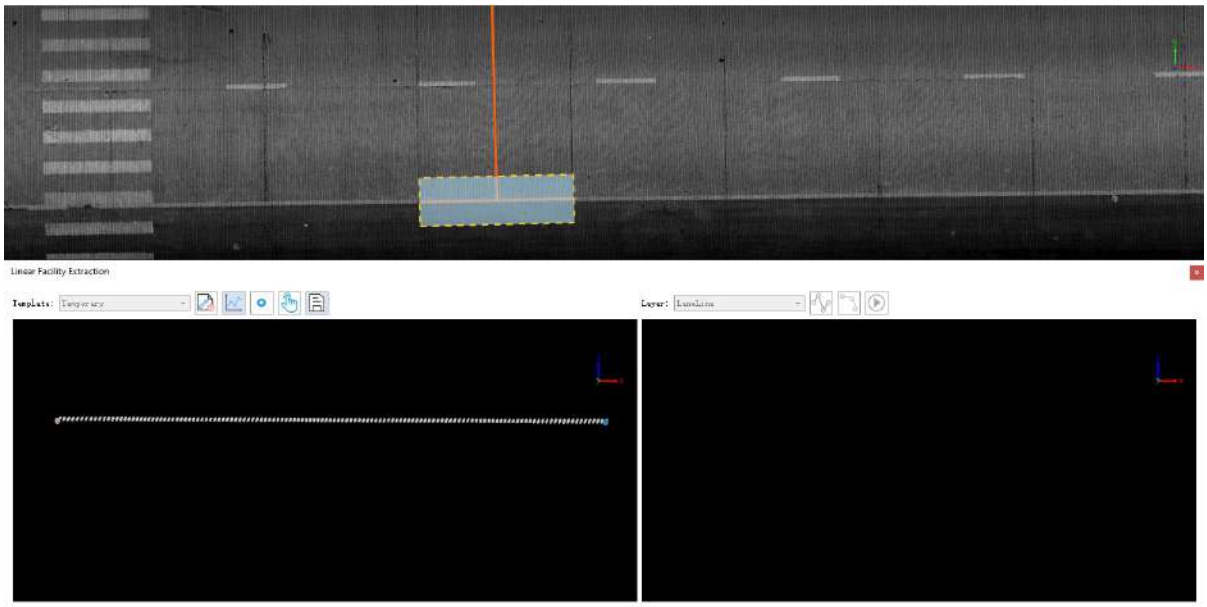
- **Add key points**: After creating the template point cloud, select key points on the template point cloud to serve as the starting points for the growth of vector lines for strip-shaped objects.
- **Selection key points**: Used for key point editing. After selection, use the 'Delete' key to remove key points. Drag with a long left-click to modify the position of the key point.
- **Save Template**: Temporarily drawn templates are placed in the 'Temporary' section. After saving the template, it can be used in the next extraction.
- **Delete Template**: Delete the current non-'Temporary' template.

Procedure:

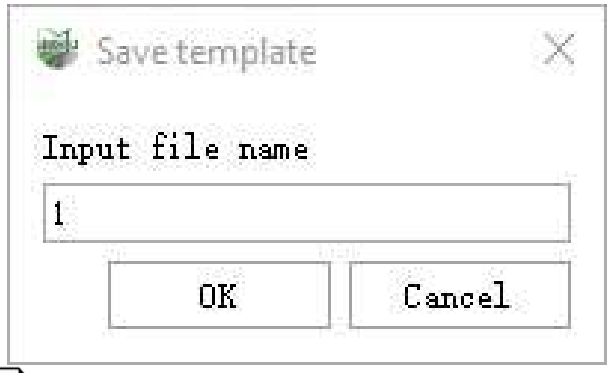
1.Click the **Linear Facility Extraction** button in the road facilities panel to enter edit mode.

2.Click the **Start Edit** button to enter template editing mode.

3.Click the **Draw Template** button, select the profile to draw in the point cloud, and add Key Point 1 and Key Point 2.



4.Click the **Save Template** button, a dialog box will appear for saving the template. Set the file name and click 'OK'.



5.Click the Fi**nish Editing** button to close the editing.

Strip Feature Extraction Operation

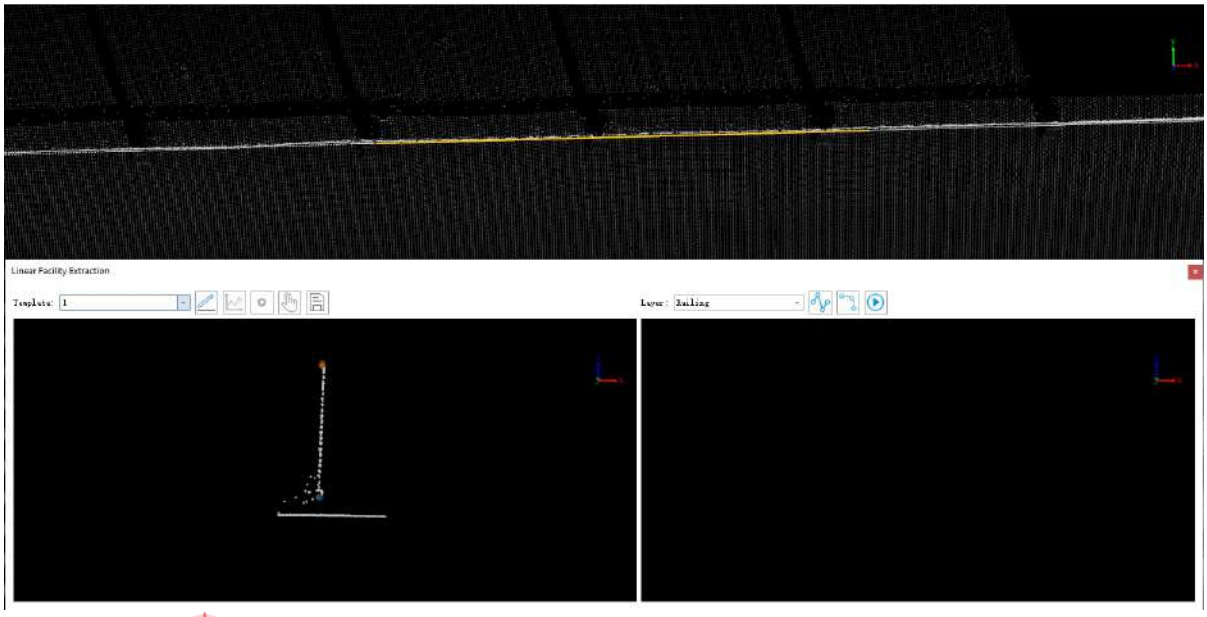
- **Result View**: The result view is located on the right side, used to display the results of each extraction, and supports adjustment of key points in the extraction results.
- **Layer Selection**: Select the result saving layer (line layer).
- **Draw a straight line**: After activation, draw a straight line in 3D view along the strip-shaped object. The first segment of the line is used to provide the initial detection direction. Extraction starts from the second point, and the extraction result is displayed in the right result view.
- **Draw an arc**: Draw an arc path, suitable for curved objects. Supports the 'S' shortcut key to switch between 'Line' and 'Arc' shapes, right-click to go back to the previous point.
- **Automatic Detection**: After completing the drawing of a line segment path (two points), automatic detection can be initiated every 1.5 seconds. If the detection results do not meet the requirements, you can use the spacebar to stop detection, use the right mouse button to backtrack to points that do not meet the requirements, or manually adjust the extraction point positions in the result view. Then, press the spacebar again to continue forward detection. After detection is complete, press Enter to finish.

Procedure:

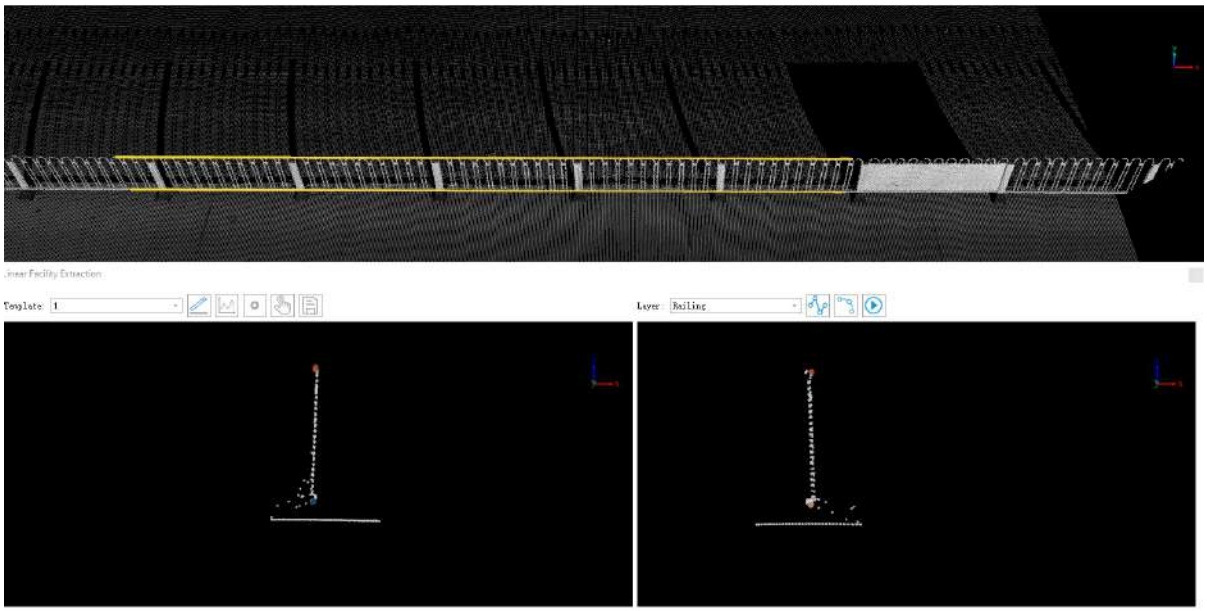
1.In the left dropdown box for strip feature extraction, select a template. In the right layer dropdown, choose the layer you need.

2.Click the **Line** button, draw the vector line of the object to be extracted in the point cloud. Right-click to undo a node. Click the **automatic detection** button for automated detection.

> You can also choose the **Arc** button according to your actual needs.



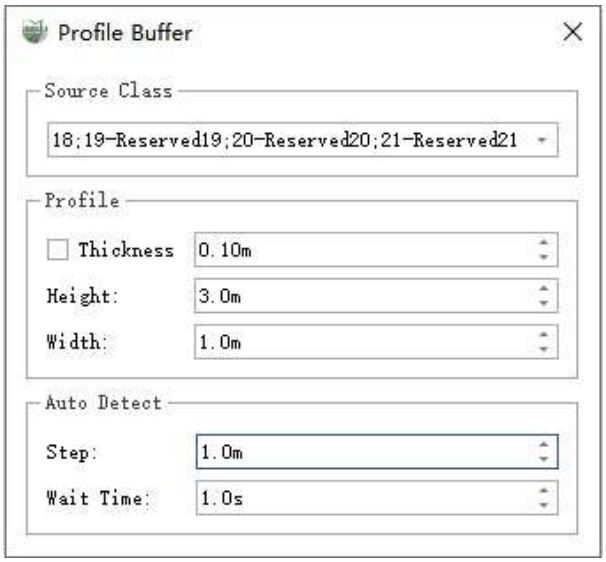
3.Click the **Stop** button to end automatic detection. Double-click with the left mouse button to finish editing.



Strip Feature Extraction Results

Important Notes

- **Profile point cloud**: First, try to ensure that the size and geometric features of the template point cloud are similar to the extraction point cloud. Second, try to include ground points.
- **Template Switching**: During the extraction process, you can switch templates for continuous detection, but the number of key points in the two templates must be the same.
- **Category and Profile Size**: Used to adjust the point cloud involved in the extraction.
- **Step**: Set the automatic detection forward step length. A shorter step length results in finer detection, and it should be reduced at turns.
- **Wait Time**: Wait time between automatic detections of two frames, used for manual adjustment of extraction results.



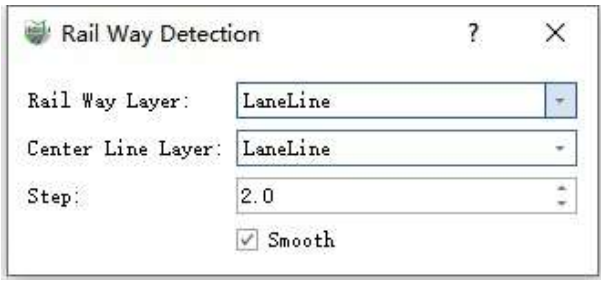
Detect Railway

Function Description：Automatically detect railway tracks along a line based on seed points and direction vectors, and generate a centerline.

Steps

1.In the road facilities panel, select the **Detect Railway** button.

2.Dialog Box：



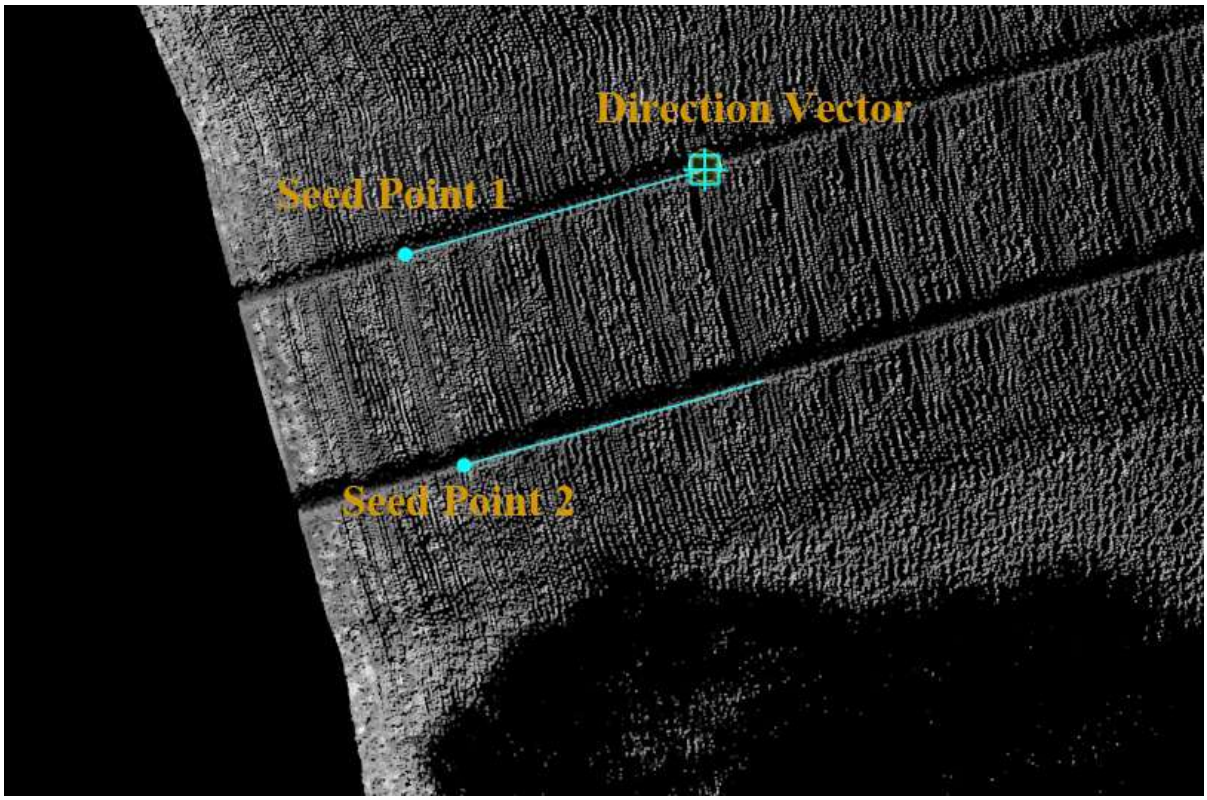
Dialog Box

Parameter Settings

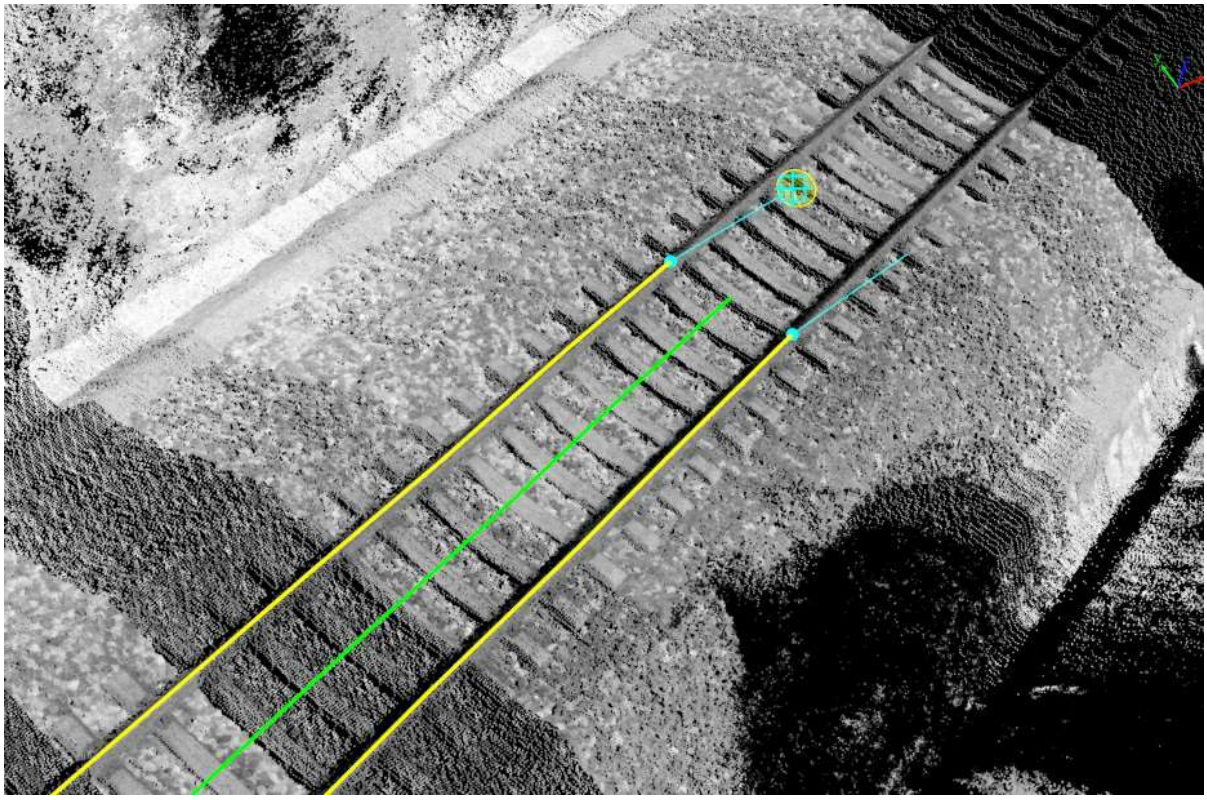
- **Rail Way Layer**：Layer to Store Railroad Vector
- **Center Line Layer**：Layer to Store Railroad Centerline.
- **Step**：The detection step during the detection process. A longer step increases efficiency but may result in less accurate fitting. A shorter step provides better fitting but may lead to slower efficiency. Recommended step: 2.0 meters.
- **Smooth**：Checking this option will apply smoothing to the vector results in the x, y, and z directions. It is recommended to check this option.

Shortcut Key Name	Function
Space	Interrupt Growth
Enter	Confirm Growth Results

3.Left-click to select the two growth starting points of the railroad, as shown in the details in the picture. After selecting the growth direction, growth begins automatically.



Click on Key Points/p>



Click on Key Points

Pole Seed Editor

Function Description：This feature supports adding, deleting, editing, and batch fitting seed points for pole objects. It also supports seed point updates and ground alignment.

Seed Pole Conversion

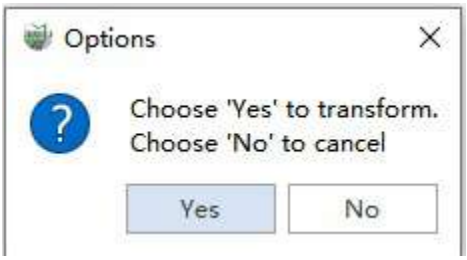
Function Description：This is used to convert general vector points in the 'pole' layer into 'pole' individual points, allowing for individual segmentation and parameter measurement operations.

Functional Description

This functionality is used to convert externally imported or manually drawn general vector points into individual points, enabling individual segmentation and parameter measurement operations.

Steps

1.Click the **Pole Seed Editor** button, and the following popup window will appear. Select 'Yes' to complete the conversion of all general vector points in the 'pole' layer.



Updata

Function Description：It supports updating new seed points after deleting existing seed points, with individual points already present.

Pole Seed Editor

Function Description：This feature supports adding, deleting, editing, batch fitting seed points, updating seed points, and ground alignment.

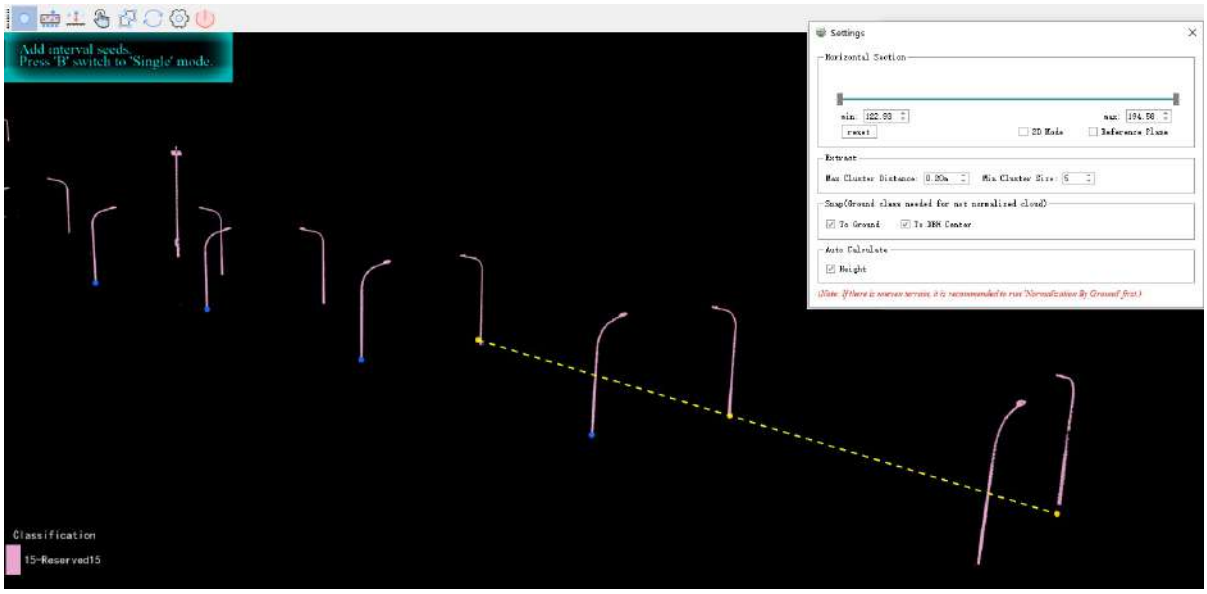
1.First, click the **Seed Point** button.

2.In the 3D window, you can see options for adding seed points 、batch extracting seed points、global extract 、moving seed points 、polygonal selection of seed points 、 seed point drape to ground 、updating seed points 、settings 、and closing .

Furthermore, by default, the dialog boxes for section settings and georeferencing settings will pop up. It is the default behavior that when adding seed points, drape to ground is performed, especially when the point cloud has already undergone ground point classification. This way, when performing subsequent seed point fitting, the DBH (Diameter at Breast Height) positions will be accurate. Therefore, it is recommended to check the drape to ground option.

（1）The addition of add seed points supports two modes: single-point mode and equidistant mode, which can be switched using the 'B' (or 'b') key. By default, it's in single-point mode, where you can directly add seed points for the current pole at suitable locations on the trunk.

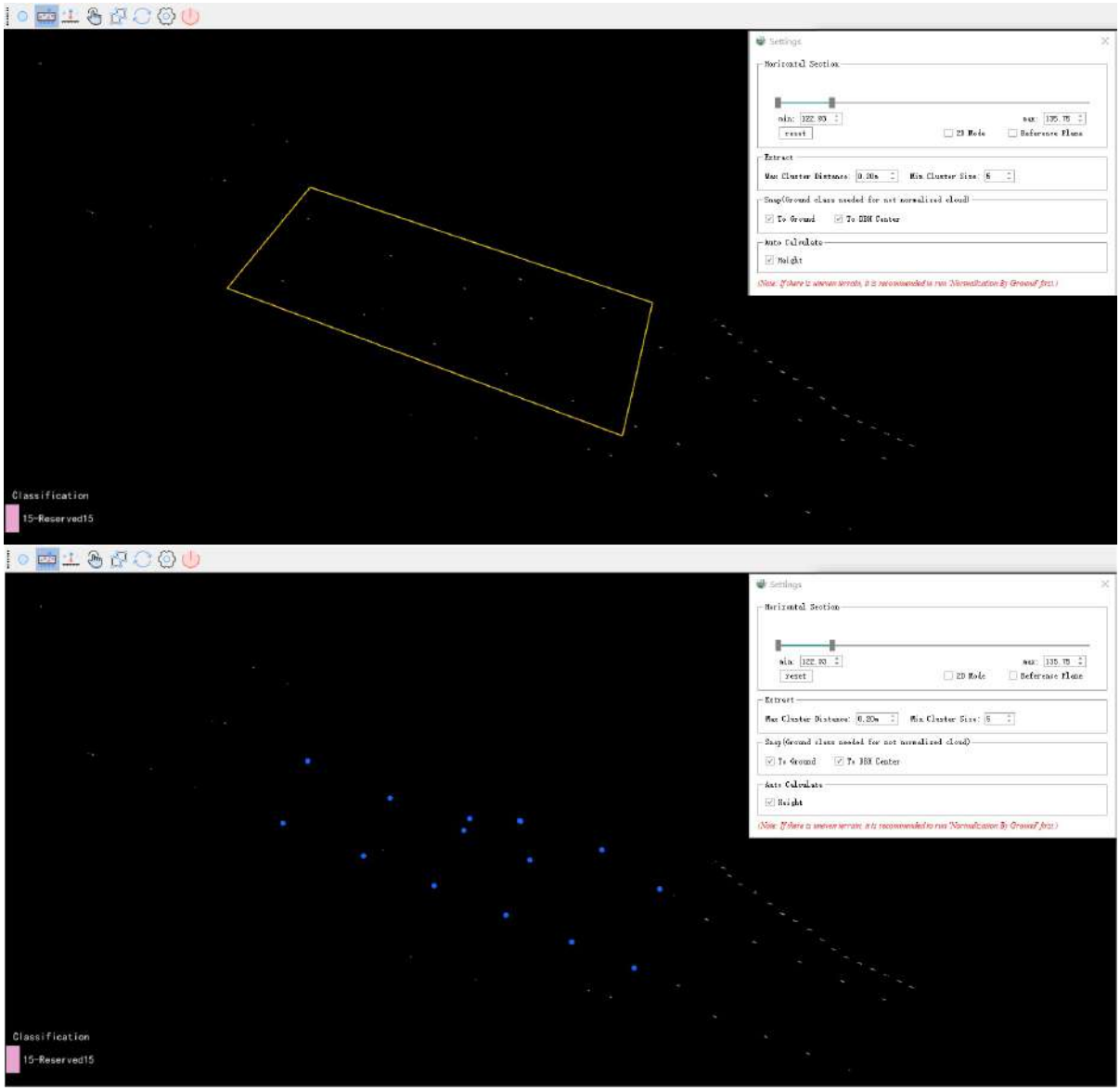
To switch to equidistant mode, press the 'B' key. In this mode, you'll start by clicking an initial seed point on the trunk of the first pole, which serves as the starting point. As you move the mouse, you'll see a dashed line indicating the distance between two poles. Then, select a seed point on the trunk of the second pole, defining the desired spacing. While moving the mouse, you can see in real-time how well the template matches the pole trunk. When you believe the fit is satisfactory, double-click the mouse to complete the batch addition of seed points.



Add seed points

（2）The batch seed point extraction function is more suitable for normalized data and can be used in conjunction with the horizontal cross-section tool. First, adjust the point cloud slicing to a suitable range, for example, setting it between 1.2 to 1.5 meters. Then, select the area to be extracted by drawing a box around it and double-click to confirm the selection. The software will automatically start the batch seed point extraction process.

> - Please note: If the point cloud data has been classified, it is recommended to switch to displaying by category and only show the category containing the poles when performing batch extraction. This allows you to filter out categories that do not need extraction and provides a clear view of the data.



Batch seed point

(3)The global seed point extraction functionality operates in batch extraction mode.

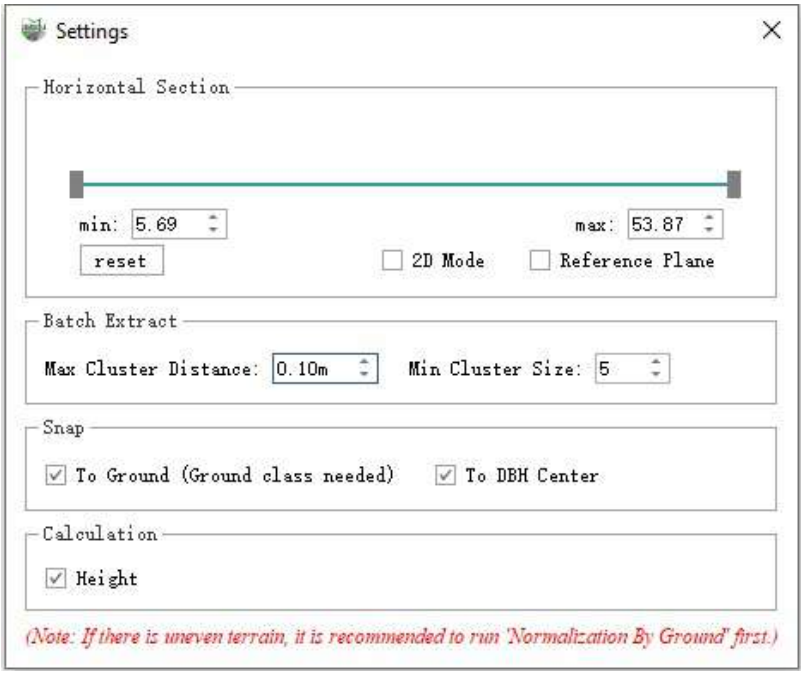
(3)Move Seed Point Function : Click on a seed point that needs modification, and then you can directly move the selected seed point's position with the mouse. You can also use 'Delete' to remove the selected seed point.

(4)Polygonal Seed Point Selection : Click on seed points that you want to select, and then use the mouse to create a selection area (box) around the desired seed points. Hold down the Ctrl key to add more seed points to the selection. The selected seed points will be displayed with their IDs in the selection dialog and will also be highlighted in the attribute table. You can directly use 'Delete' to remove the selected seed points.

(5)Select seed points for drape to ground : if the data is unclassified, ground points have not been separated, or ground points are not in category 2, the initially added seed points will not undergo ground fitting. After changing the point cloud categories, you can use the ground fitting function to select seed points that have not been fitted to the ground and perform ground fitting.

(6) The update seed points function is designed for scenarios where seed points are represented as vectors within the 'pole' vector layer. If you have cleared all the seed points from the directory pole layer, but your point cloud data has already undergone segmentation without seed points, you can use the update button. After clearing the seed points, you can choose whether to perform an update. Since the seed points have been cleared, the update will recreate the corresponding seed points for each individual.

(7) Setting Function , you to set the minimum and maximum values of the horizontal cross-section using dual sliders, or you can set custom minimum and maximum values. The 3D window will display the results in real-time, and you can reset the current settings by clicking the reset button. Checking the '2D mode' box allows you to switch between 2D and 3D modes in the 3D window, and checking the 'reference plane' box enables or disables the reference plane for the current 3D window, as shown in the diagram.



Setting

(8) Close the seed point functionality.

> Note: When adding and moving seed points, it is advisable to use the 'Hide Unselected Categories' and 'Quick Horizontal Section' features in combination for easier operation.

Object Editor

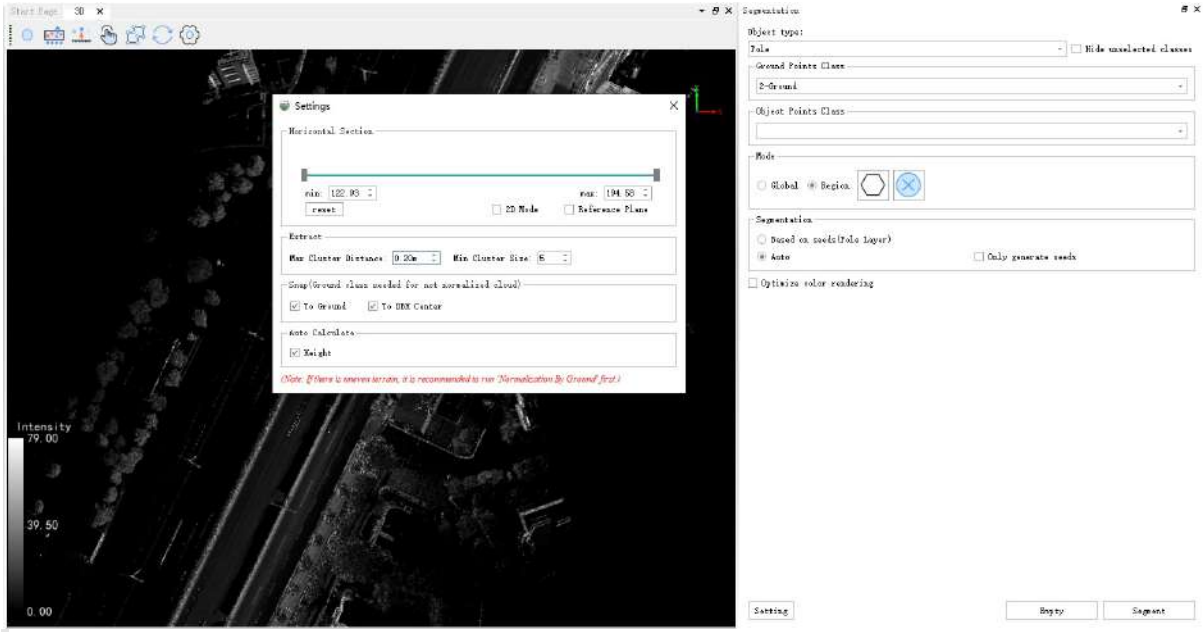
- Segmentation
- Editor
- Measurement

Segmentation

Function Description: Supports segmentation and removal of three types of individual entities: poles, vehicles, and buildings. A detailed explanation will be given below.

Steps

1.Click on the **Segmentation** button, a pop-up box will appears:



> Note: Seed point editing is reused in the segmentation function. Please refer to the specific usage.PoleSeedEditor

Parameter Description:

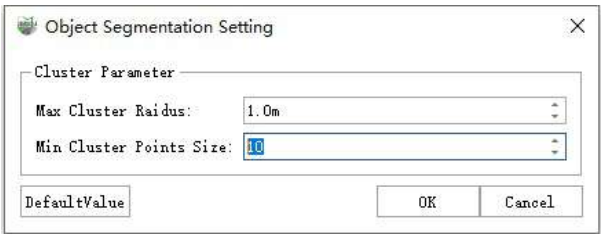
- **Object type**: Currently, there are three options: poles, vehicles, and buildings.
- **Hide unselected classes**: If checked, points from unselected categories will be hidden, which can be used in combination with any rendering method to facilitate seed point operation and point cloud observation.
- **Ground Points Class**: The category where the ground points are located (the ground point category is checked to assist segmentation and improve algorithm robustness).
- **Object Points Class**: The category where the individual to be segmented is located (for example, when segmenting 'vehicle', check the point cloud category where 'vehicle' is located).

> Tip: When adding and moving, you can combine with 'Hide Unselected Categories' and 'Quick Horizontal Section' for easier operation.

- **model**: Divided into global and Region modes.
  - **Global**: During processes such as seed point generation, segmentation based on seed points, or clearing segmentation, operations are performed on all point clouds of the set monomer point type.

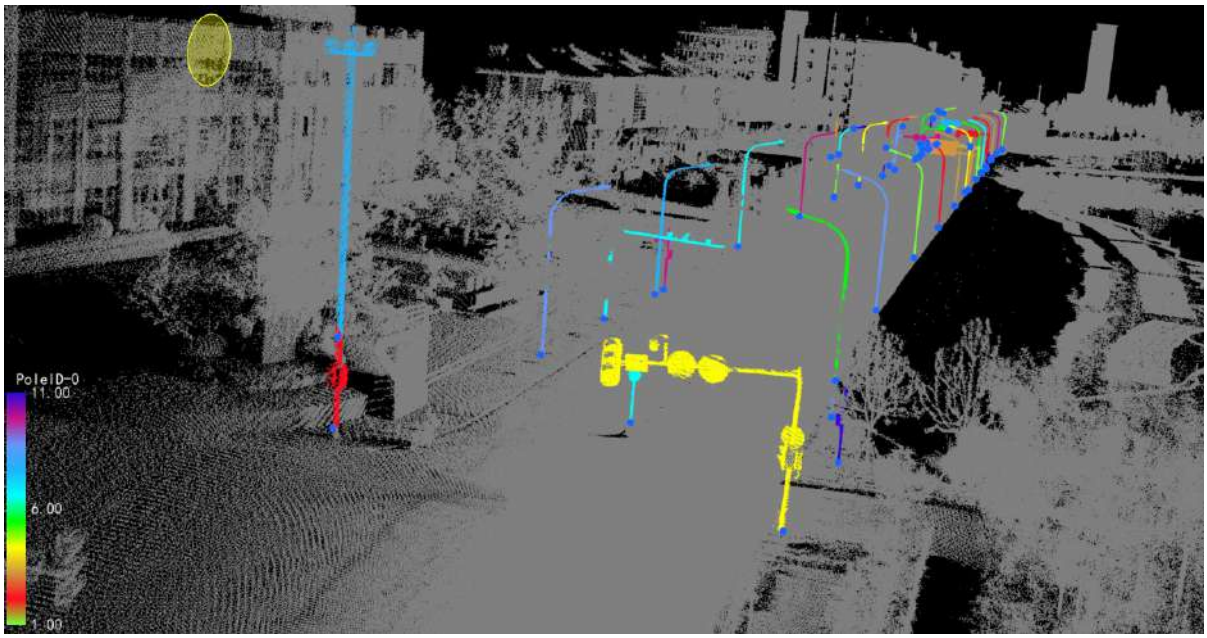
- **Region**：Using the polygon tool, you can customize the area for analysis and processing. Processes such as seed point generation, segmentation based on seed points, or clearing segmentation are performed only in the selected area.

- **Segment**：For the 'pole' segmentation process, it is divided into two steps: seed point generation and growing from seed points. The process is controlled by checkboxes. For example, if only 'Generate Seed Points' is selected, after segmentation, the result will only contain seed points. At this point, you can manually add seed points or modify their positions, suitable for scenarios where users want to perform secondary editing on seed points. Another example is, after seed points are determined, only checking 'Grow,' after segmentation, the final segmentation result is generated, ensuring the correctness of the pole
- **Optimize color rendering**：Optimize the rendering of adjacent individual trees, ensuring that the colors of adjacent individual trees are different.
- **Setting**：Used to set clustering parameters.



  - **Min Cluster Points Size**: Minimum number of points in a single cluster.
  - **Max Cluster Rai dus**: Minimum distance between clusters.

2.After clicking **Segment** the segmentation result is as shown in the figure:



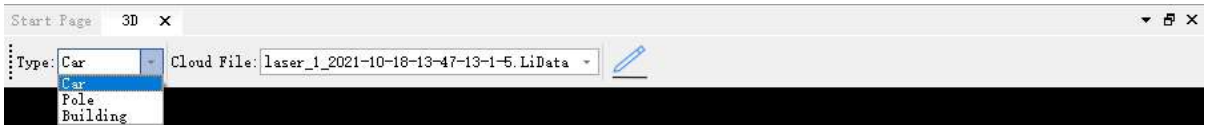
3.Click **Empty**, all objects of the specified type within the target area will be removed, and the point object ID will be reset to zero.

Editor

Function Description: Supports the manual editing of three types of entities: poles, vehicles, and buildings, including the creation, deletion, and setting of entity points. Details will be given below.

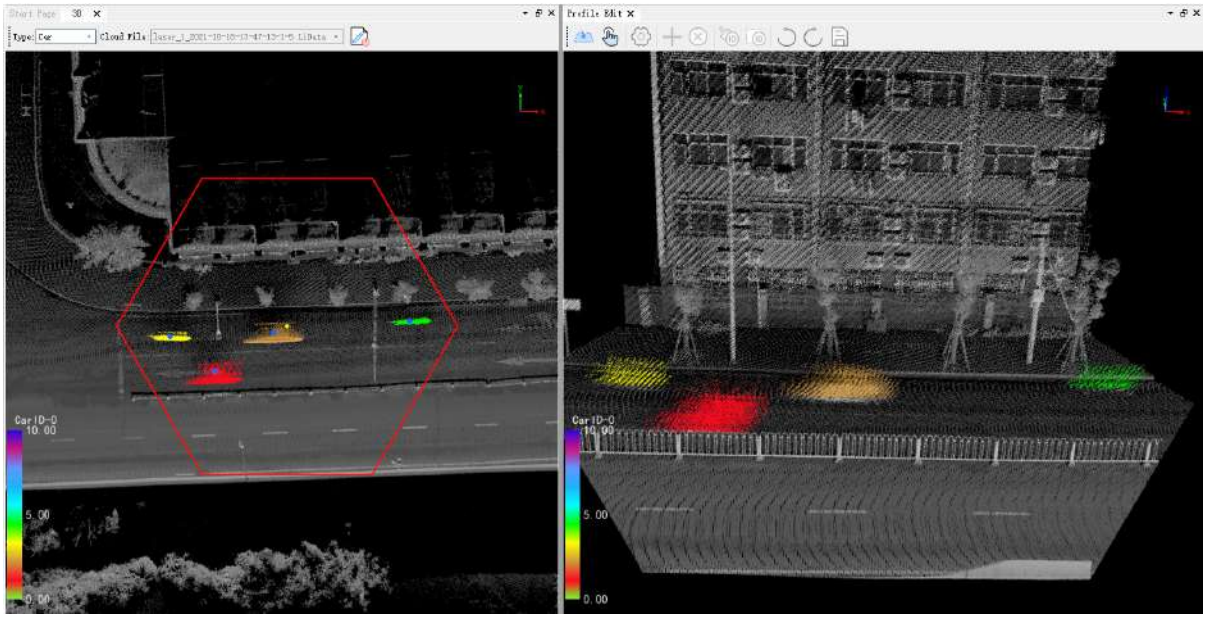
Steps

1.Click the **Editor** button, select the category and point cloud file to be edited, then click the **Start Editing** button. The point cloud in the 3D window will then be displayed according to the selected entity type ID.



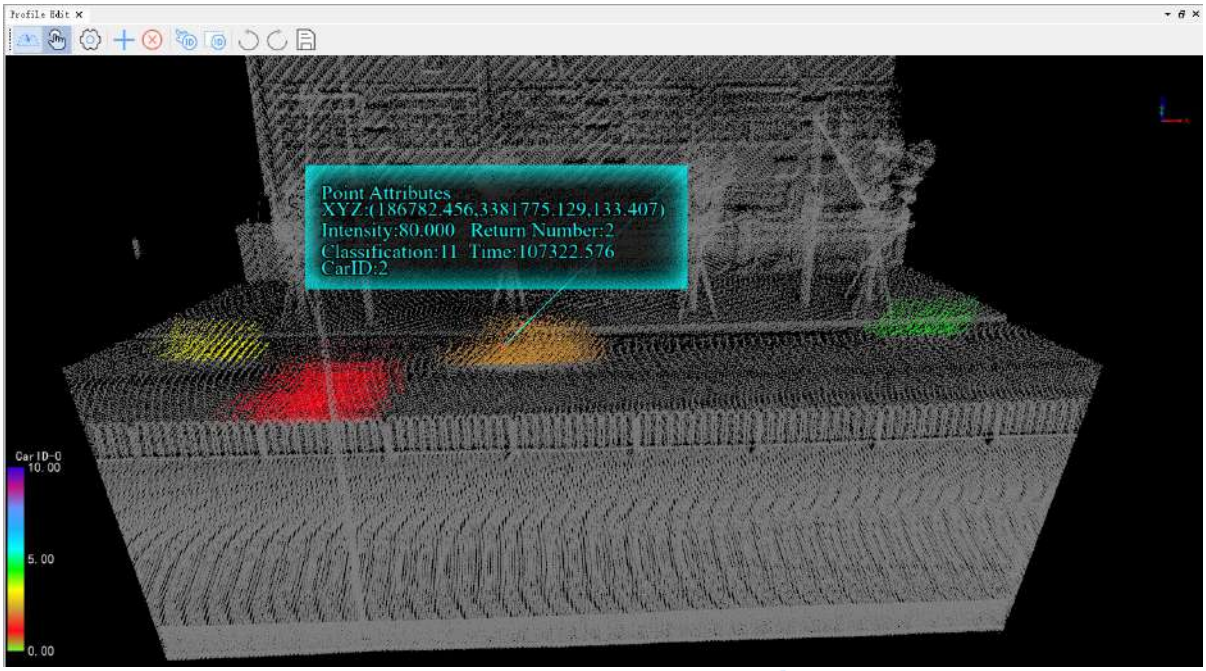
2.After starting editing, the individual editing window will automatically pop up. Within this window, you can perform operations to add, delete, and reduce entities.

3.In the 3D window, click the left mouse button to draw a hexagonal editing area. At this point, the points in this area will be displayed in the individual editing window.

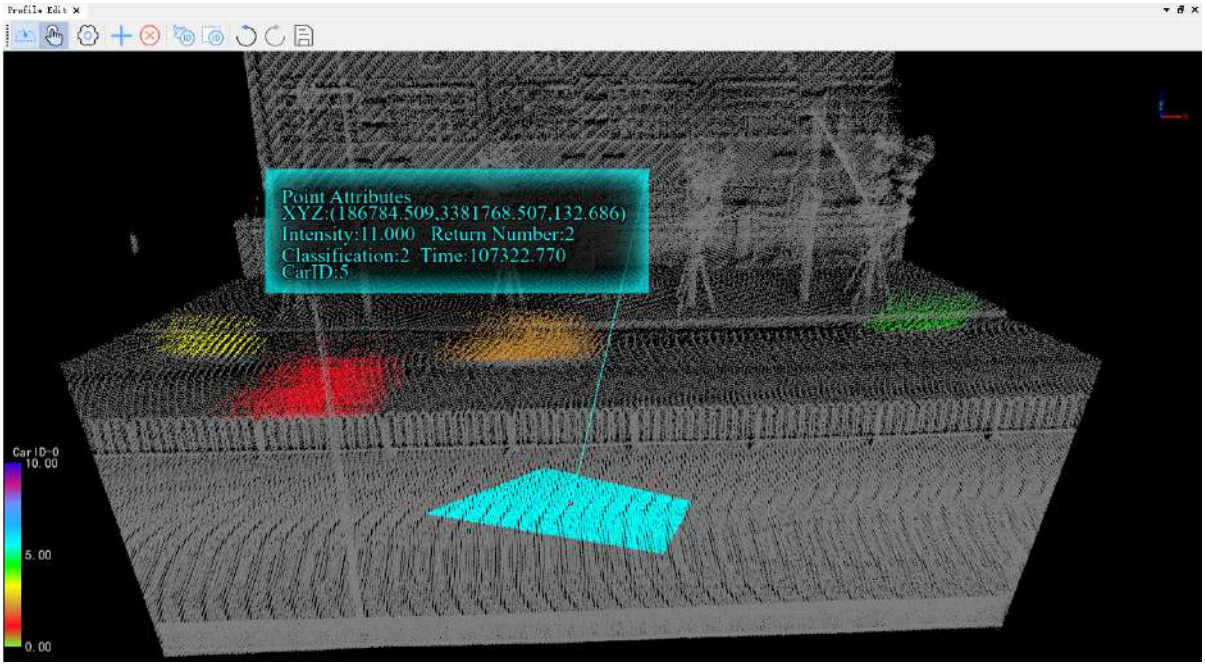


4.Click the **Load Edit Region** button in the object editing window to add the points of the editing area to the memory for editing. Without loading, editing is not supported.

5.Click the **Selection** button in the object editing window to view the attribute information of the selected points, including 3D coordinates, GPS time, intensity, RGB, object ID, etc.



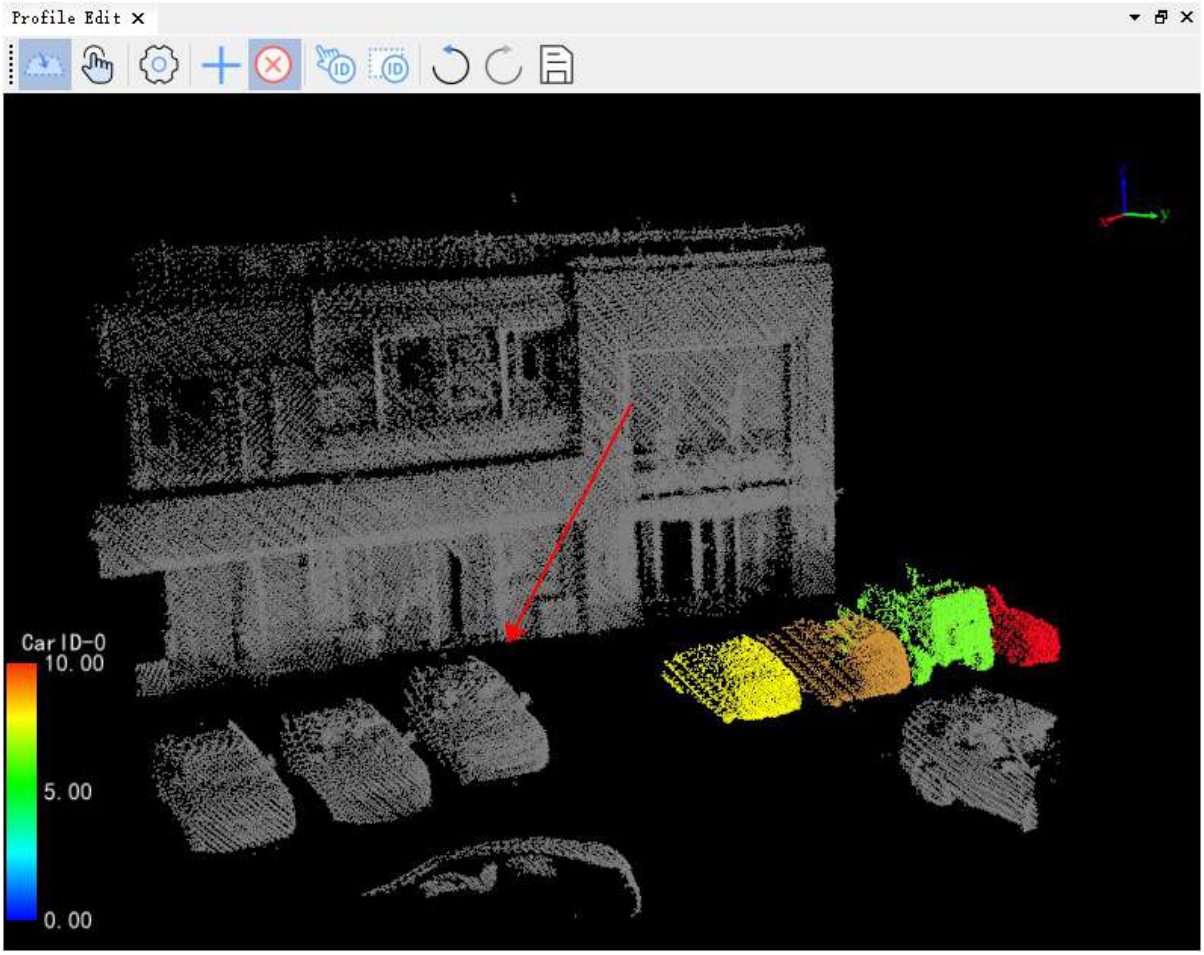
6.**Create Object**: In the object editing window, click the **Create Object** button, and create an object by selecting the point cloud of the object. At this point, the object ID of the points in the selected area will be set to the new object ID, and an object record will be automatically added to the attribute table. If you want to finely edit the point cloud of the object, you can use the [Set ID by Point Selection] and [Set ID by Frame Selection] operations.



Creating New Object

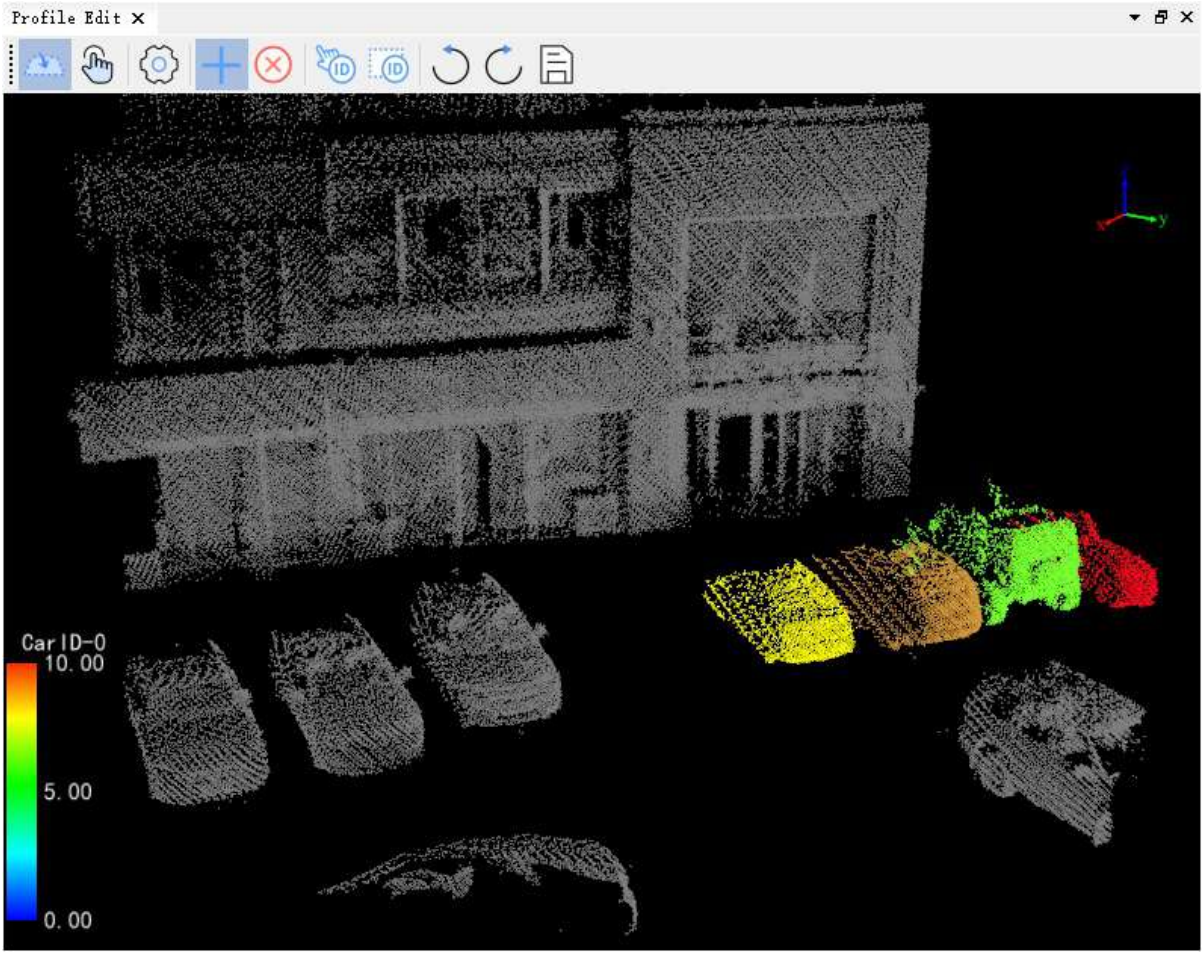
7.**Setting**: In the individual editing window, click the **Setting** button. In the setting window, by checking the box, you can display the points that need to be edited from two dimensions: category and entity ID, reduce the interference of the background point cloud on the editing process, and enhance operational convenience.

8.**Delete Object**: In the object editing window, click the **Delete Object** button. In the setting window, by checking the box, you can display the points that need to be edited from two dimensions: category and object ID, reduce the interference of the background point cloud on the editing process, and enhance operational convenience.

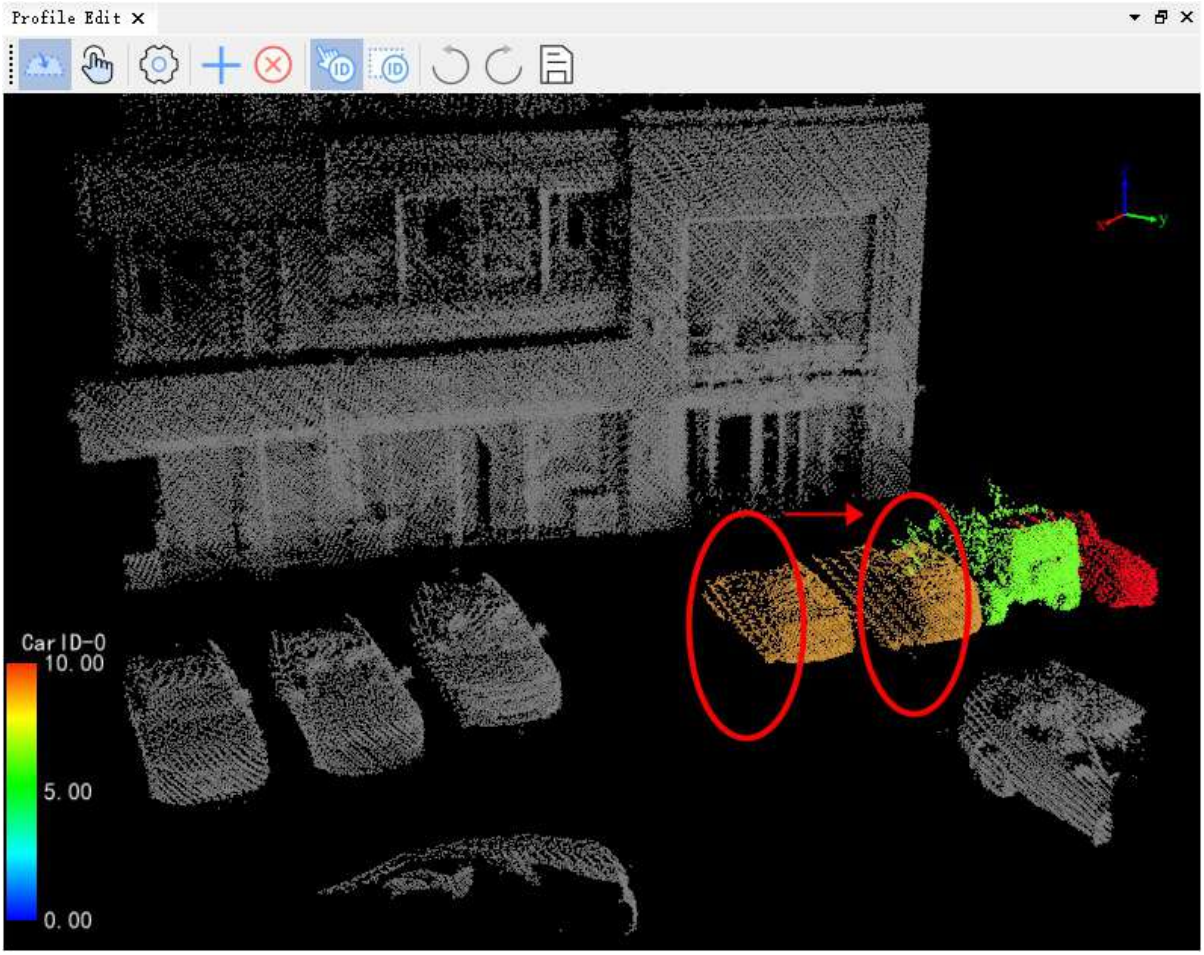


Deleting Object

9.**Set ID by Selection**: In the object editing window, click the **Set ID by Selection** button, select two points in succession by clicking. The ID of the object clicked first will be set to the ID of the object clicked second. It can be used to merge two point clouds in the editing area or delete a certain point cloud.

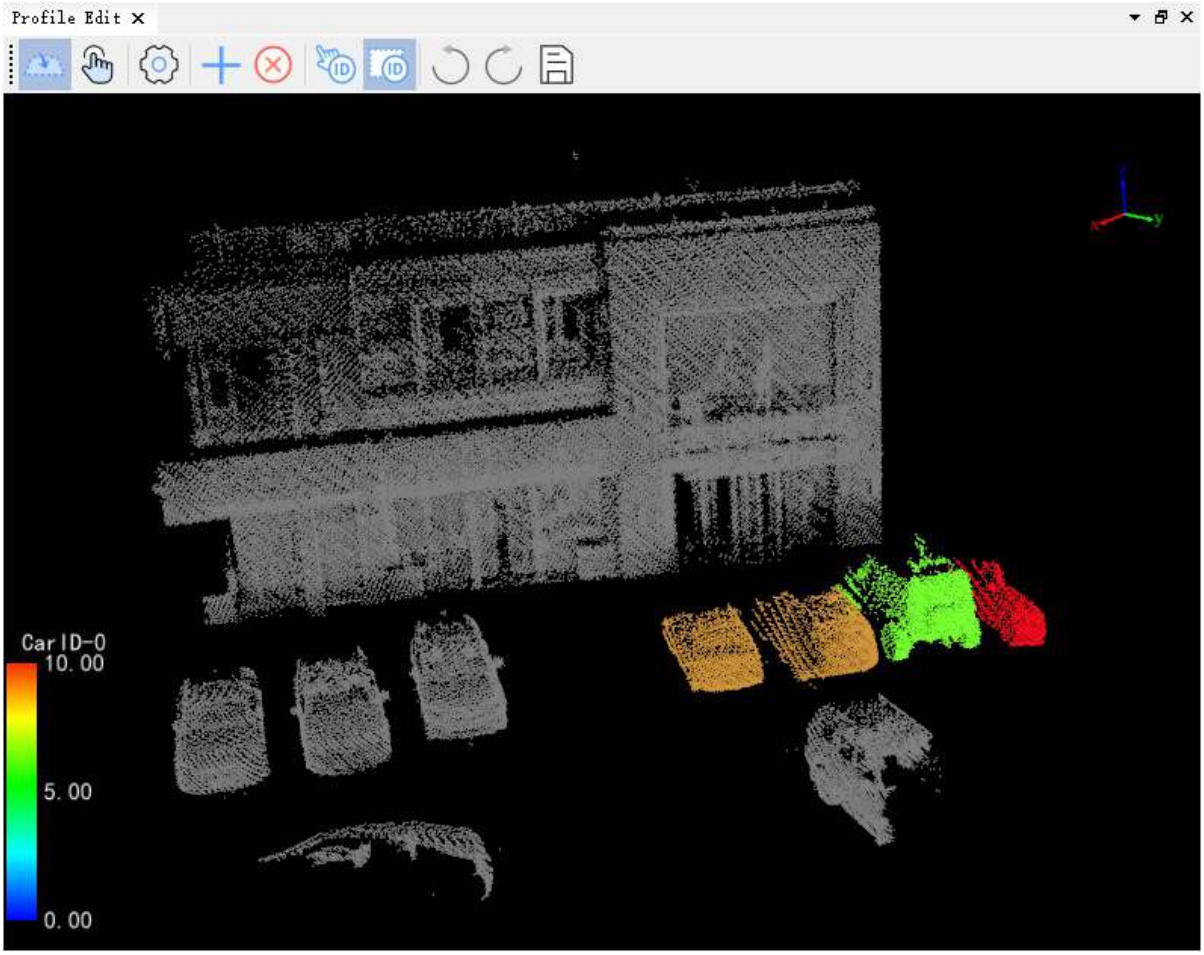


Before Setting ID by Point Selection

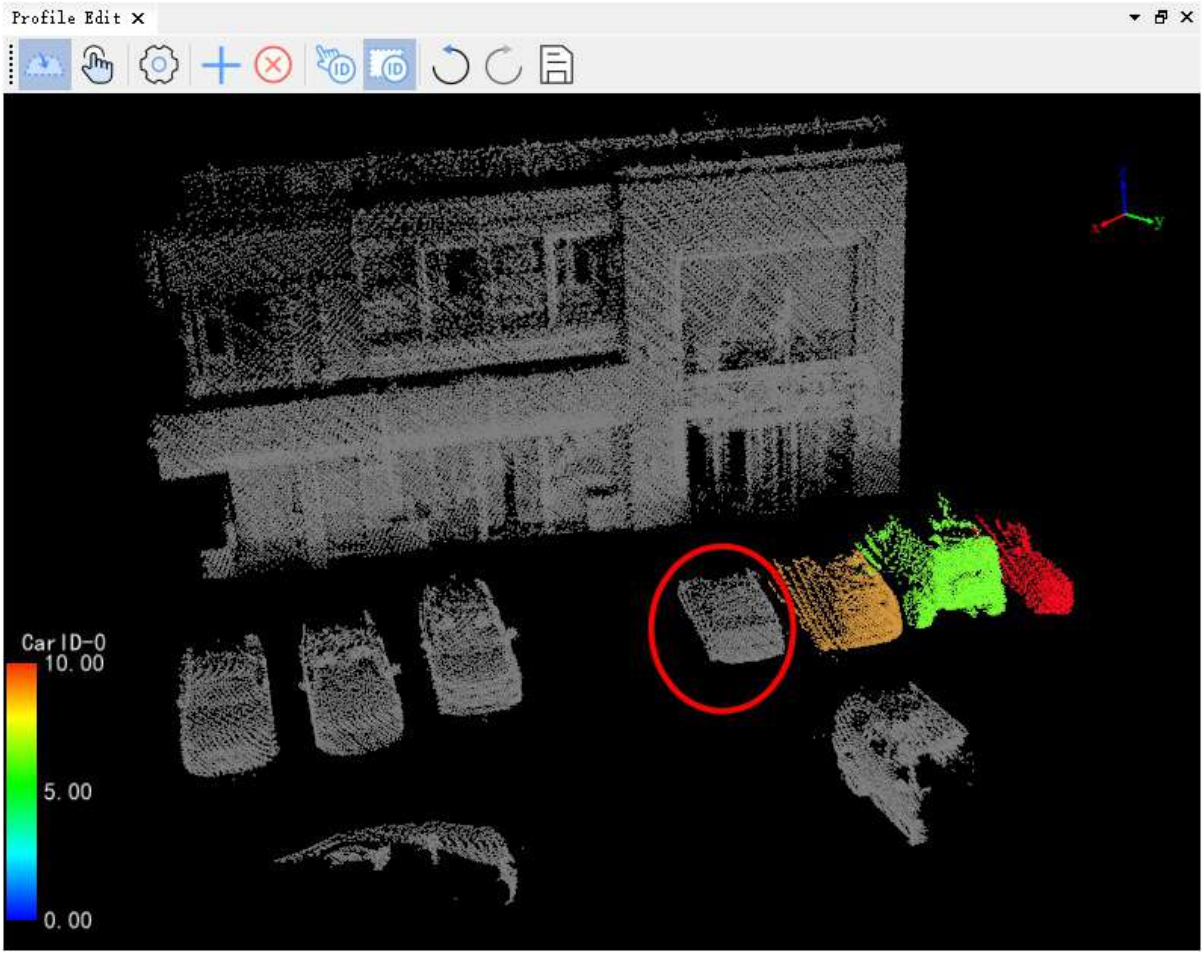


After Setting ID by Point Selection

10.**Set ID by Frame Selection**：In the object editing window, click the **Set ID by Frame Selection** button, first select a part of the points by framing, then click an object (or non-object point), and the object ID of the points in the frame will be set to the clicked object ID. It can be used for point refinement and point deletion by framing.



Before setting ID in the box



After checking the Set ID box

11.**Undo**: In the object editing window, click the **Undo** button to revert to the previous step, supports "Ctrl+Z" shortcut key.

12.**Redo**: In the object editing window, click the **Redo** button to restore to the next step, supports "Ctrl+Y" shortcut key.

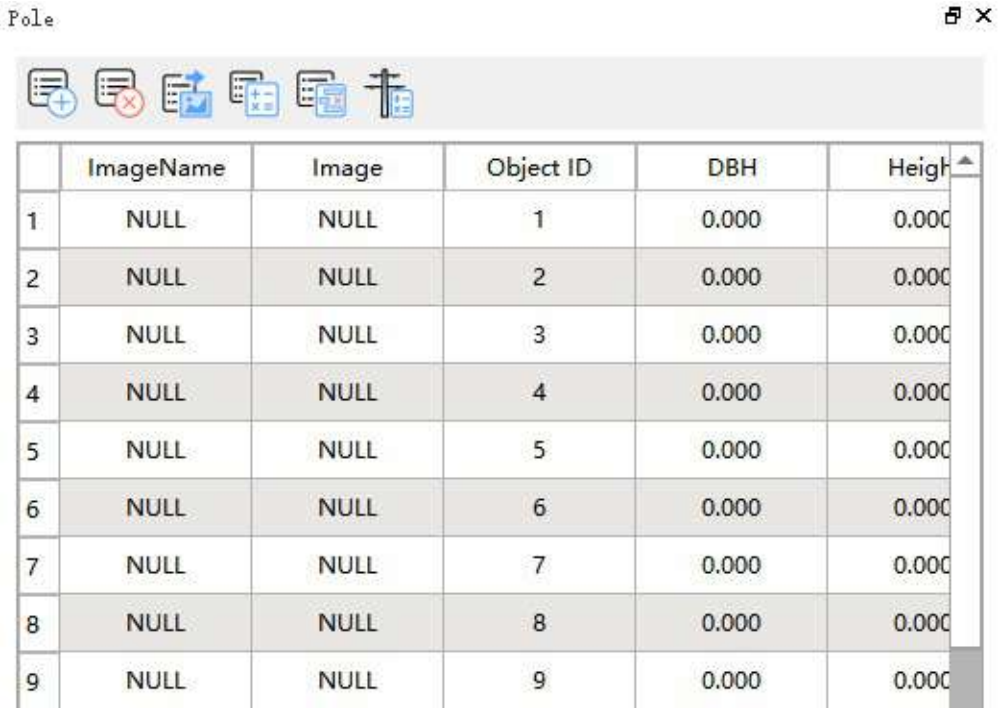
13.**Save**: In the object editing window, click the **Save** button to save the editing to a file, the 3D view on the left will refresh the result, supports "Ctrl+Shift+S" shortcut key.

Measurement

Function Description: Supports automatic and manual measurement of 'pole' parameters, including DBH (Diameter at Breast Height), height, and tilt angle.

Steps

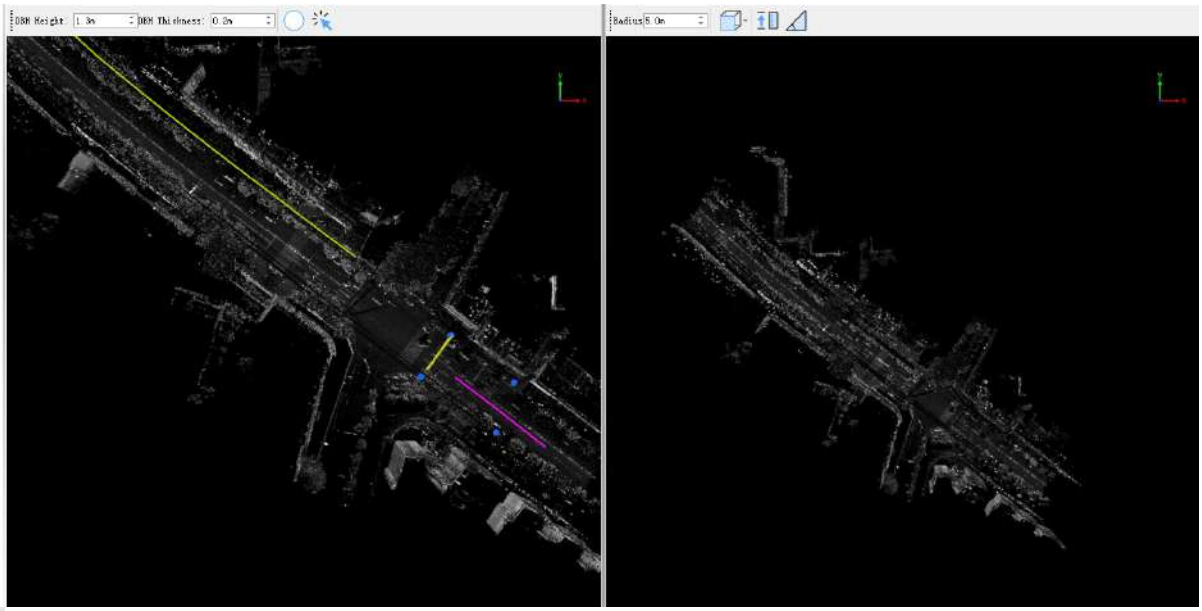
1.In the attribute table, select one or more rows (hold down ctrl to select multiple), then click the **Calculate Pole Parameter** button to start calculating parameters. The results are displayed in real-time in the table as shown below:



	ImageName	Image	Object ID	DBH	Heigh
1	NULL	NULL	1	0.000	0.000
2	NULL	NULL	2	0.000	0.000
3	NULL	NULL	3	0.000	0.000
4	NULL	NULL	4	0.000	0.000
5	NULL	NULL	5	0.000	0.000
6	NULL	NULL	6	0.000	0.000
7	NULL	NULL	7	0.000	0.000
8	NULL	NULL	8	0.000	0.000
9	NULL	NULL	9	0.000	0.000

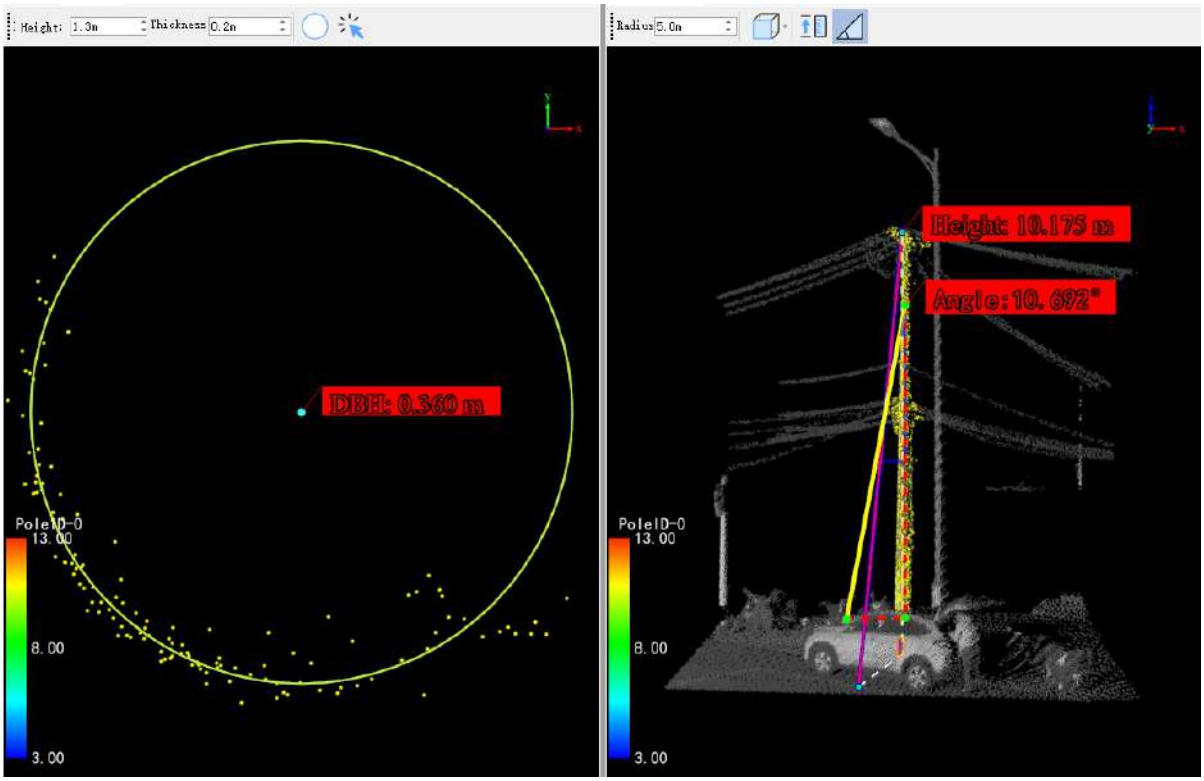
> Note: Clicking on the upper-left white box will select all individual IDs.

2.Click the **Measurement** button to enter the function interface:



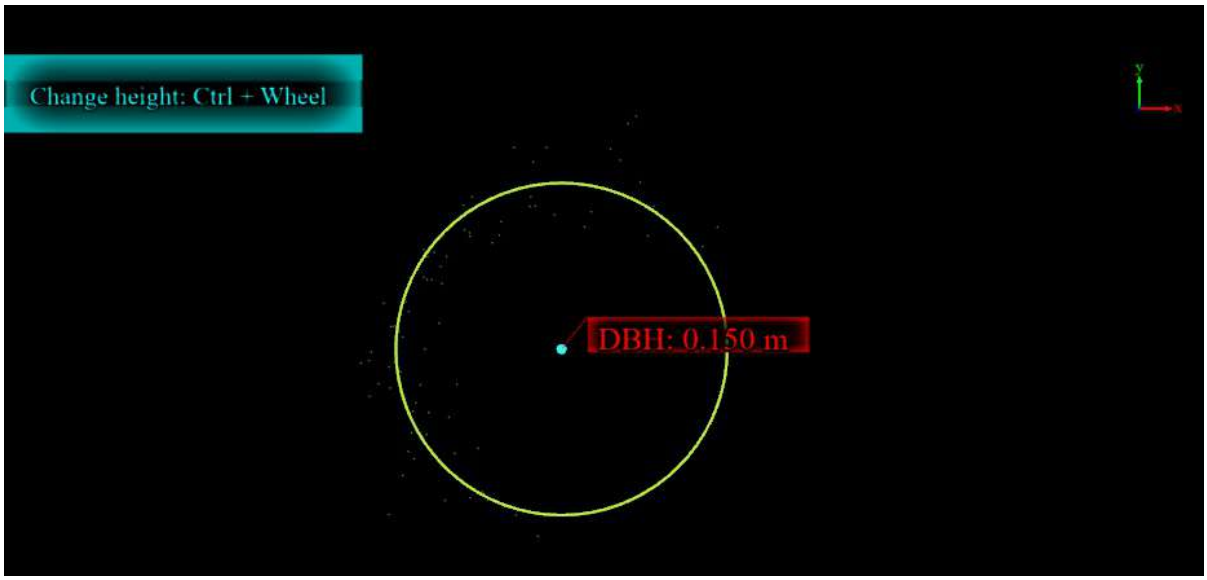
Note: You must first complete the individual segmentation and editing operations

3.Once the parameters are calculated, select a row in the table, the 3D view on the left will display the DBH parameter, and the Object View window on the right will display parameters such as height, crown width, angle, etc. The left view toolbar has two parameters: slice height and slice thickness, which indicate the points involved in the calculation of the DBH, the default values are 1.3m and 0.2m.

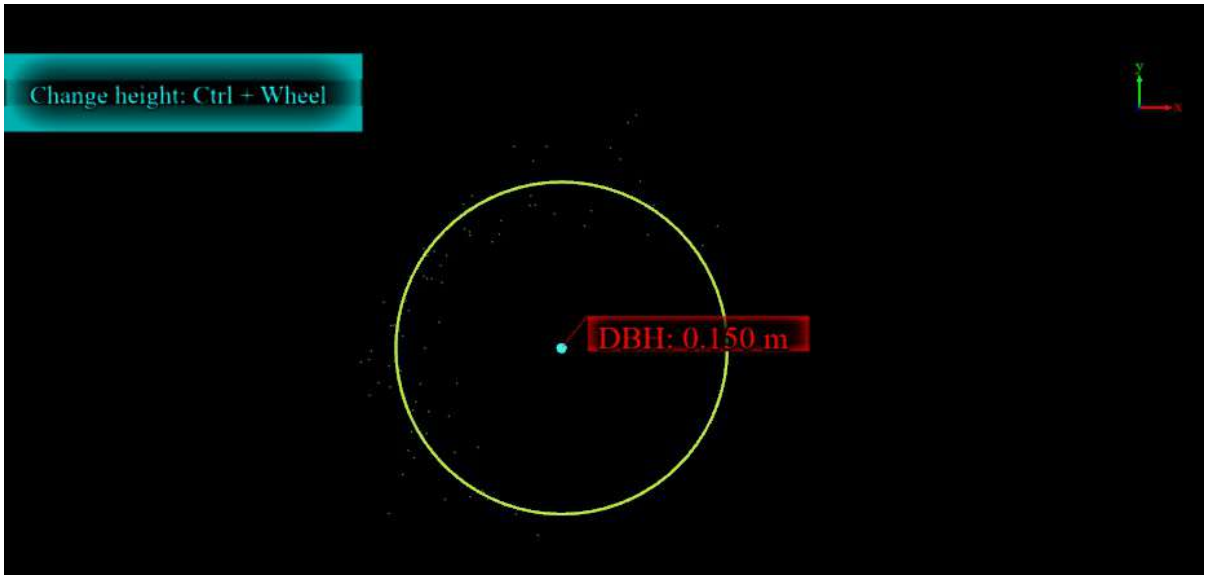


4.**DBH Measurement**：There are two ways to refit DBH:

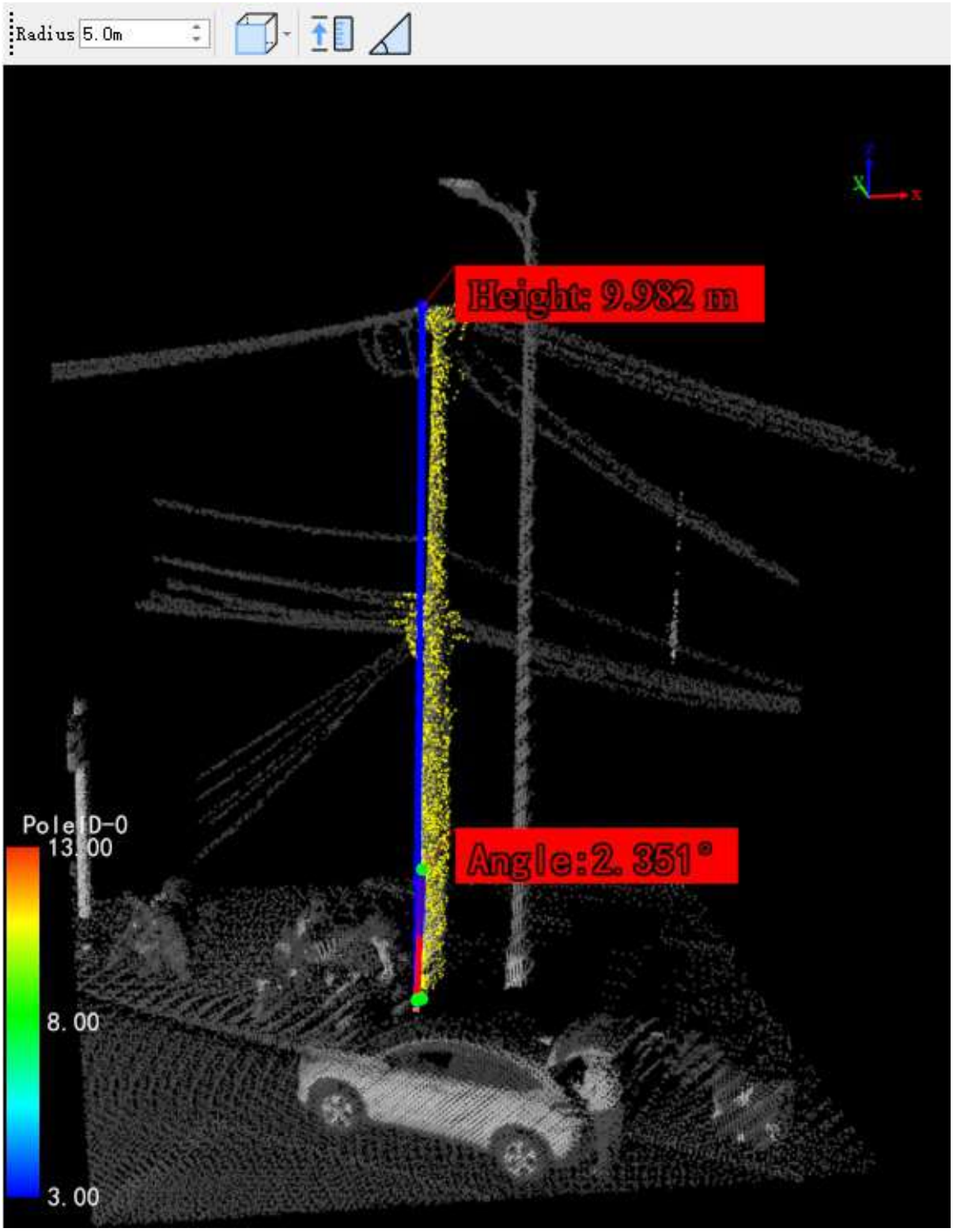
- **DBH Three-Point Fitting**：First click on the individual ID you want to view, then in the 3D view window, click the**DBH Three-Point Fitting** button. Using the left mouse button, select three points to determine the refitted circle, as shown in the figure:



- **DBH Frame Selection Fitting**：After switching to the individual view, in the 3D window click the **DBH Frame Selection Fitting** button, and select the point cloud that needs to refit DBH:



5.**Height Measurement**：In the Object View window, click the **Height Measurement** button and use two points on the point cloud to remeasure the height of the individual. The new measurement value will be updated in real-time in the corresponding individual table list.



The value after remeasuring the height

6.**Angle Measurement**: In the Object View window, after switching the point cloud to a suitable angle using the view switch button (Top View, Front View, Back View, Left View, Right View), click the **Angle Measurement** button to remeasure the angle and update the result in the parameter table.

Vector Editor

Vector Editor allows the user to edit vector data through modules such as layer management, vector drawing, element editor, and edit vertices.

Draw Vector

Element Editor

Edit Vertices

Auxiliary Lines

Drawing

The function of the drawing module is mainly to add shape vectors such as **point**, **line**, **surface**, **circle**, and **arc**.

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

Drawing Elements

- Add Point
- Add Multi-point
- Add Line
- Add Line Segment
- Add Arc
- Add Polygon
- Add Rectangle
- Add Parallelogram
- Addw Circle
- Add Hexagon
- Add Octagon

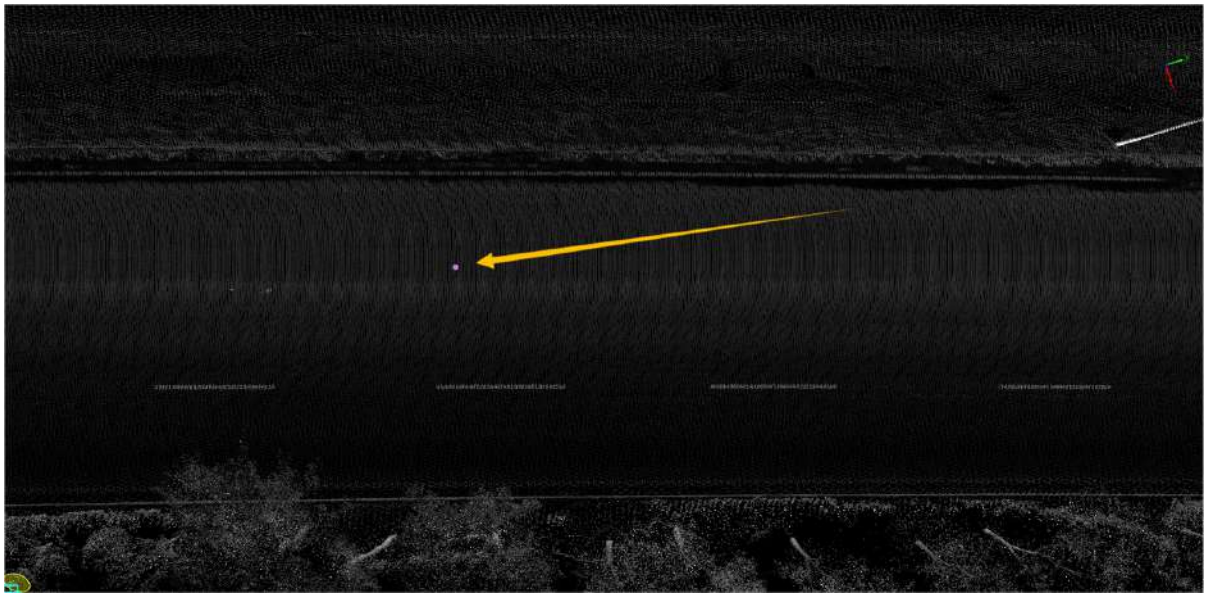
Add Point

Function Description：This tool allows the user to add vector points to objects.

Steps

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

2.Click **Add Point** button in the Vector Editor, use the mouse to select a point in the point cloud, and a dot will be drawn at that position according to the current layer.



Draw point interface

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

Add Multi-point

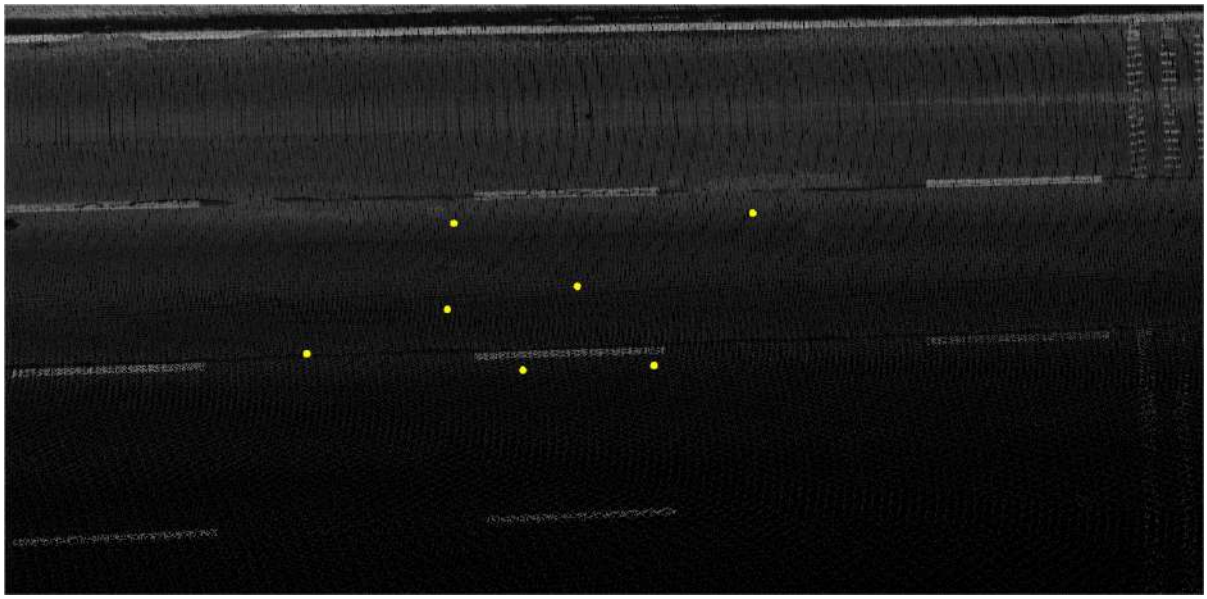
Function Description：This tool allows the user to add multiple vector points to objects.

Steps

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

2.Click **Add Multi-point** button in the Vector Editor, use the mouse to continuously select multiple points in the point cloud, and quickly double-click the mouse to end the drawing.

> Note: After drawing, multiple points are stored as an object.



Draw multiple point interface

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

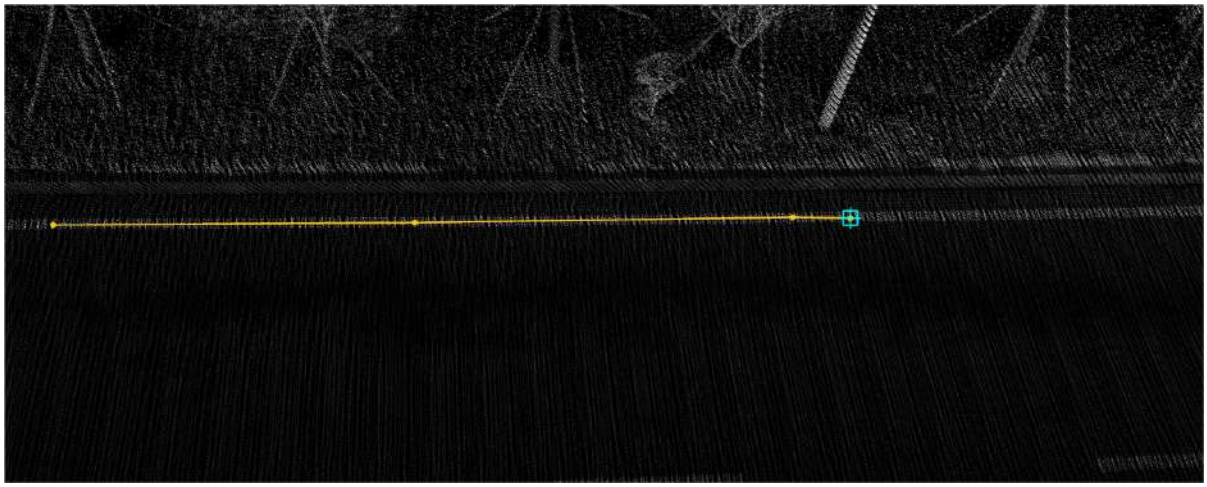
Add Line

Function Description：This tool allows the user to add vector lines to objects.

Steps

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

2.Click **Add Line** button in the Vector Editor. Use the mouse to select the starting point of the line in the point cloud, click the left mouse button continuously to confirm multiple nodes of the line, and double-click the left mouse button to end the drawing of the current line.



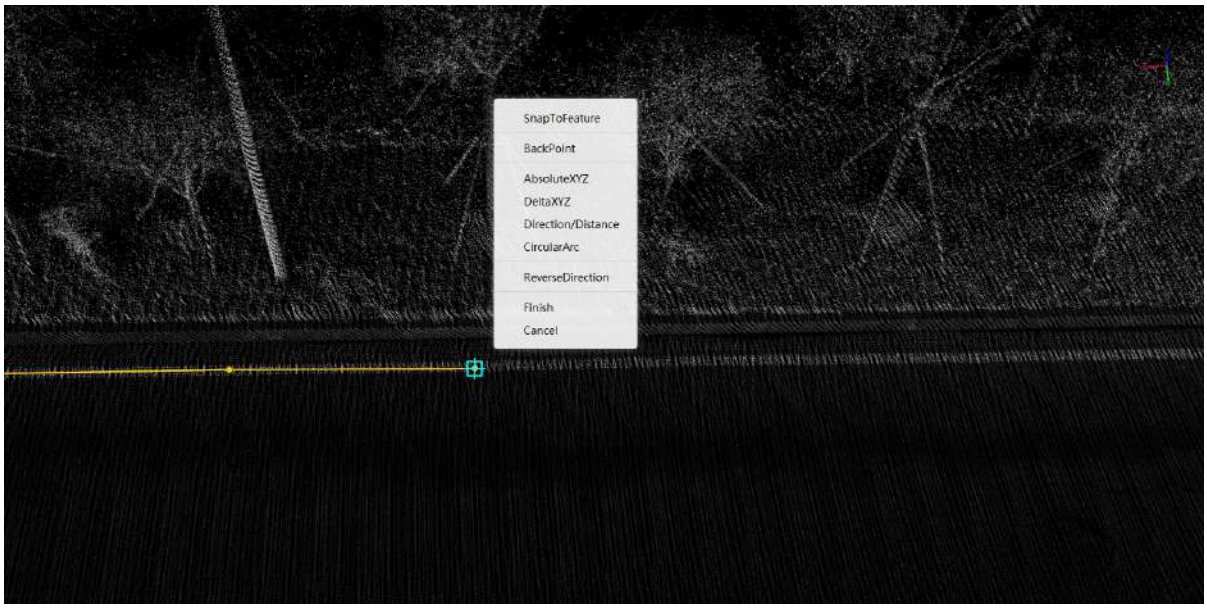
Draw line interface

3.During the drawing process, you can use different shortcut keys to switch between different modes:

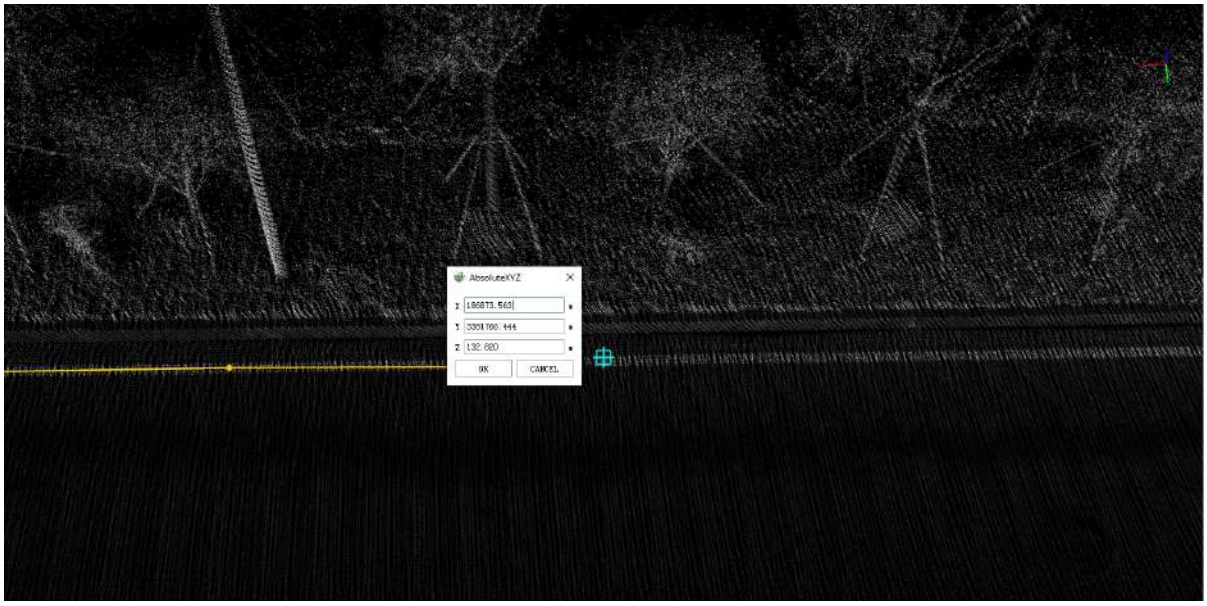
Function	Shortcut	Descripción
Drawing Arc	S/s	Short press the s key to switch the drawing mode from drawing a straight line to drawing an arc. The arc adopts the three-point mode. The first point is the last point before short press s, the second point is the end point of the arc, and the third point controls the arc
Backing off a point	B/b	Press b briefly to back out of the drawn node
Tracking Mode	T/t	In the area where vector tracking needs to be activated, press the "T" key briefly to switch to tracking mode. Once in tracking mode, click on a starting point on the object to be tracked. While moving the mouse along the object to be tracked, vector nodes will be automatically collected without the need for manual node placement. During the tracking process, you can cancel the current tracking by right-clicking the mouse. Note that after cancellation, not only will the tracked nodes be cleared, but the tracking mode will also be exited by default. If you only want to cancel part of the tracking, you can reverse the mouse movement while tracking to undo the tracked nodes. When you need to stop tracking, simply click the left mouse button once at the ending position, and it will automatically switch back to normal drawing mode.
Plot Spline Curve	F/f	During the drawing process, if you need to create a Spline curve, you can press the "F" key to switch to Spline curve drawing mode. In this mode, the added nodes will be calculated and added according to the Spline

		curve.
Drawing Bezier Curves	G/g	During the drawing process, if you need to create a Bézier curve, you can press the "G" key to switch to Bézier curve drawing mode. In this mode, the added nodes will be calculated and added according to the Bézier curve.

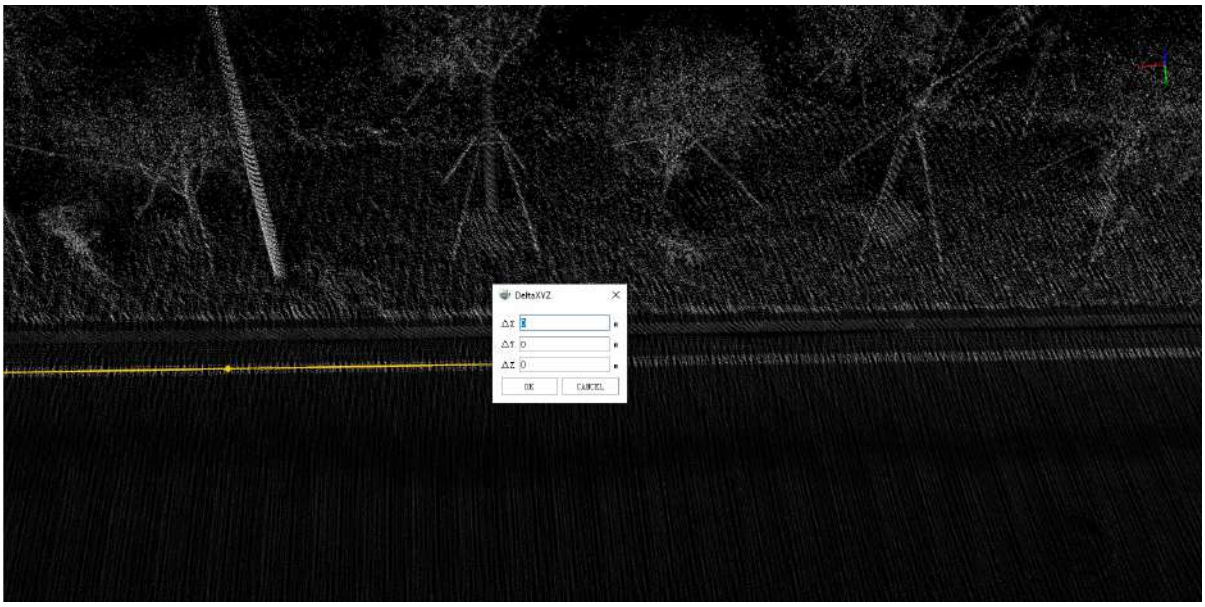
4.During the drawing process, right-clicking the mouse can include the following operations:



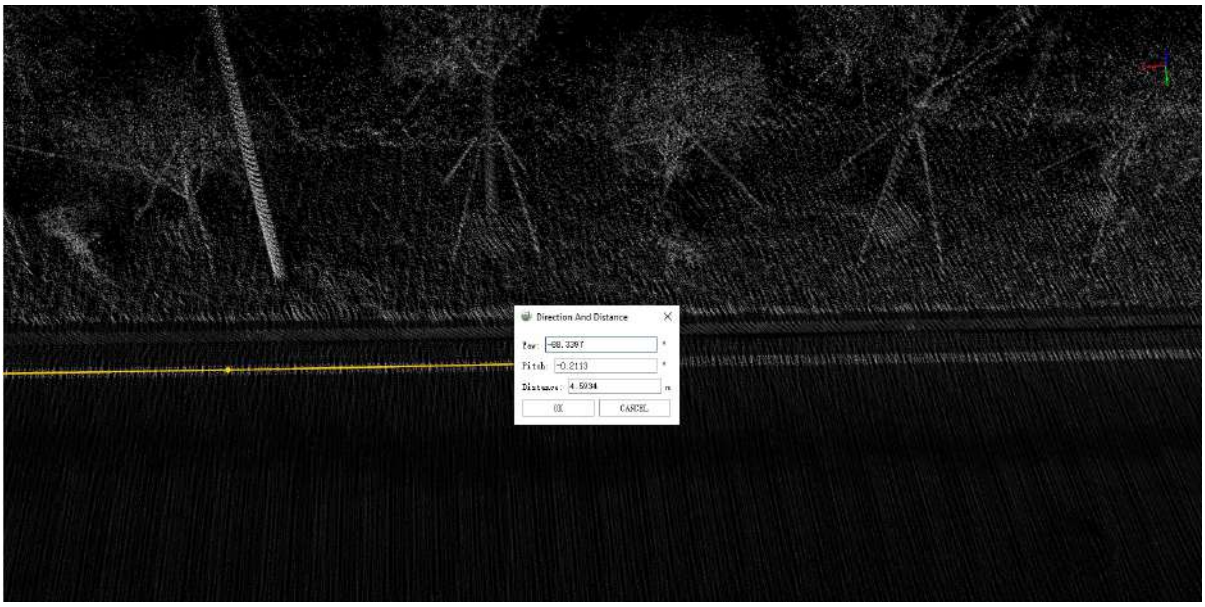
- **SnapToFeature**: You can switch various capture modes at any time in the SnapToFeature menu.
- **BackPoint**: Click BackPoint to fallback nodes in order. (the first node cannot be rolled back). During drawing, the shortcut key of BackPoint is b/B.
- **AbsoluteXYZ**: With the AbsoluteXYZ function, you can specify the coordinates of AbsoluteXYZ during the drawing process. The coordinate value entered in the pop-up box is the position drawn by the node.



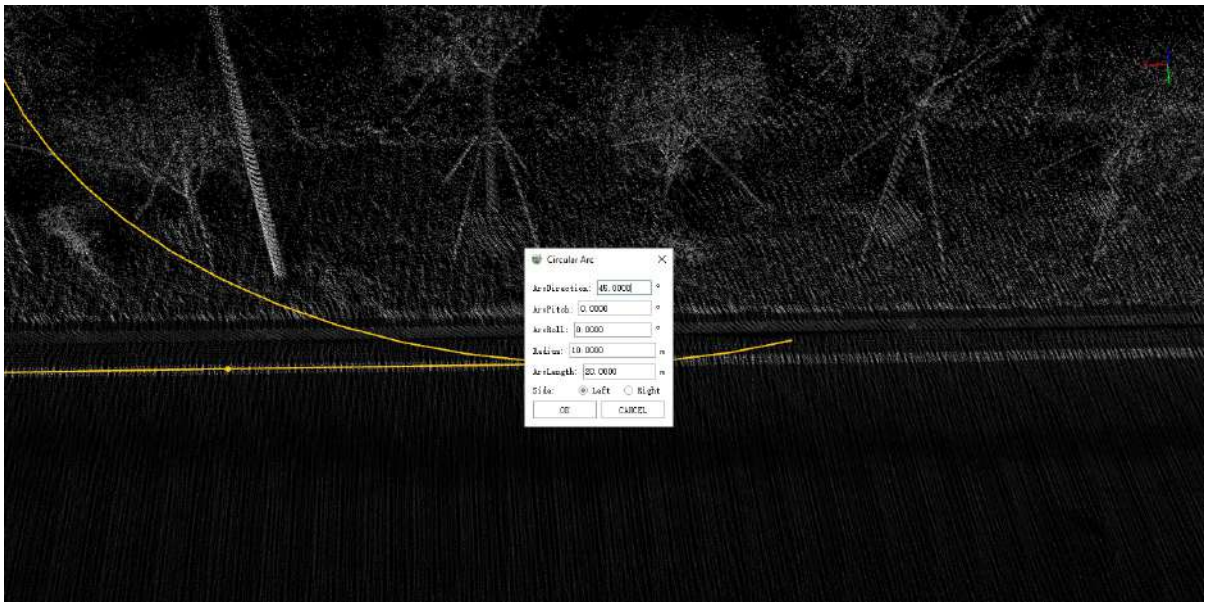
- **DeltaXYZ**: With the DeltaXYZ function, you can specify the coordinate position of the next node relative to the previous node in the drawing process. The coordinate value entered in the pop-up box is the position relationship between the next node and the previous node.



- **Direction/Distance**: With the Direction/Distance function, you can specify the direction and distance of the next node relative to the previous node in the drawing process. The yaw angle is the included angle with the Y axis of the coordinate system, the pitch angle is the included angle with the XY plane of the coordinate system, and the distance is the distance between the next point and the next point in the above angular direction. In the profile window, there is no need to modify the yaw angle.



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



- **ReverseDirection**: During the line drawing process, the direction of the drawn vector line can be reversed with the ReverseDirection function, and the drawing can continue based on the new direction.
- **Finish**: During the drawing process, you can double-click the last node with the left mouse button to complete the drawing, or you can click the Finish button in the pop-up box with the right mouse button to end the drawing of the current object.
- **Cancel**: During the drawing process, you can click the right mouse button to pop up the Cancel button to exit the current drawing.

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

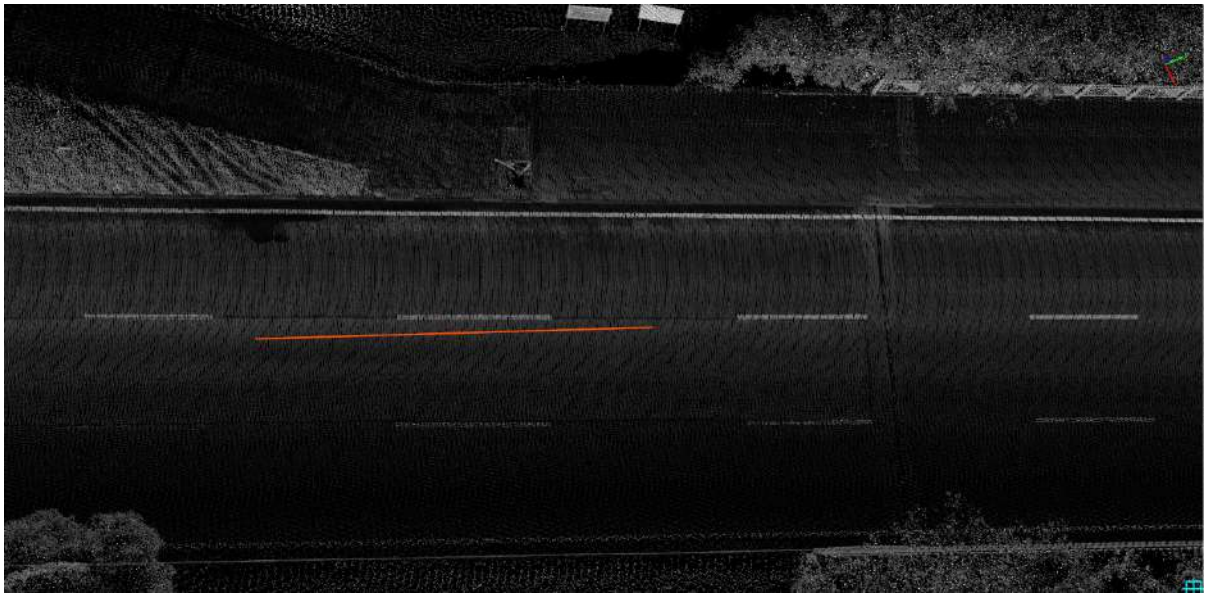
Add Line Segment

Function Description：This tool allows the user to add a vector line segments to objects.

Steps

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

2.Click **Add Line Segment** button in the Vector Editor. Use the mouse to select the starting point of the selection in the point cloud, move the mouse, and click the left mouse button again to confirm the other end of the line.



Draw Line segment interface

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

Add Arc

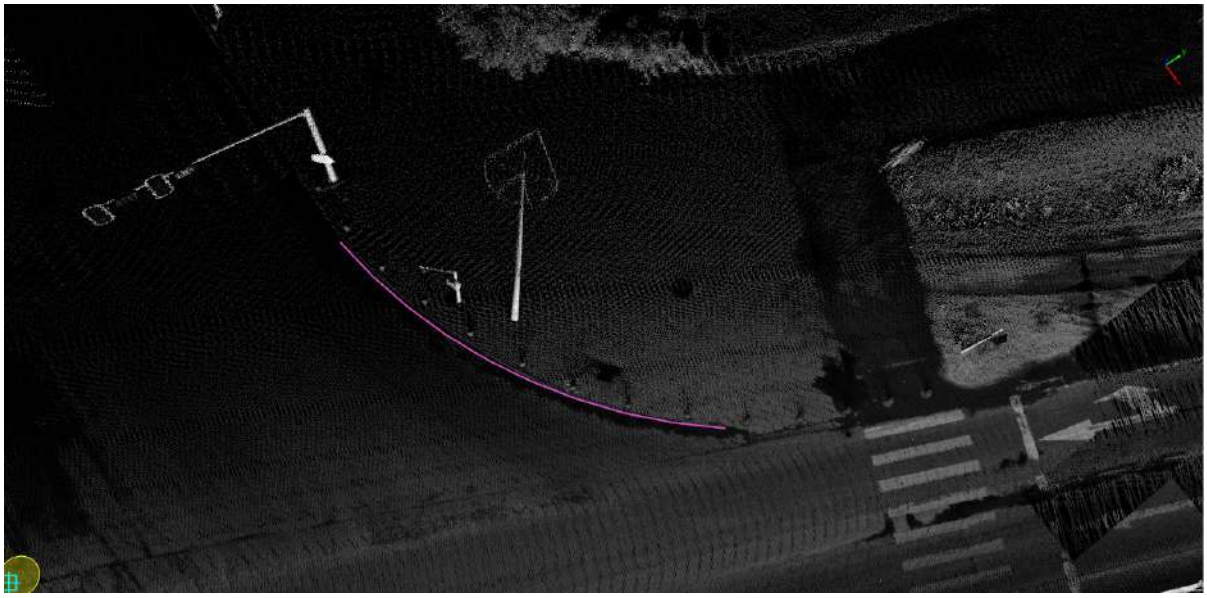
Function Description：This tool allows the user to use the three key point method to add arc-shaped vector objects.

Steps

1.Select a surface layer, such as a lane centerline layer.

2.Click **Add Arc** button in the Vector Editor:

- Left click the point cloud to select the first key point.
- Left click the point cloud to select the second key point.
- After clicking two key points, an arc-shaped real-time display box will appear. Left-click on the point cloud and select the third key point to determine an arc-shaped vector object.



Draw arc interface

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

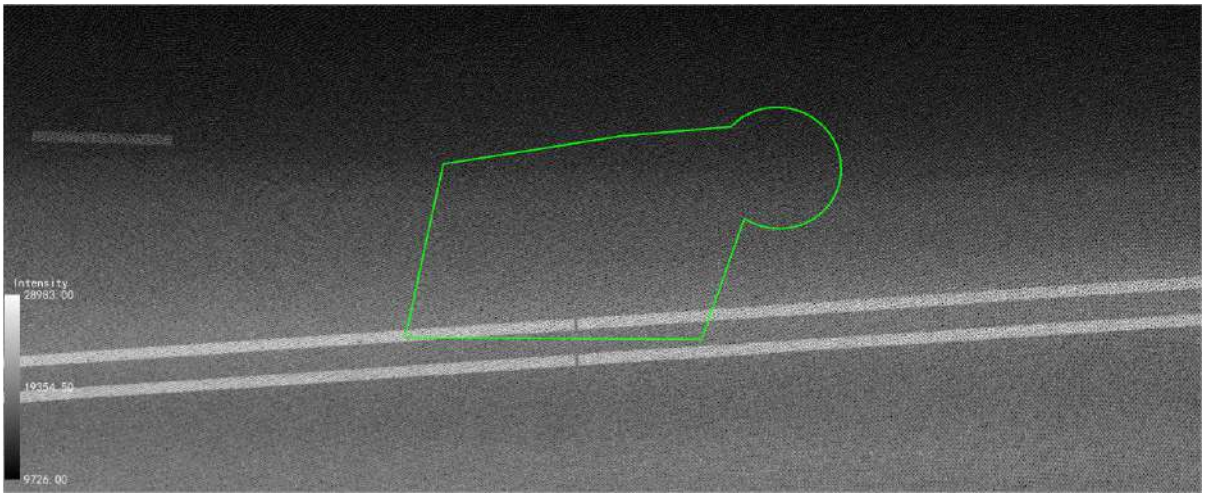
Add Polygon

Function Description：This tool allows the user to add vector polygons to objects.

Steps

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

2.After click **Add Polygon** button in the Vector Editor, use the mouse to click multiple points in succession to confirm the nodes of the polygon, The shortcut key B can be used to roll back the drawn node, and the keyboard key s can be used to switch the mode of drawing the arc in the polygon. The current point is the first point of the arc, the second point is the end point of the arc, and the third point determines the radian of the arc, which can complete the arc drawing. During the drawing process, the line and arc drawing can be switched several times, and the last point can be double-clicked to complete the drawing.



Draw polygon interface

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

Add Rectangle

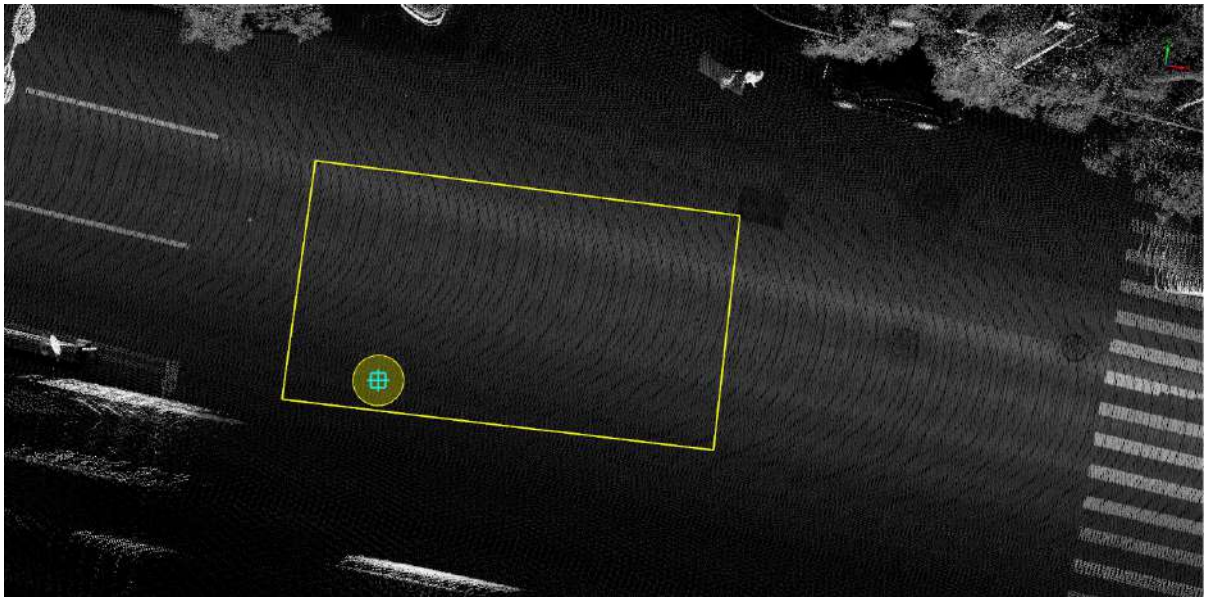
Function Description：This tool allows the user to use the three key point method to add vector rectangular objects.

Steps

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

2.Click **Add Rectangle** button in the Vector Editor:

- Left click the point cloud to select the first key point as the starting point of the rectangle.
- Left click the point cloud to select the second key point, and determine the width of the rectangle.
- Left click the point cloud to select the third key point, determine the length of the rectangle, and complete the drawing of the rectangle object.



Draw rectangle interface

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

Add Parallelogram

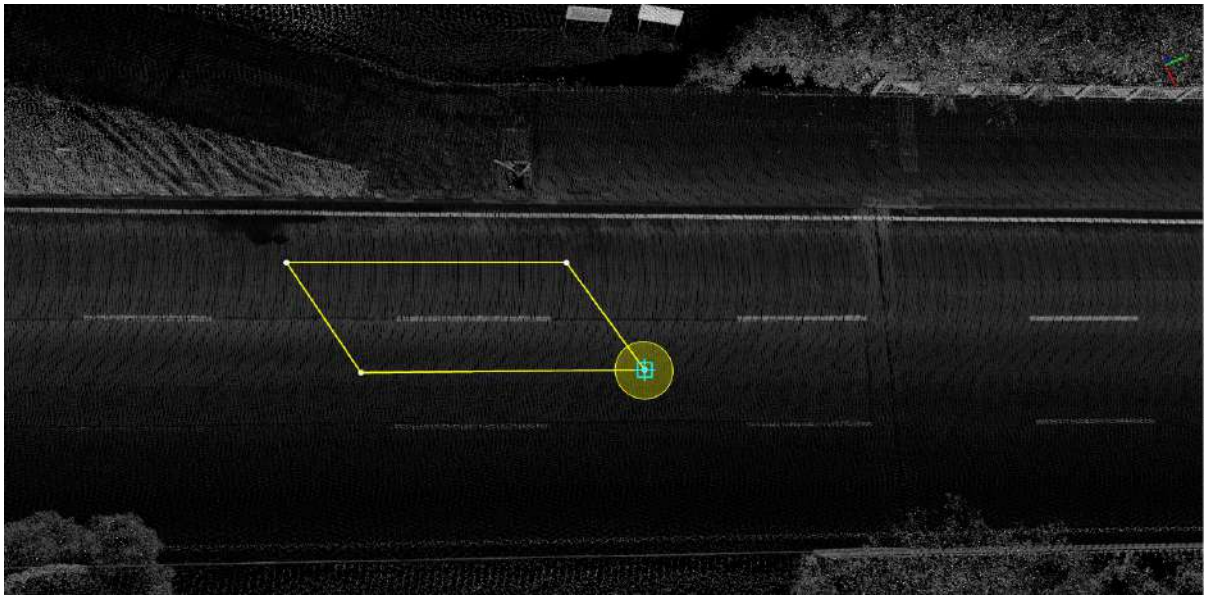
Function Description：This tool allows the user to use the three key point method to add a parallelogram vector objects.

Steps

1.Select a surface layer, such as a surface facility layer.

2.Click **Add Parallelogram** button in the Vector Editor:

- Left click the point cloud to select the first key point as the starting point of the parallelogram.
- Left click the point cloud to select the second key point, and determine one side of the parallelogram.
- Left click the point cloud, select the third key point, confirm the other side of the parallelogram, and then complete the parallelogram object drawing.



Draw parallelogram interface

> Note: The geometric characteristics of the vector result must be consistent with the geometric characteristics of the layer to be added. For example, the point-like vector result must be added to the point-like layer, otherwise it will be prompted that it cannot be drawn. For the description of layers, see Layer Management

Add Circle

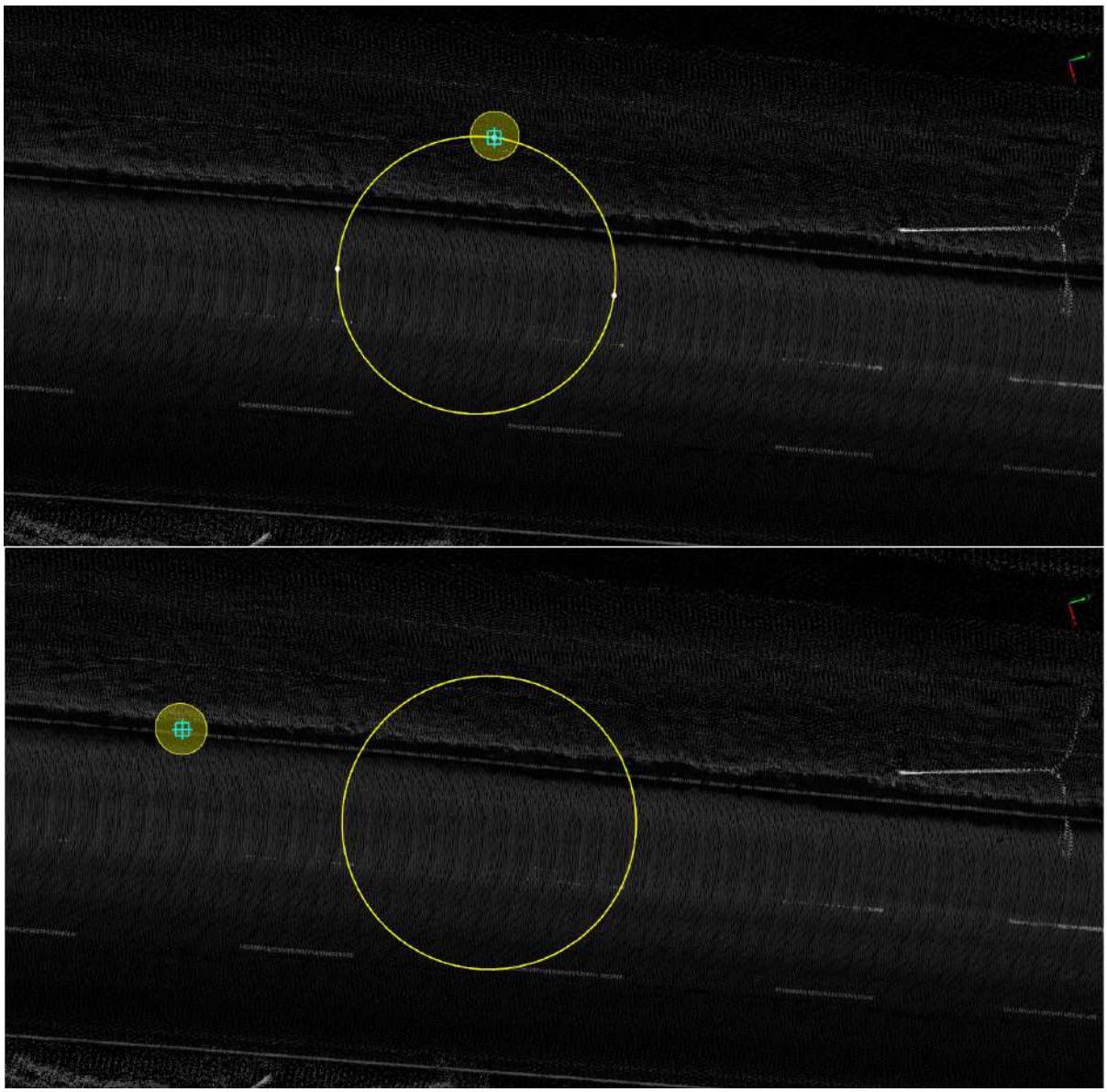
Function Description：This tool allows the user to use the three key point method to add circle-shaped vector objects.

Steps

1.Select a surface layer, such as a surface facility layer.

2.Click **Add Circle** button in the Vector Editor:

- Left click the point cloud to select the first key point.
- Left click the point cloud to select the second key point.
- After clicking two key points, an arc-shaped real-time display box will appear. Left click on the point cloud and select the third key point to determine a circle-shaped vector object.



Draw circle interface

> Note: The geometric characteristics of the vector result must be consistent with the geometric characteristics of the layer to be added. For example, the point-like vector result must be added to the point-like layer, otherwise it will be prompted that it cannot be drawn. For the description of layers, see Layer Management

Add Hexagon

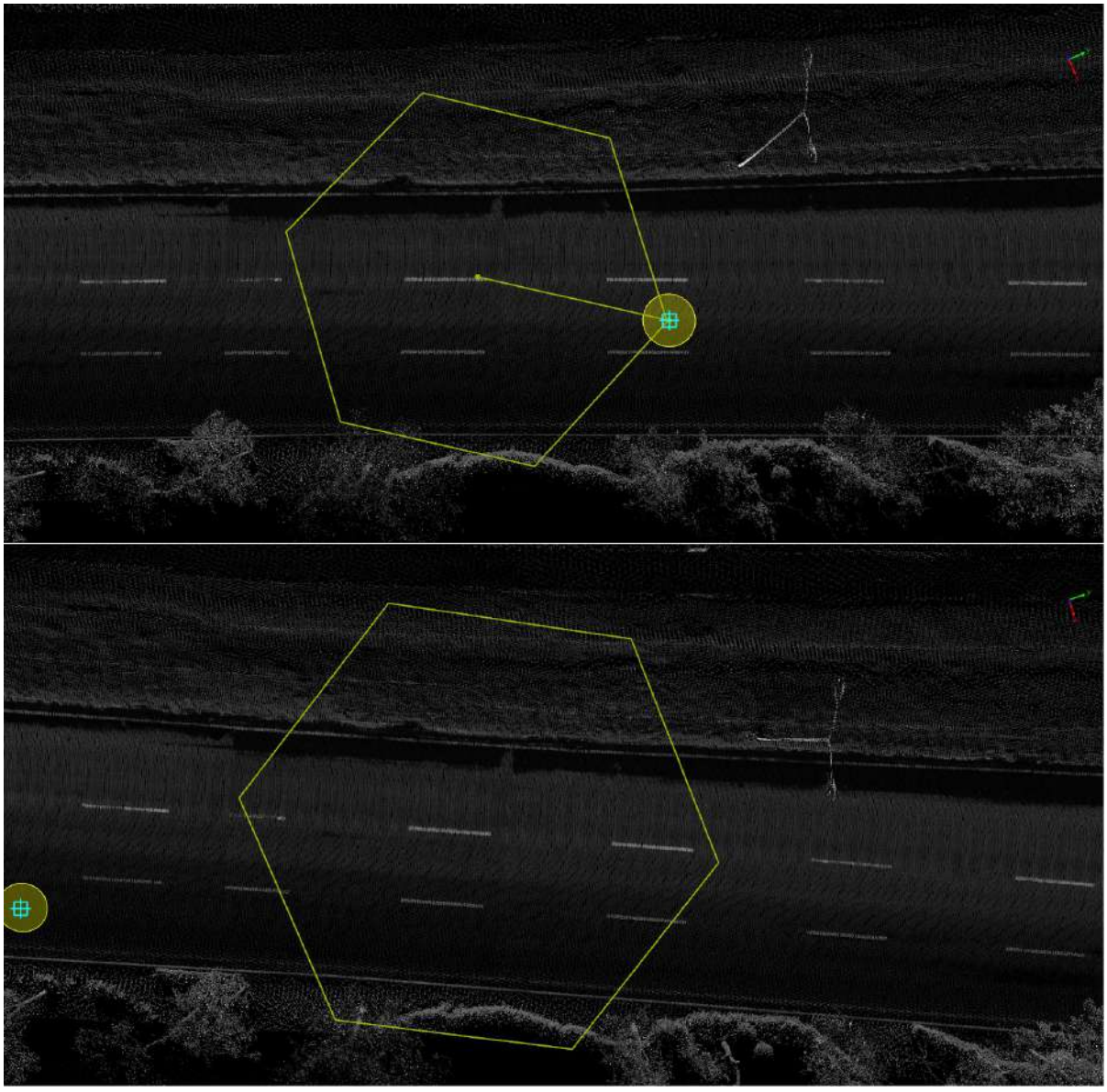
Function Description: This tool allows the user to add a hexagonal vector object by using the center point and dragging the mouse.

Steps

1.Select a surface layer, such as a surface facility layer

2.Click **Add Hexagon** button in the Vector Editor:

- Click the point cloud with the left mouse button, select the center point of the hexagon, and then drag the mouse, a real-time display box of the hexagon will appear. After determining the size and direction, click the point cloud with the left mouse button to determine a hexagon vector object.



Draw hexagonal interface

> Note: The geometric features of the vector result must be consistent with the geometric features of the layer to be added. For example, the point-like vector result must be added to the point-like layer, otherwise it will prompt that it cannot be drawn. For the description of layers, see Layer Management

Add octagon

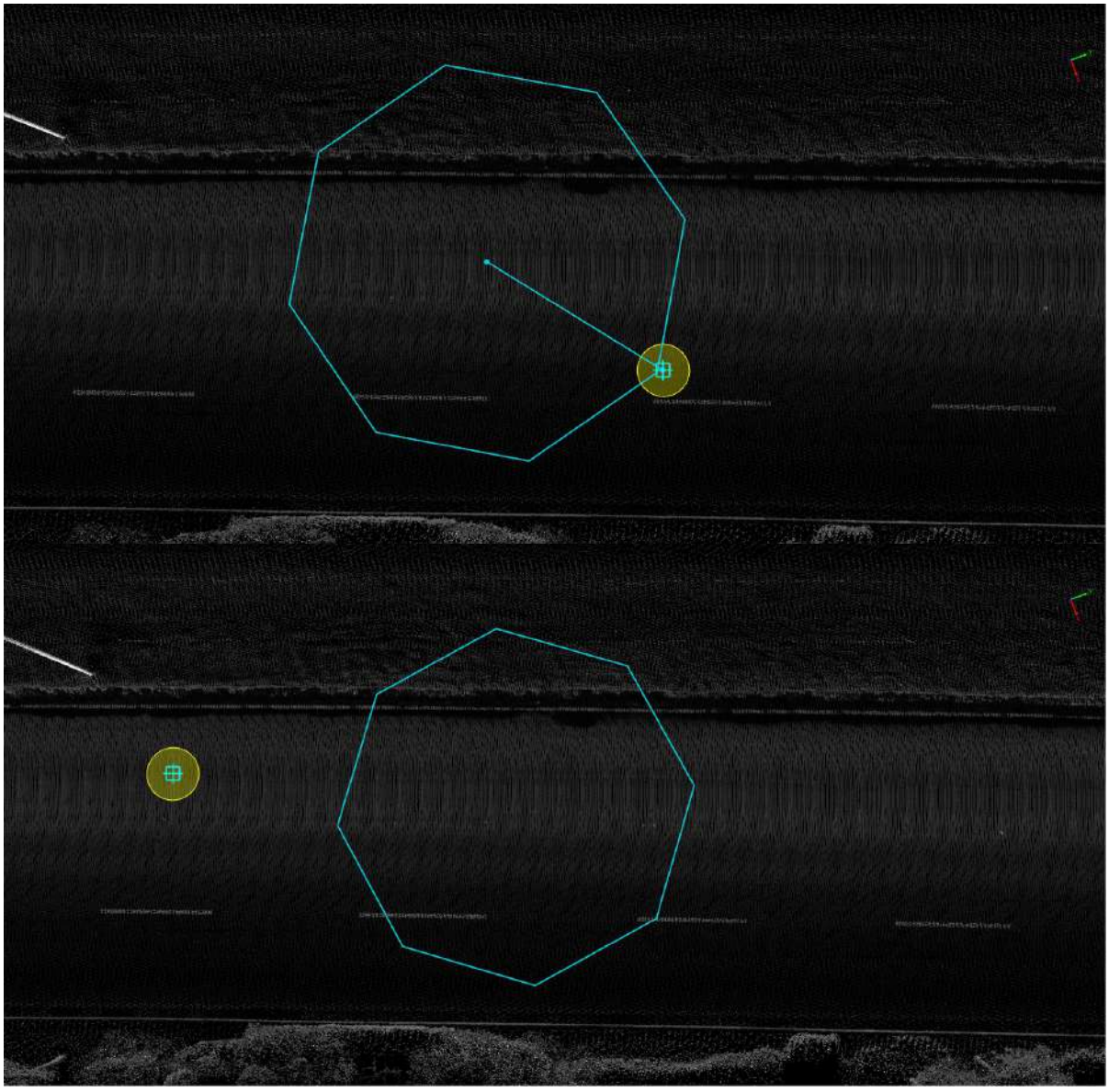
Function Description: This tool allows the user to add an octagon vector object by using the center point and dragging the mouse.

Steps

1.Select a surface layer, such as a surface facility layer.

2.Click **Add Octagon** button in the Vector Editor:

- Click the point cloud with the left mouse button, select the center point, and then drag the mouse, an octagonal real-time display box will appear. After determining the size and direction, click the left mouse button on the point cloud to determine an octagonal vector object.



Drawing octagonal interface

> Note: The geometric features of the vector result must be consistent with the geometric features of the layer to be added. For example, the point-like vector result must be added to the point-like layer, otherwise it will prompt that it cannot be drawn. For the description of layers, see Layer Management

Element Editor

The Element Editor module mainly performs operations such as **moving**, **copying**, **deleting**, and **editing** on vector objects to improve the efficiency of user drawing. It mainly includes the following specific functions:

- Select
  - Pick
  - Rectangle
  - Line
  - Polygon
  - Lasso
  - Circle
  - Box
  - Sphere
  - Cylinder
- Interrupt
  - Point Split Line
  - Line Split Line
  - Line Split Polygon
  - Line Split Objects
  - Layer Split Layer
- Alignment
  - Move
  - Parrallel Copy
  - Rotate
  - Scale Polygon
  - Mirror
- Reshape
  - Reshape Line
  - Reshape Rect
  - Merge Lines
  - Break Line
  - Split Line

    - Extend or Trim
    - Split or Merge Polygon
    - Link Line
    - Extend Line
    - Fillet
    - Combine Polygons or Points
    - Smooth
    - Point To Line
    - Line To Points
    - Construct Polygon
    - Intersection Line
    - Multipart To Singleparts
- Auxiliary
    - Drape
    - Line To Polygon
    - Polygon To Line
    - Draw By Intersection
    - Draw By Perpendicular

Select

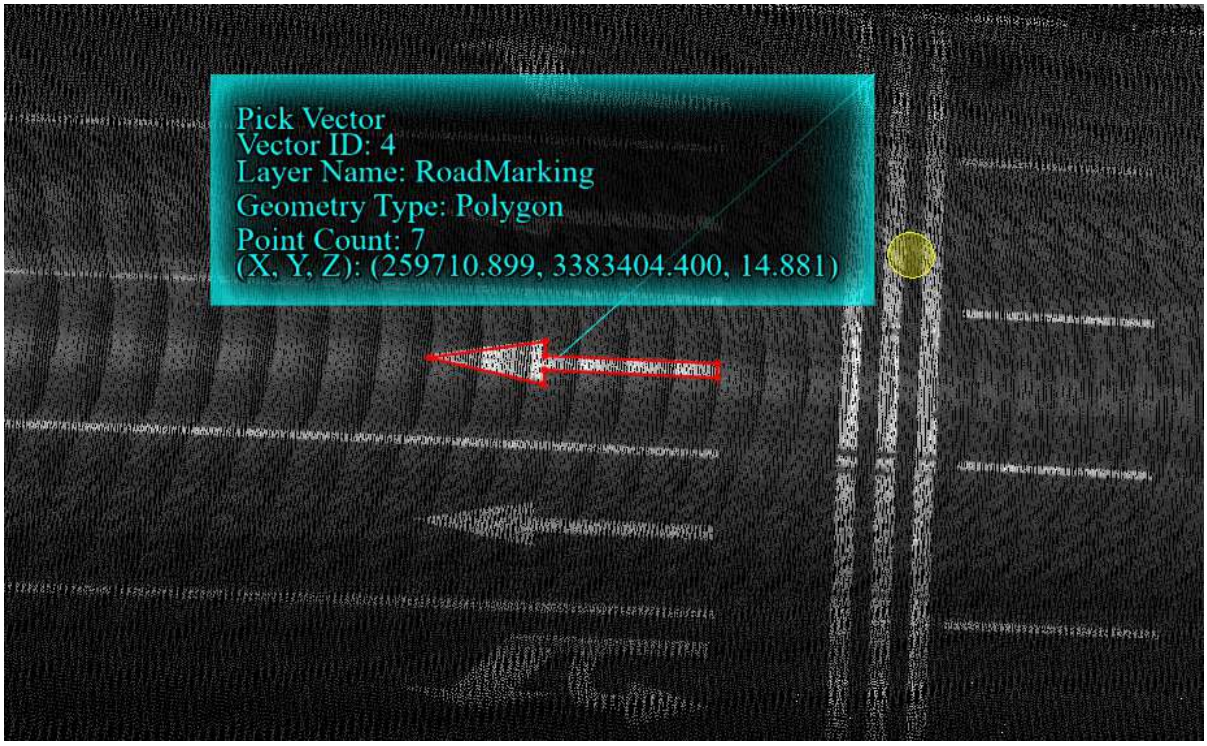
- Pick
- Rectangle
- Line
- Polygon
- Lasso
- Circle
- Box
- Sphere
- Cylinder

Pick

Function Description：The select vector function allows the user to select a vector object and obtain its detailed information.

Steps

1.After clicking the **Pick** button in the Vector Editor, move the mouse to the vector and click the left mouse button, the selected vector will be highlighted, and the main viewport will display the ID number of the current vector. Layer, geometry information and the attribute table window will locate to the row where the attribute field of the currently selected vector is located.



Select vector

2.After clicking, you can directly use the shortcut keys to copy, delete and other operations.

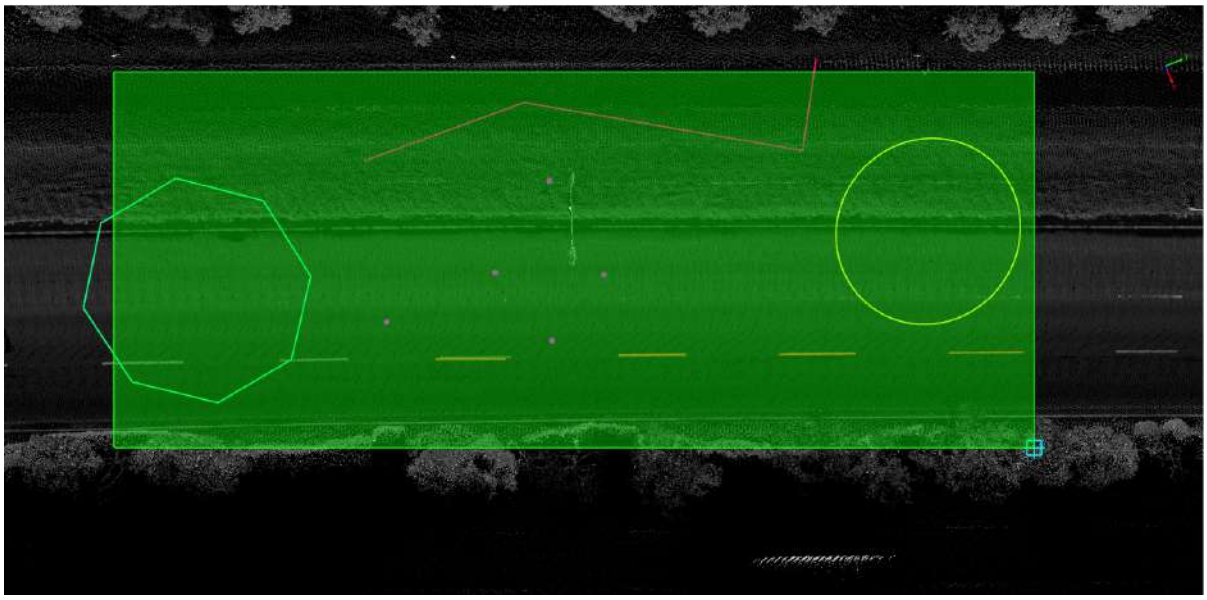
Shortcuts	Function
Ctrl+C	Copy
Ctrl+V	Copy to the same layer
Ctrl+Alt+V	Copy to any layer with the same geometry (pop-up layer is optional)
Delete	Delete
Shift+R	For line vector objects, change the direction of the line
Ctrl+Shift+R	For unselected vector objects, it becomes selected; for selected vector objects, it switches to unselected status

Rectangle

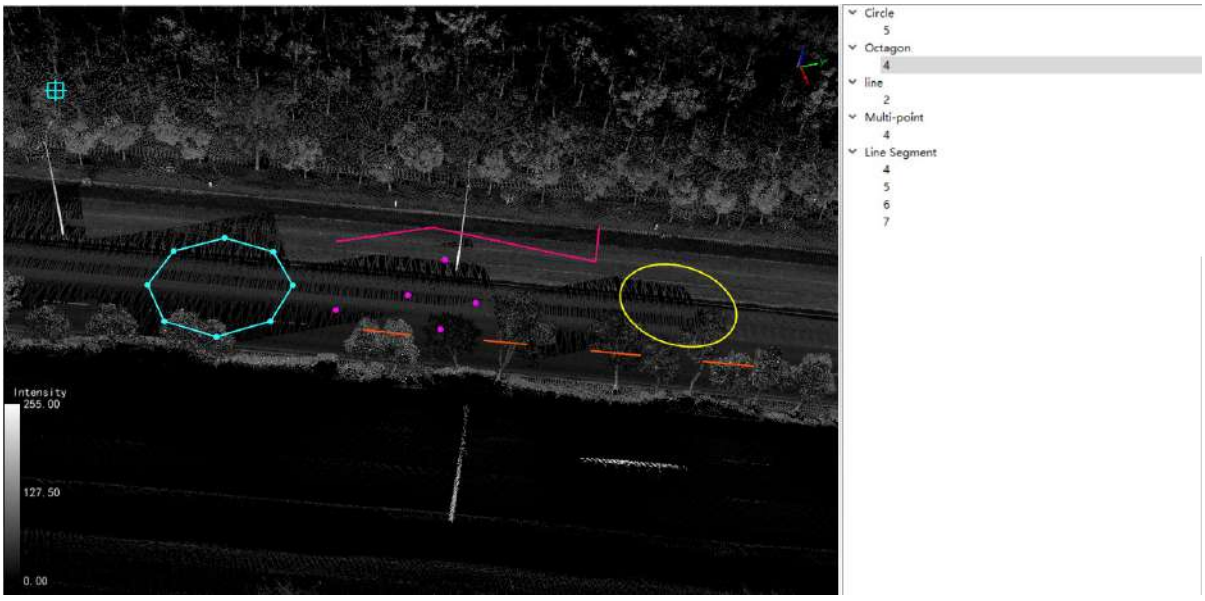
Function Description：The Rect Pickup Vector function allows users to select a series of vector objects by using a rectangle frame.

Steps

1.Click **Rectangle** button in the Vector Editor, Left click for the first time, draw the upper left point of the frame selection rectangle, drag the mouse, draw the rectangle frame, left click again, the frame selection rectangle is drawn, and the polygon inside the frame and the polygon that intersects with the frame will be selected and highlighted show.



Section selection vector



Section selection vector - vector tree

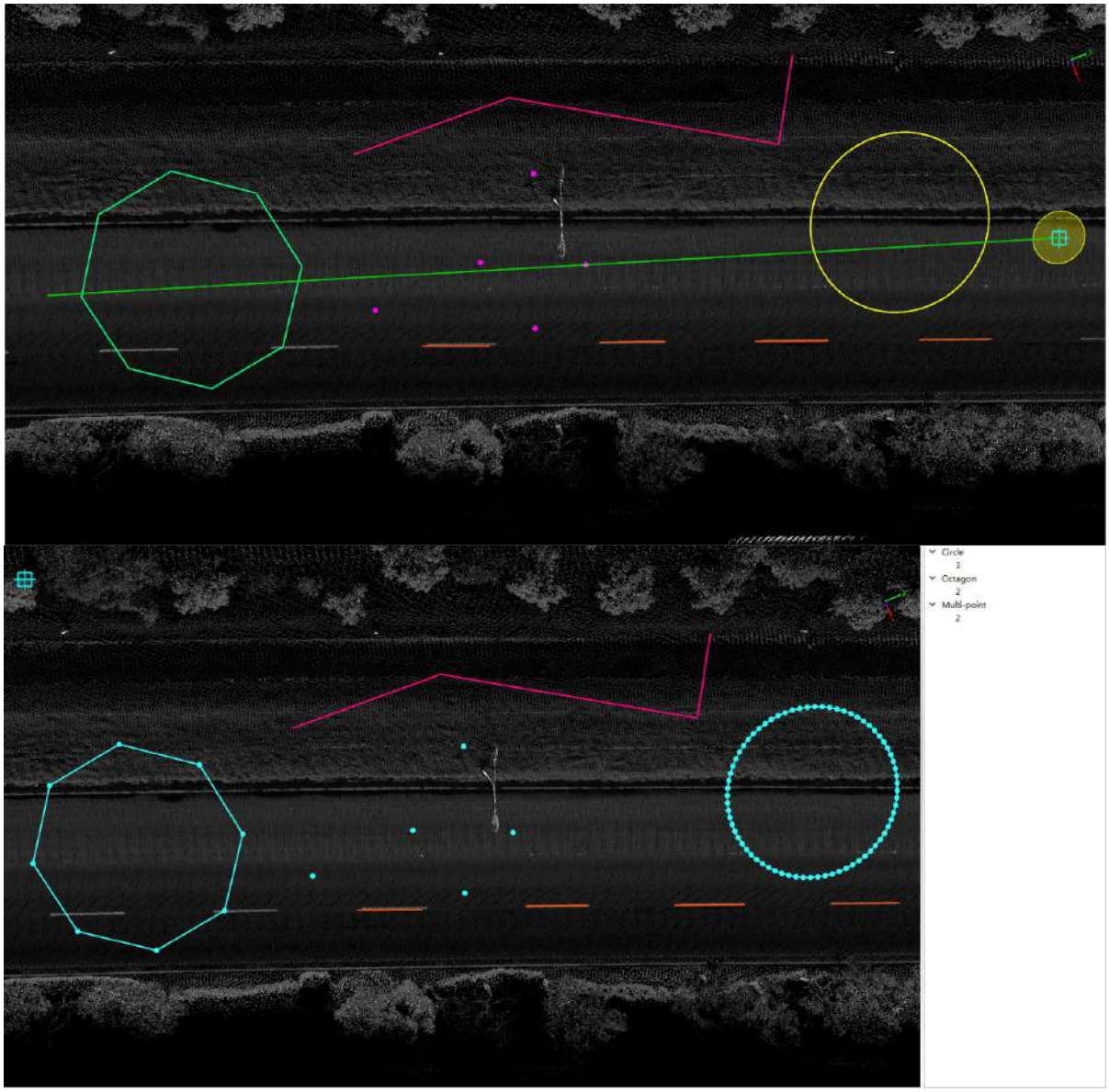
> Note: Select the shortcut key to copy, delete and other operations are the same as Pickup Vector.

Line

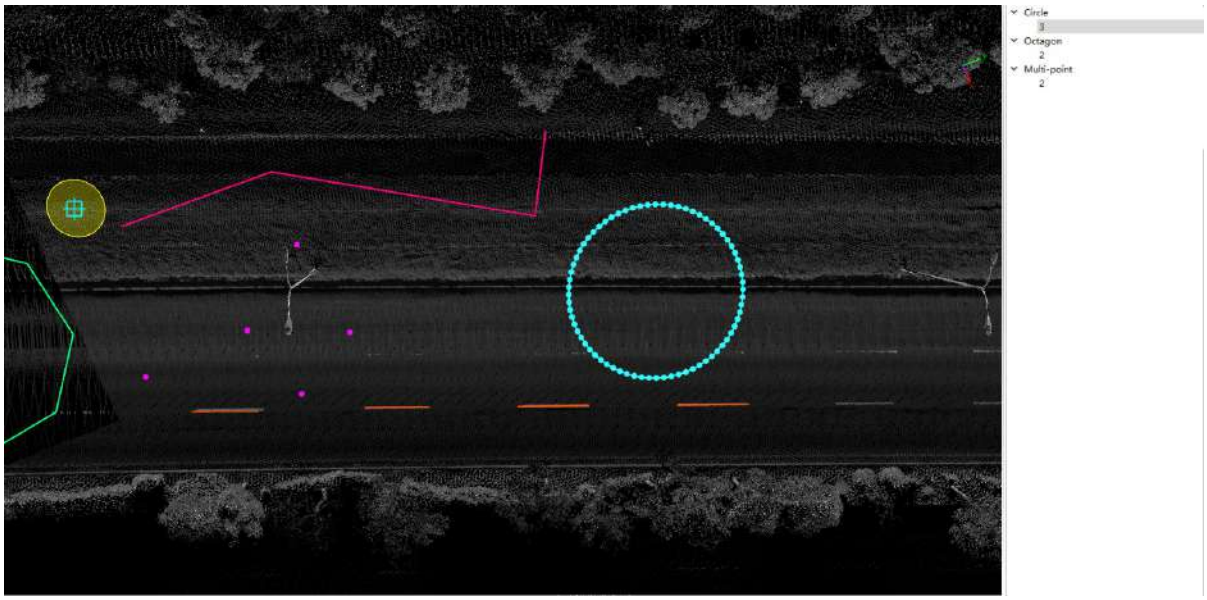
Function Description：The Pickup by Line function allows users to select multiple objects by drawing lines and operate them.

Steps

1.After clicking the **Line** button in the Vector Editor, click the left mouse button to draw a line segment, the vector objects intersecting with the line segment will be selected and highlighted, and the selected elements in the attribute table window will also be highlighted.



Line selection vector



Framed Vector - Vector Tree

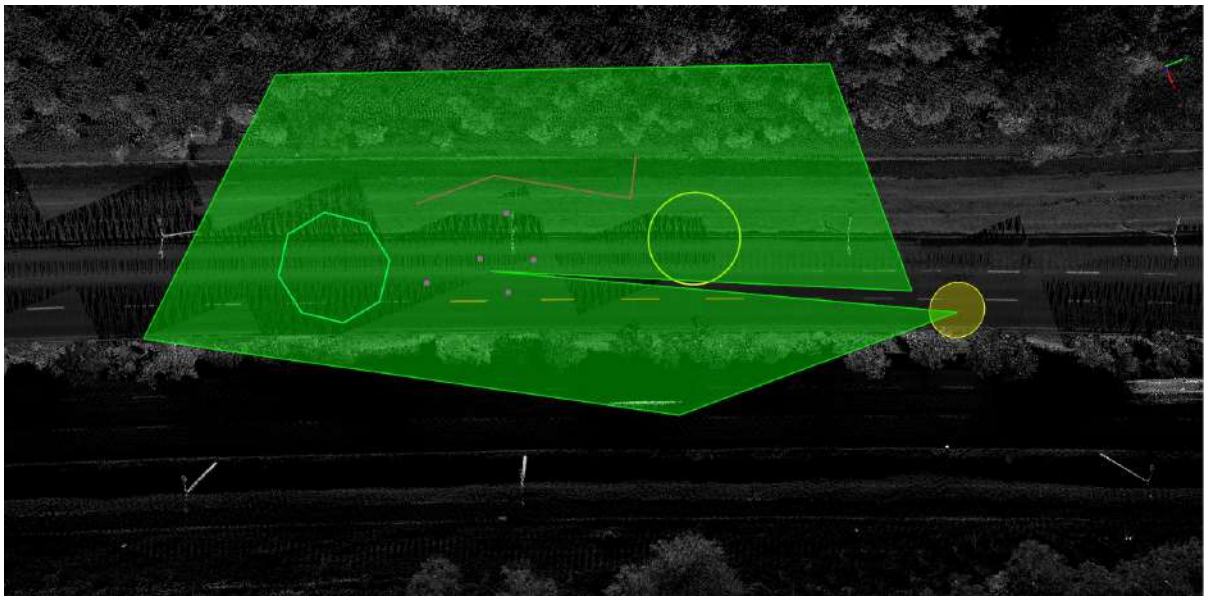
> Note: Select the shortcut key to copy, delete and other operations are the same as Pickup Vector.

Polygon

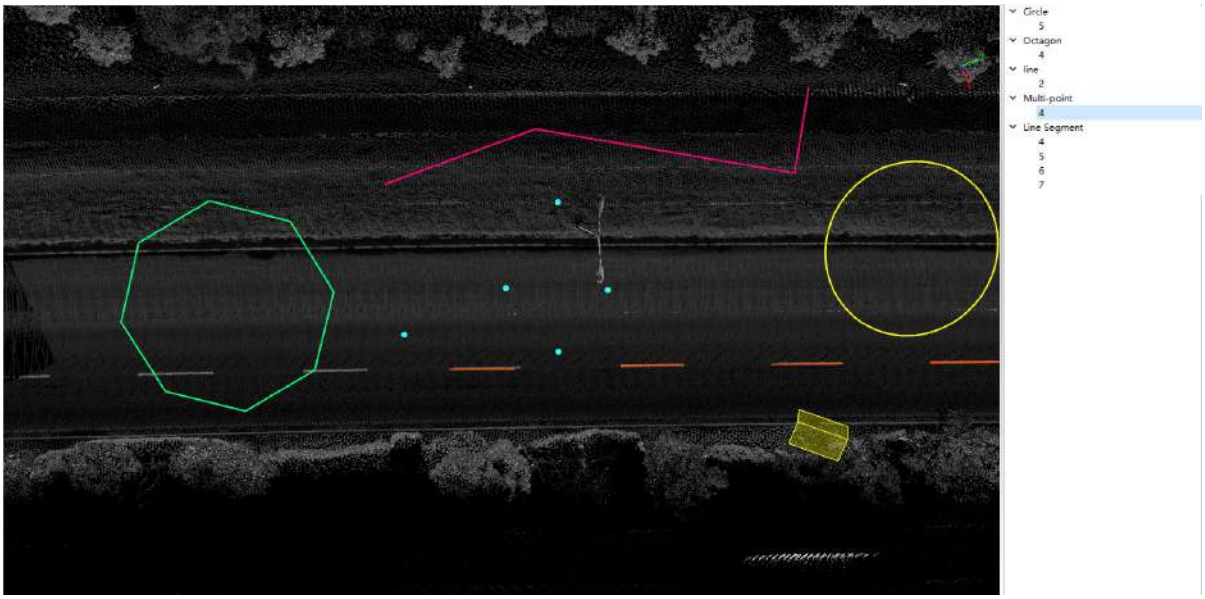
Function Description: The Pickup by Polygon function allows users to select a series of vector objects by using a polygon frame.

Steps

1.Click **Polygon** button in the Vector Editor, use the mouse to continuously click multiple points to determine the nodes of the polygon box, double-click the left button of the mouse to select the polygon box and draw it, and the vectors in the box and intersecting with the box will be selected and highlighted. If multiple vectors are selected, a tree-structured dialog box will pop up on the right side of the view, including all currently selected vectors. Clicking the node with the left mouse button will deselect other vectors, and only select the corresponding vector and highlight it.



Polygon selection vector



Polygon frame vector - vector tree

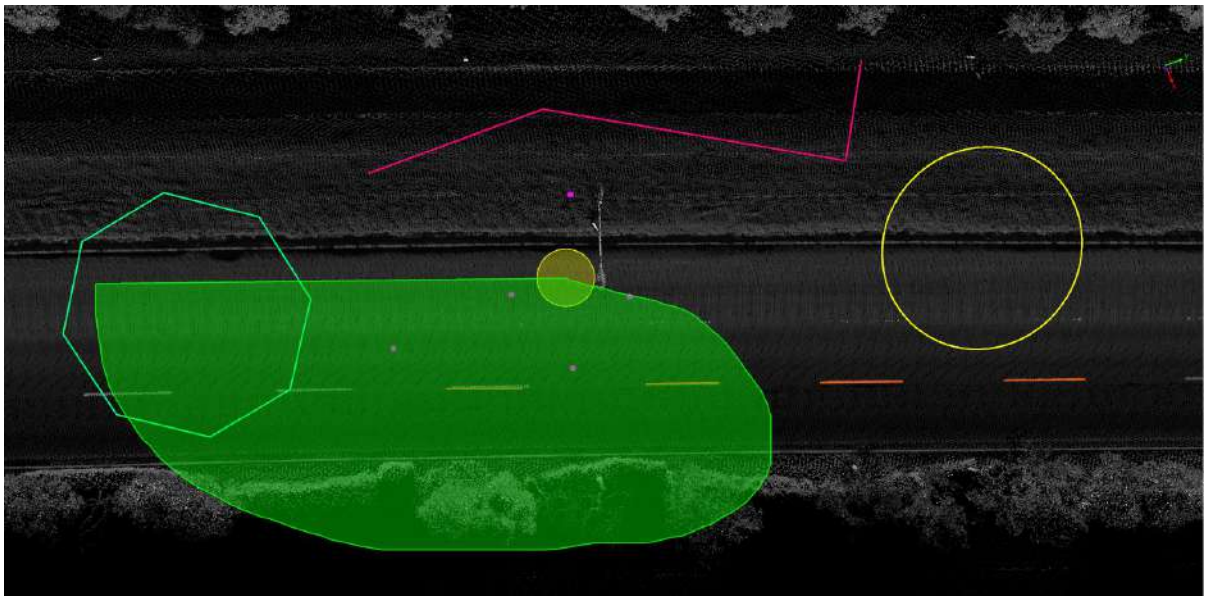
> Note: Select the shortcut key to copy, delete and other operations are the same as Pickup Vector.

Lasso

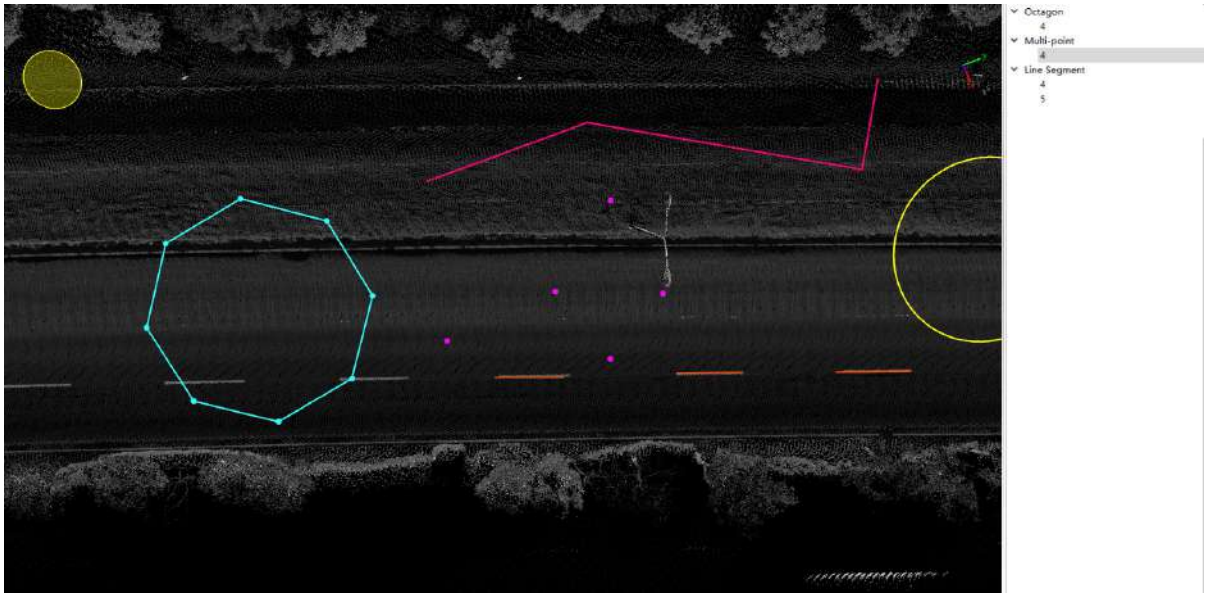
Function Description: The Pickup by Lasso function allows users to move the mouse sleeve to select a series of vector objects.

steps

1.Click **Lasso** button in the Vector Editor, select a key point with the left mouse button, hold down the left mouse button and drag to form an area, and finally release the left mouse button, the vector in the area and its intersection will be selected and highlighted. If multiple vectors are selected, a tree-structured dialog box will pop up on the right side of the view, including all currently selected vectors. Clicking the node with the left mouse button will deselect other vectors, and only select the corresponding vector and highlight it.



Lasso selection vector



Lasso selection vector - vector tree

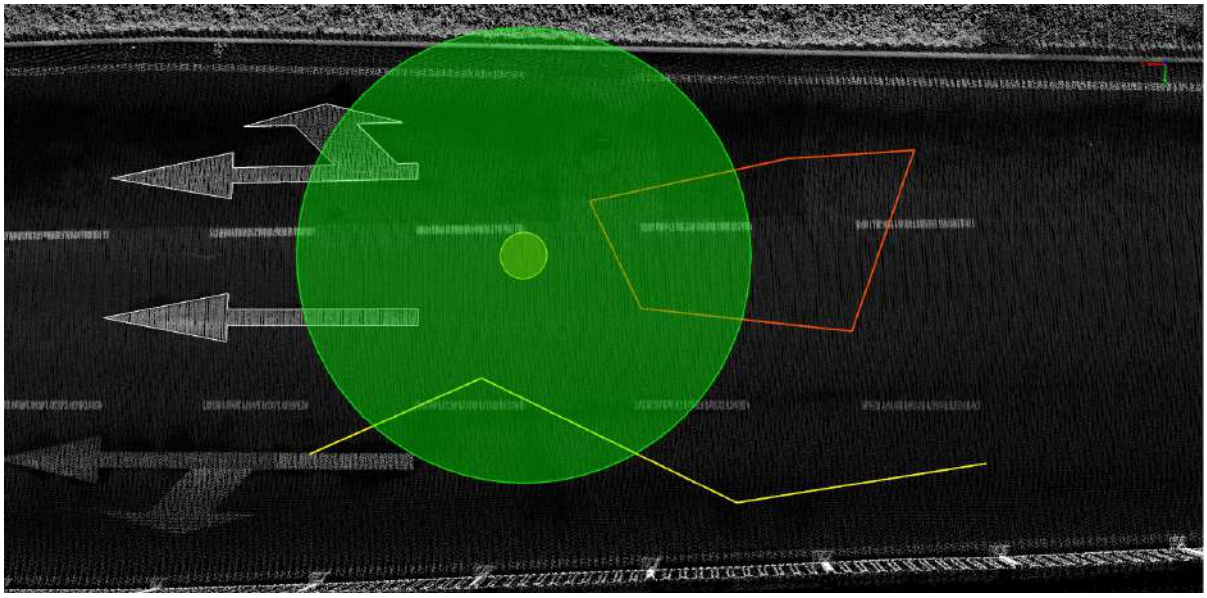
> Note: Select the shortcut key to copy, delete and other operations are the same as Pickup Vector.

Circle

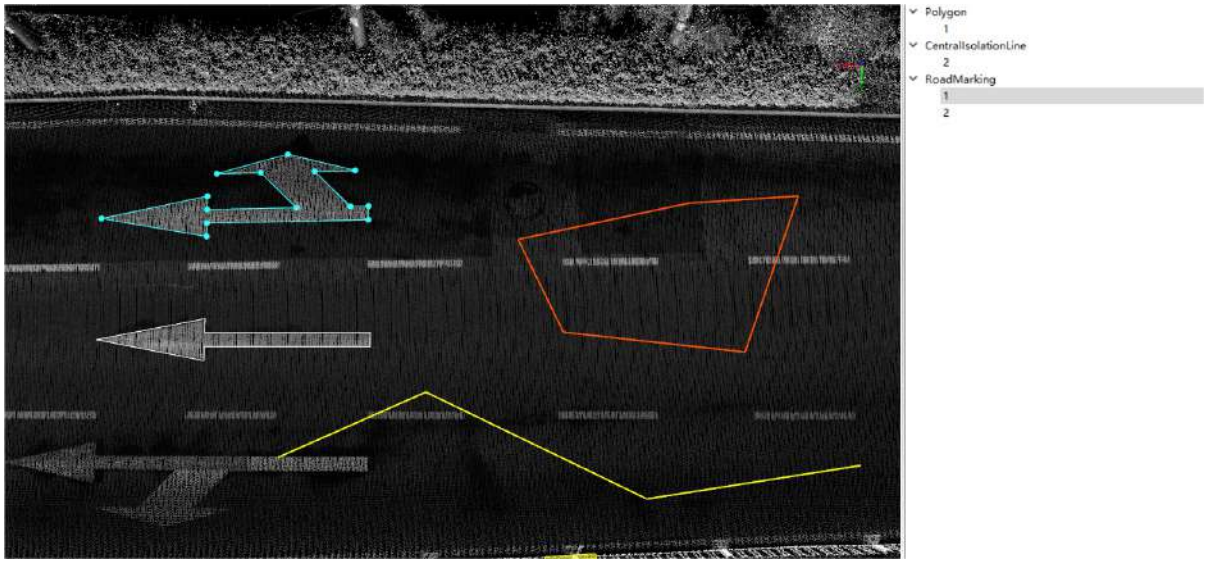
Function Description: The Pickup by Circle function allows users to use circles to select a series of vector objects.

steps

1.Click **Cicle** button in the Vector Editor, select a key point with the left mouse button, hold down the left mouse button and drag to form a circular display box, and finally release the left mouse button, the vector within the circle and intersecting with the circle will be selected and highlighted. If multiple vectors are selected, a tree-structured dialog box will pop up on the right side of the view, including all currently selected vectors. Clicking the node with the left mouse button will deselect other vectors, and only select the corresponding vector and highlight it.



Circle selection vector



Circle selection vector-vector tree

> Note: Select the shortcut key to copy, delete and other operations are the same as Pickup Vector.

Box

Function description: The Pickup by Box function allows users to select a series of vector objects by using a rectangle.

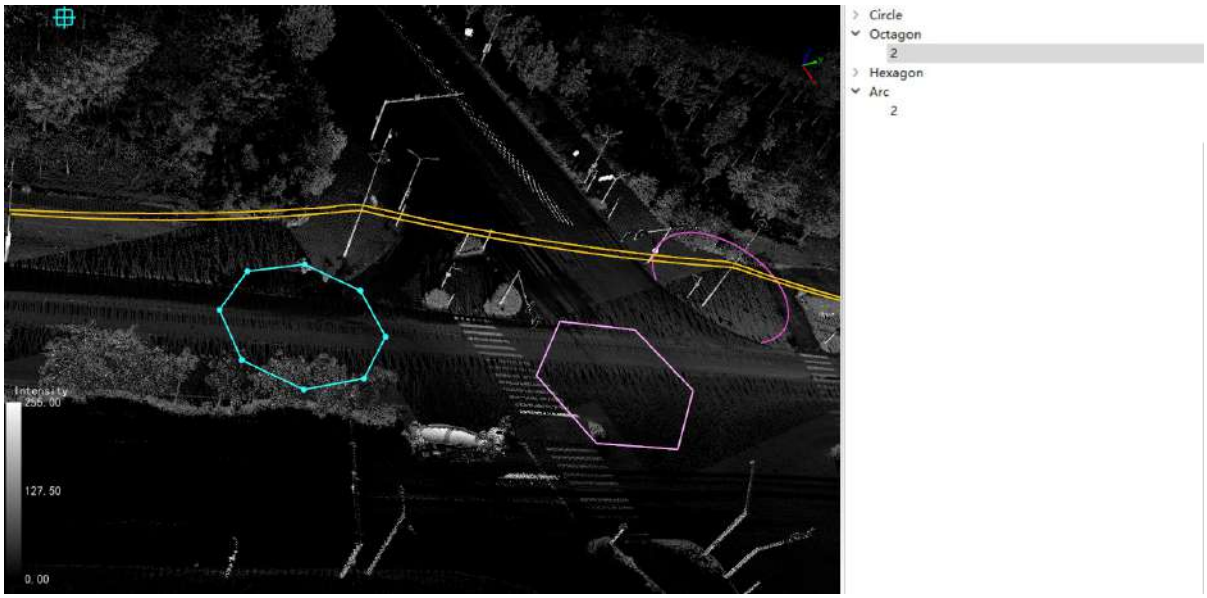
steps

1.Click **Box** button in the Vector Editor:

- Click the point cloud with the left mouse button and select the first key point as the starting point of the cuboid.
- Click the point cloud with the left mouse button, select the second key point, and define an edge of the bottom surface of the box.
- Click the point cloud with the left mouse button, select the third key point, and determine the other side of the bottom of the box.
- Click the point cloud with the left mouse button, select the fourth key point, and determine the height of the cuboid to complete the drawing of the cuboid

2.Vectors inside and intersecting the bounding box will be selected and highlighted. If multiple vectors are selected, a tree-structured dialog box will pop up on the right side of the view, including all currently selected vectors. Clicking the node with the left mouse button will deselect other vectors, and only select the corresponding vector and highlight it.

Bounding box selection vector



Bounding box selection vector-vector tree

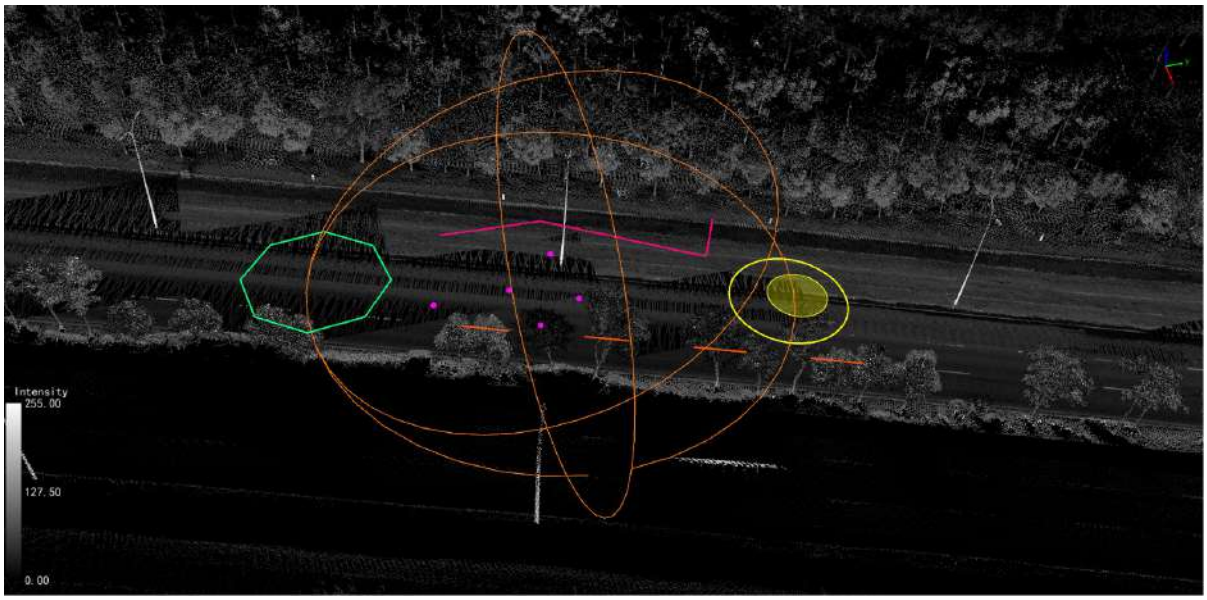
> Note: Select the shortcut key to copy, delete and other operations are the same as Pickup Vector.

Sphere

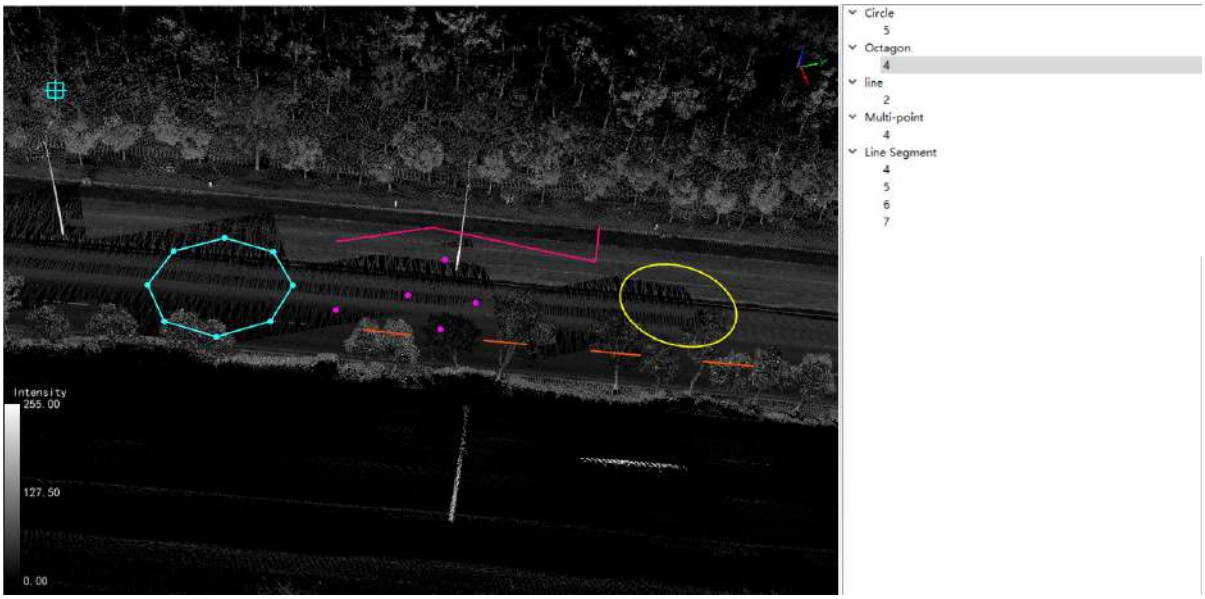
Function Description: The Pickup by Sphere function allows users to select a series of vector objects using spheres.

Steps

1.Click **Sphere** button in the Vector Editor, click and drag the mouse with the left mouse button to display a spherical display. Finally, after clicking the left mouse button, the vectors within the sphere and intersecting with the ball will be selected and highlighted. If multiple vectors are selected, a tree-structured dialog box will pop up on the right side of the view, including all currently selected vectors. Clicking the node with the left mouse button will deselect other vectors, and only select the corresponding vector and highlight it.



Spherical selection vector



Spherical selection vector - vector tree

> Note: Select the shortcut key to copy, delete and other operations are the same as Pickup Vector.

Cylinder

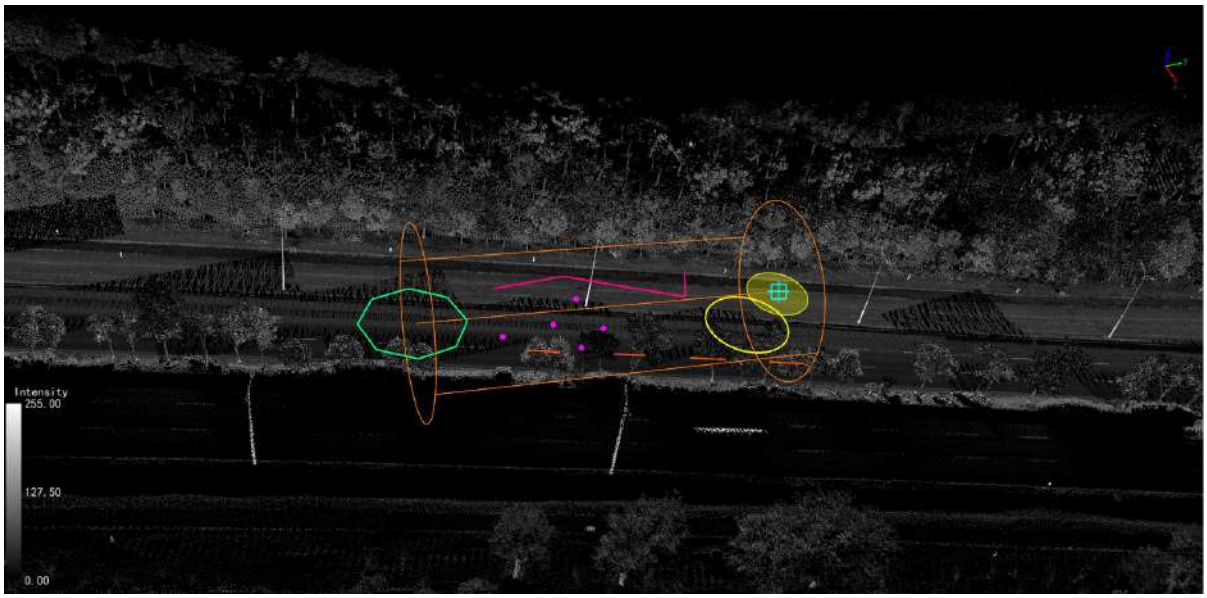
Function Description: The Pickup by Cylinde function allows users to select a series of vector objects using cylinders.

steps

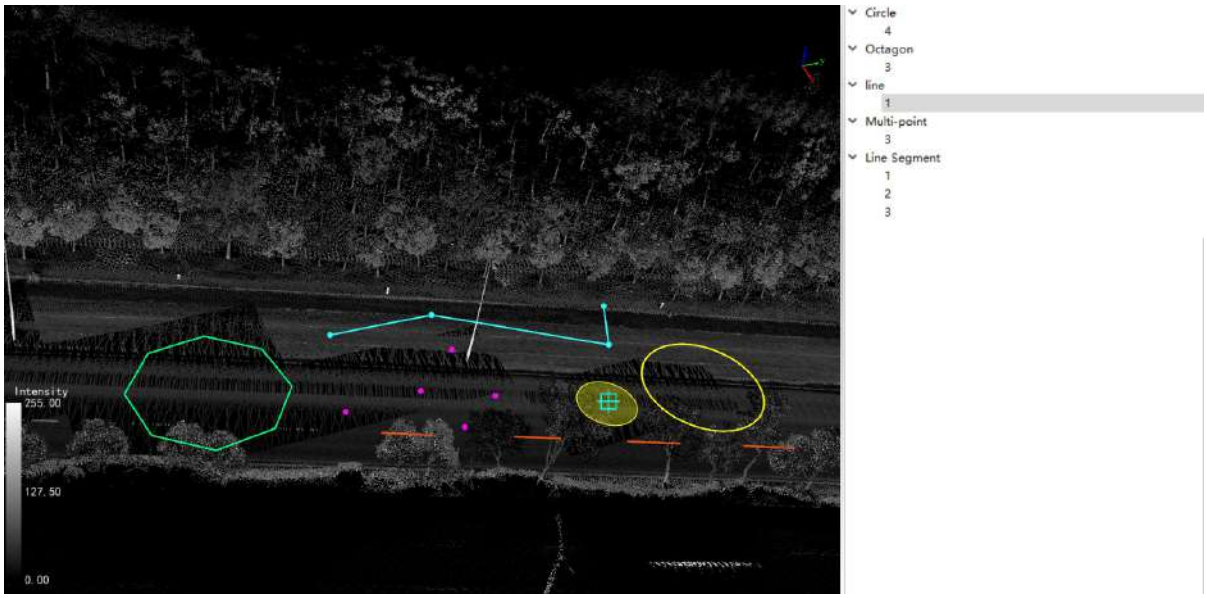
1.Click **Cylinder** button in the Vector Editor:

- Click the point cloud with the left mouse button and select the first key point as the center point of the bottom surface of the cylinder.
- Click the point cloud with the left mouse button, select the second key point, and determine the center point of the other base circle of the cylinder.
- Click the point cloud with the left mouse button, select the third key point, and determine the size of the bottom circle of the cylinder to complete the drawing of the cylinder.

2.Select a key point with the left mouse button, hold down the left mouse button and drag to form a circular display box, and finally release the left mouse button, the vectors within the circle and intersecting with the circle will be selected and highlighted. If multiple vectors are selected, a tree-structured dialog box will pop up on the right side of the view, including all currently selected vectors. Clicking the node with the left mouse button will deselect other vectors, and only select the corresponding vector and highlight it.



Cylinder selection vector



Cylinder selection vector-vector tree

> Note: Select the shortcut key to copy, delete and other operations are the same as Pickup Vector.

Interrupt

- Point Split Line
- Line Split Line
- Line Split Polygon
- Line Split Objects
- Layer Split Layer

Point Split Line

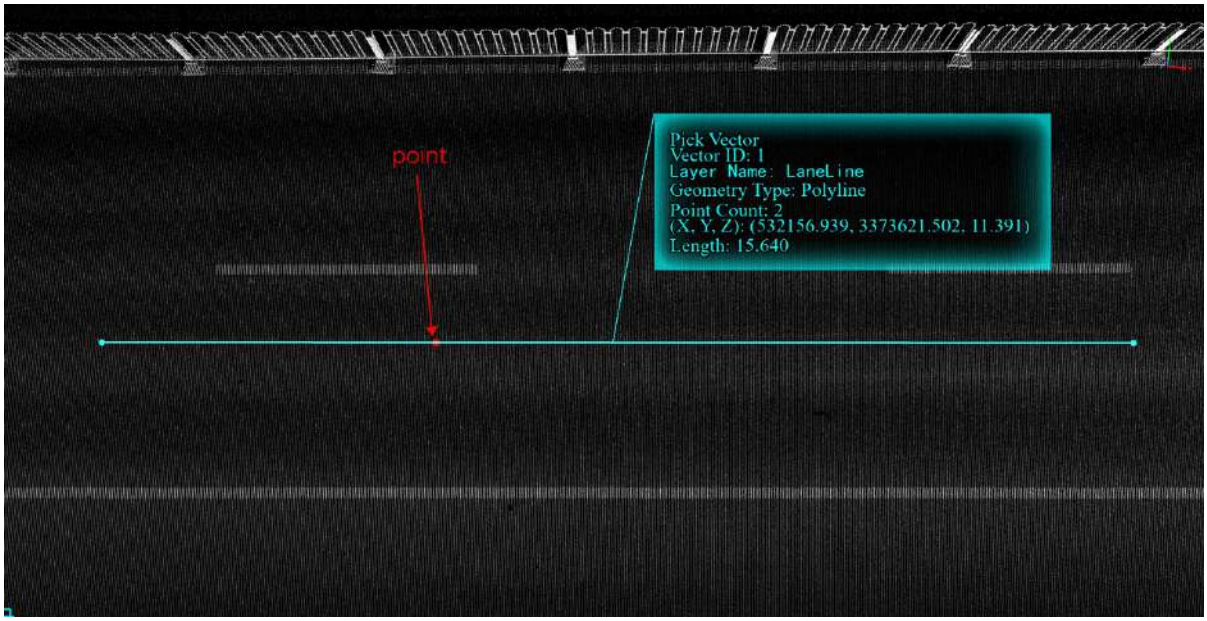
Function Description：The Point Split Line function allows the user to break a line object into two end-to-end line objects through a point on the line.

Steps

1.After clicking the **Point Split Line** button in the Vector Editor, move the mouse to click the point object on the line first, and then click the line object to be broken again, the line object will be broken into two line objects connected end to end.

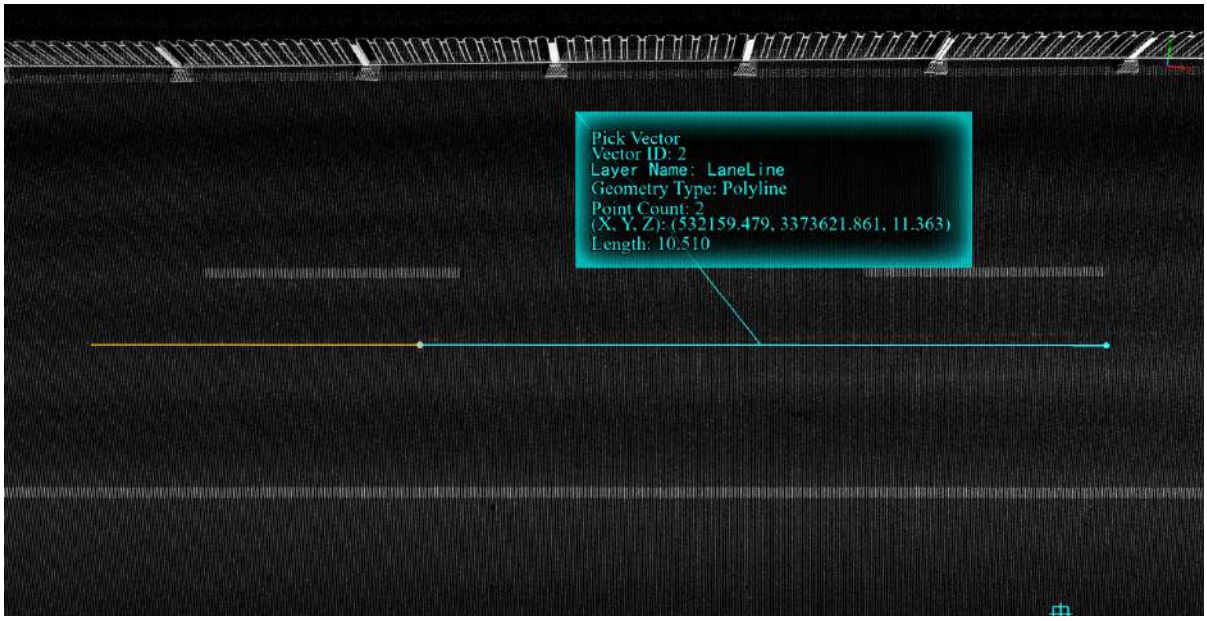
> If the point object is not on the line object, interruption is not allowed, and there will be a prompt in the output window.

- **Original line segment**: a single line object.



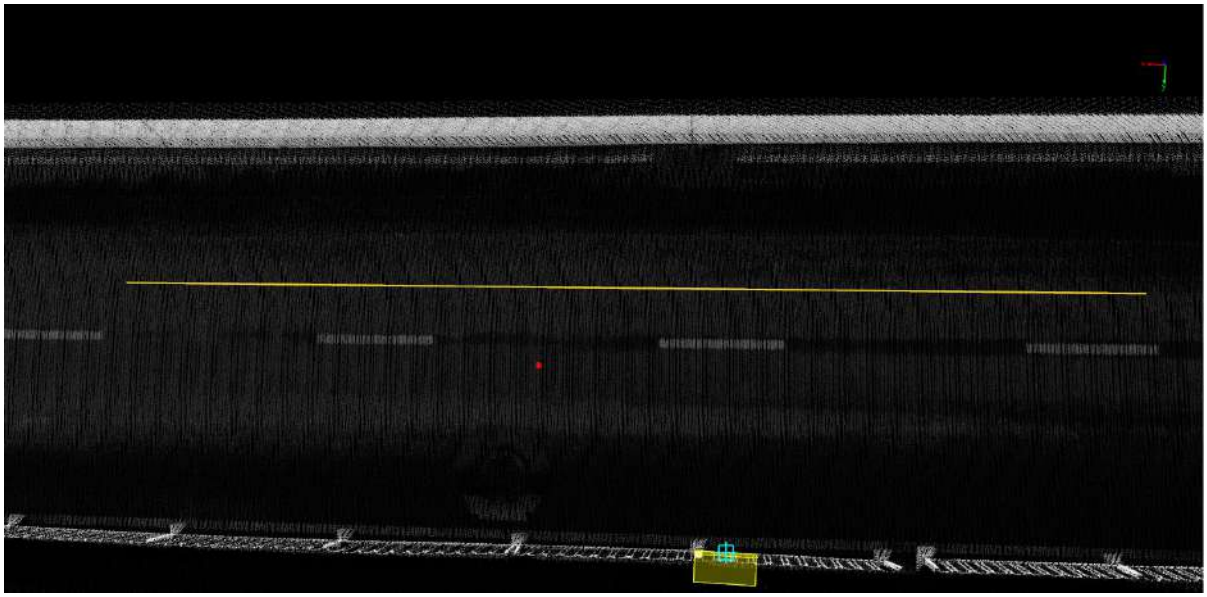
Original line segment

- **Break at the point object on the line segment**: The two lines after the break are connected end to end at the point object.

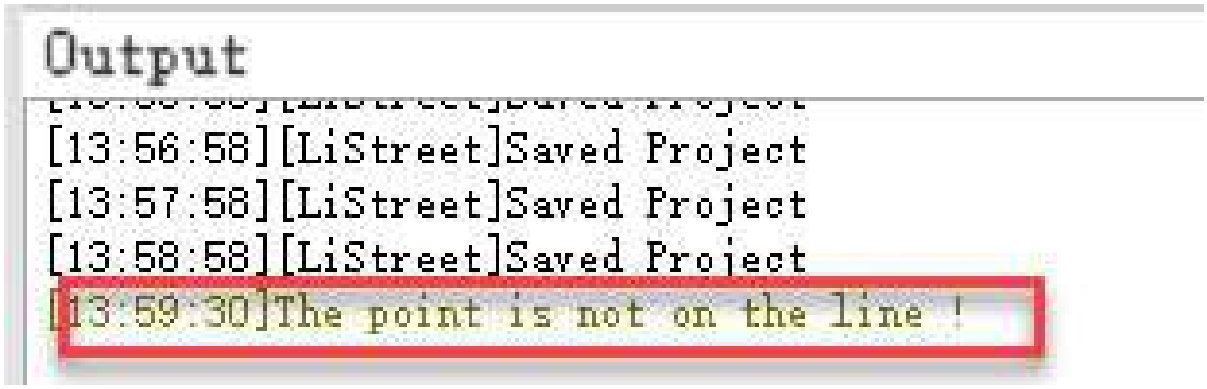


Line segment after break

- **The point is not on the line**: Break will not be allowed, and there will be a prompt in the output window.



Break point not on line



Error prompt

Line Split Line

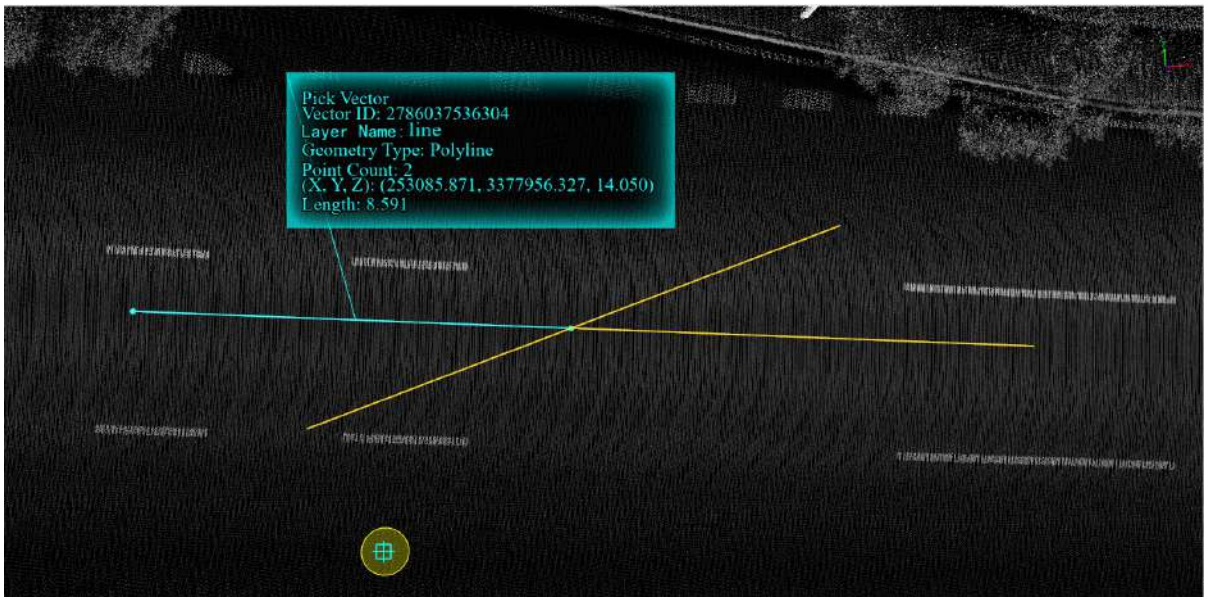
Function Description：The Line Split Line function supports by intersecting one of the two lines, the other line is broken into two lines connected end to end.

Steps

1.Click **Line Split Line** button in the Vector Editor, move the mouse to click one of the intersecting line objects first, and then click the line object to be broken again, the line object will be broken into two line objects connected end to end.

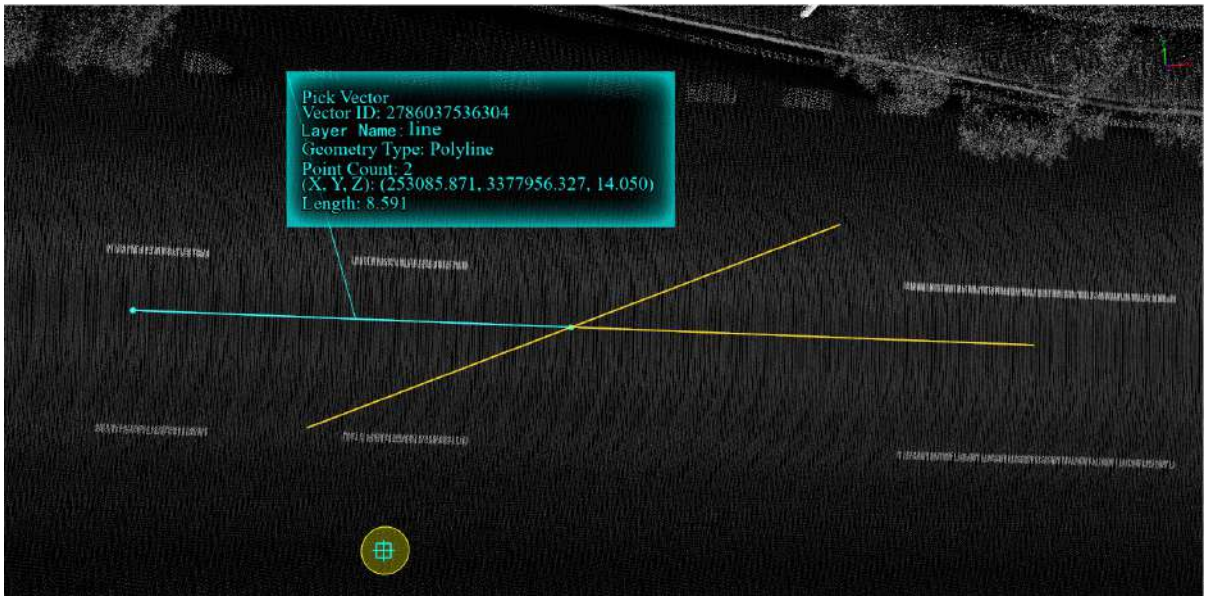
> If the two line objects do not intersect, line break will be invalid, and there will be a prompt in the output window.

- **Original intersecting lines**: Two intersecting line objects.



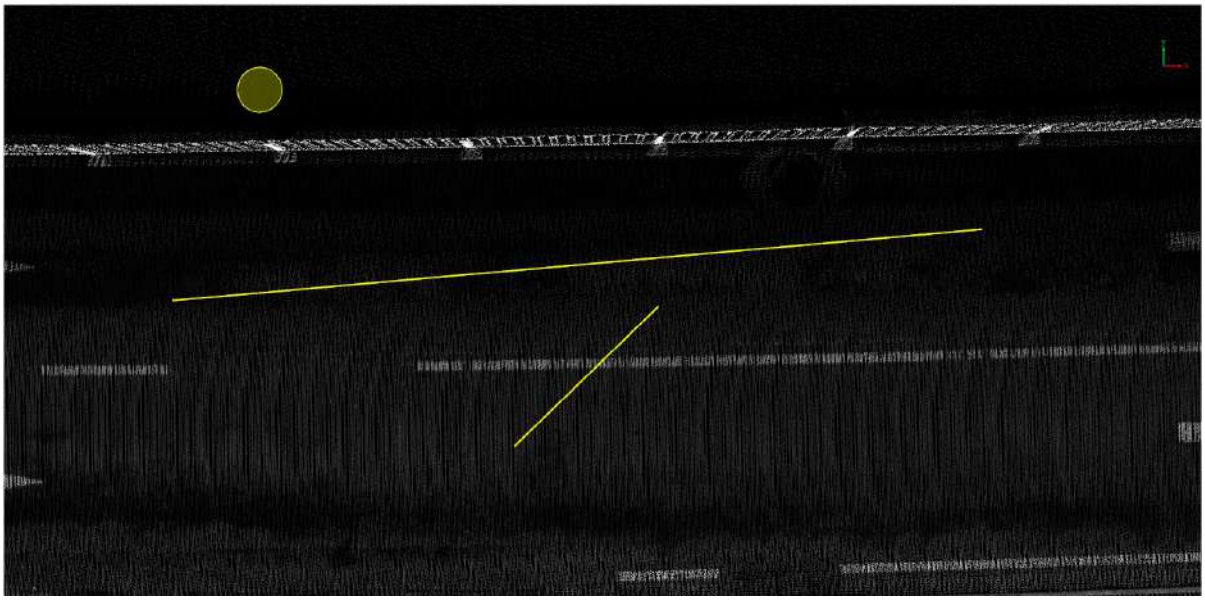
Original line

- **Break at intersection point**: The two lines after the break are connected end to end at the intersection point.

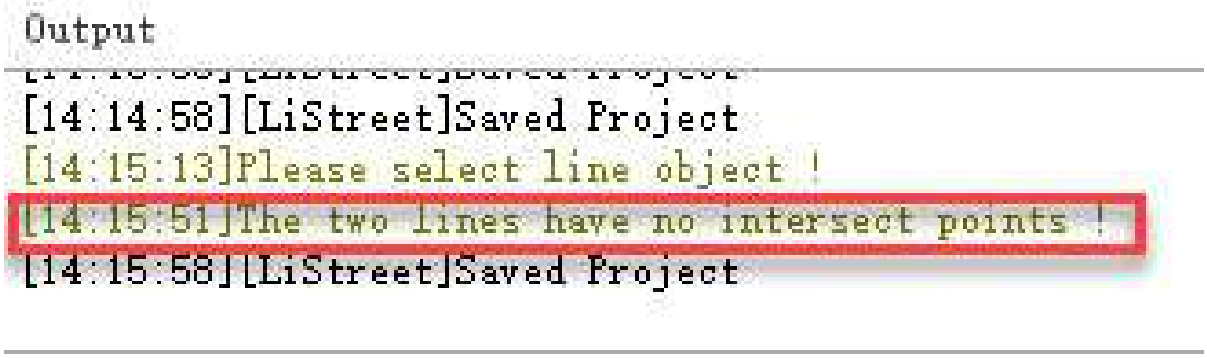


Line segment after break

- **Two line segments do not intersect**: line break is not allowed, and there will be a prompt in the output window.



Two line segments do not intersect



Error prompt

Line Split Polygon

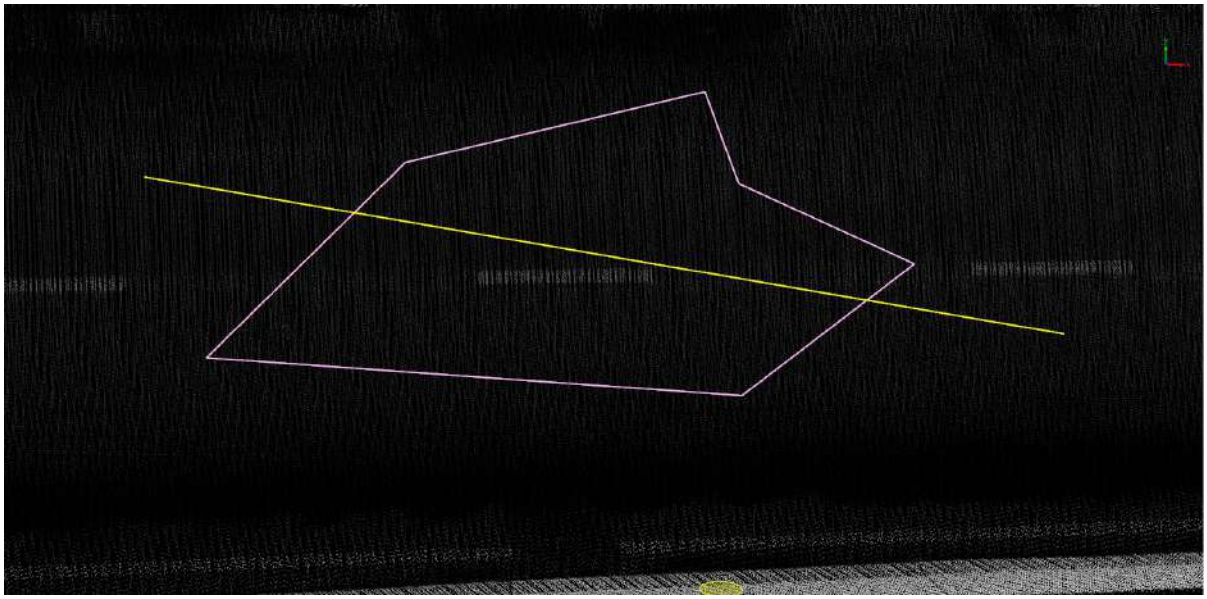
Function Description：The Line Split Polygon feature allows the user to break an area object into two area objects by intersecting a line in the area object.

Steps

1.After clicking the **Line Split Polygon** button in the Vector Editor, move the mouse to first click the line object in the intersecting object, and then click the surface object to be interrupted again, and the surface object will be broken into two connected surface objects.

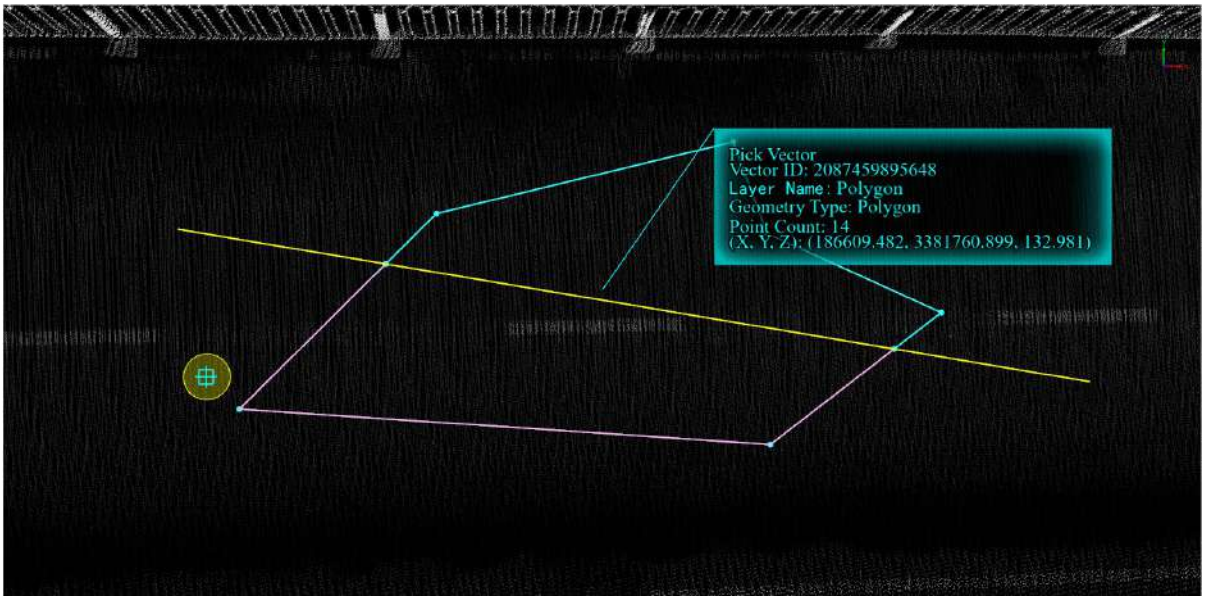
> If the intersection of the line and the polygon is less than two, line break will not be allowed, and there will be a prompt in the output window.

- **Original intersecting lines and polygon**: Two intersecting line and area objects.



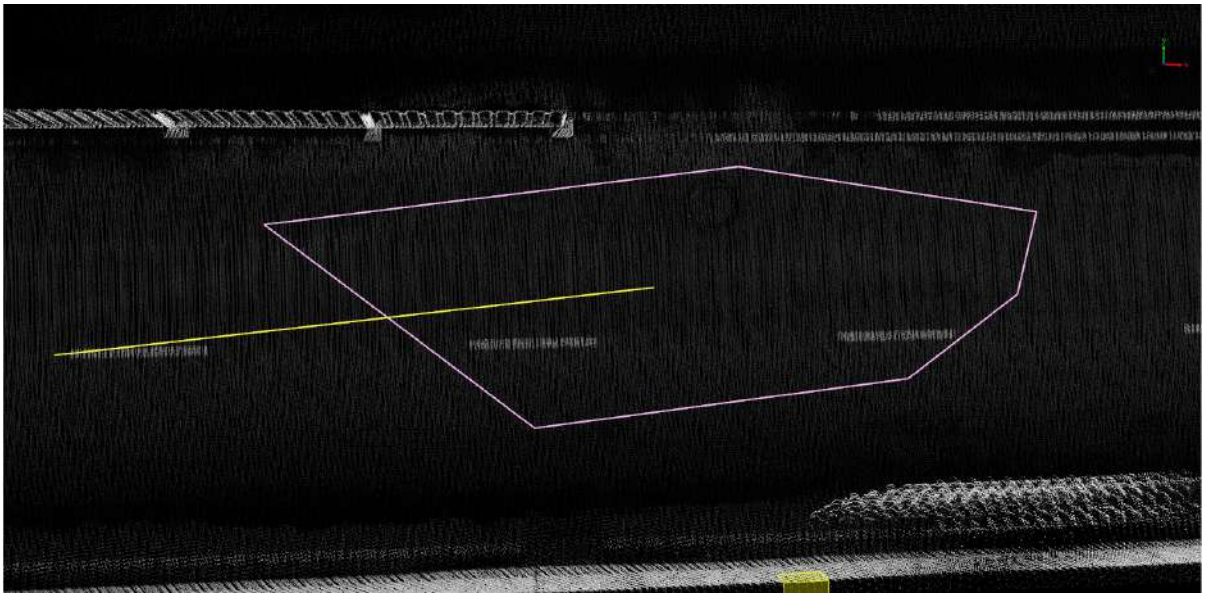
Original lines and polygon

- **Broken polygon**: Two polygons that are connected after being broken.

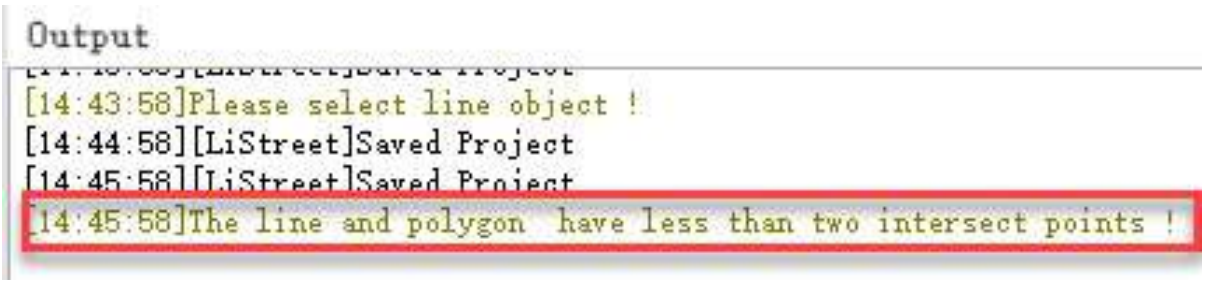


Two polygons connected after break

- **The break points of the polygon is less than 2**: No break is allowed, and there will be a prompt in the output window.



Less than 2 break points



Line-polygon intersection is less than two points

Line Split Objects

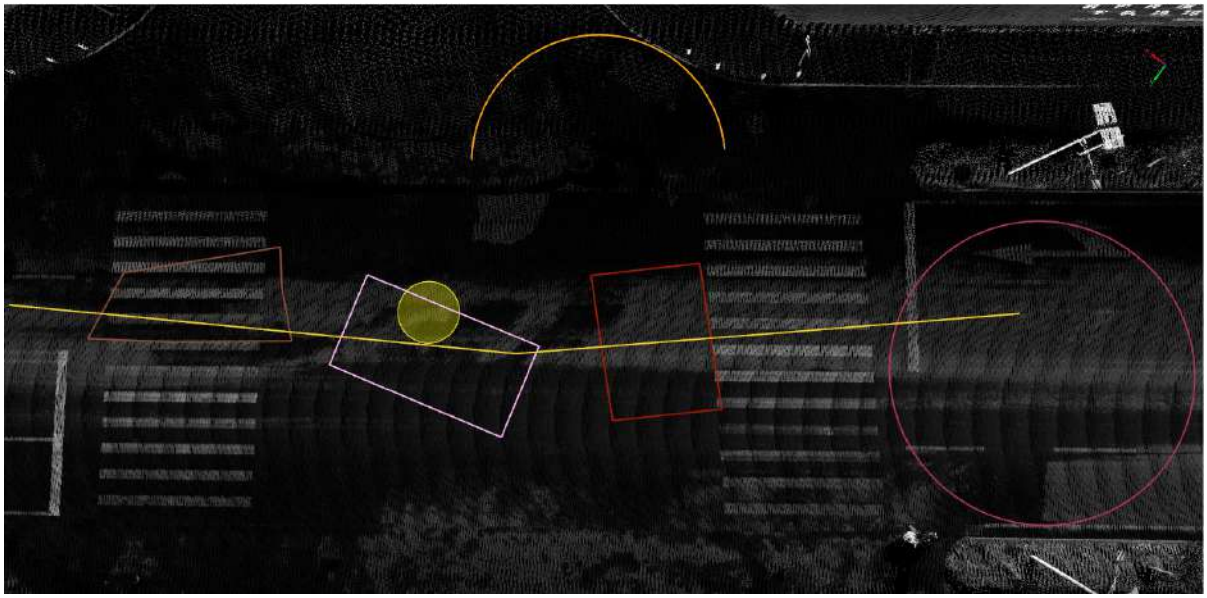
Function Description: This tool allows the user to break the intersecting objects through a line, and break a line or area object into two objects.

Steps

1.Click **Line Split Objects** button in the Vector Editor, click the left mouse button for the first time, draw the upper left point of the rectangle, drag the mouse, determine the range, click the left mouse button again, the rectangle is drawn, and the objects inside the box and intersecting with the box will be selected and highlighted. Move the mouse and click the line object in the intersecting object first, then the object intersecting with the line object will be interrupted into two connected objects.

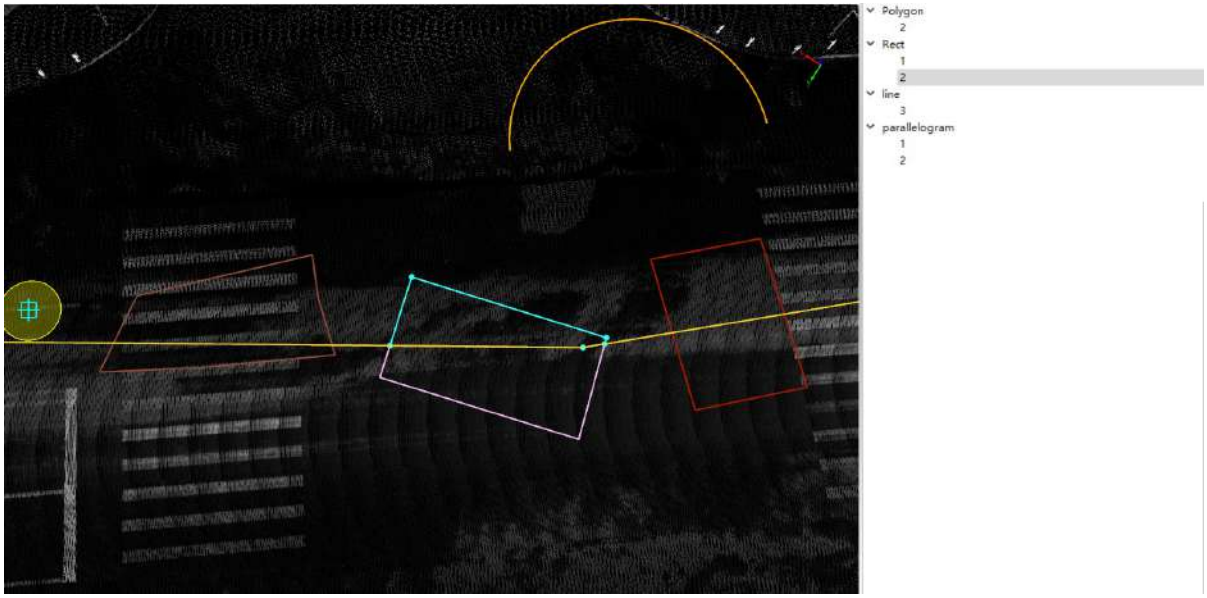
> If the intersection of the line and multiple objects is less than two, interruption is not allowed, and there will be a prompt in the output window.

- **Original intersected line with multiple objects**: Line object intersects with multiple objects.



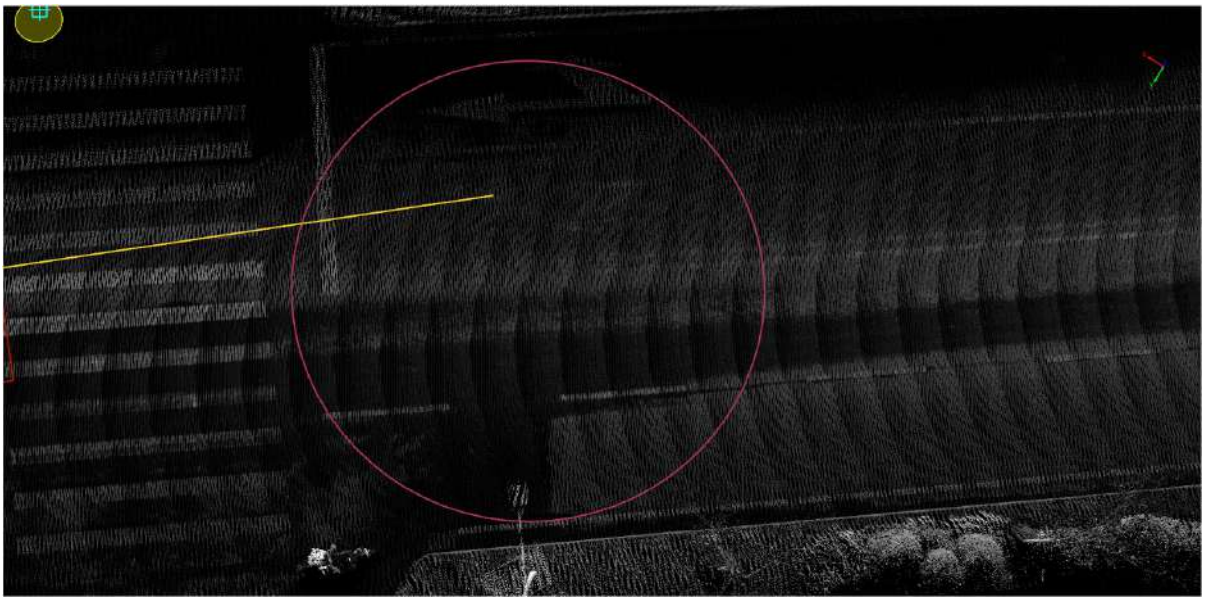
Original intersecting lines with multiple objects

- **Interrupted Objects**: Two objects connected after being interrupted. Use box selection to view the results.



Two objects connected after interruption

- **The intersection of the line and the object is less than two**: Interruption is not allowed, and there will be a prompt in the output window.



The intersection of the line and the object is less than two

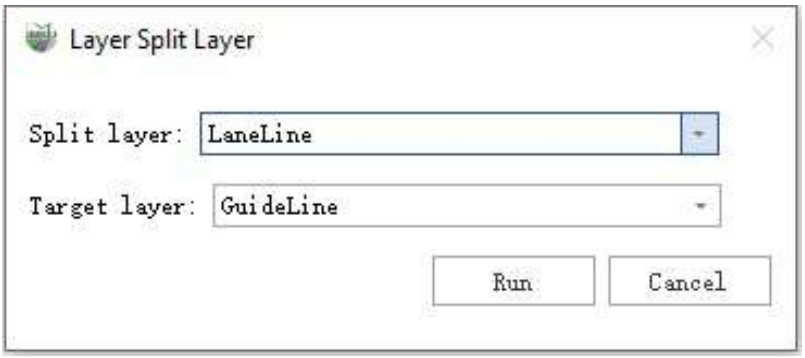
The intersection of the line and the object is less than two prompts

Layer Split Layer

Function Description：This function involves selecting one source layer and one target layer to achieve batch breaking of the target layer at intersecting locations by using the source layer as a reference.

Steps

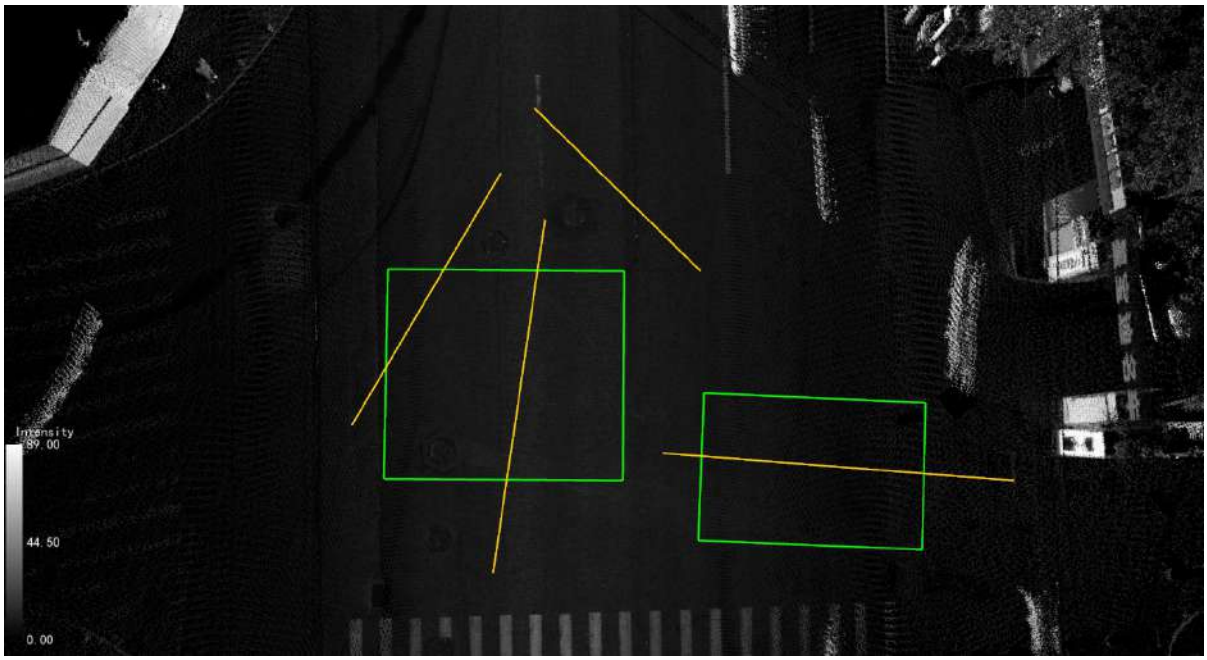
1.After clicking the **Layer Break** button in the vector editing menu, a settings dialog box will appear.



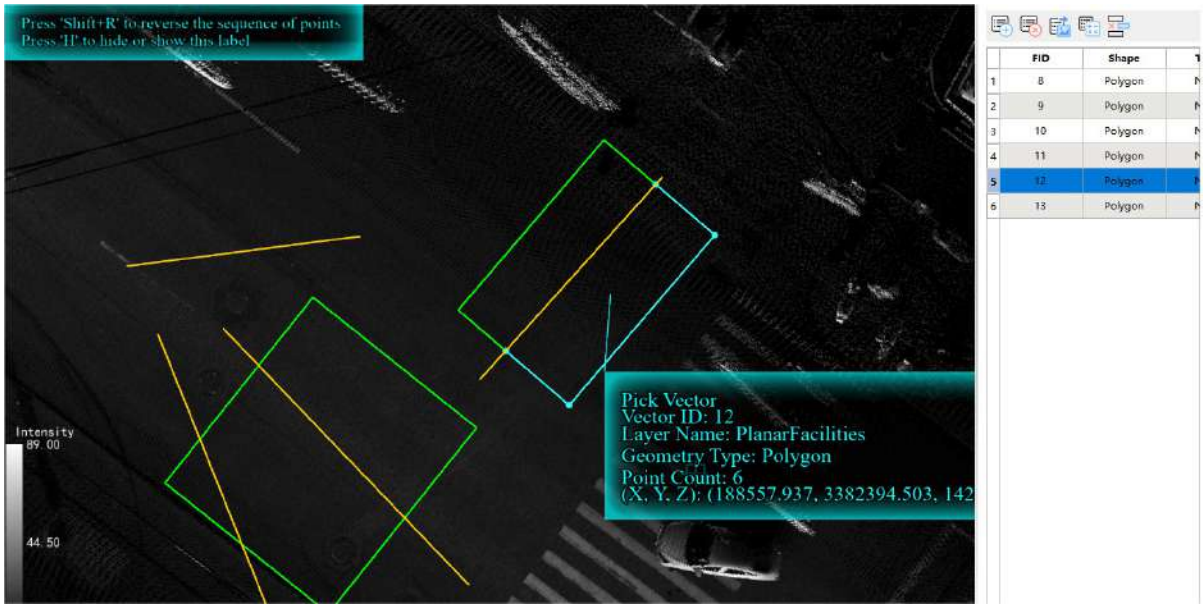
Parameter Description：

- **Split layer**：Refers to the layer used to perform the breaking operation.
- **Target layer**：Refers to the layer that is being broken.

2.Clicking "Run" will cause the software to automatically traverse the features in the source layer and determine whether they intersect with features in the target layer. By default, it will break all intersecting locations, as shown in the image below:

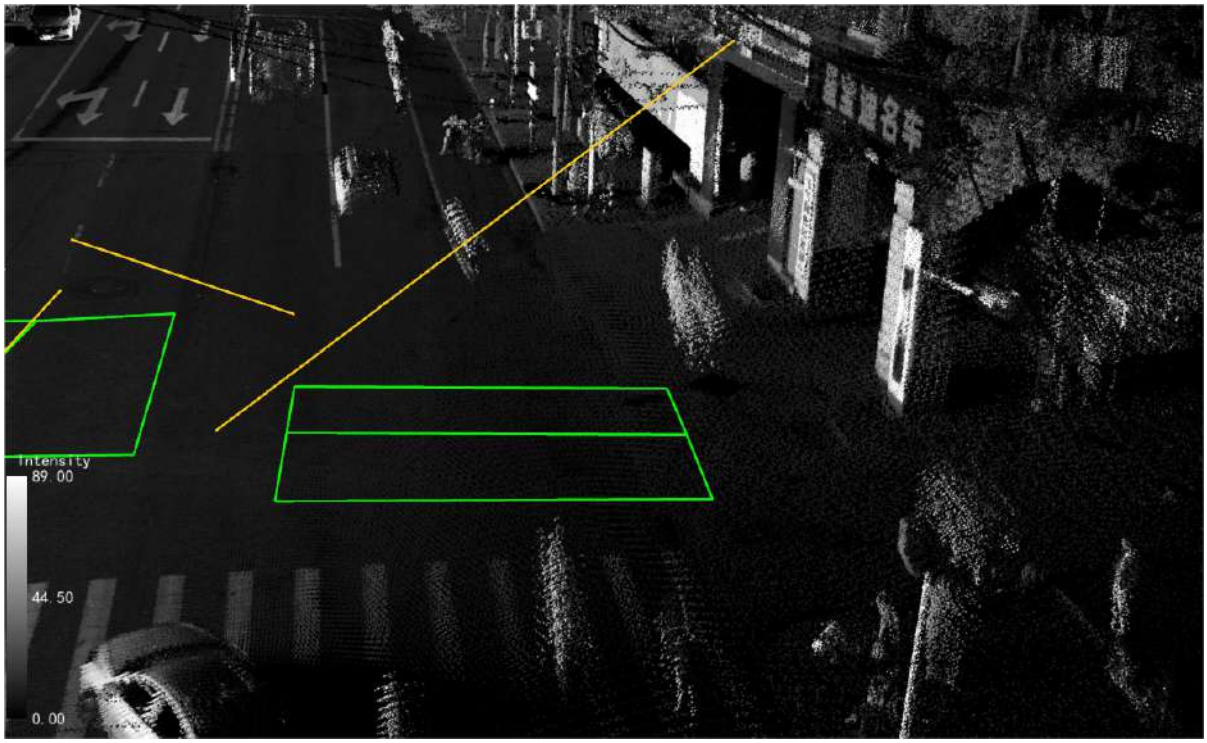


pre-interruption



after the break

3.As shown in the image below, if the source layer and target layer are not coplanar, the software will perform the breaking based on the XY-plane position, and the Z-value will be taken as the average height of the target objects.



coplanar

Alignment

- Move
- Parrallel Copy
- Rotate
- Scale Polygon
- Mirror

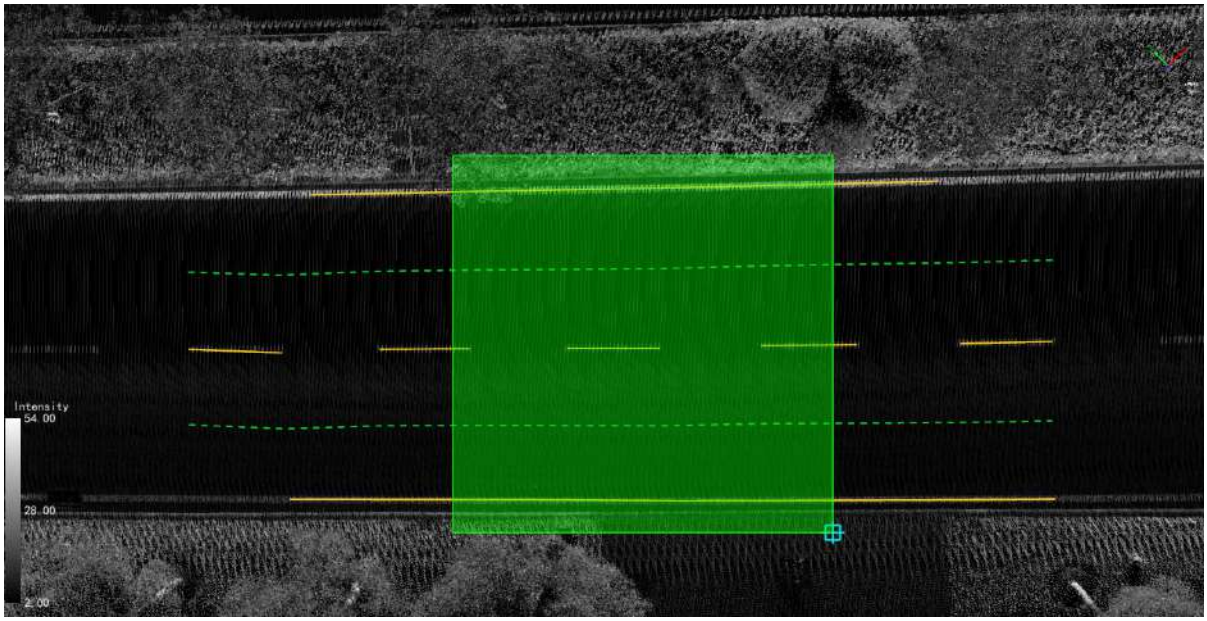
Move

Function description: The Move feature allows users to move a vector object to a new location.

> Note：
>
> 1.This function supports the selection and movement of obscured vectors.
>
> 2.Upon entering the move operation, left-clicking to move vector objects no longer simultaneously responds to rotating the point cloud view. If you need to rotate the point cloud view, you need to simultaneously press and hold the V key on the keyboard while left-clicking with the mouse for the rotation to take effect.

Steps

1.Click the **Move** button in the Vector Editor, then select the area where the vector to be moved is located.

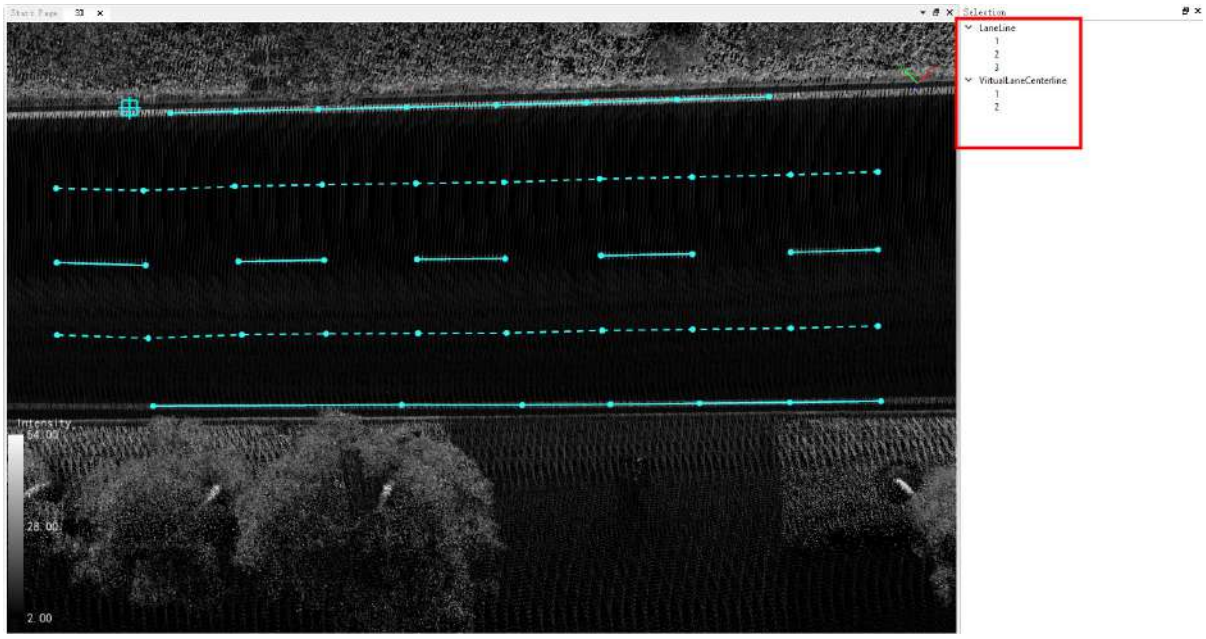


Select the vector

（1）Shortcuts：

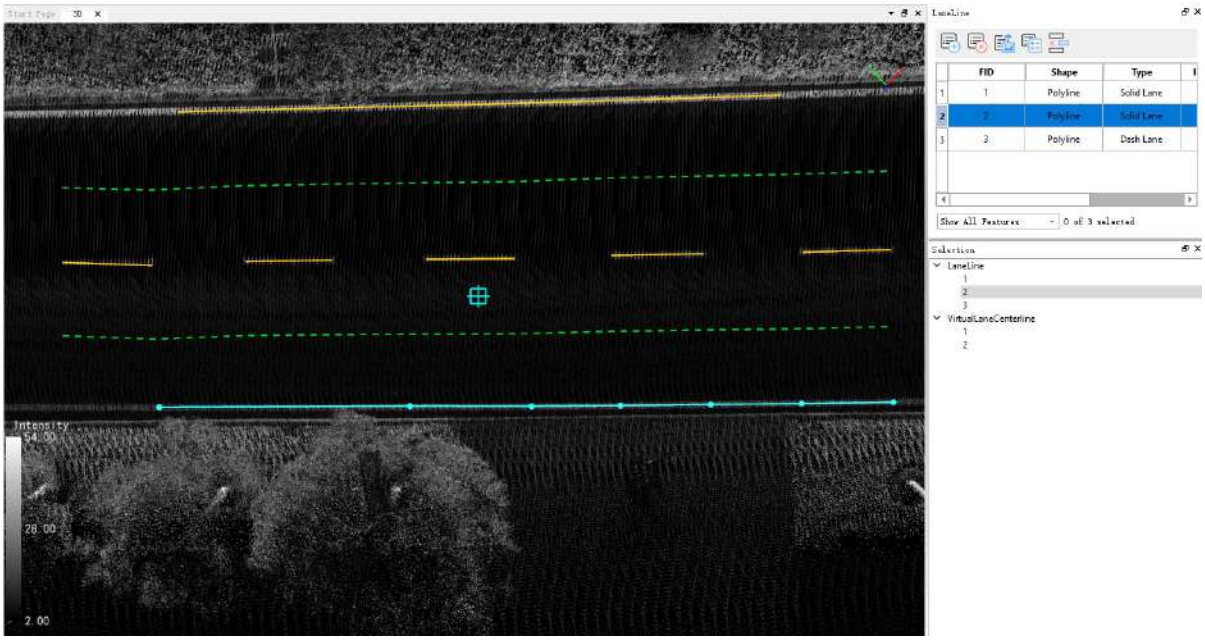
Function	Shortcut Key	Description
ReversE the sequence of points	Shift+R	For line vector objects, change the direction of the line
Invert Selection	Ctrl+Shift+R	For unselected vector objects, it becomes selected; for selected vector objects, it switches to unselected status.
Rotate Point Clound	V/v + drag the left mouse button	While moving vectors, you can use V/v + drag the left mouse button to move the point cloud.

2.After selection, the selected vector will be highlighted, and the detailed information of all the vectors selected will be displayed on the right side of the main window, including the vector layer and ID.



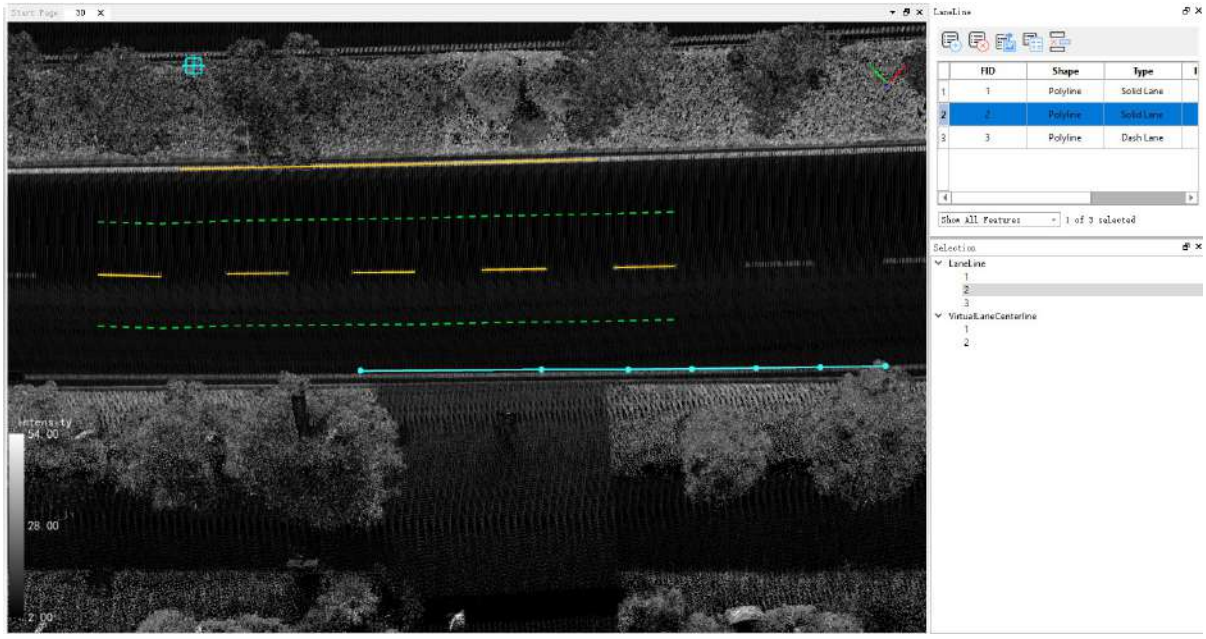
After vector selection

3.Click on the detailed information of a vector on the right side, the highlight of other vectors will be cleared, and the clicked vector will be highlighted.



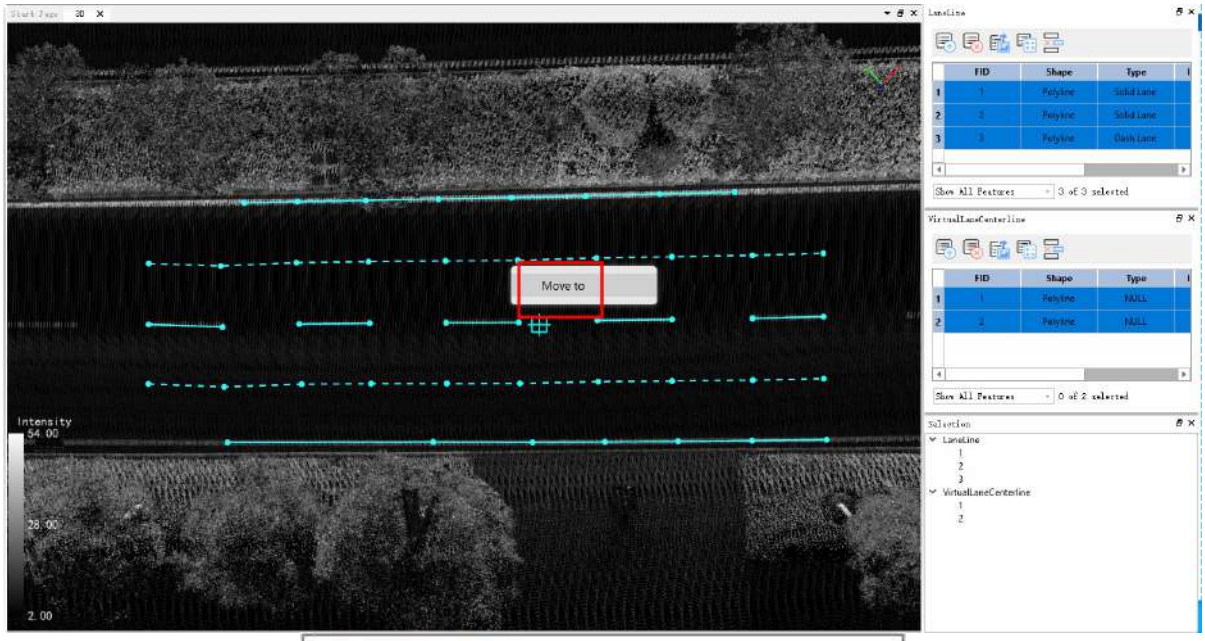
Select a vector

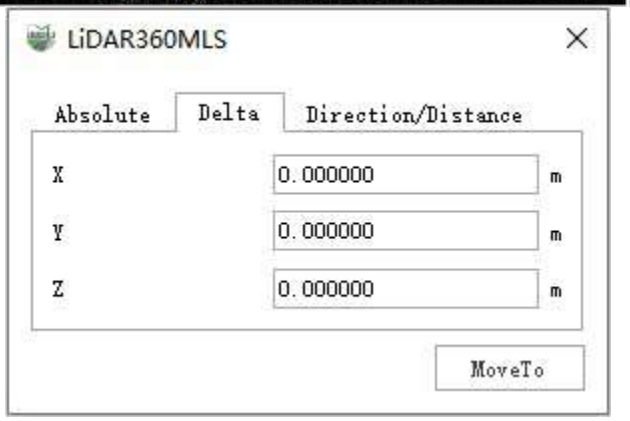
4.Click on the highlighted vector in the 3D window and hold down the left mouse button, drag the vector to the new position to be moved, and after lifting the left mouse button, the vector will be moved to the new position.



After vector moving

5.After the selection is completed, the selected vector will be highlighted. Right-click to pop up the drop-down box 'Move to', and click to pop up the dialog box.

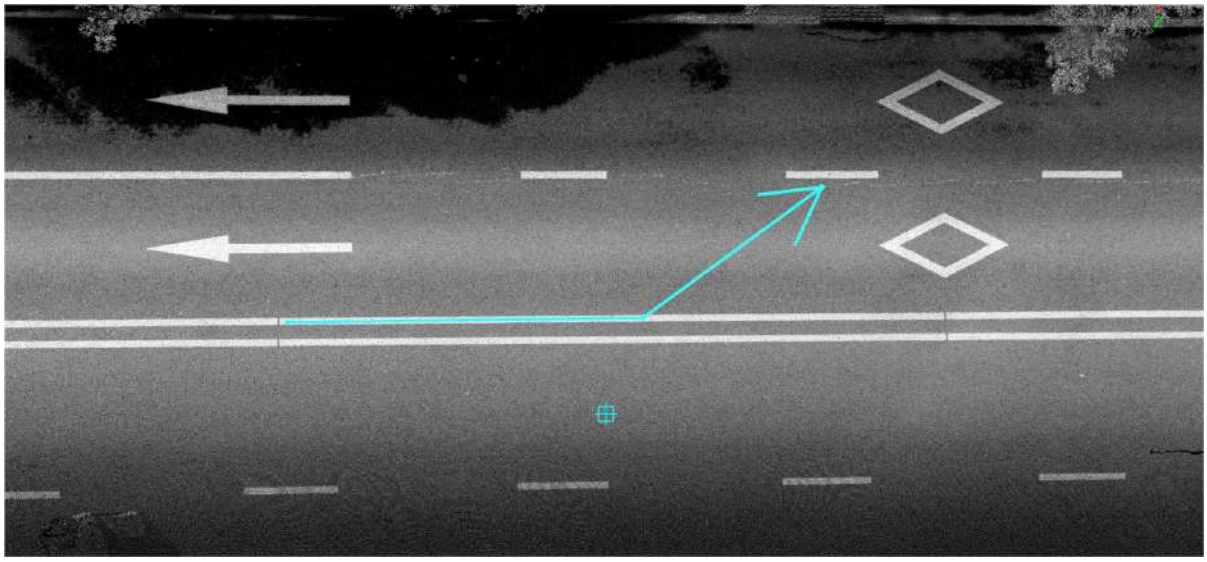




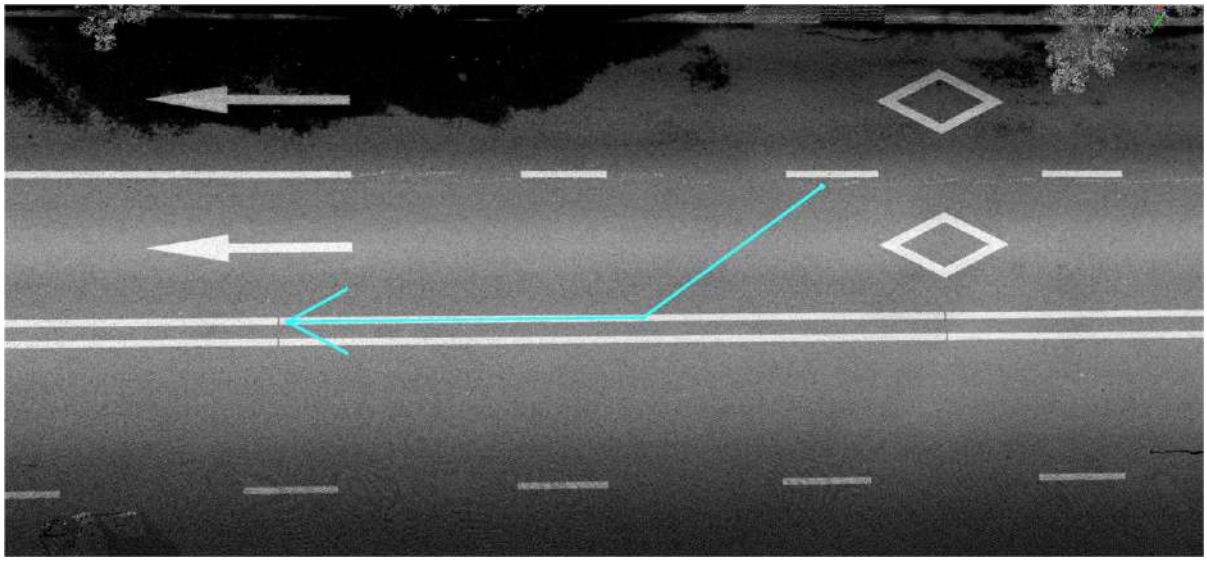
- **Absolute**: Set the coordinates of X, Y, and Z for absolute pinpoint movement.

- **Relative**: Define the coordinates of X, Y, and Z for relative pinpoint movement.
- **Direction/Distance**: Set the direction angle, pitch angle, and distance to move.

6.Shortcut Shift+R: For line vector objects, change the direction of the line;



Before transformation



After transformation

> Note: After a vector object is selected, the selected vector will be highlighted, but the detailed information of all selected vectors, including the vector layer and ID, will not be displayed on the right side of the main window.

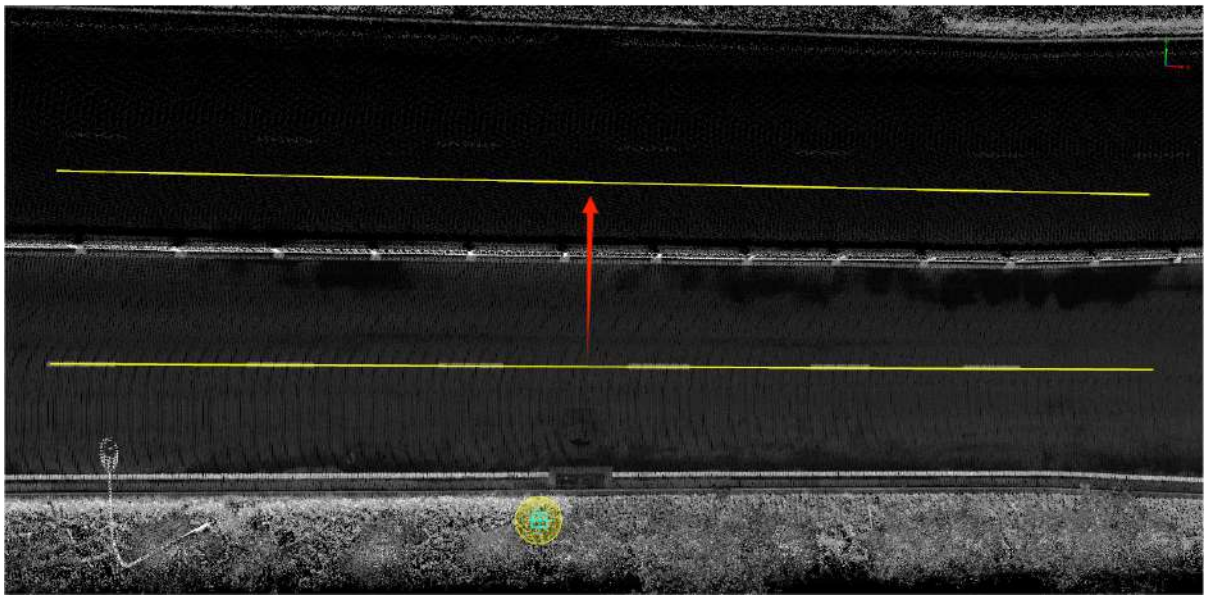
Parallel Copy

Function Description：The Parallel Vector Copy function allows users to copy a vector object in parallel.

> Note：Upon entering the parallel copy operation, left-clicking to copy vector objects no longer simultaneously responds to rotating the point cloud view. If you need to rotate the point cloud view, you need to simultaneously press and hold the V key on the keyboard while left-clicking with the mouse for the rotation to take effect.

Steps

1.After clicking the **Parallel Copy** button in the Vector Editor, move the mouse to the vector and click and hold the left click, drag the mouse to copy the selected vector to the current mouse position.



Parallel vector copy

Rotate

Function Description：The Rotate function allows users to rotate vector objects with a certain point as the center.

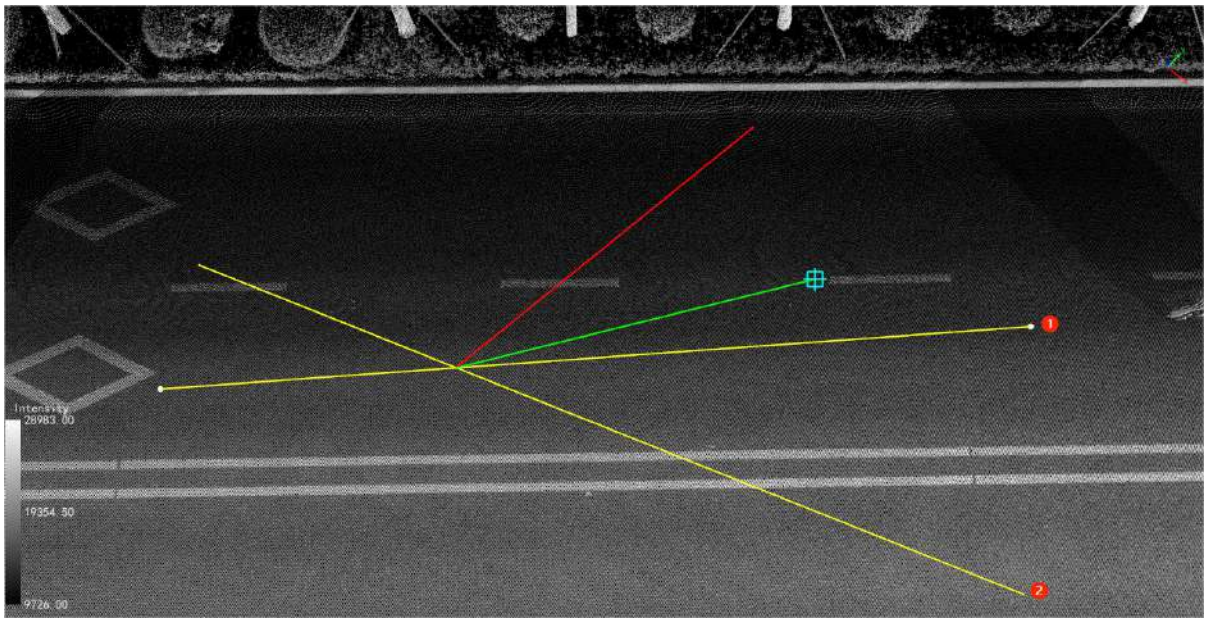
Steps

1.After clicking **Rotate** button in Vector Editor, move the mouse to the vector and left click to confirm the center of rotation;

2.Left click again to select the second point to confirm the rotation auxiliary line, drag the mouse to rotate in the window, and the real-time preview will be at the position where the vector is rotated;

3.After the position is determined, finally left click, and the vector will be rotated to the position where it was previewed.

> In the figure below, the yellow line No. 1 is the initial vector object, the red line is the rotation auxiliary line determined by the second key point, the yellow line No.2 is the position after rotation, and the green line is the corresponding line of the red line after the rotation. The angle between the green line and the yellow line No.2 is equal to the angle between the red line and the yellow line No.1.



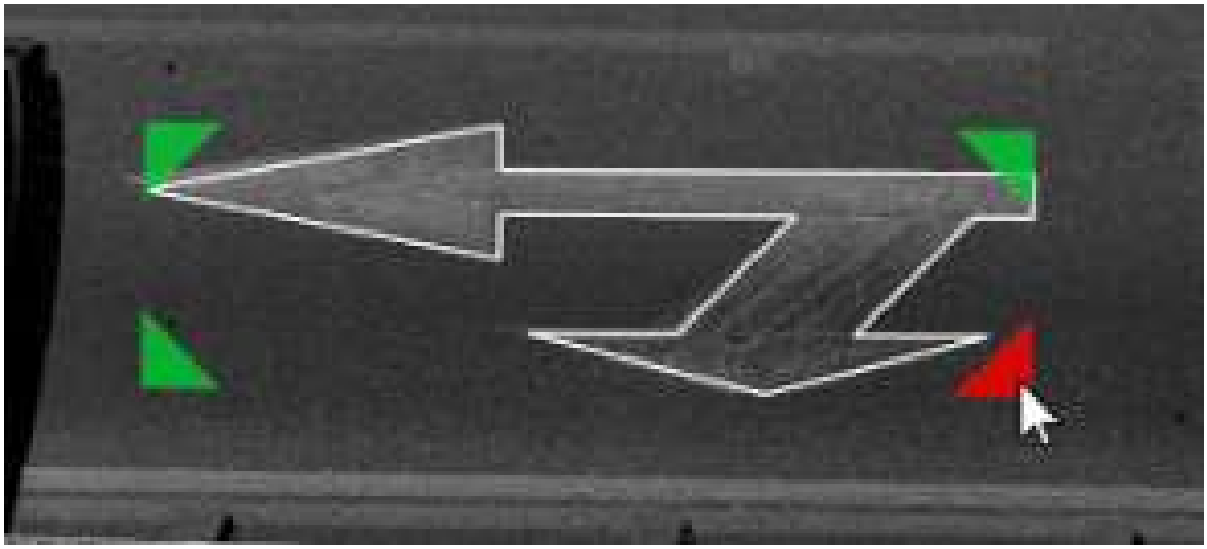
Rotate vector

Adjusting Vectors

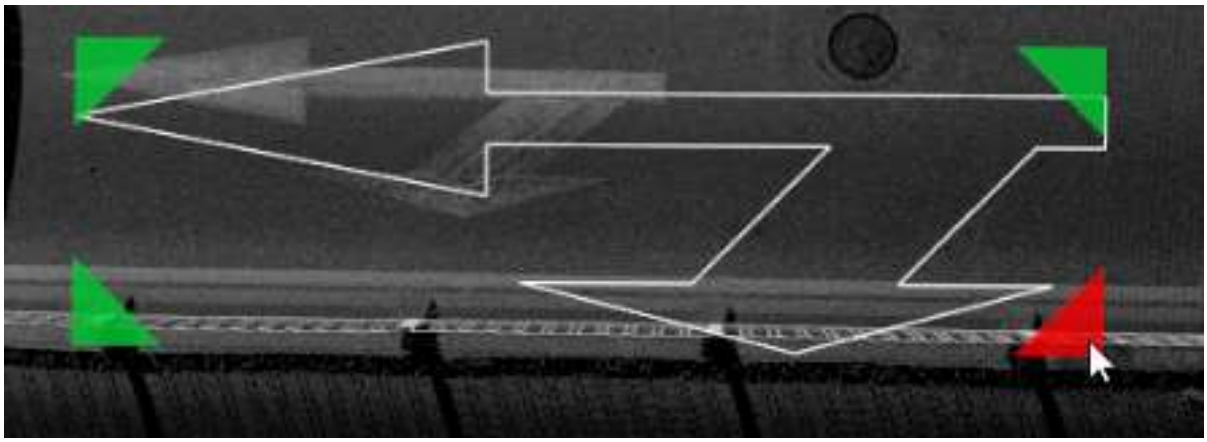
Functionality: The scaling functionality allows you to scale polygons proportionally on their respective planes.

Steps

1.Click on the **Adjust Vectors** button in the Vector Editor. In the view, select a key point on the border of the vector. It will turn green to indicate that it is selected. You can then perform rotation. The four dashed lines of the border will turn green to indicate that they are selected, and you can move them. The key points on the border will turn green when selected, and you can perform scaling, flipping, and other operations to make the vector fit the point cloud as much as possible.



Before Adjusting Vectors



After Adjusting Vectors

Mirror

Function Description：The Mirror Vector function allows the user to move a vector object to a new position in a mirror-symmetrical manner.

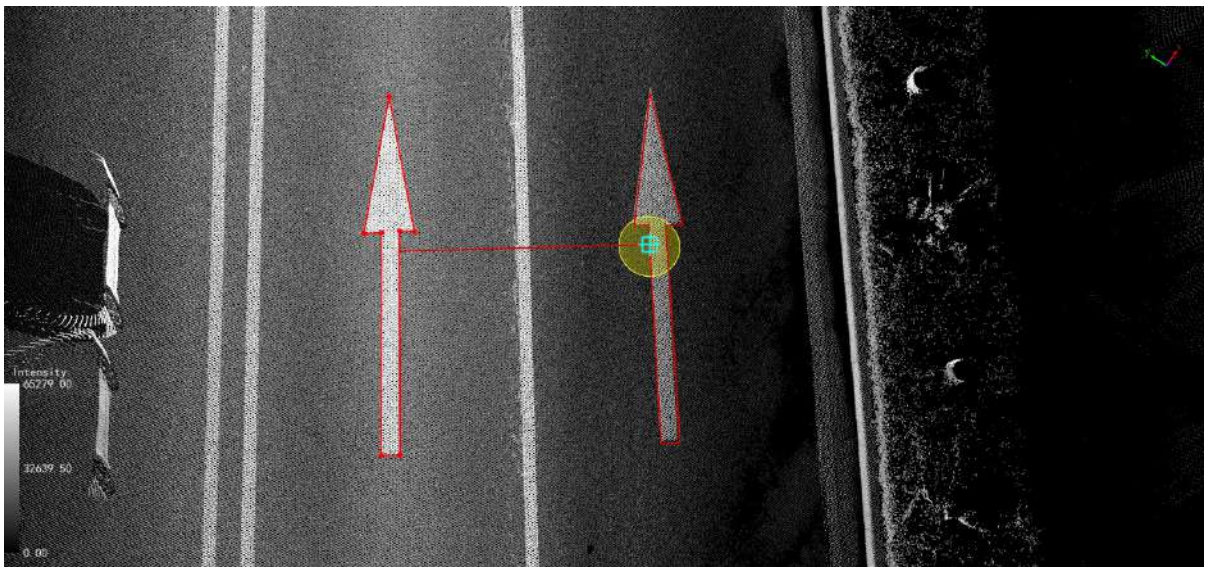
Steps

1.Click **Mirror** button in the Vector Editor;

2.Move the mouse to the vector and click the left mouse button to determine the reference point for generating the mirror image;

3.Then move the mouse, a mirror preview will be generated at the position where the mouse moves;

4.After confirming the position where the vector will be mirrored, finally click the left mouse button, and the vector object will be mirrored and moved here.



Mirror Vector

Reshape

- Reshape Line
- Reshape Rect
- Merge Lines
- Break Line
- Split Line
- Extend or Trim
- Split or Merge Polygon
- Link Line
- Extend Line
- Fillet
- Combine Polygons or Points
- Smooth
- Point To Line
- Line To Points
- Construct Polygon
- Multipart To Singleparts

ReShape Line

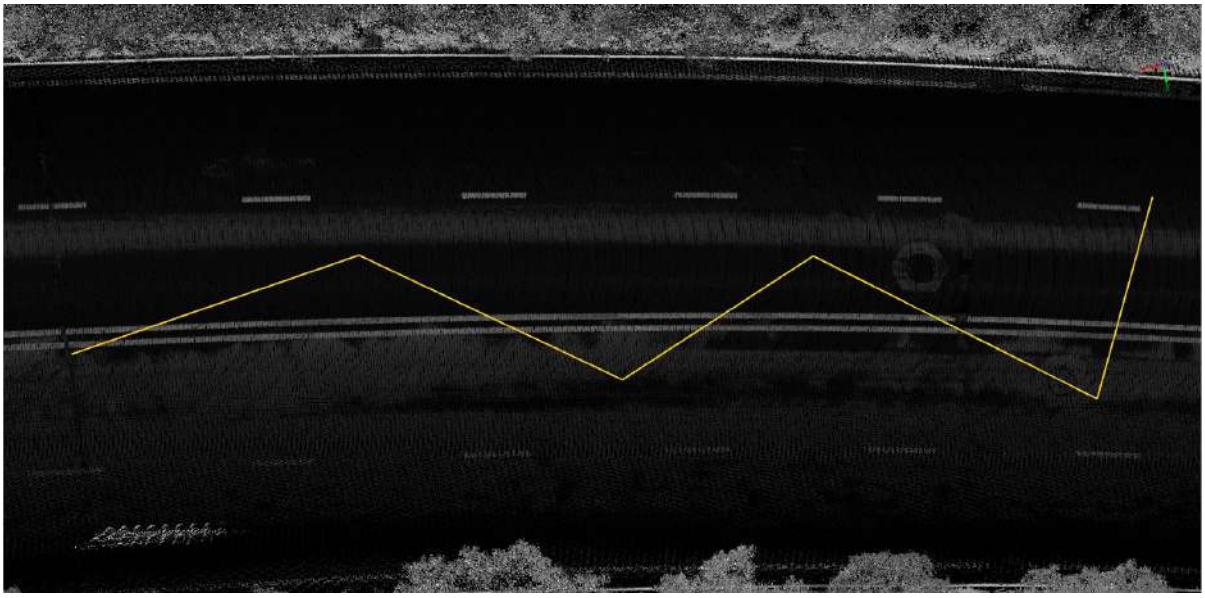
Function Description：ReShape Line allows the user to adjust and repaint of existing line objects.

Steps

1.After clicking the **ReShape Line** button in the Vector Editor,move the mouse to the starting position of the line to be modified, left click to start redrawing, and double-click to end after drawing, and the line object will be redrawn.

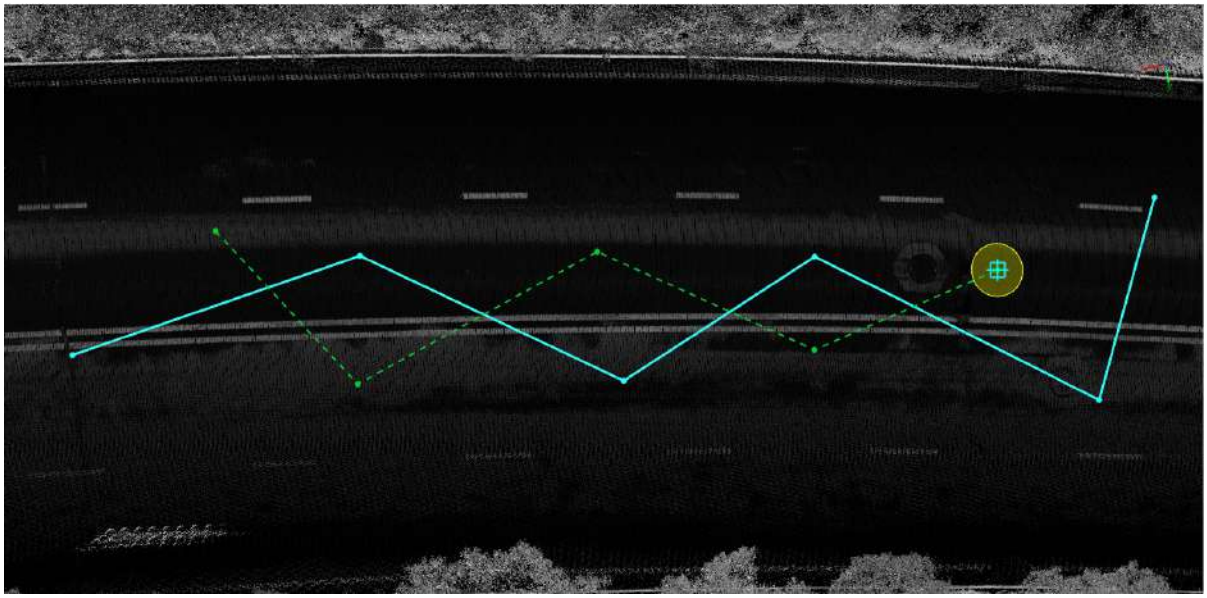
> Please ensure that the newly drawn polyline has more than two intersection points with the original line object, otherwise it will not be redrawn.

- **Original line object**: The object to be redrawn.



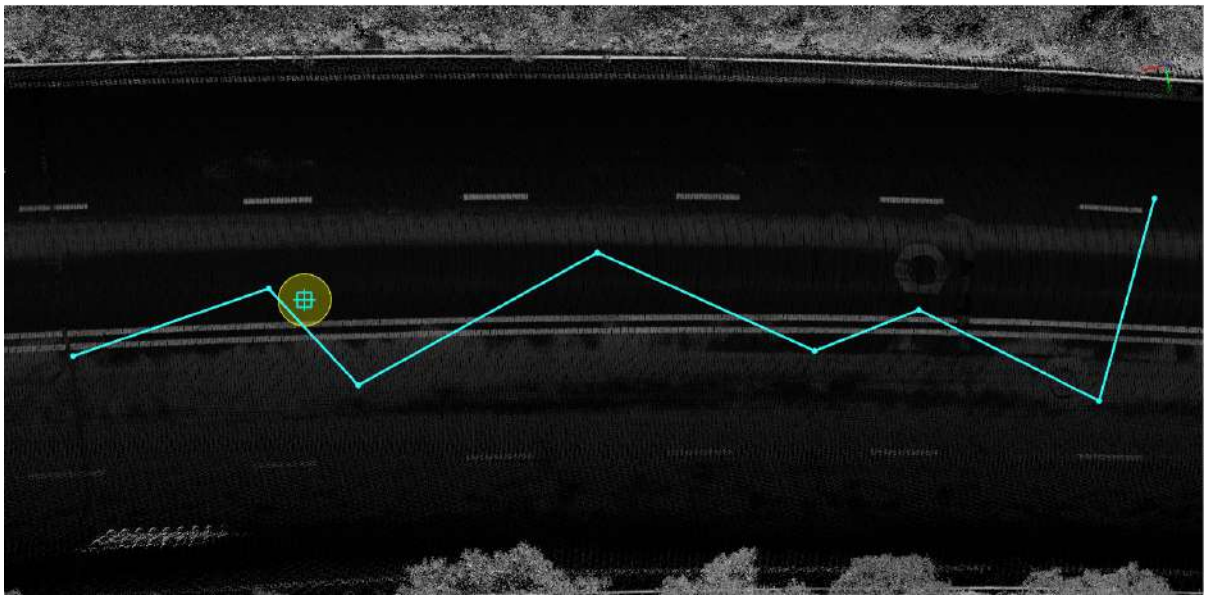
Original line object

- **Drawing process**: The same steps as drawing a polyline, consecutively click on each point to be redrawn , and double-click to end.



Reshape process

- **Reshaped object**: After the reshape is complete, the original line object will be regenerated based on the drawn line shape.



After reshape

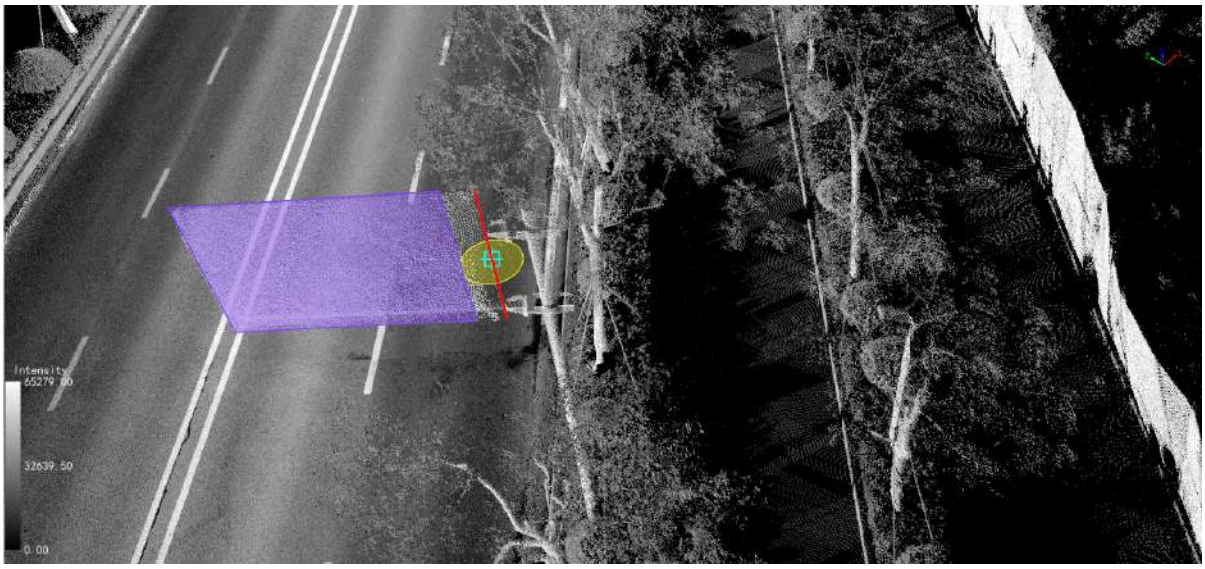
ReShape Rect

Function Description: The ReShape Rect function allows the user to move one edge of the quadrilateral object in parallel to the position of the mouse.

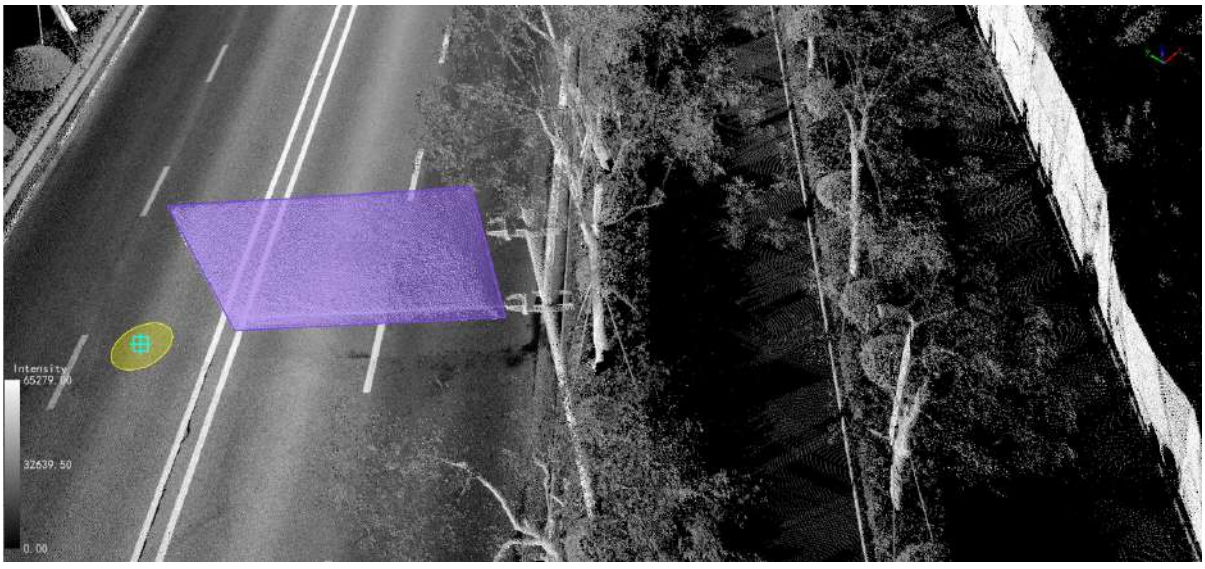
Steps

1.Click **ReShape Rect** button in the Vector Editor;

2.Move the mouse to an edge of the face vector and click the left mouse button to determine the key edge;



3.Then move the mouse to the desired position, double-click the left button of the mouse to determine the edge position, and form a new vector, as shown in the following figure;



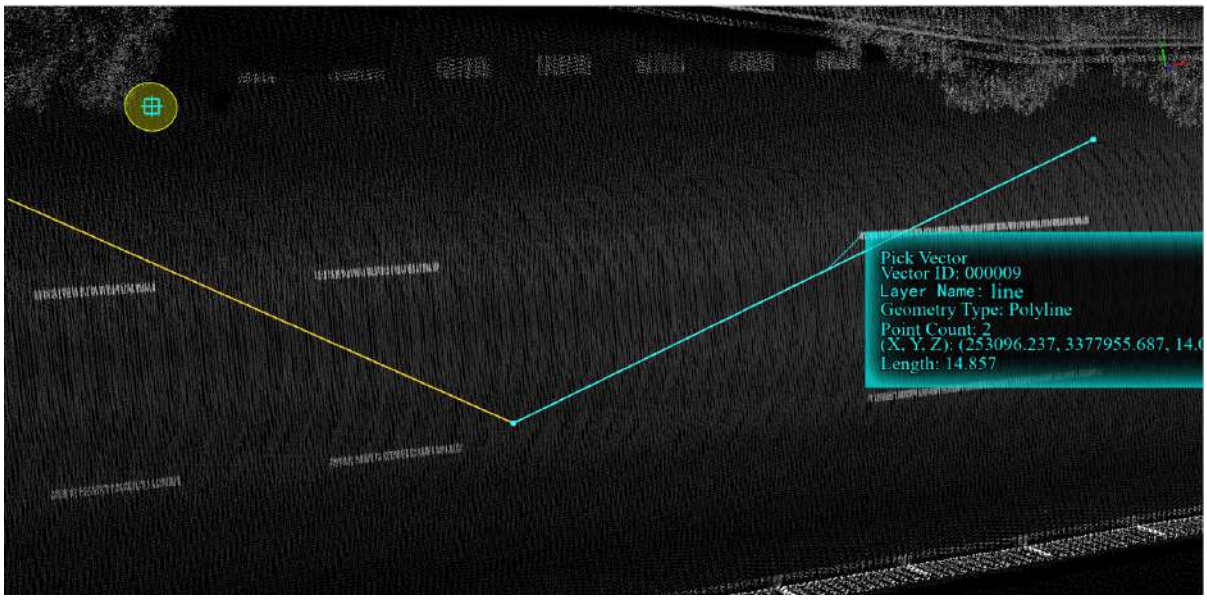
Merge Lines

Function Description：The Merge Lines function allows the user to merge two lines connected end to end into one line object.

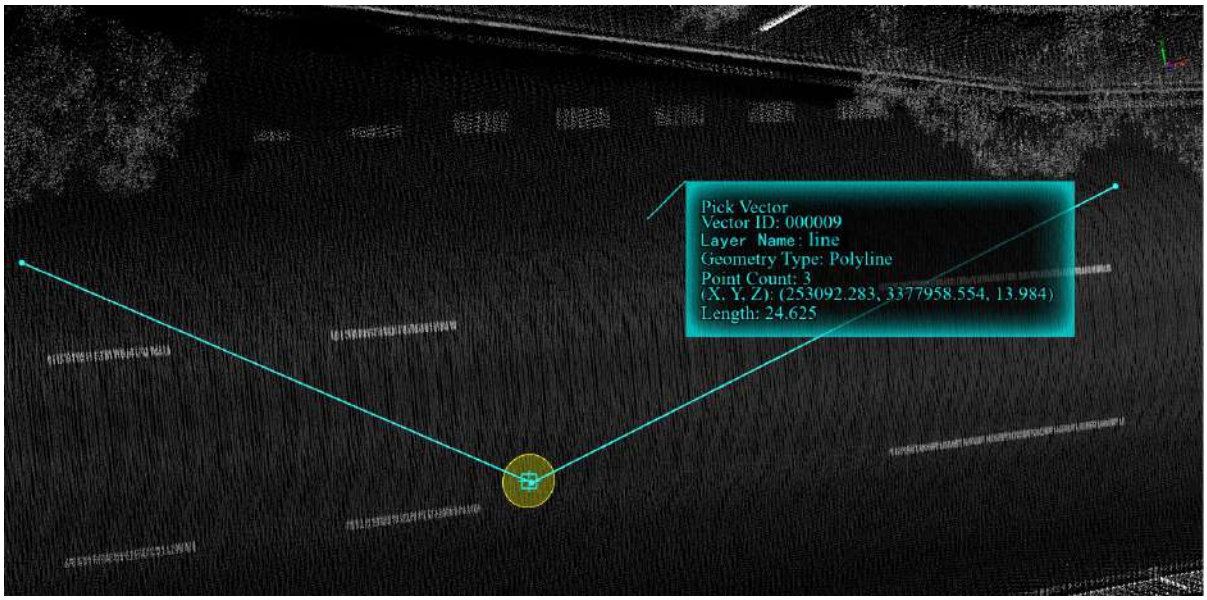
Steps

1.Click **Merge Lines** button in the Vector Editor, click the two lines that are connected end to end and need to be merged with the left mouse button.

> This function can only process two lines with the same point at the beginning and end. For example, the node 1-2 of line 1 and the node 2-3 of line 2 will be merged into a line string with nodes 1-2-3. The details are as follows:



Two lines to be connected



Two lines after connection

Break Line

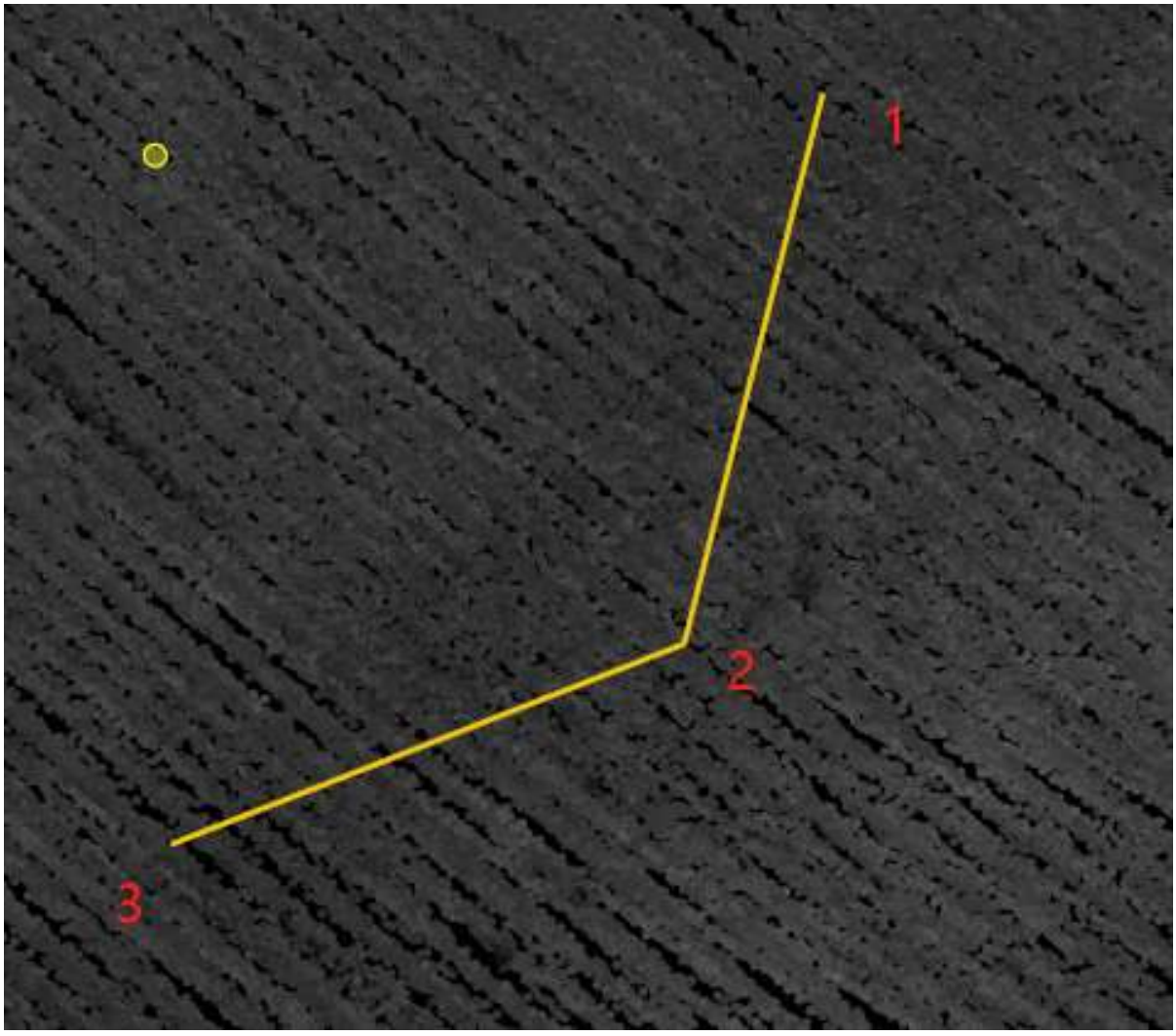
Function Description：The Break Line function allows the user to break a line object into two line objects connected end to end.

Steps

Click **Break Line** button in the Vector Editor, move the mouse to the vector, select the place you want to break, and left click to complete the break of the line.

> If the node is selected, it will break from the node and change from a line string to two lines; if the mouse selected is not a node, insert a node at that position and break it into two lines.

- **Original polyline**: A polyline composed of three points.

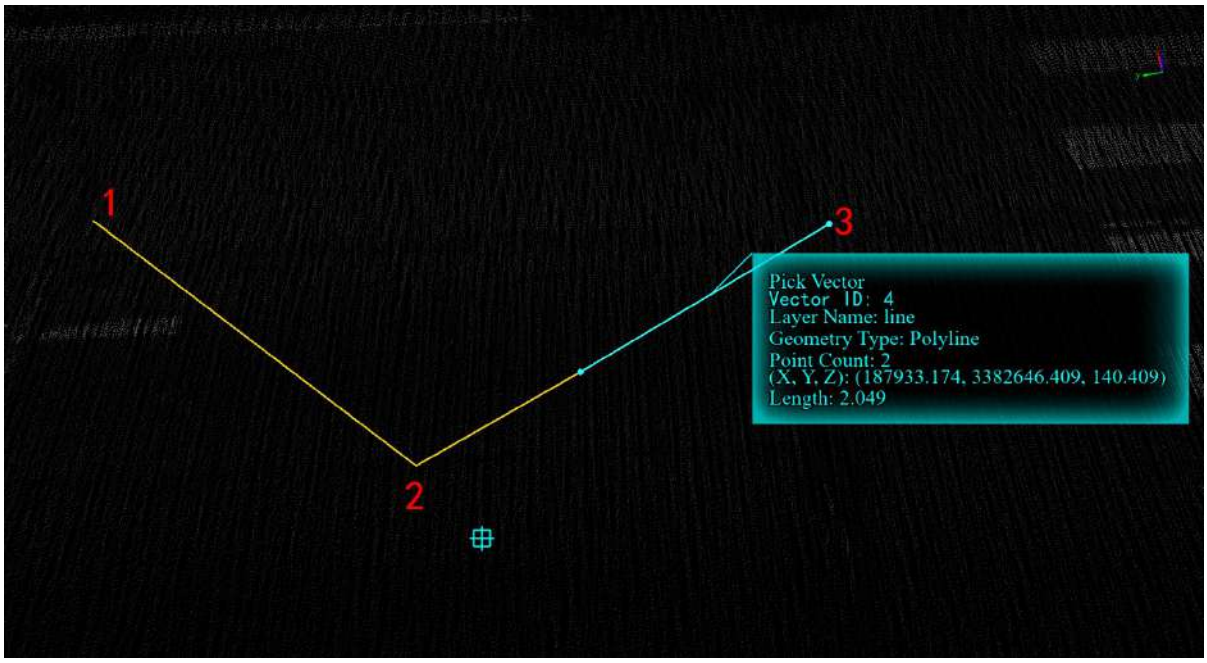


Original polyline

- **Break at original node 2**: The two lines after the break are connected end to end at node 2.

Original polyline

- **Break at non-nodes**: Insert a node, and the two lines after the interruption are connected end to end at the inserted node.



Original polyline

Split Line

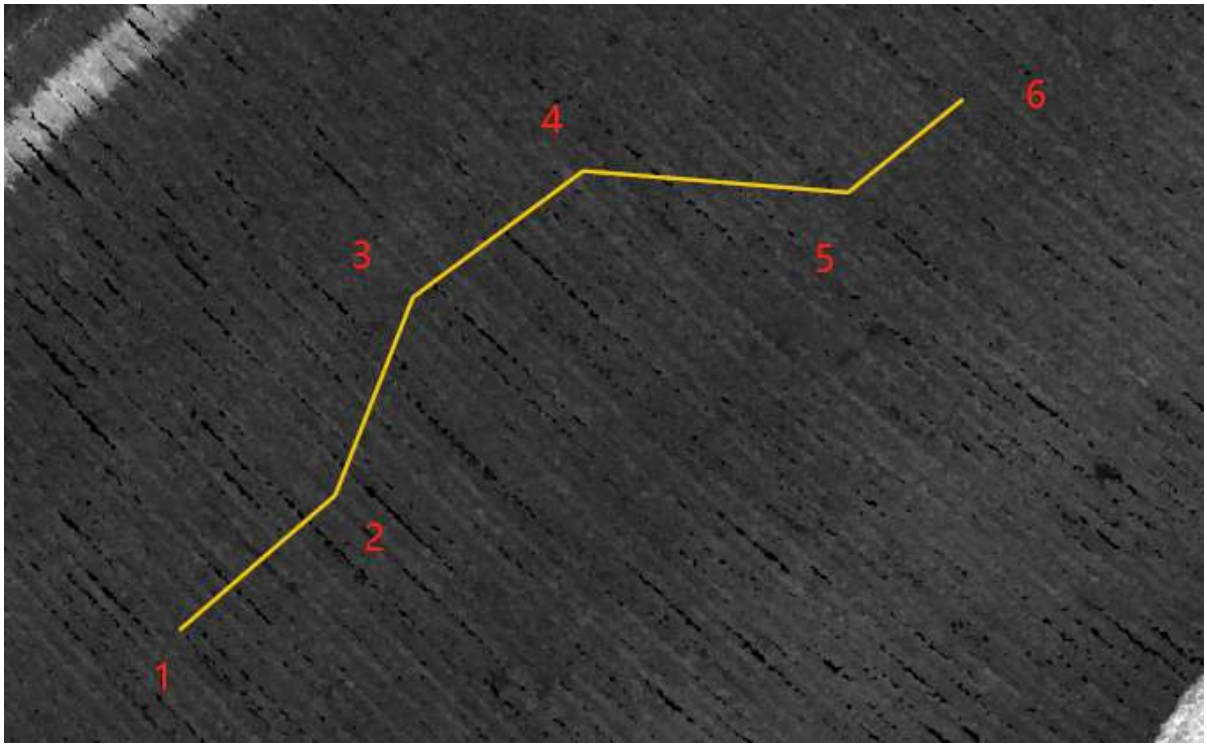
Function Description：The Split Line function allows the user to split a solid line object into a dashed line object. See the picture below for details.

Steps

1.Click **Split Line** button in the Vector Editor.

2.Left click the line you want to split to complete the line segment splitting.

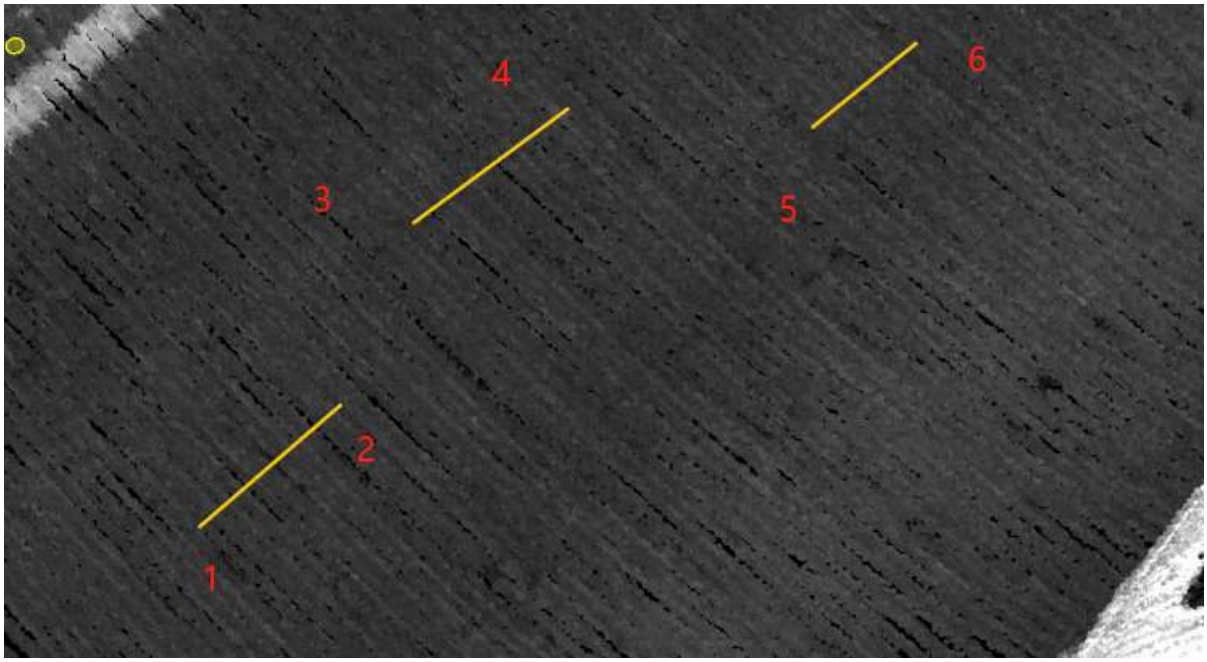
- **Before vector line splitting**: The original solid line consists of 6 nodes.



Original solid line

- **After the vector line is split**: The even-numbered node -> odd-numbered node part of the original solid line is interrupted and disappeared, thus realizing the function of splitting the original solid line into dashed lines.

> In this example: 1-2 nodes form a line segment, 3-4 nodes form a line segment, 2-3 parts are interrupted and disappear.



Split into segments

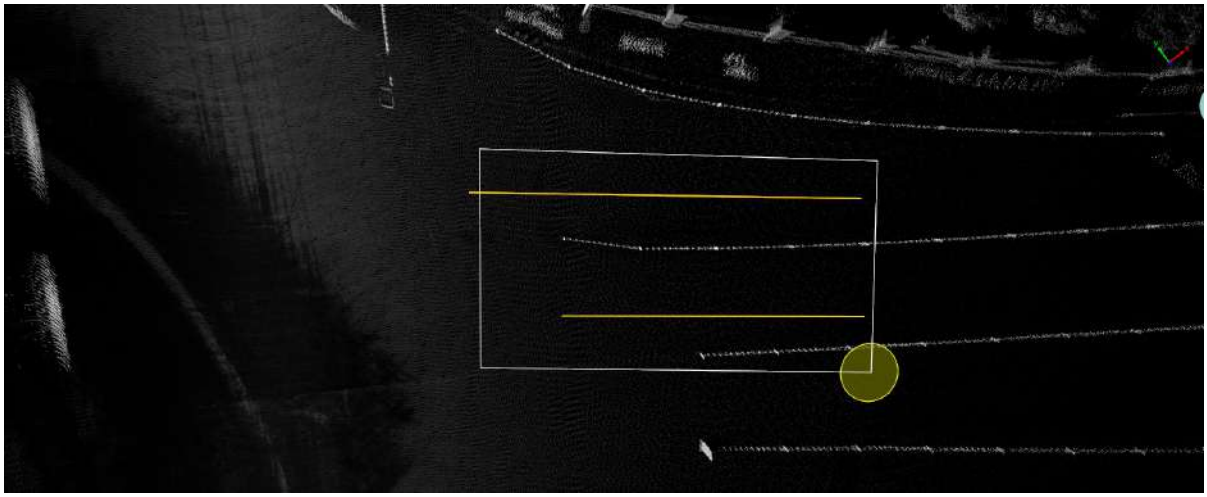
Extend or Trim

Function Description：Allows the user to modify a line, wherein a target line can be drawn by the user to stretch or cut the start or end of a series of lines.

Steps

1.Click **Extend or Trim** button in the Vector Editor, the left mouse button uses a three-key point method on the screen to select the line object to be edited, as shown in the figure below.

> Note: The white quadrilateral is the selection box, and the line objects located inside the selection box or intersecting the selection box will be selected (highlighted in red).



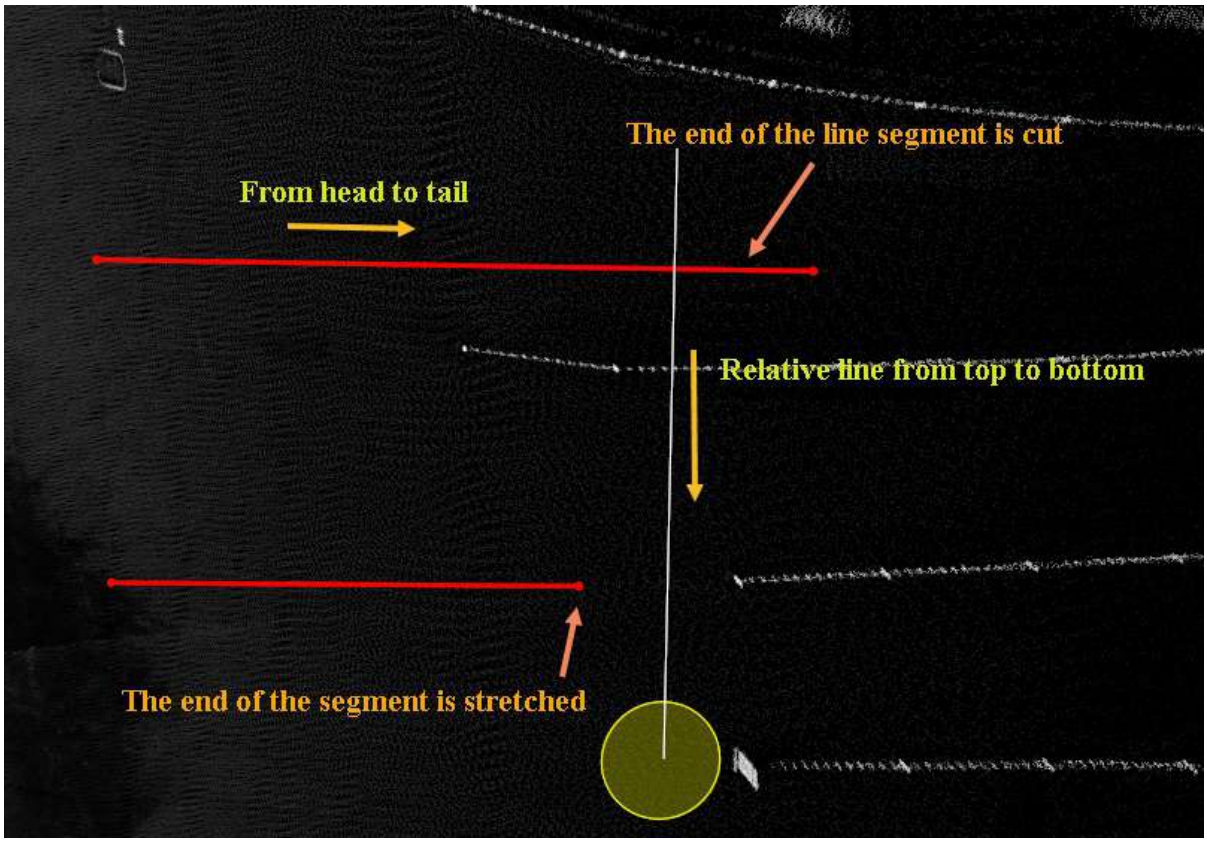
Frame selection of line objects

2.Right-click to select two points one after the other to form a **cutting/stretching** reference line to complete the cutting or stretching. Refer to the stretching/cutting principle below for details.

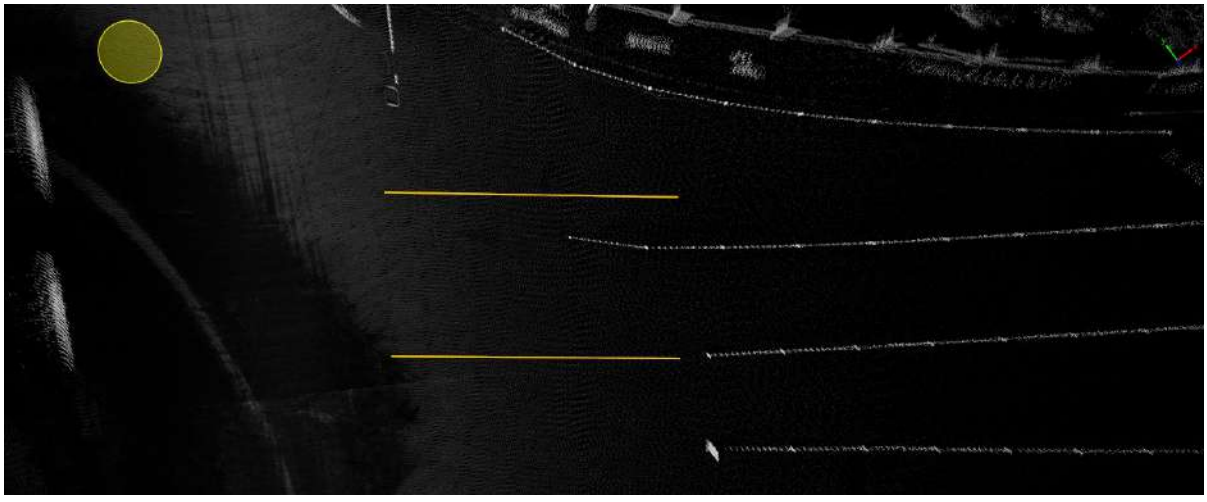
3.By clicking the shortcut key G and then performing the cutting/stretching operation, the line after cutting/stretching can be affixed to the ground.

Cutting/Stretching Principle

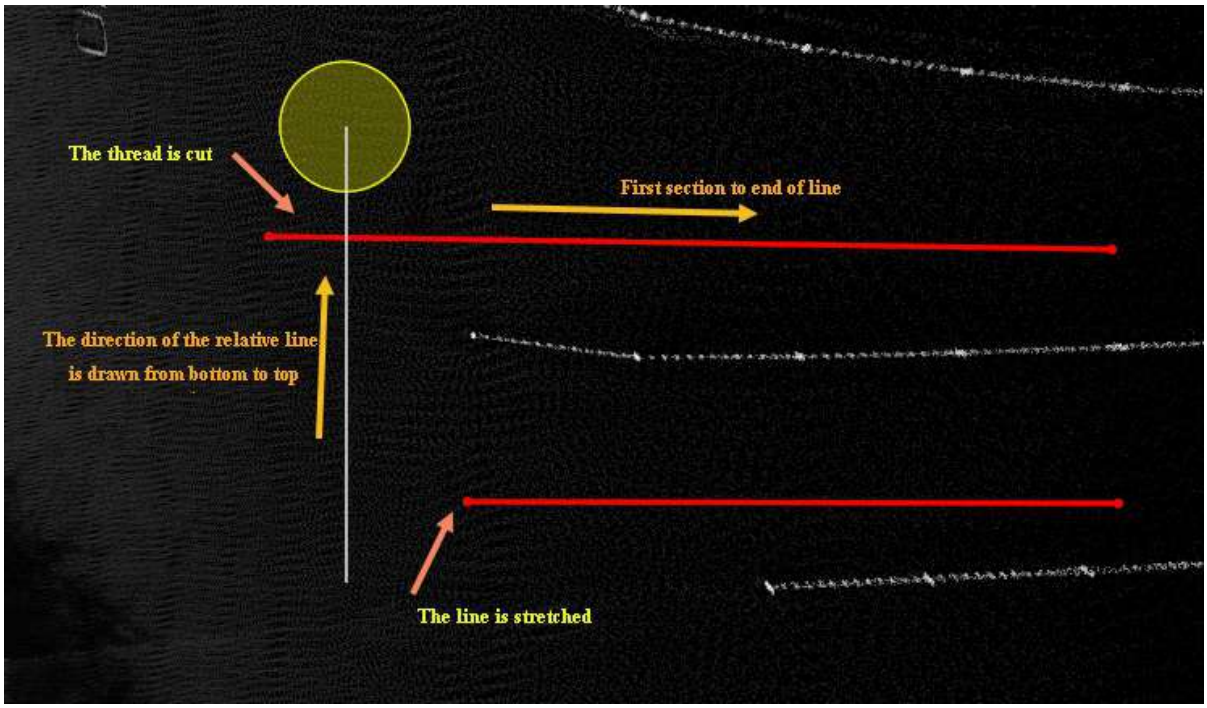
- Draw from left to right relative to the direction of the selected line.



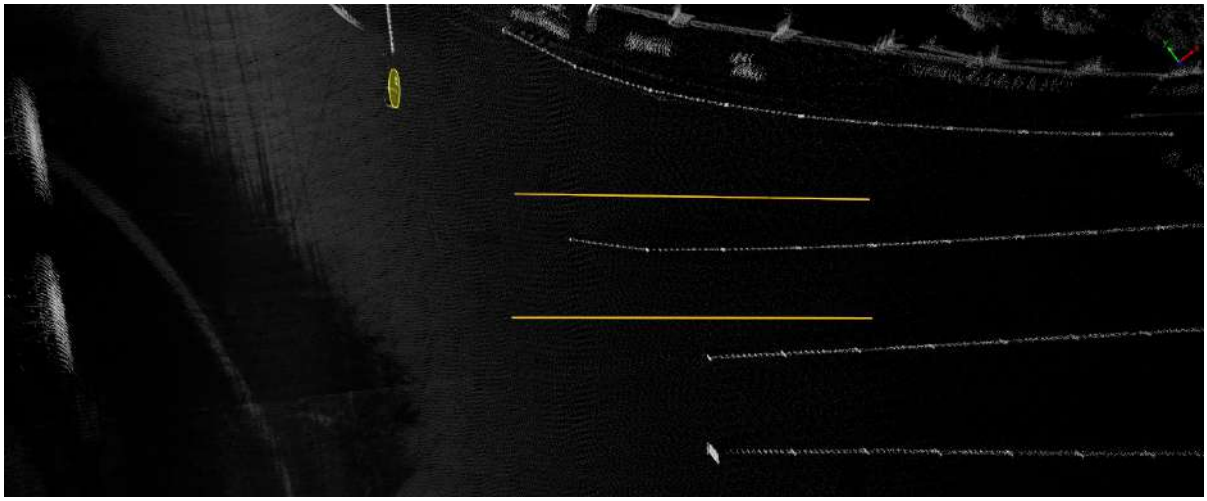
Draw the target line from left to right


Cutting/stretching result

- Draw from right to left relative to the direction of the selected line.



Draw the target line from right to left



The target line is drawn from right to left, the first end of the selected line is cut or stretched as a result

Split and Merge Polygon

Function Description：This function allows the user to split a convex polygon into two convex polygons, or merge two convex polygons into a convex polygon vector object.

> Note: Only the polygon objects under the road surface layer are supported

Steps

1.Click **Split and Merge Polygon** button in the Vector Editor, the operation mode selection dialog box will pop up.

The pop-up window selects the current operation as cutting or merging

Parameter Description:

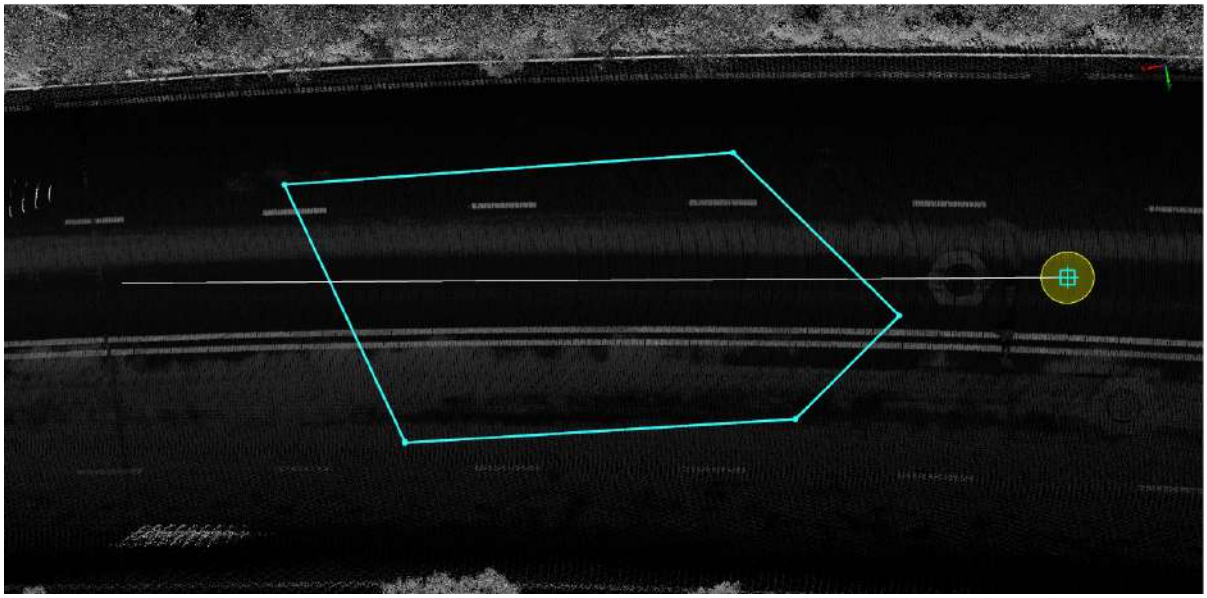
- **SplitPolygon**: Split mode.
- **MergePolygon**: Merge mode.

SplitPolygon

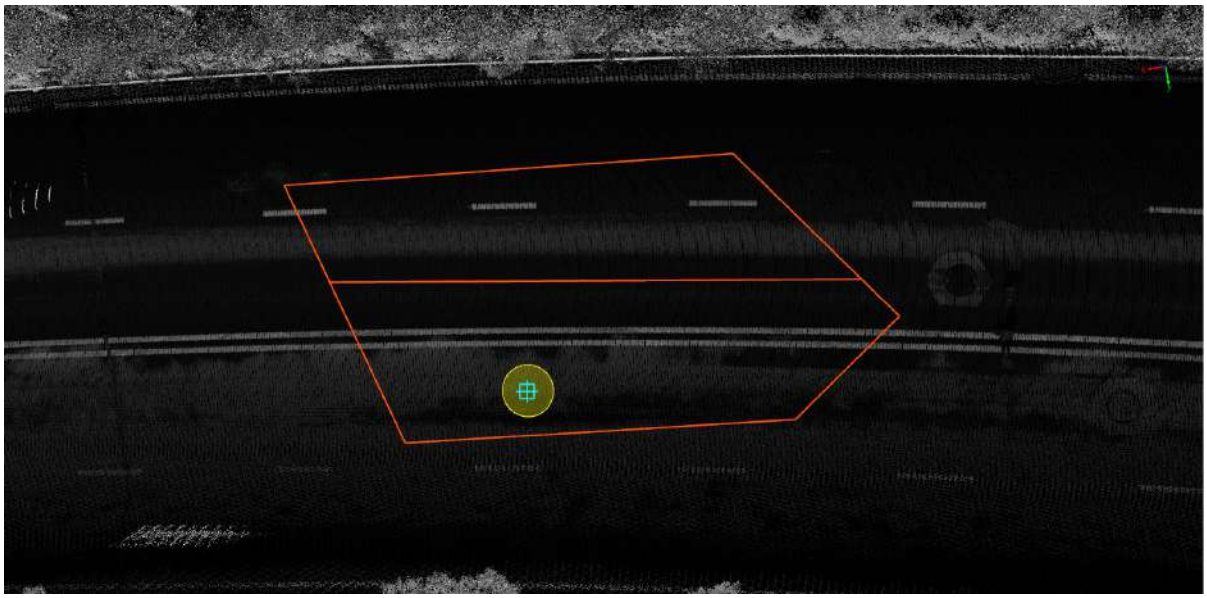
1.Left click to select the polygon to be split, and the polygon will be highlighted in blue.

2.Right click two key points to form a cutting line, as shown by the white line in the figure below, to complete the cutting of the polygon.

> Note: The cutting line needs to intersect the two sides of the polygon.



Draw cutting line

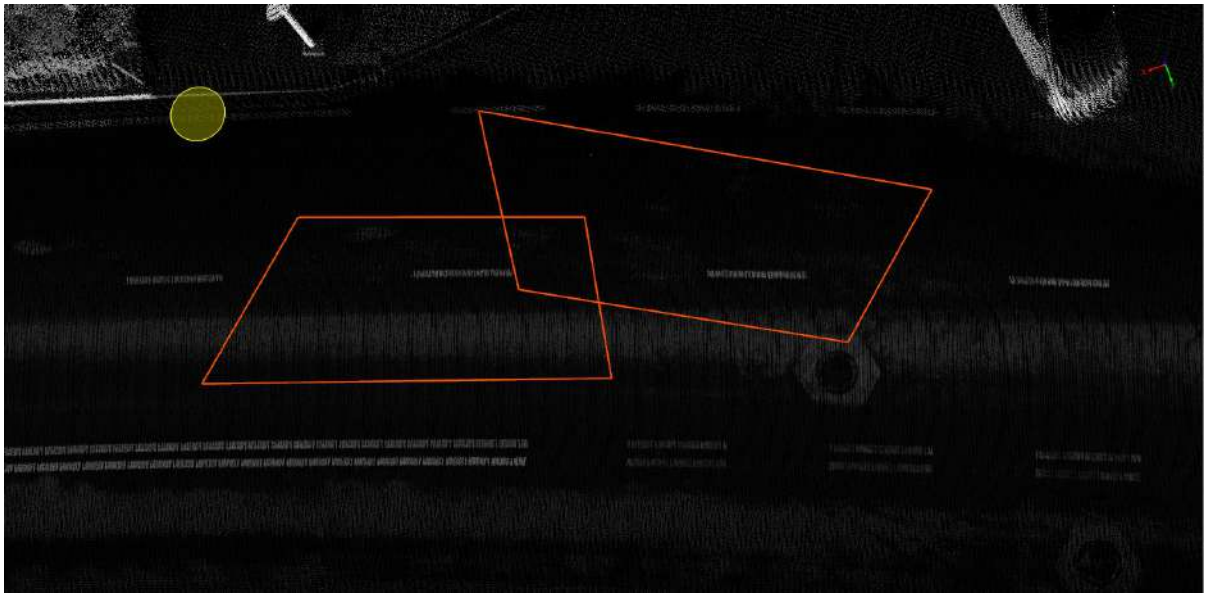


Cutting result

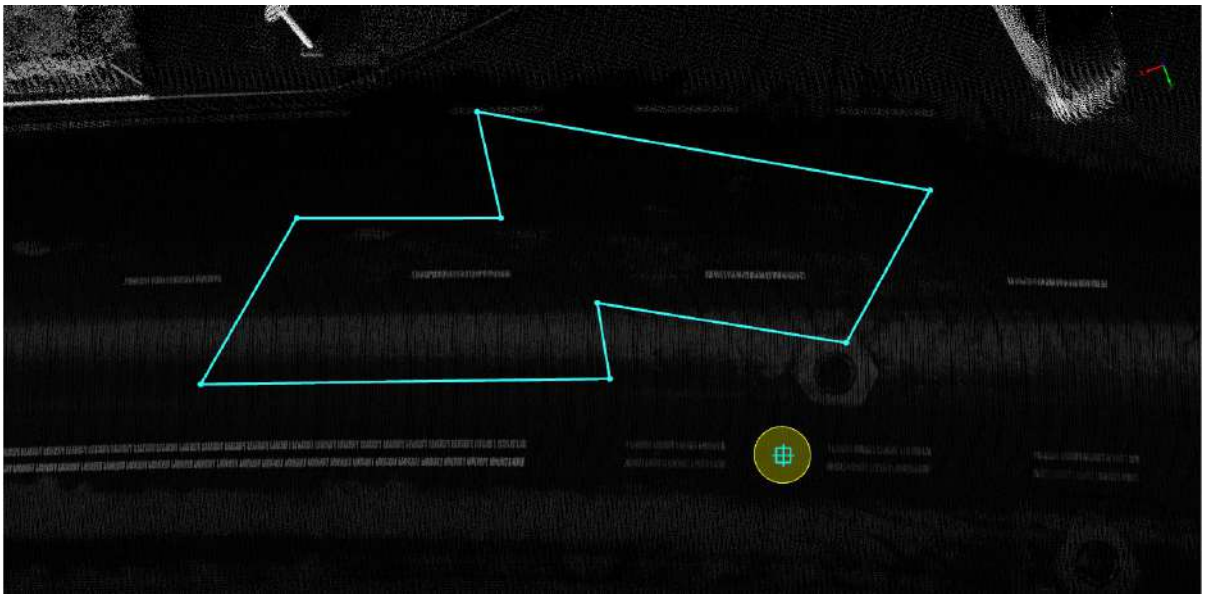
MergePolygon

1.Left click on the two polygons intended to be merged to select, and complete the merging of the polygons.

> Note: The polygons that need to be merged need to have intersections.



The two polygons before the merge



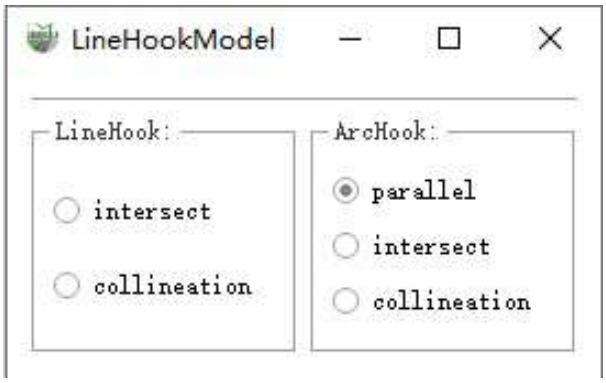
Merge result

Link Line

Function Description：This function allows the user to link lines by way of circular arc or straight line connection, where the relative position relationship of the vectors are collinear, parallel or intersecting.

Steps

1.Click **Link Line** button in the Vector Editor, LineHookModel pop-up window will appear. Select the corresponding mode to perform the link operation and see the detailed description below for details.



The pop-up window selects the connection mode and the relative relationship of the lines

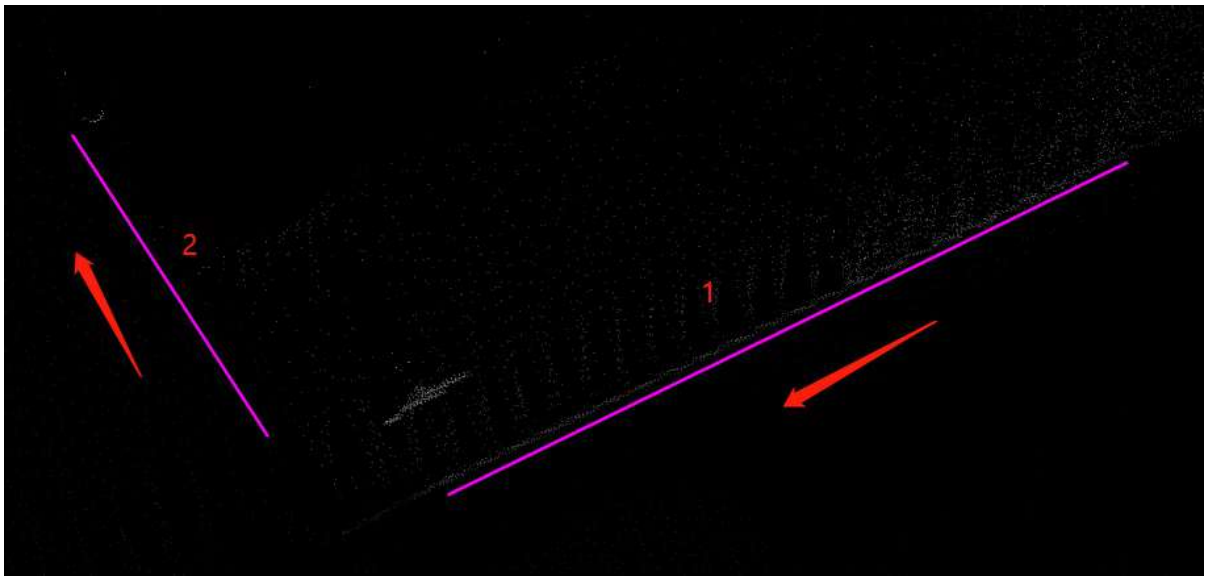
Parameter Description

- **LineHook**: According to the spatial geometric relationship of the two lines, they are connected end to end or extended to intersect.
- **ArcHook**: According to the spatial geometric relationship of the two lines, the end points of the two lines plus a user-defined point are selected, a total of three points are used to determine a circular arc; or a point is selected by user-defined and connect the two lines in a circular arc tangent to the two lines.

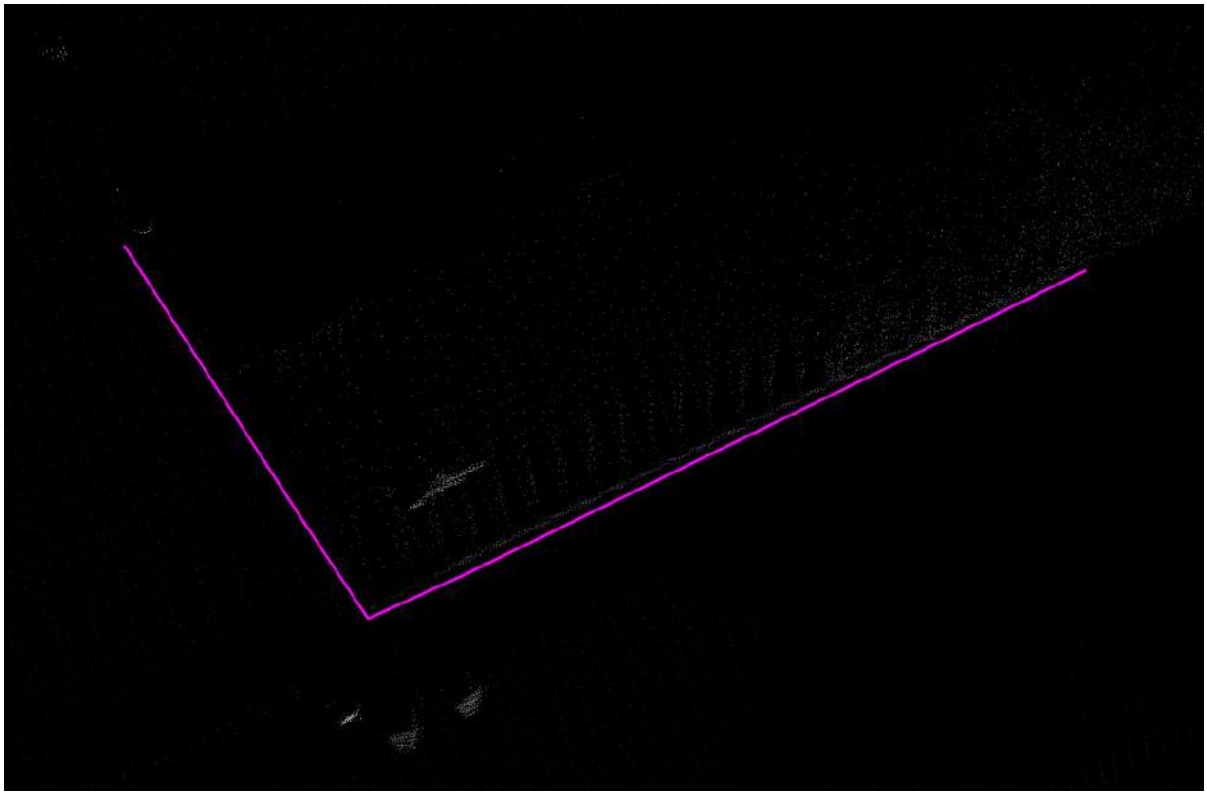
2.**LineHook**

（1）**intersect**: Select **LineHook**, and the relative relationship of the lines is **intersect**, click the left mouse button one by one, and select the two lines that need to be linked to complete the automatic linking. Among them, the order of clicking must be consistent with the direction of the line. As shown in the figure below, you need to **first click** line 1 and then **click** line 2 to achieve the desired effect.

> Note: Intersect means that the straight lines passing through two line segments can intersect at one point.

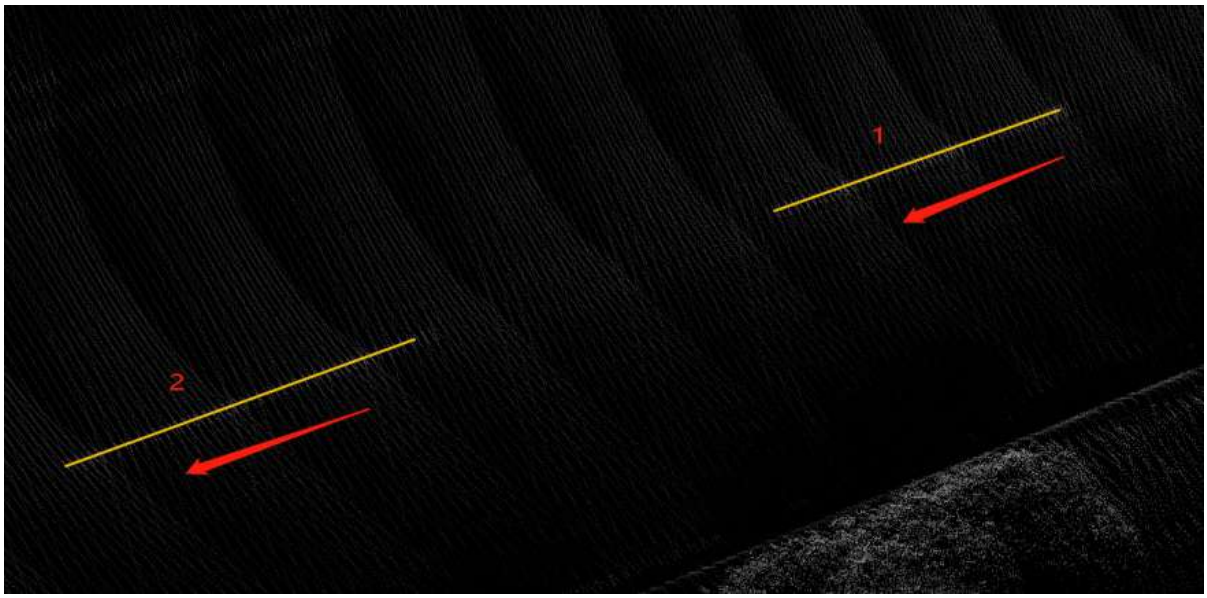


The two lines before linking

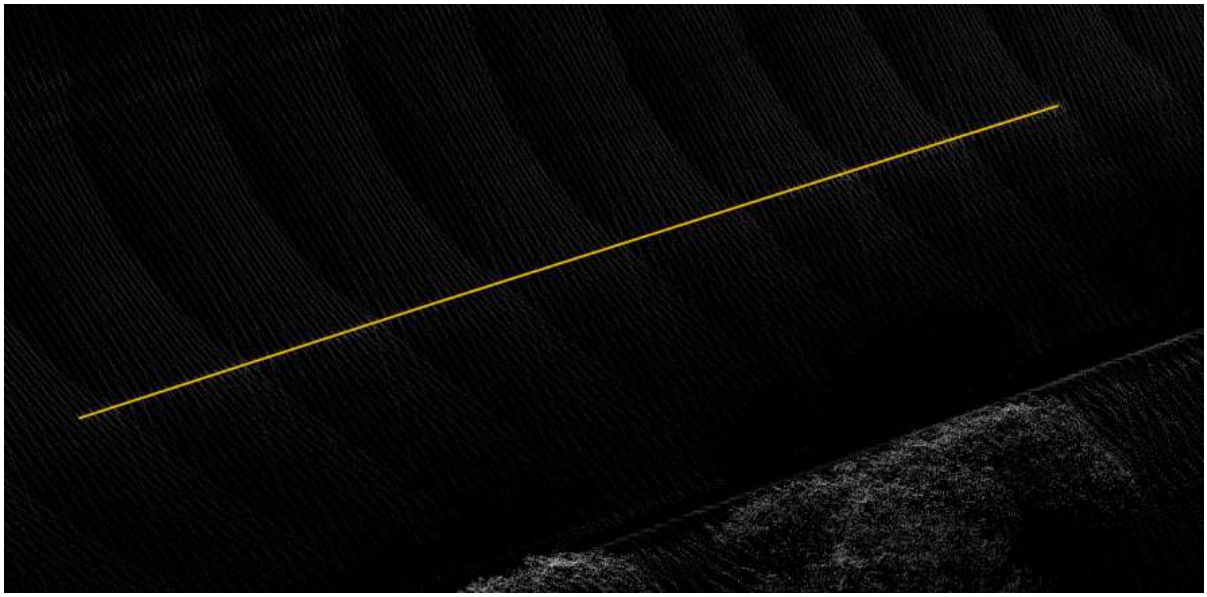

Linking result

（2）**collineation**:Select **LineHook**, and the relative relationship between the lines is **collineation**, click the left mouse button one by one, and select the two lines that need to be connected to complete the automatic connection. Among them, the order of clicking must be consistent with the direction of the line. As shown in the figure below, you need to **first click** line 1 and then **click** line 2 to achieve the desired effect.

> Note: Collineation means that the straight line passing through the two line segments is approximately on a straight line.



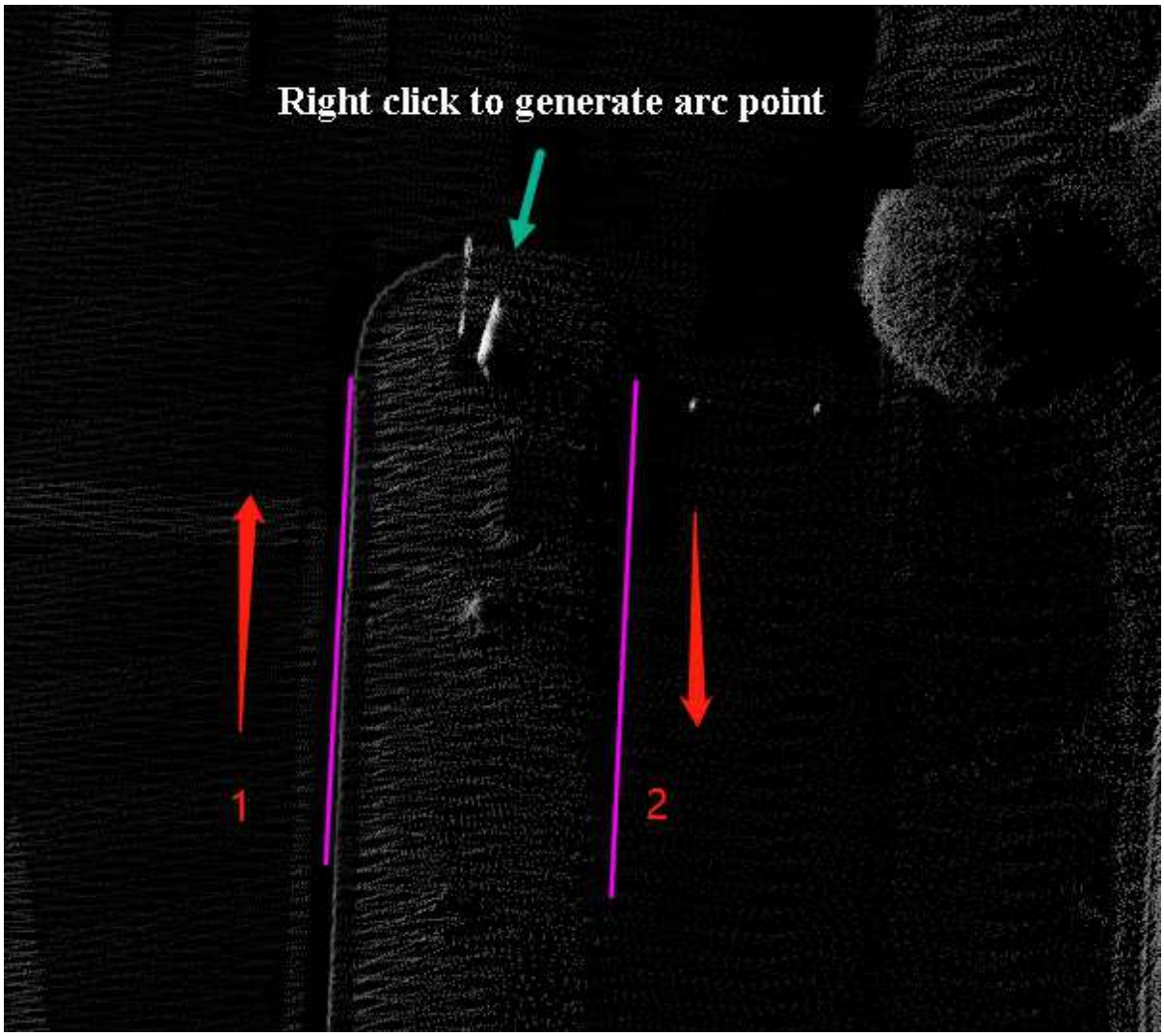
The two line before linking


Linking result

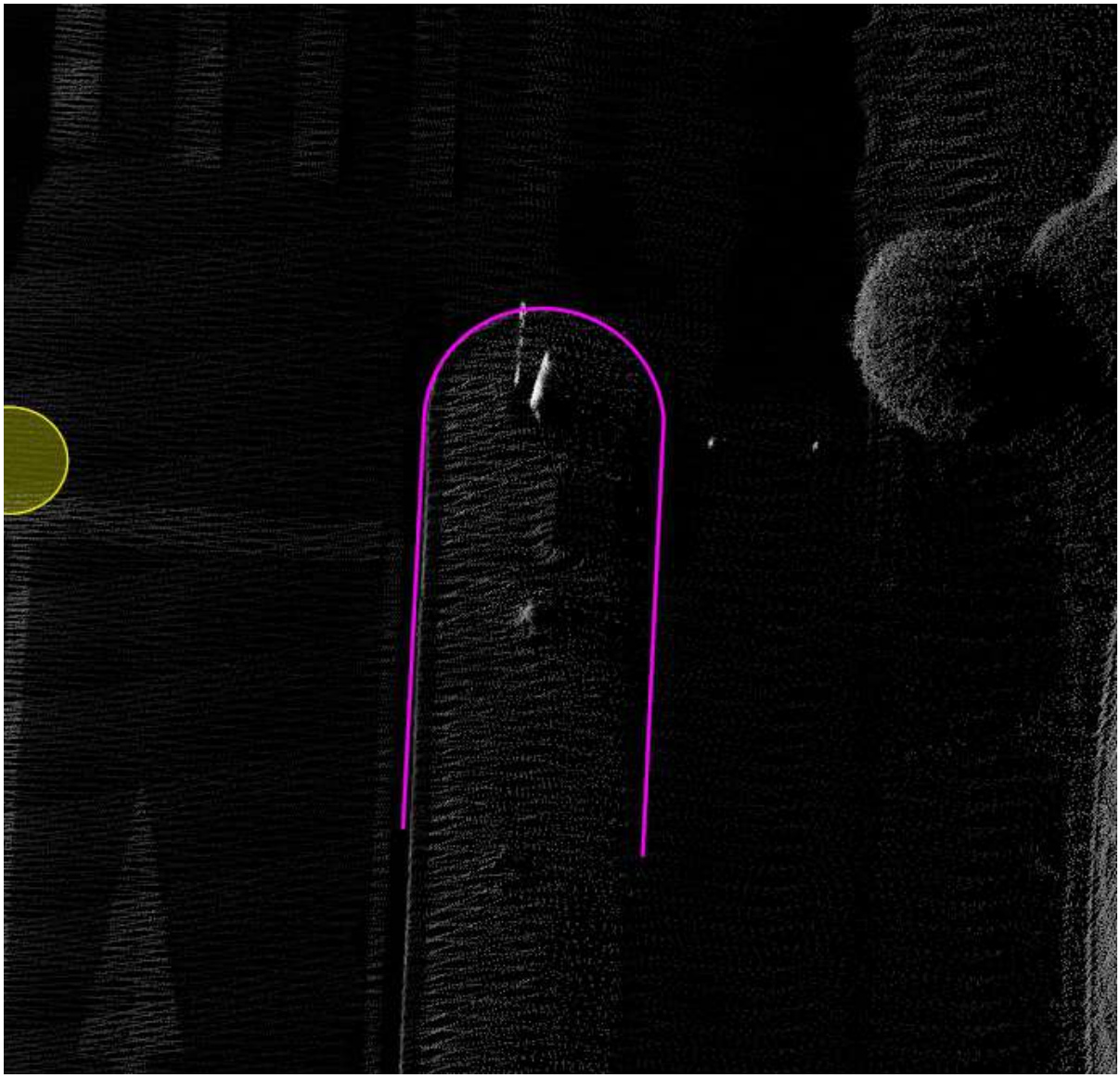
3.**ArcHook**

（1）**parallel**: Select **ArcHook**, and the relative relationship of the lines is **parallel**, click the left mouse button in turn to select the two lines to be hooked, and the order of clicking must be consistent with the direction of the line, as shown in the figure below, you need to **first click** line 1 and then line 2 to achieve the desired effect. Finally **right-click** to select the key point of the arc to be generated, and the arc hook in **parallel mode** can be completed.

> Note: Parallel means that two lines are approximately parallel in space.



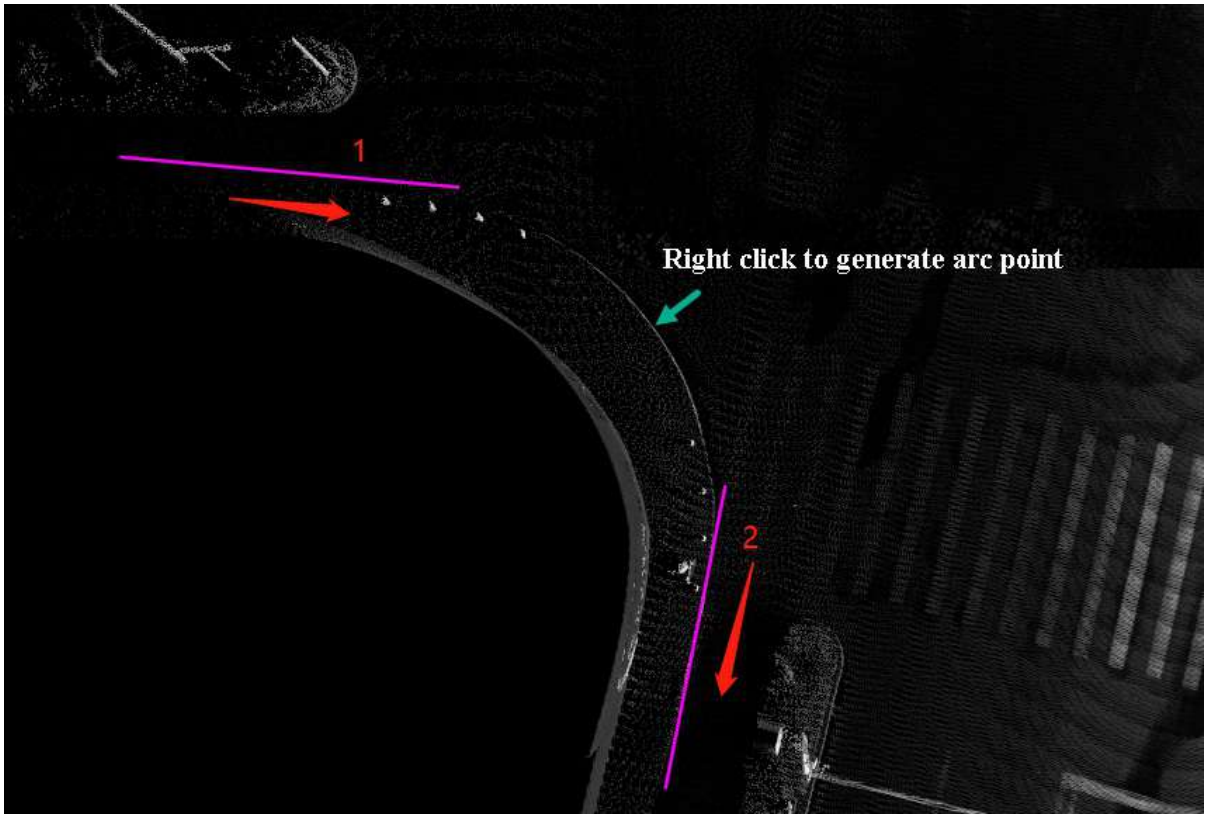
The two lines before linking



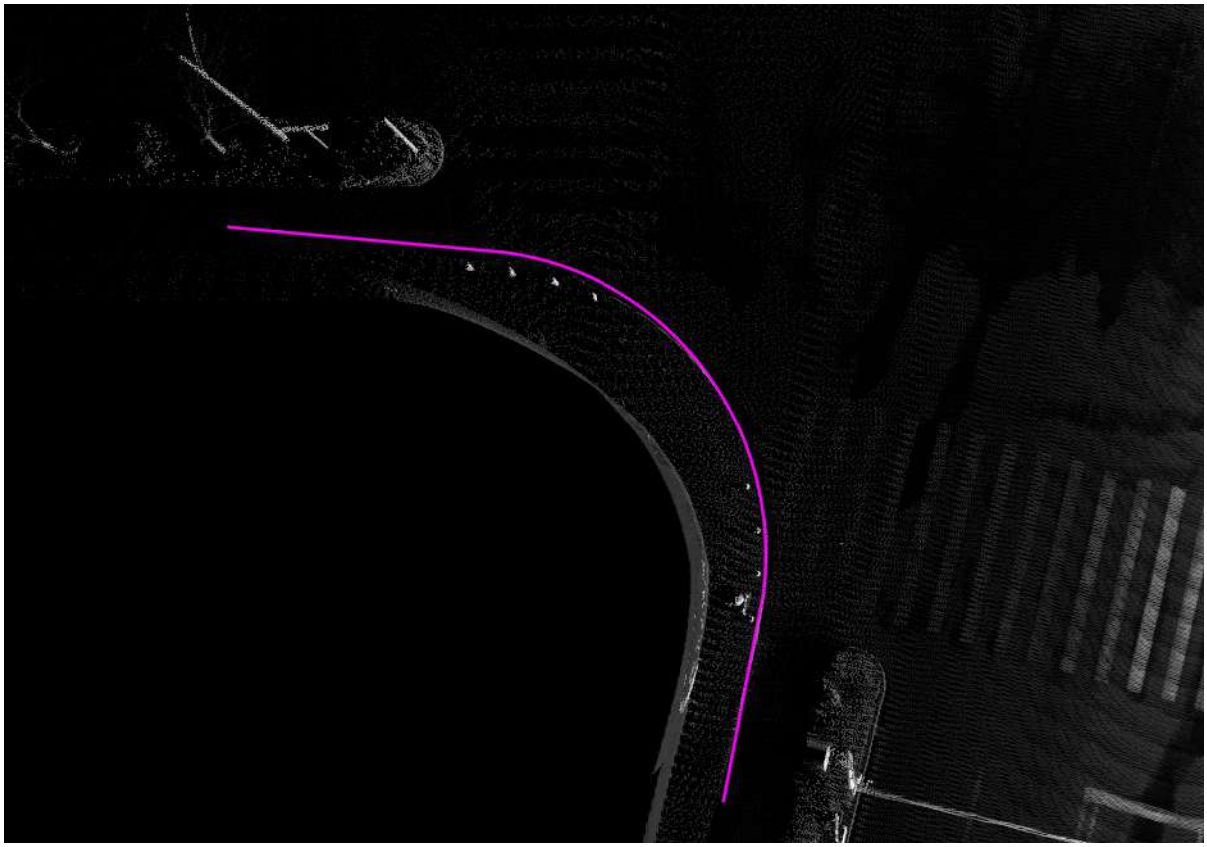
Linking result

（2）**intersect**: Select **ArcHook**, and the relative relationship of the lines is **intersect**, click the left mouse button in turn to select the two lines to be hooked, and the order of clicking must be consistent with the direction of the line, as shown in the figure below, you need to **first click** line 1 and then line 2 to achieve the desired effect. Finally **right-click** to select the key point of the arc to be generated, and the arc hook in **intersection mode** can be completed.

> Note: Intersect means that the straight lines passing through two line segments can intersect at one point.

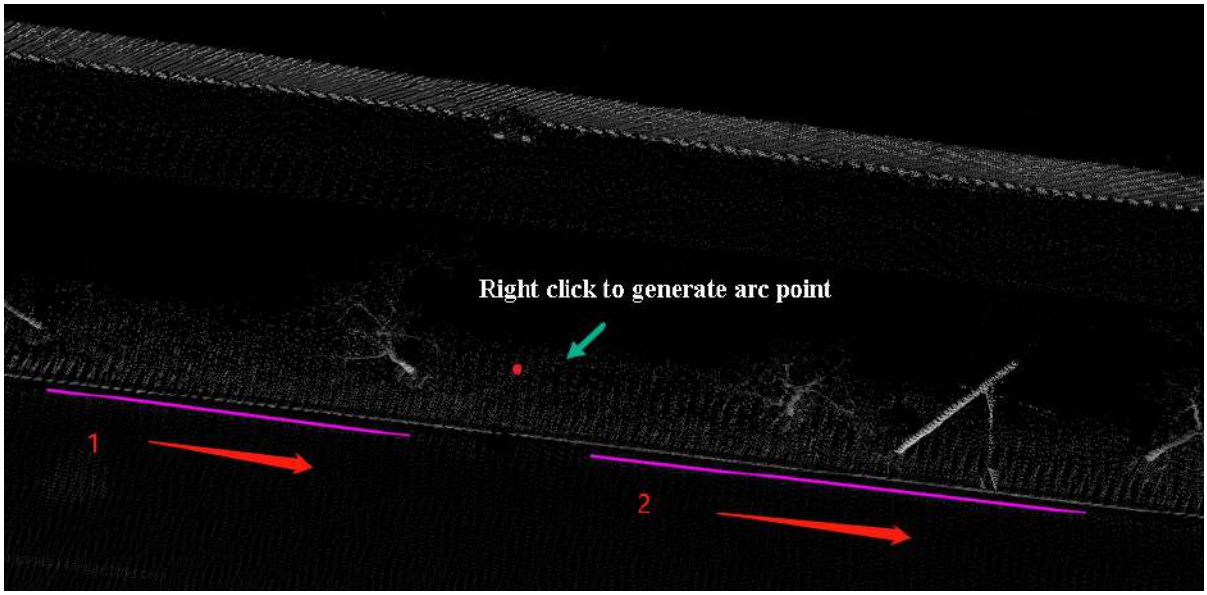


The two lines before linking

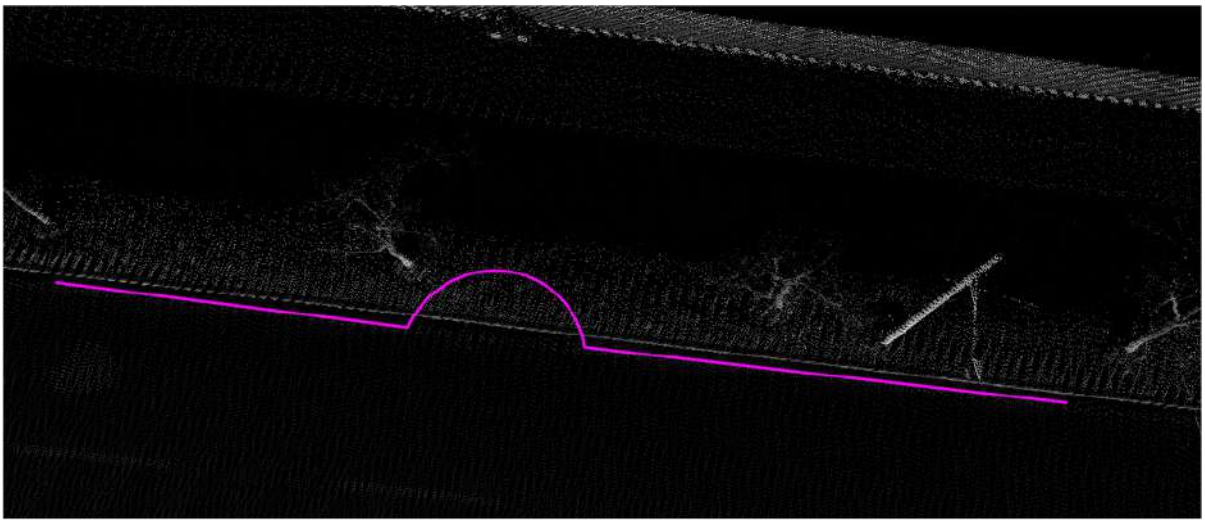

Linking result

（3）**collineation**：Select **ArcHook**, and the relative relationship of the lines is **collineation**, click the left mouse button in turn, and select the two lines to be hooked. The order of clicking must be consistent with the direction of the line. As shown in the figure, you need to **first click** line 1 and then line 2 to achieve the desired effect. Finally **right-click** to select the key point of the arc to be generated, and the arc hook in **collineation mode** can be completed.

> Note: Collineation means that the straight line passing through the two line segments is approximately on a straight line.



The two lines before linking



Linking result

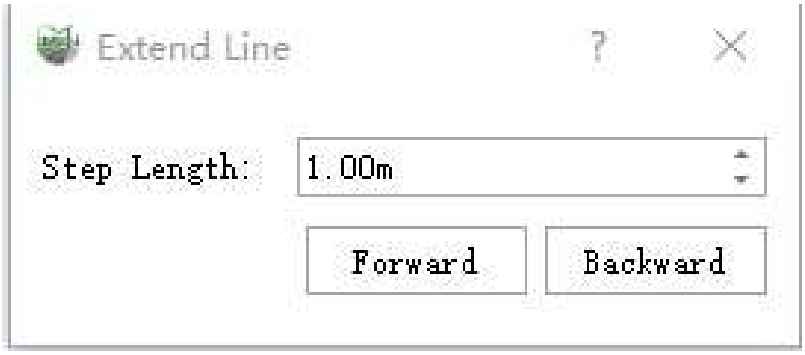
Extend Line

Function Description：The Extend Lines function allows the user to adjust the vector length to make the line object extend in the direction of the head or tail.

Steps

1.Click the **Extend Line** button in the Vector Editor.

2.**Extend Lines** dialog box pops up.



Extend Lines dialog box

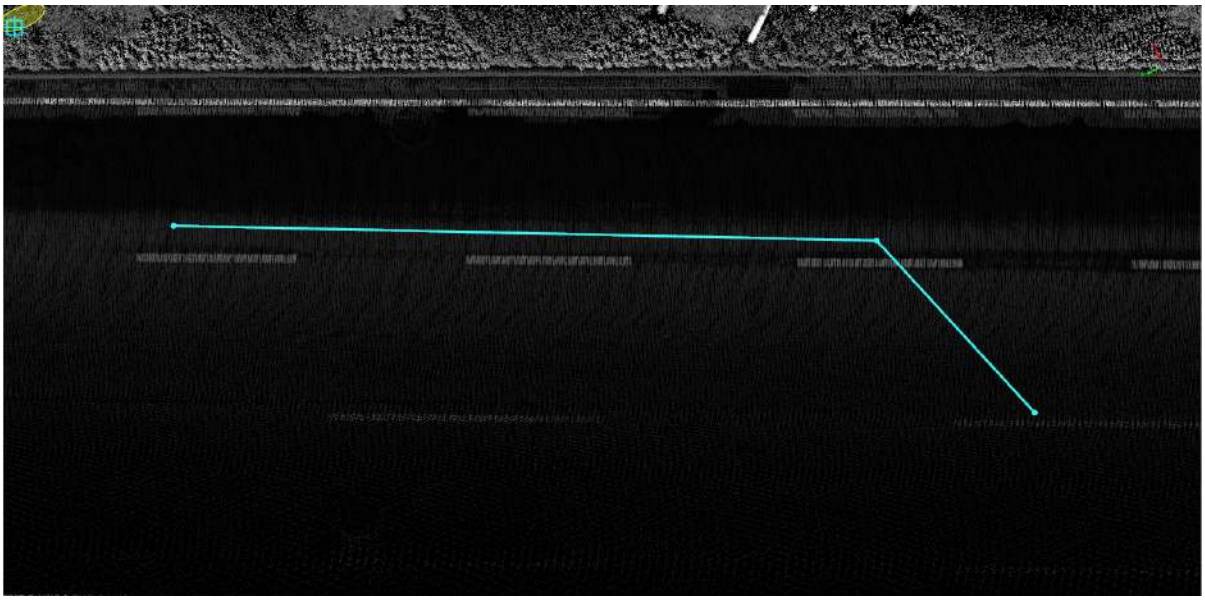
Parameter description:

Step Length: The default is 1.0 meters, which means the length of the line object to extend forward or backward each time.

> Note: When using shortcut keys, make sure that the 3D window is the current active window.

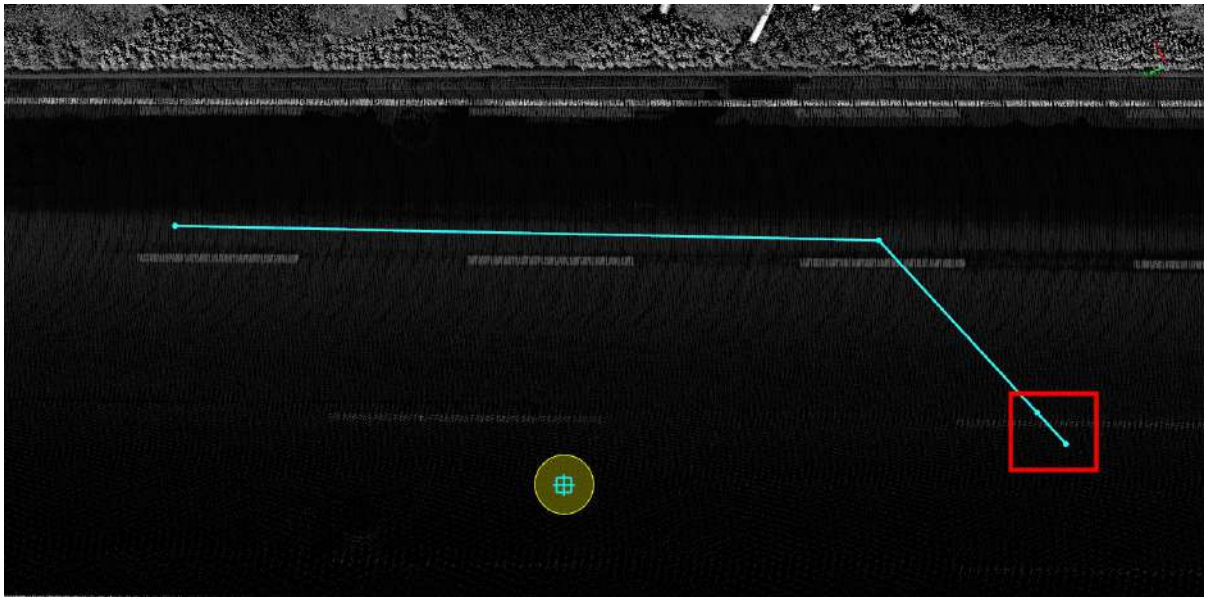
Shortcut keys	Function
F	Forward
B	Backward
Ctrl+Z	Fallback extension point
H	Hide shortcut key pop-up dialog box

3.Click the mouse to select the line object to be extended.

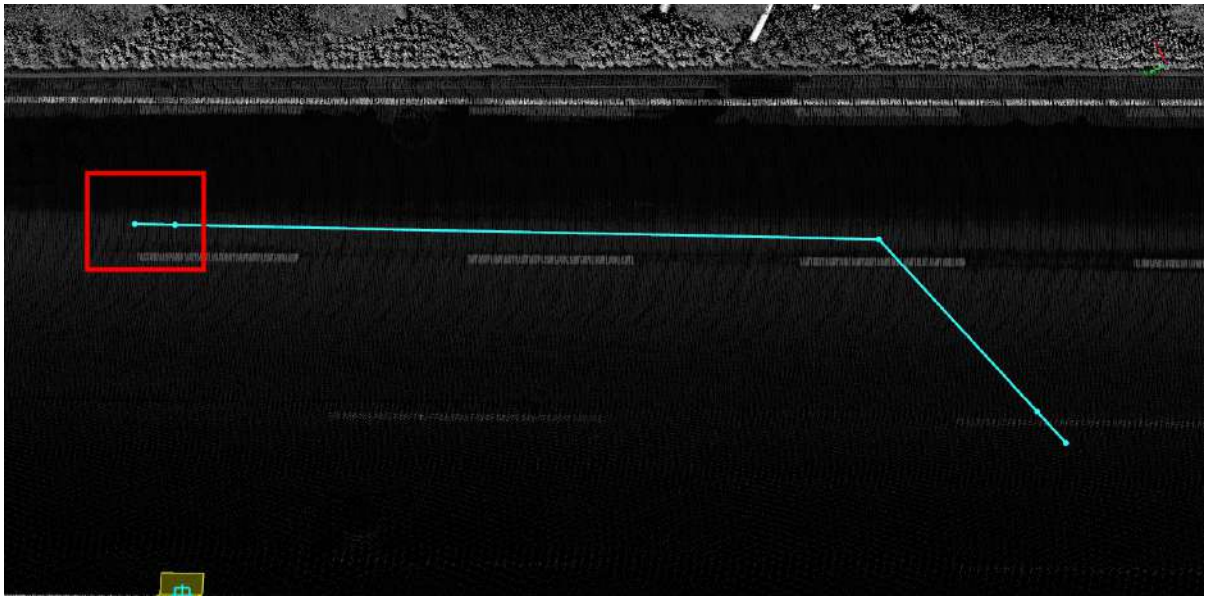


Select the line to be extended

4.Click the Forward or Backward button in the dialog box, and the line object will be extended in the corresponding direction.



Extend forward



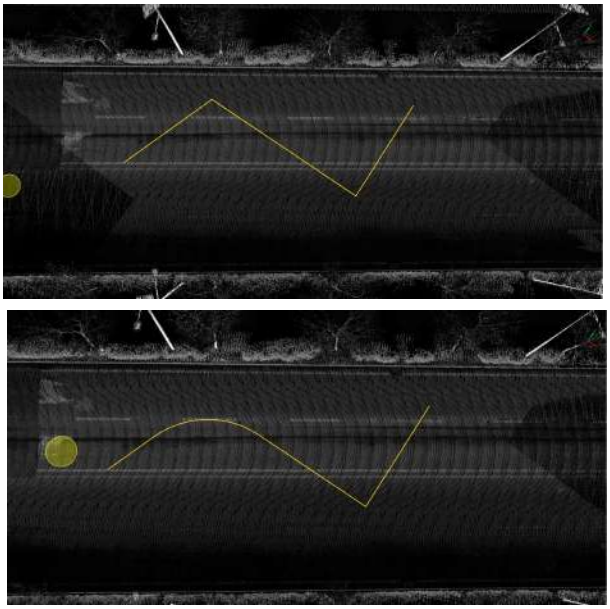
Rendering of extend forward

Fillet

Function Description: The Fillet function allows the user to adjust the size of the angle through the edge of the same node.

Steps

1.Click **Fillet** button in the Vector Editor, click the left mouse button to click the two intersecting line segments consecutively, move the mouse, determine the needed angle according to the requirements, and double-click the mouse to complete.



Fillet

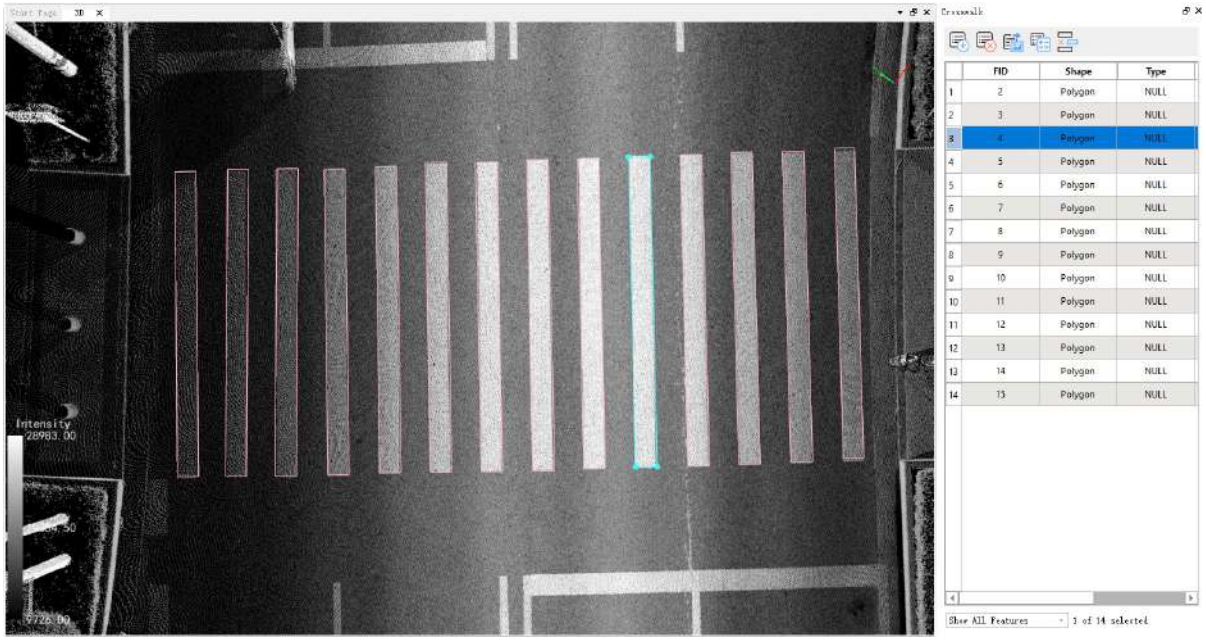
Combine Polygons or Points

Function Description: This function allows the user to combine different vectors in the same layer to form a vector object.

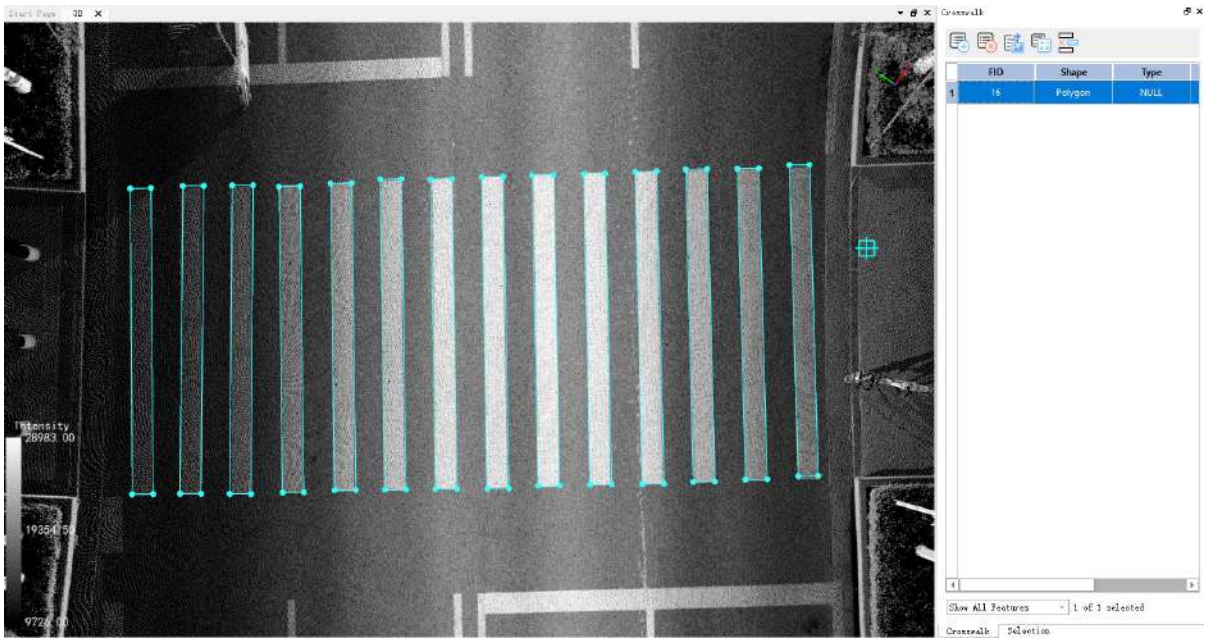
Steps

1.Click **Combine Polygons or Points** button in the Vector Editor, click the left mouse button, drag the mouse to select the vector objects that need to be combined, and click Enter to complete the object combination.

> This function can only process the vectors in the same layer, the details are as follows:



Vectors to be combined



The combined vector

Smooth

Function Description：Make unsmooth vector objects spreading and smooth.

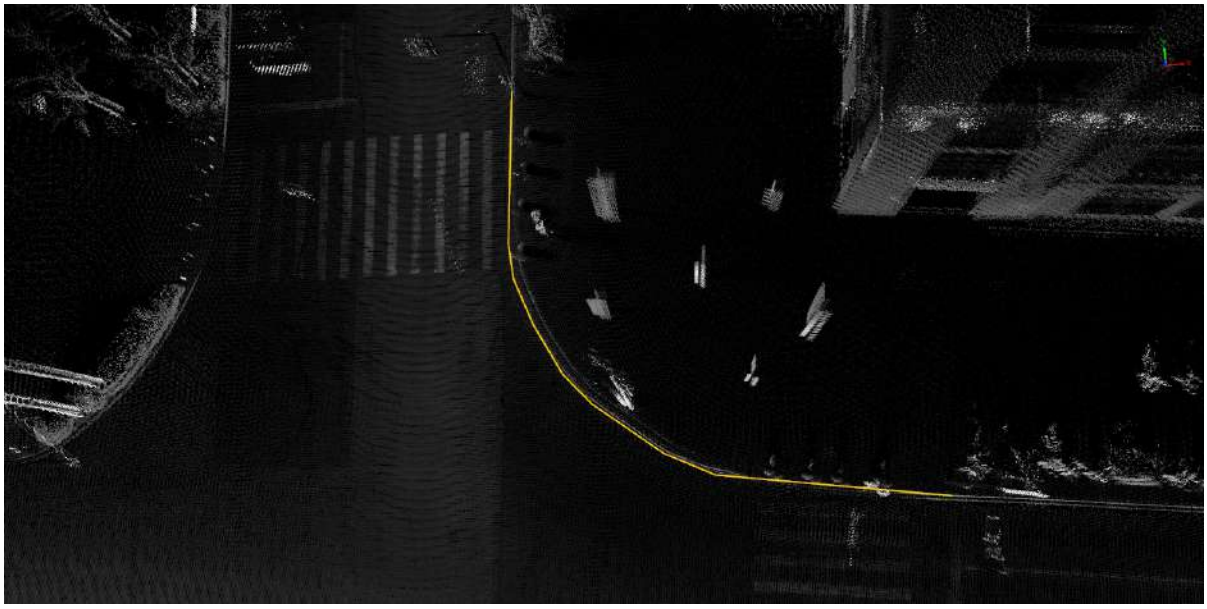
Steps

1.After clicking the **Smooth** button in the vector editor, Pop-up box for setting vector smoothing parameters.

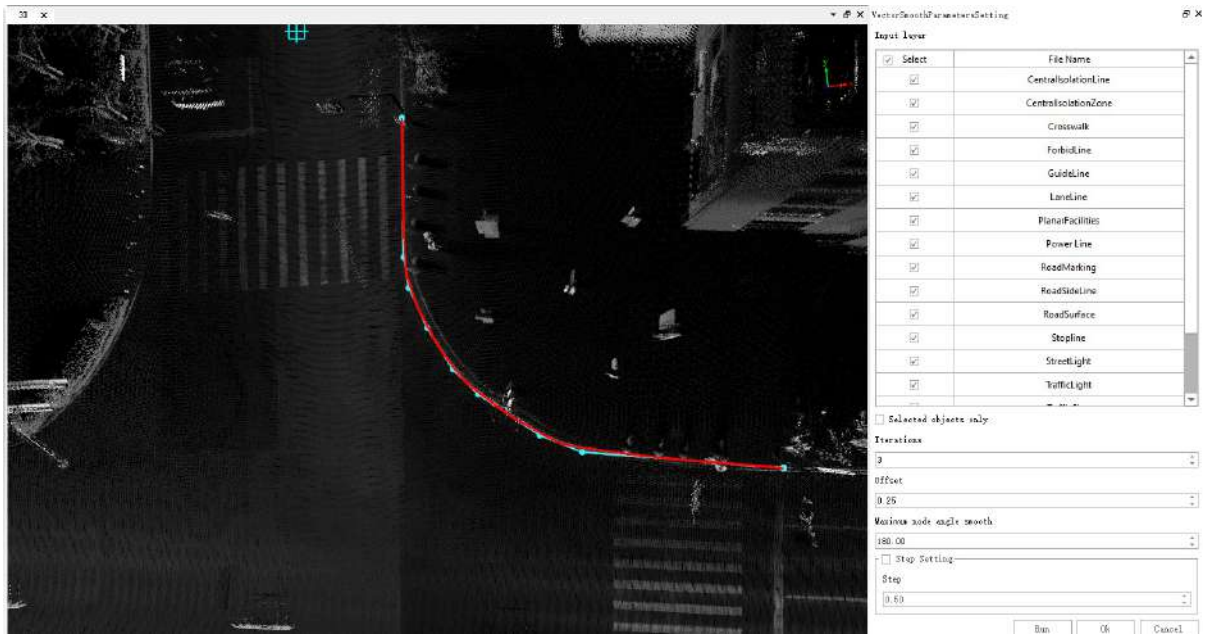
Parameter Description

- **Input layer**：Select layers according to requirements
- **Selected objects only**：After checking the box, click on the object you want to change
- **Iterations**：Default iteration 3. Up to 100 iterations. Set the number of iterations as required
- **Offset**：Default offset 0.25. up to 10. the distance between the actual address of the memory cell and the segment address of the segment it is in
- **Maximum node angle smooth**：The default maximum smoothing angle is 180°. The minimum smoothing angle is 1°. Set the smoothing angle according to the requirements
- **Step Setting**：Whether to sample nodes, while smoothing, set the interval between two sampled nodes. For example, if the interval is set to 0.5, all nodes will be sampled with a spacing of 0.5 after smoothing.
- **Run**：Preview of the effect after setting parameters
- **OK**：After setting the parameters and previewing the effect, click the OK button to save the settings
- **Cancel**：Cancel Settings

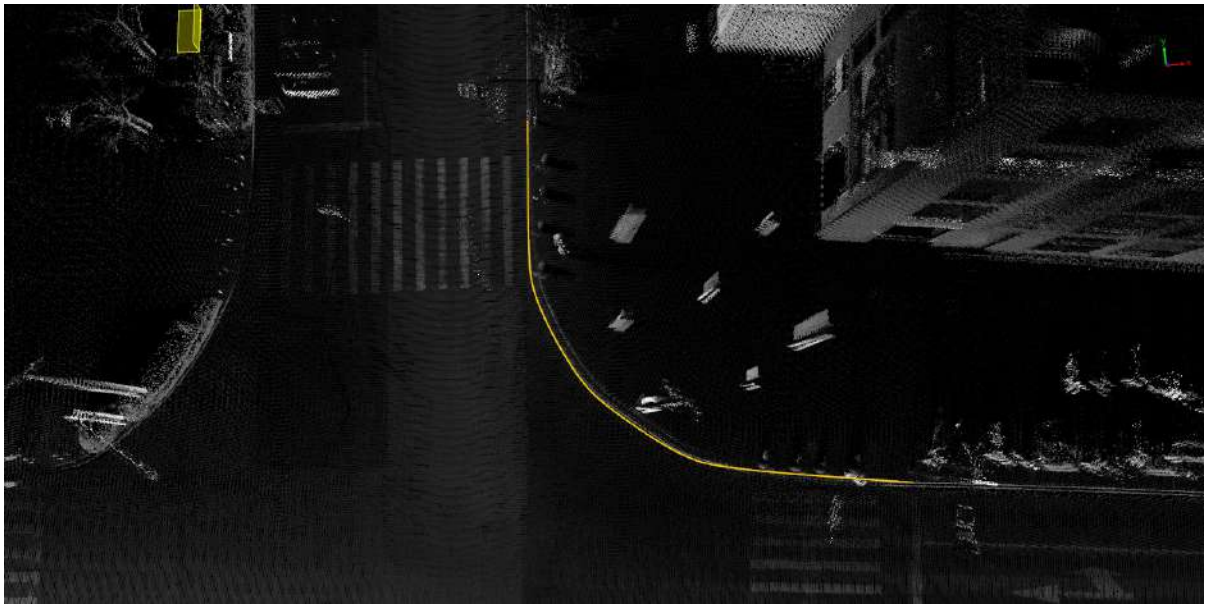
2.The result of smoothing is shown in the following figure.



Vector to be smoothed



Preview effect



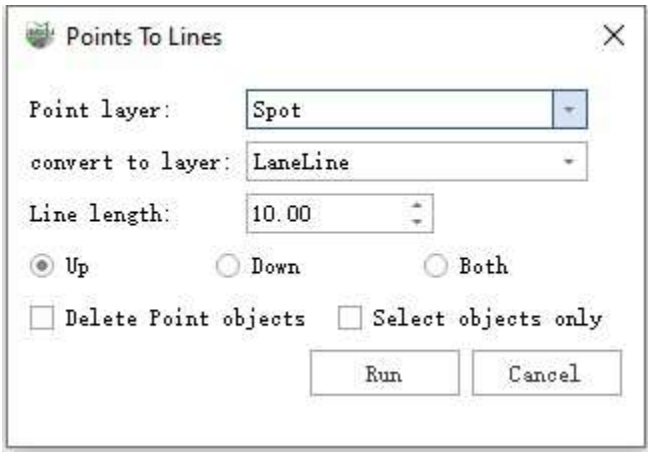
Vector after smoothing

Point to Line

Function Description: Extends the selected point vector object to create a line vector object in the positive or negative direction of the Z-axis of the point vector object, or in both directions, with a specified length.

Steps

1.Click the **Point to Line** button in the vector editor. Select the point vector object that needs to be converted to highlight it, then press "Enter" to confirm the selection, and a dialog box will pop up.



Function Description

- **Point Layer**: Select the point layer that you want to convert based on your requirements.
- **Convert to layer**: Select the line layer based on your requirements.
- **Line length**: Set the length of the converted line vector object based on your requirements.
- **Up**: The line vector object extends in the positive Z-direction from the selected point based on the set length.
- **Down**: The line vector object extends in the negative Z-direction from the selected point based on the set length.
- **Both**: The line vector object extends in both the positive and negative Z-directions from the selected point based on the set length.
- **Delete Point objects**: After converting to a line vector object, do you want to delete the point vector object.
- **Select objects only**: Select the point vector object that you want to convert based on your requirements.
- **Run**: After selecting the layer, click "OK" to complete the conversion.
- **Cancel**: Cancel the conversion.

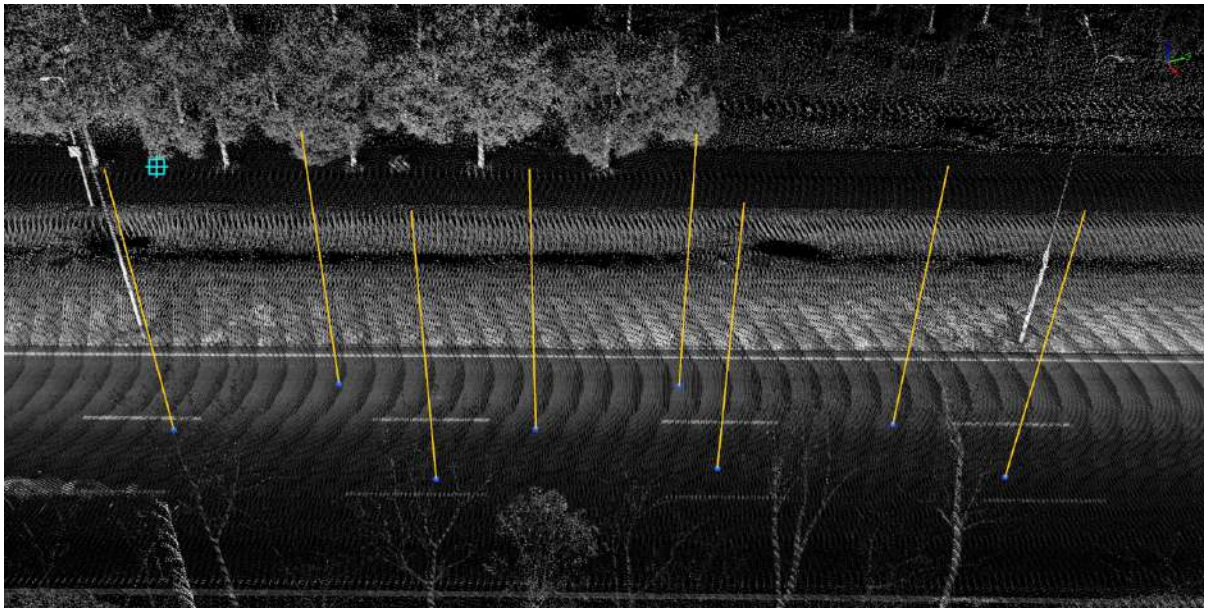
2.There are two ways to perform the conversion:

2.1 First conversion method: Through layer conversion:

（1）First, select the point layer you want to convert, then choose the layer where the lines will be stored. Next, set the length and direction of the converted lines. You can also choose whether to retain the line vector objects based on your requirements.

（2）After setting the parameters, click "Run" to start the conversion.

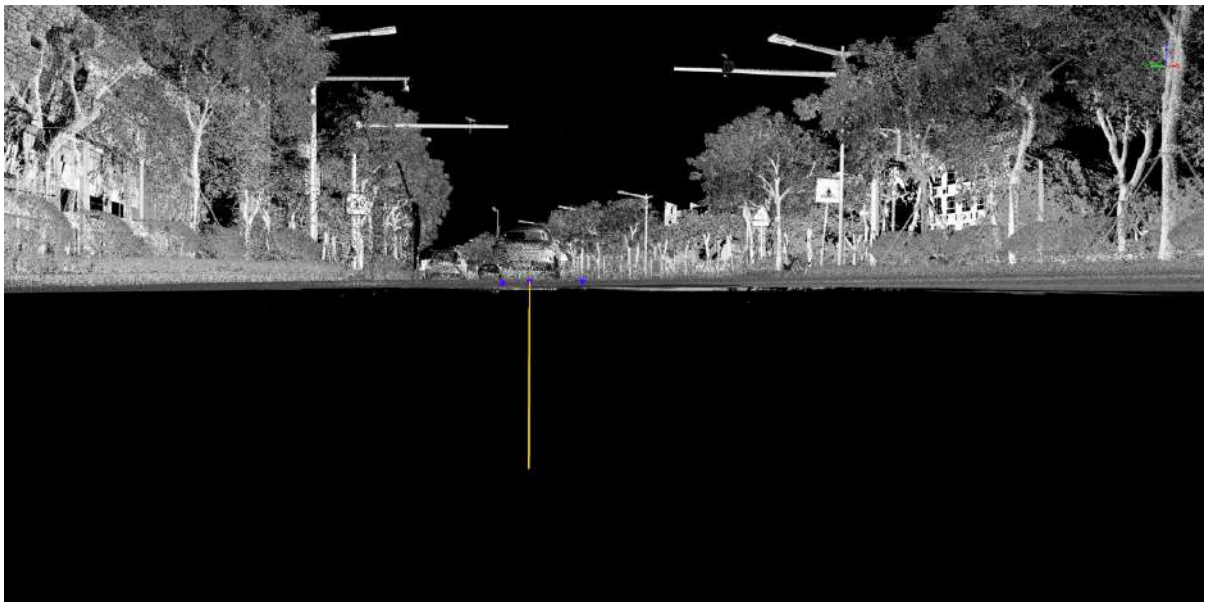
（3）Using the example of generating upward, as shown in the following figure.



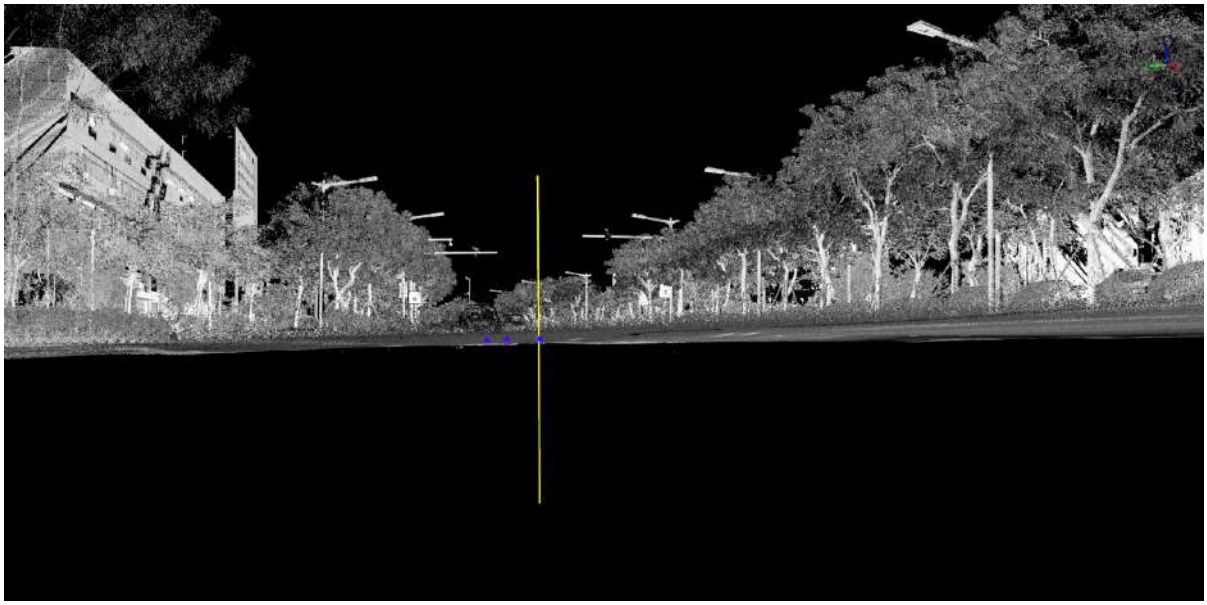
Up

2.2 The second method is: Select the line vector object for conversion. Choose the line vector object you want to convert to highlight it, then set the parameters. Finally, click "Run" to complete the conversion.

Up



Down



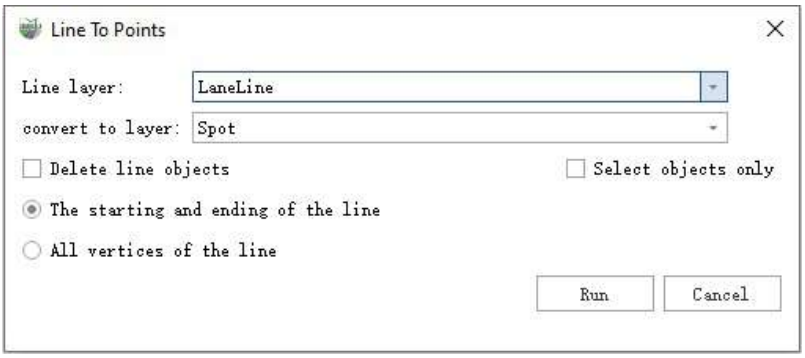
Both

Line to Points

Function Description Converts projection points on both ends of a line and the center of its bounding box into point vector objects.

Steps

1.Click the **Line to Points** button in the vector editor. Select the line vector object that needs to be converted to highlight it, then press "Enter" to confirm the selection, and a dialog box will pop up.



Function Description:

- **Line layer**: The line layer to be converted.
- **Convert to layer**: Select the point layer where the converted data will be stored based on your requirements.
- **Delete line objects**: After converting to a point vector object, do you want to delete the line vector object?
- **Select objects only**: Select the line vector object that you want to convert by clicking on it.
- **The starting and ending of the line**: Only convert the start and end points of the line vector object.
- **All vertices of the line**: Convert all nodes of the line vector object.
- **Run**: After selecting the layer, click "OK" to complete the conversion.
- **Cancel**: Cancel the conversion.

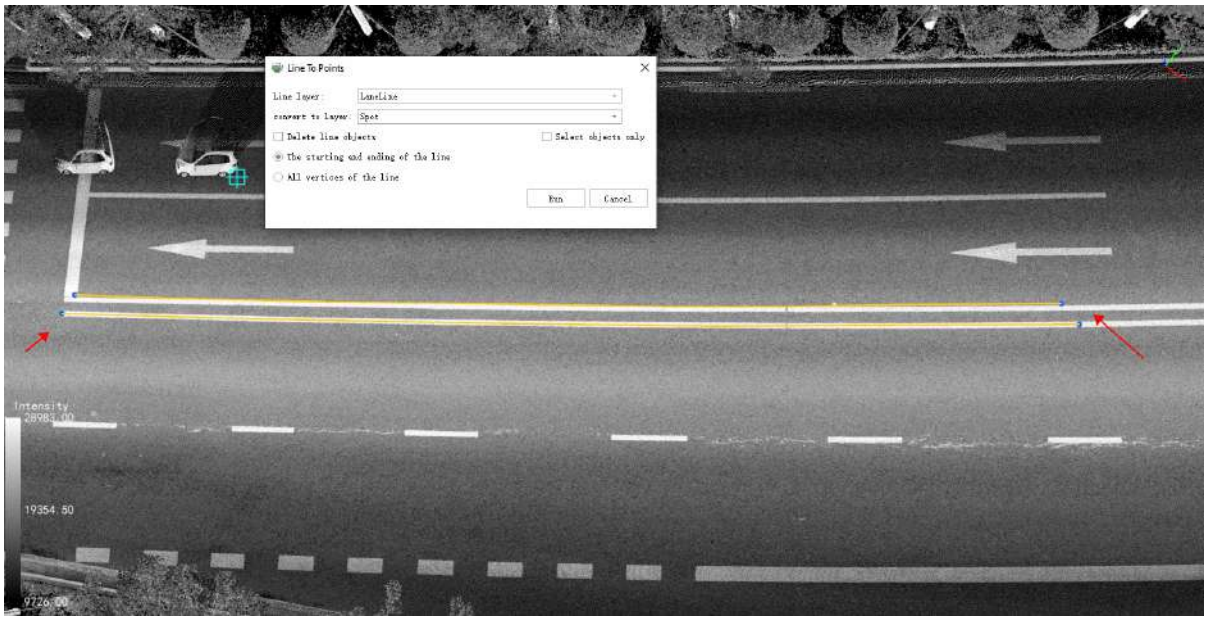
2.There are two methods of conversion, and there are also two results of the conversion. You can choose the conversion method based on your needs.

2.1The first conversion method: Through layer conversion:

（1）First, select the line layer you want to convert, then choose the layer where the points will be stored. Optionally, decide whether to retain the line objects based on your requirements:

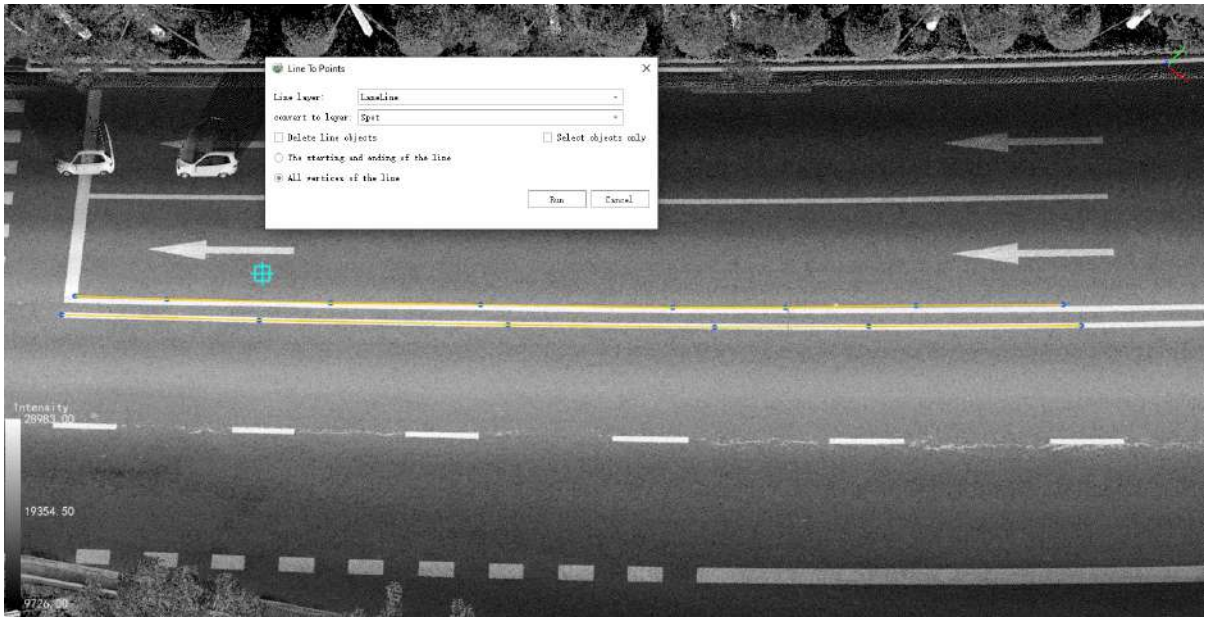
（2）There are two ways to convert the point results, and then click "Run" to complete the conversion.

- **The starting and ending of the line**: Only convert the start and end points of the line to the selected layer.



The starting and ending of the line

- **All vertices of the line**：Convert all nodes of the line vector object to the selected layer.

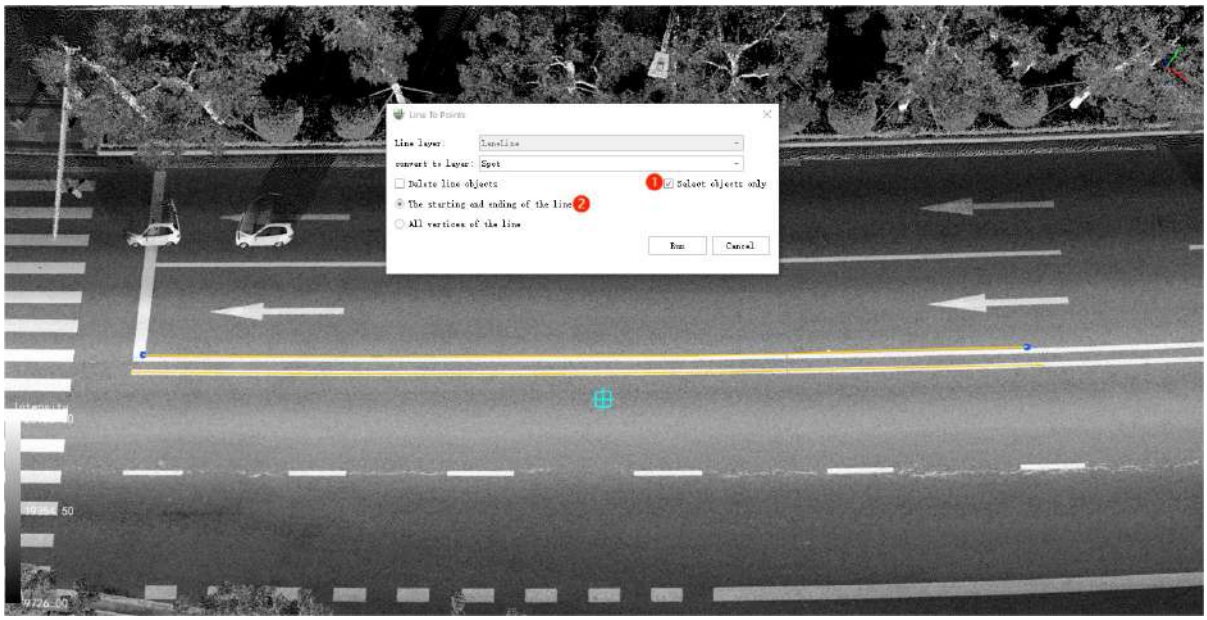


All vertices of the line

2.2 The second method is: Select the line vector object for conversion:

（1）First, select the line vector object you want to convert to highlight it. There are also two ways to obtain point conversion results. Then, click "Run" to complete the conversion.c

- **The starting and ending of the line**：Only convert the start and end points of the line to the selected layer.



The starting and ending of the line

- **All vertices of the line**: Convert all nodes of the line vector object to the selected layer.

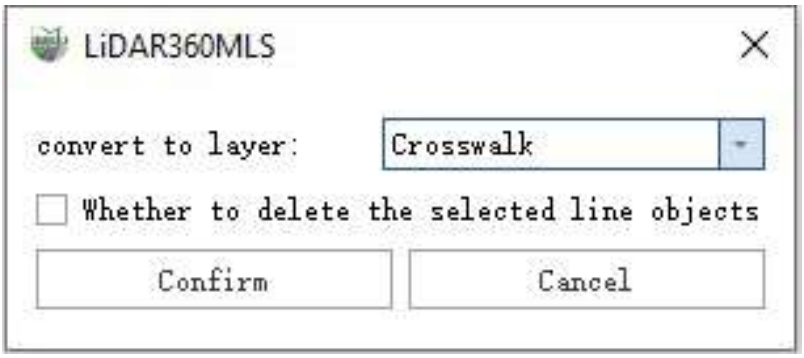
All vertices of the line

Construct Polygon

Function Description：This function constructs a polygon object by connecting specified lines end-to-end with the same direction.

Steps

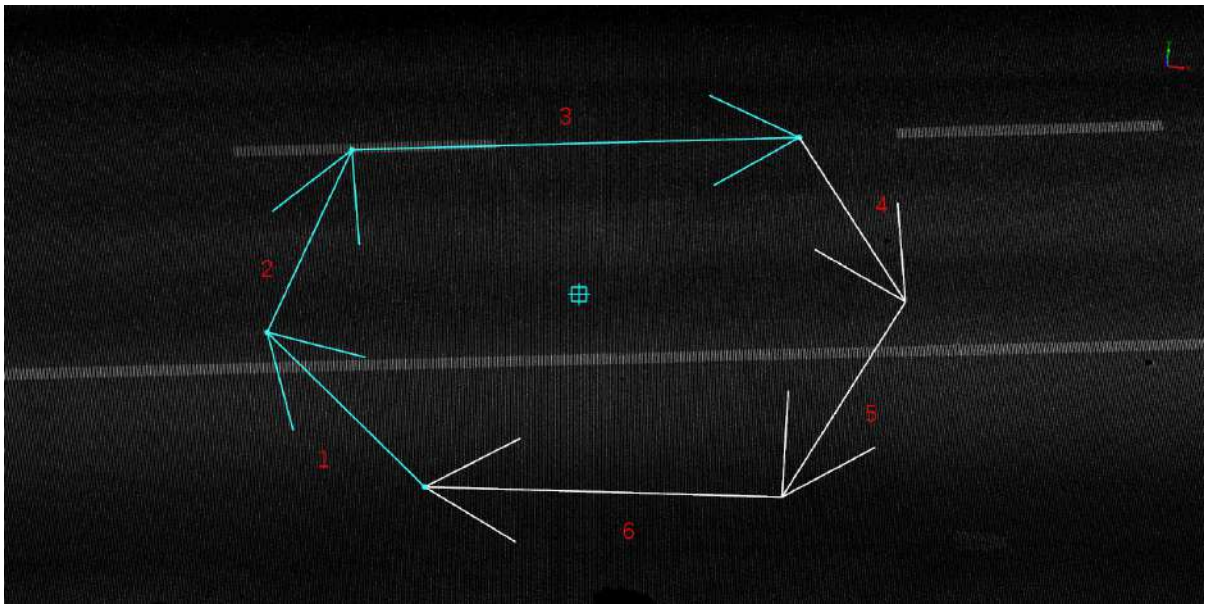
1.Click the **Construct Polygon** button in vector editing, and a dialog box will pop up:



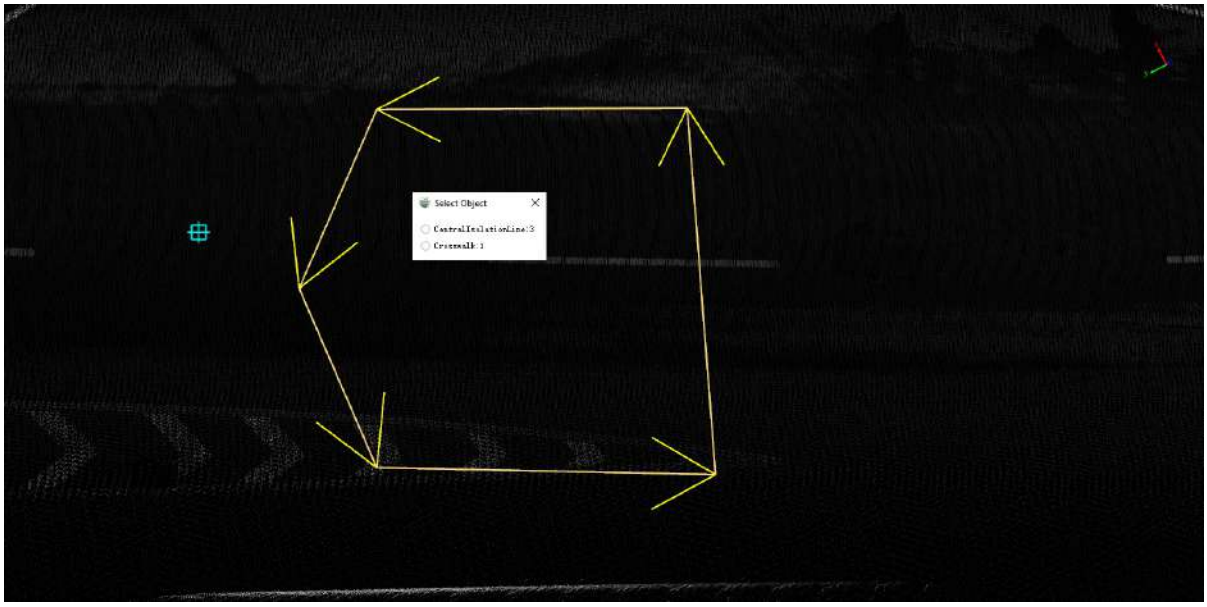
Function Description

- **convert to lay**：Choose the face layer where you want to store it based on your needs.
- **Whether to delete the select layer**：After converting to a line vector object, do you want to delete the point vector object?
- **Confirm**：After selecting the layer, click "OK" to complete the conversion.
- **Cancel**：Cancel the conversion.

2.In sequence, click on the lines that need to form the face. It's important to note that adjacent lines must be connected at their endpoints, and their directions must be the same. If these conditions are not met, you can use the node editing tool to make adjustments.

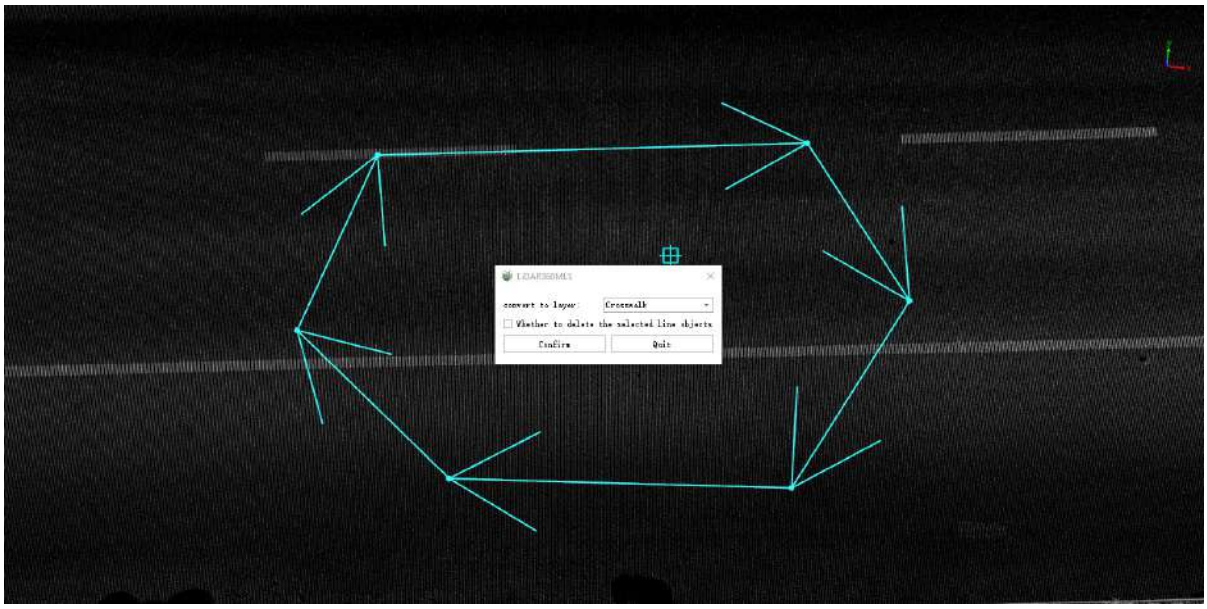


3.If there are multiple lines at the same location, a dialog box will appear when you click, allowing you to select only one of them.

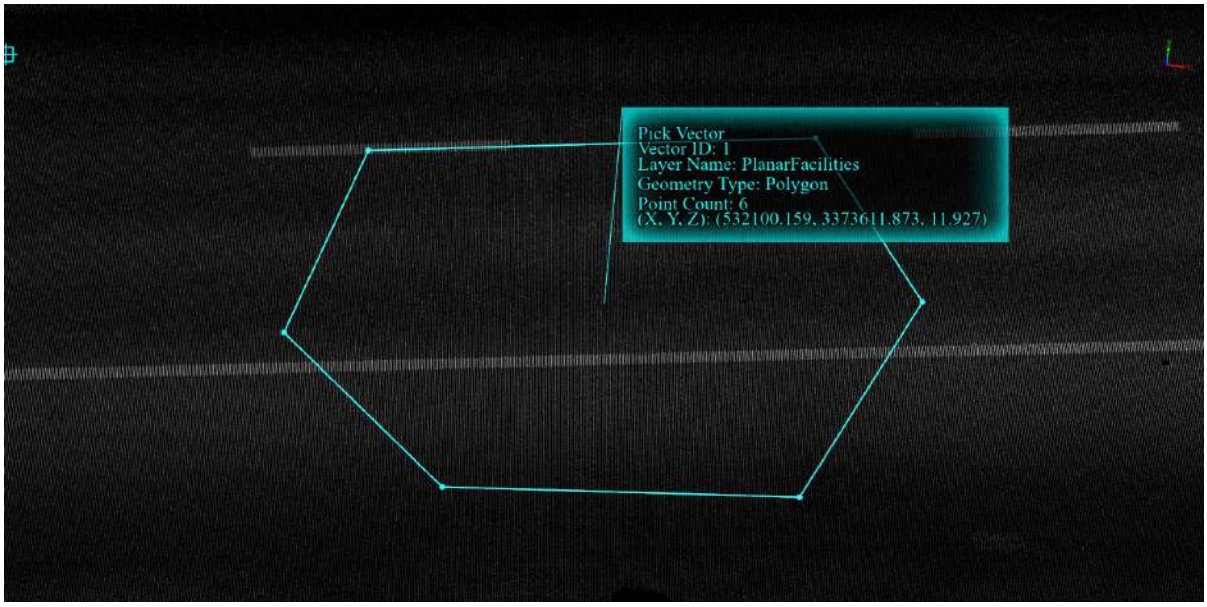


4.If you want to cancel a previously selected line, you can click on it again to deselect it.

5.Once you have selected all the lines you want to construct into a face, choose the layer where the face object will be stored in the dialog box. If you want to delete the lines used to construct the face, you can check the "Delete Selected Objects" checkbox. When constructing the face, the original lines will be deleted.



6.After confirming the layer for storing the face object, click "OK" to complete the construction of the face.


Pick Vector
Vector ID: 1
Layer Name: PlanarFacilities
Geometry Type: Polygon
Point Count: 6
(X, Y, Z): (532100.159, 3373611.873, 11.927)

Intersection Line

Function Description：The line intersection functionality allows for the splitting of lines at the positions where they intersect. Intersection points can occur at specific locations or at implicit intersection points along one or two extending lines.

Steps

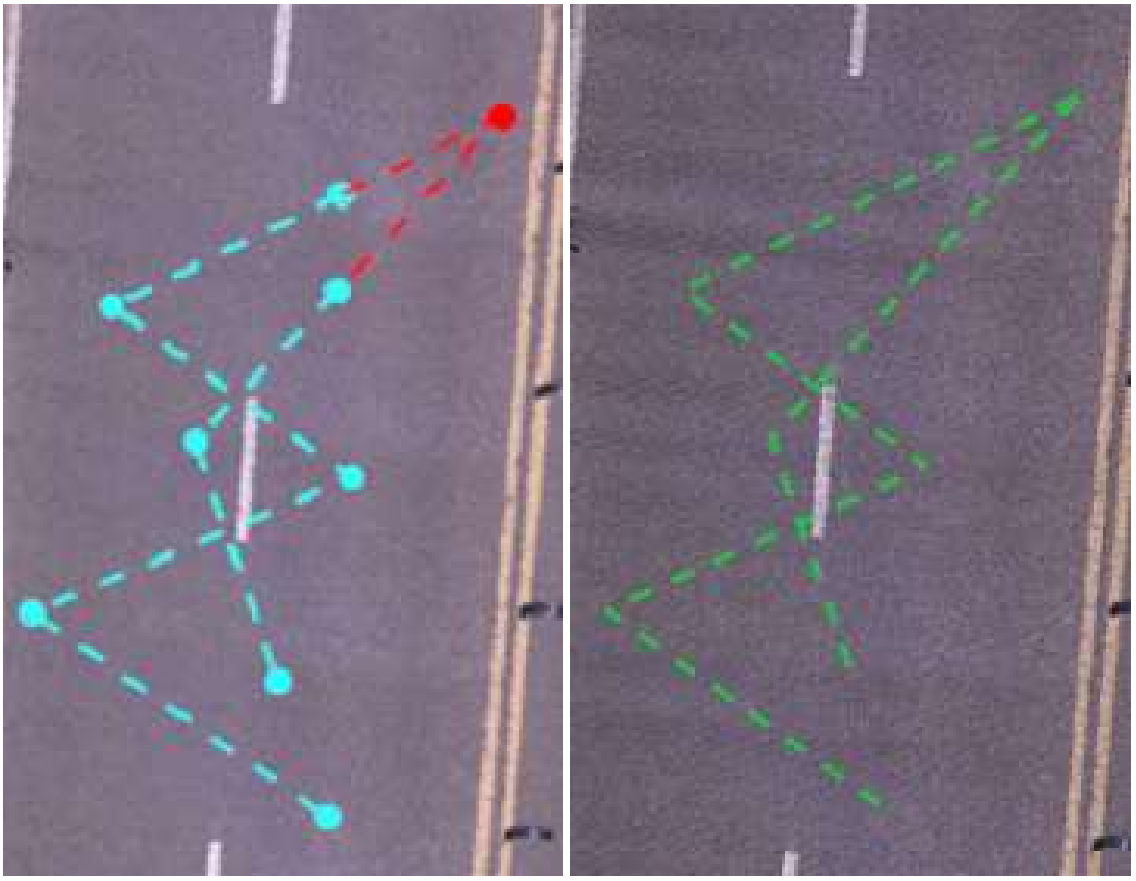
1.Clicking the **Line Intersection** button in vector editing switches the viewer to 2D mode, disabling the roaming function for dragging and rotating.

2.Left-click with the mouse sequentially on the two lines that require intersection operations.

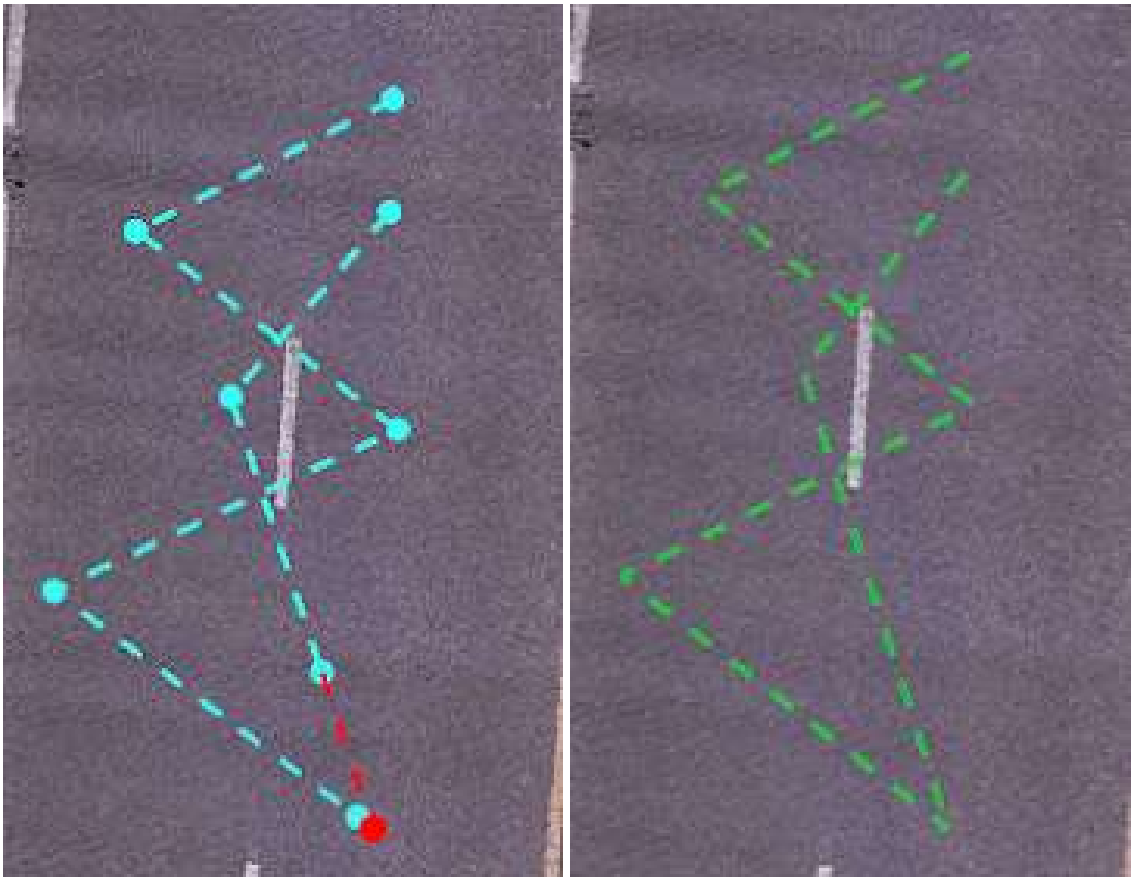
3.When the two lines have only one intersection point, the intersection operation will be executed immediately. If the intersection point lies on the line, the line object will be split into two lines at the intersection point. If the intersection point lies on the extension of the line, the line object will be extended to the intersection point. See the illustration below:

Before and after the intersection

4.When there are multiple intersection points between two lines, the intersection operation will not be executed immediately. Instead, a preview of one of the intersection scenarios will be displayed. Moving the mouse over other segments of the line object will display previews of the intersection after applying the operation to those segments as well if they intersect. Once you confirm the desired intersection operation preview, click the left mouse button to execute the intersection operation.



Before and after the intersection



Before and after the intersection

Multipart To Singleparts

Function Description：This function converts multiple objects into a single feature.

Steps

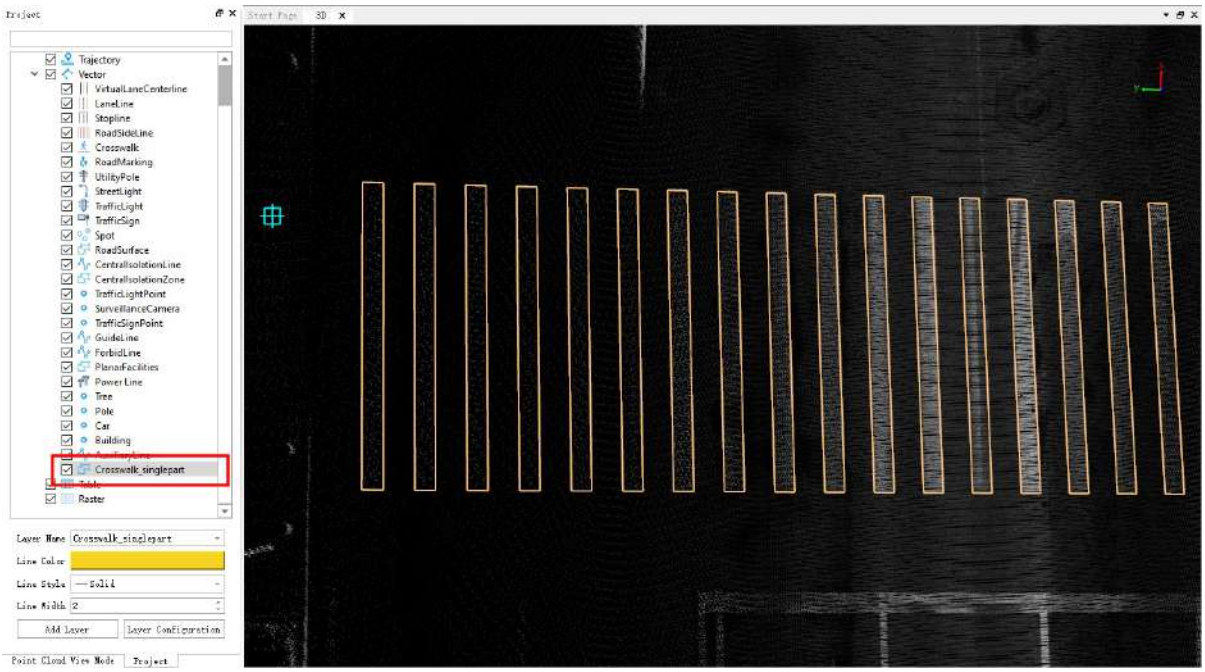
1.Click the **Multipart To Singleparts** button in the vector editing menu to open the Multi-Part to Single-Part dialog box.

Parameter

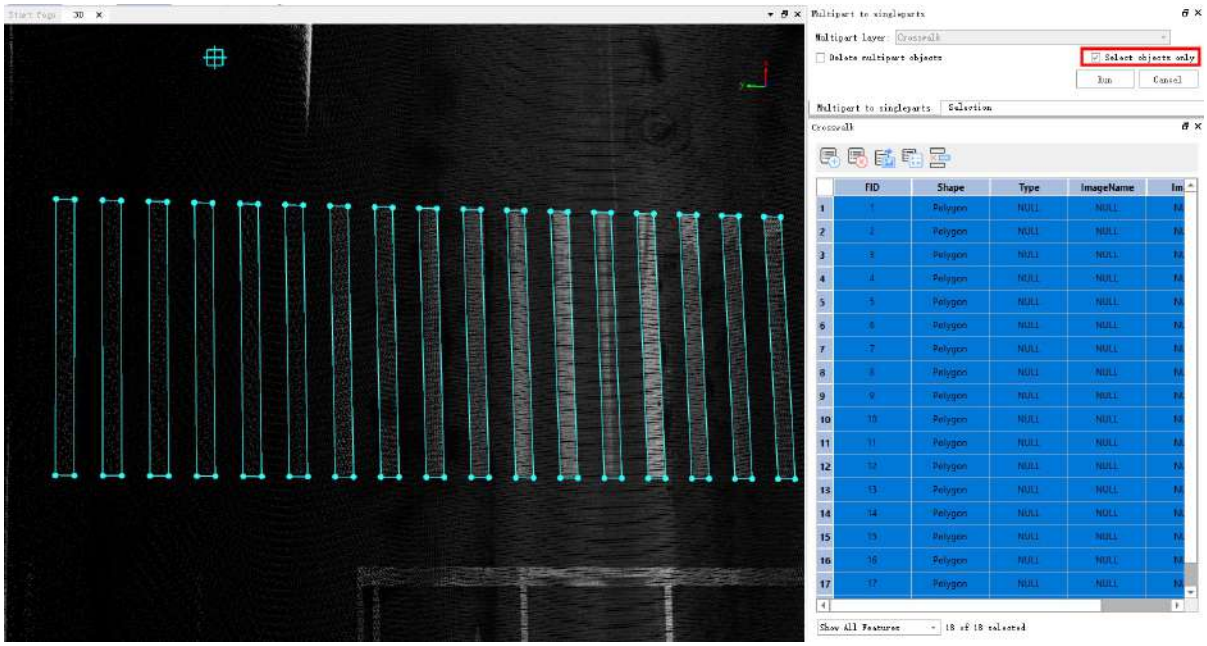
- **Multipart layer**：The layer where the multi-part objects are located
- **Delete multipart objects**：After converting to a single part, do you need to delete the multi-part objects?
- **Select objects only**：Select the multi-part objects you want to convert.
- **Run**：After selecting the multi-part objects you want to convert, click the "Run" button to initiate the transformation.
- **Cancel**：Exit function.

2.There are two ways to convert:

（1）When performing batch conversion through layers, a new layer will be created and named as follows: "Layer Name of Multi-Object Conversion_singlepart."



（2）Select the multi-objects you want to convert, and after the conversion, they will be in the original layer.


Multipart to singleparts
Multipart layer:
Crosswalk
Delete multipart objects
Select objects only
Run
Cancel
Multipart to singleparts
Selection
Crosswalk
FID
Shape
Type
ImageName
Polygon
NULL
Show All Features
18 of 18 selected

Auxiliary Functions

- Vector Draping
- Line to Polygon
- Polygon to Line
- Intersection Drawing
- Perpendicular Drawing

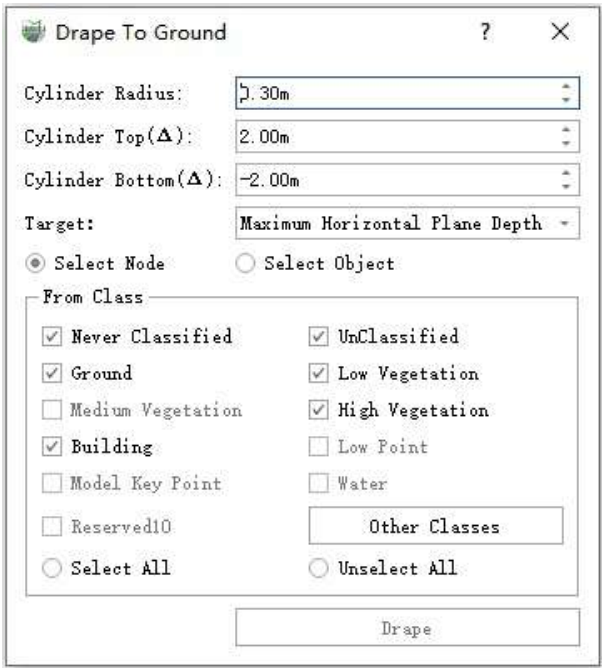
Drape To Ground

Function Description：Fit the vector object or part of the node of the vector object to the ground setting position.

Steps

1.Click the **Drape To Ground** button in the vector editor.

2.Pop up **Drape To Ground**：



Drape To Ground

Parameter Description:

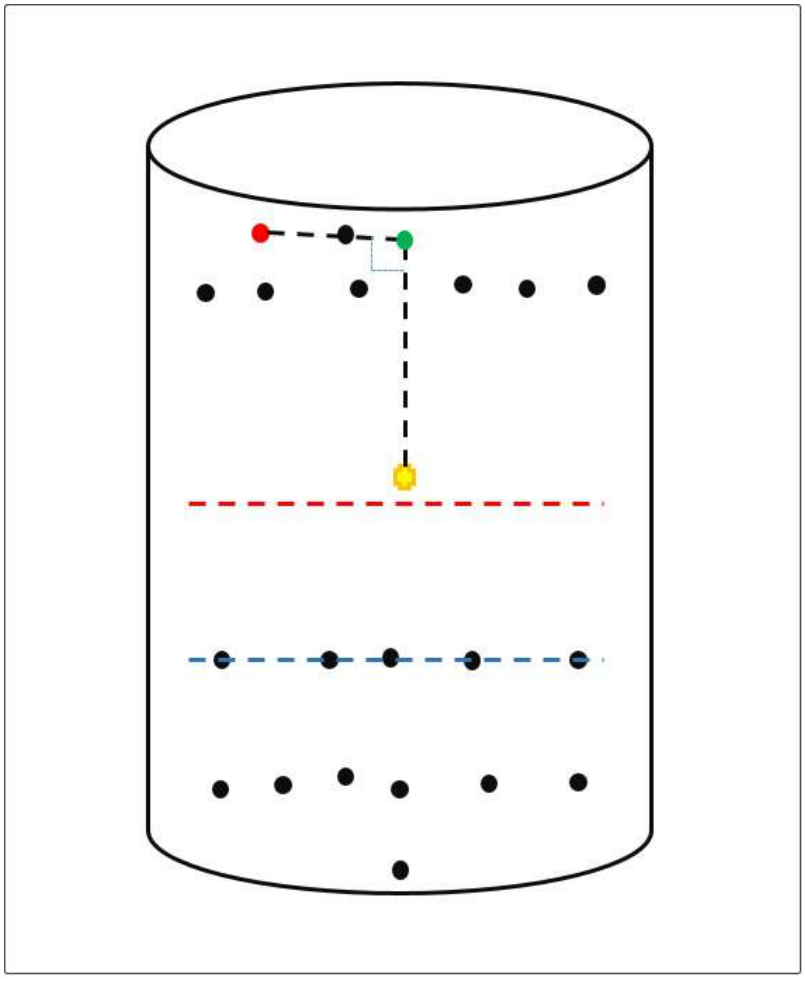
The Drape To Ground function, within a cylinder, performs the search and calculation of the point cloud to achieve the ground-fitting. The cylinder, with the principal axis direction parallel to the Z-axis and the center of the principal axis as the vector node in the current calculation, is referred to hereafter by the search cylinder as a surrogate for the cylinder.

- **Cylinder Radius**：Default is 0.3 m. Search for the radius of the column.
- **Cylinder Top(Δ)**：The default is 2.0 m, which means the upper surface is located 2.0 m above the current node, and the relative height of the upper surface of the search cylinder to the current node. Can be negative, which means the upper surface is located below the current node.
- **Cylinder Bottom(Δ)**：The default is -2.0 m, which means the lower surface is located 2.0 m below the current node, and the relative height of the lower surface of the search cylinder to the current node. Can be a positive value, which means the lower surface is located above the current node.

  > Note: The upper surface value should be greater than the lower surface value

- **Target**：Default is the lowest depth, the current node needs to fit to the surface location, physical meaning see the search column schematic below

- Minimum depth: without changing the XY value of the current node, the Z value takes the average Z value of the effective layer where the lowest point is located
- Maximum depth: without changing the XY value of the current node, the Z value takes the average Z value of the effective layer where the highest point is located
- Maximum point depth: without changing the XY value of the current node, the Z value is taken as the average Z value of the layer with the highest number of points in the search cylinder
- Average depth: without changing the XY value of the current node, the Z value takes the average of the lowest depth and the highest depth
- Median depth: without changing the XY value of the current node, the Z value takes the Z value of the middle valid layer in the search cylinder
- Lowest point: move the current node to the lowest point within the search cylinder
- Highest point: move the current node to the highest point of the search column



Schematic diagram for searching columns

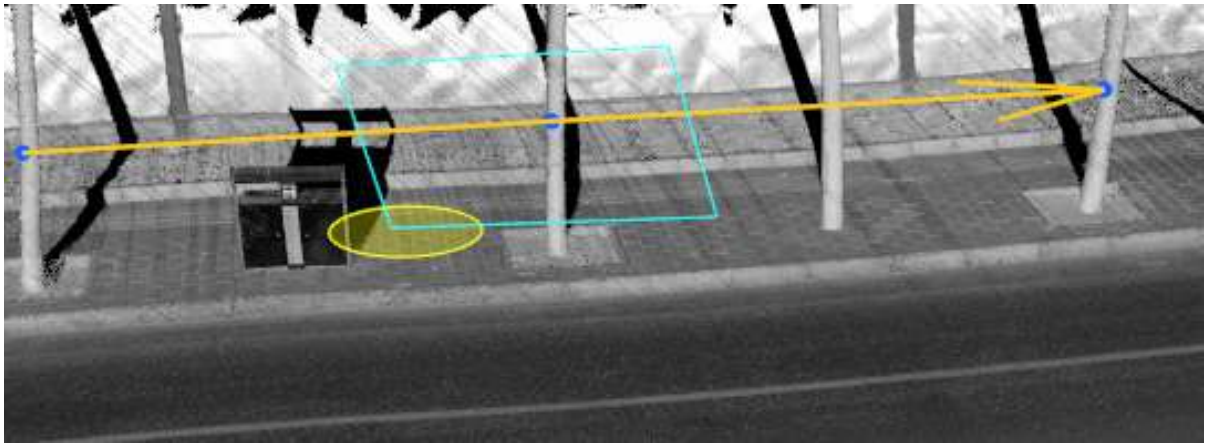
The middle yellow point is the current node, the red point is the highest point, the green point is the virtual point represented by the highest depth, the red dashed line represents the average depth, and the blue dashed line represents the median depth

- **Select Node**: Polygon box selection method, select the area to be grounded, the nodes of vector objects located in the area will be grounded

- **Select Object**：Tap to select a vector object and fit all the nodes of the object, you can tap multiple objects and calculate at the same time
- **From Class**：Select the category point cloud to be used for the calculation
- **Drape**：The node selection method is not available. After selecting the object, click the button to start the fitting operation.

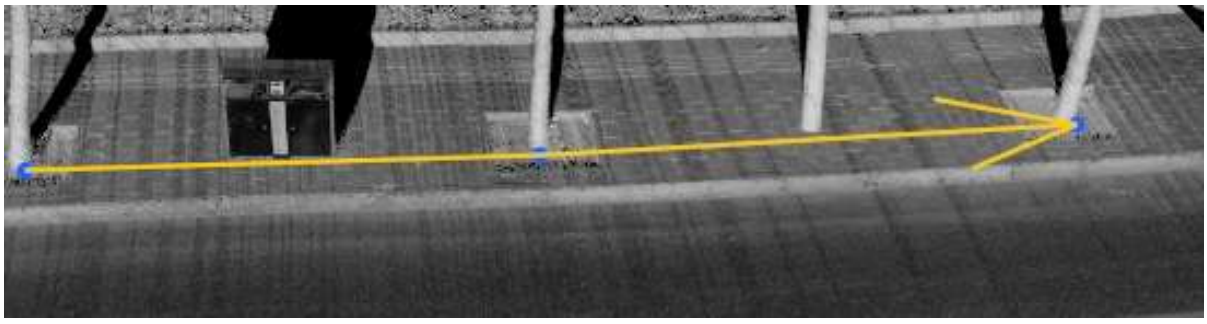
Select node mode

1.Mouse box the nodes that need to be grounded.



Check the nodes that need to be grounded

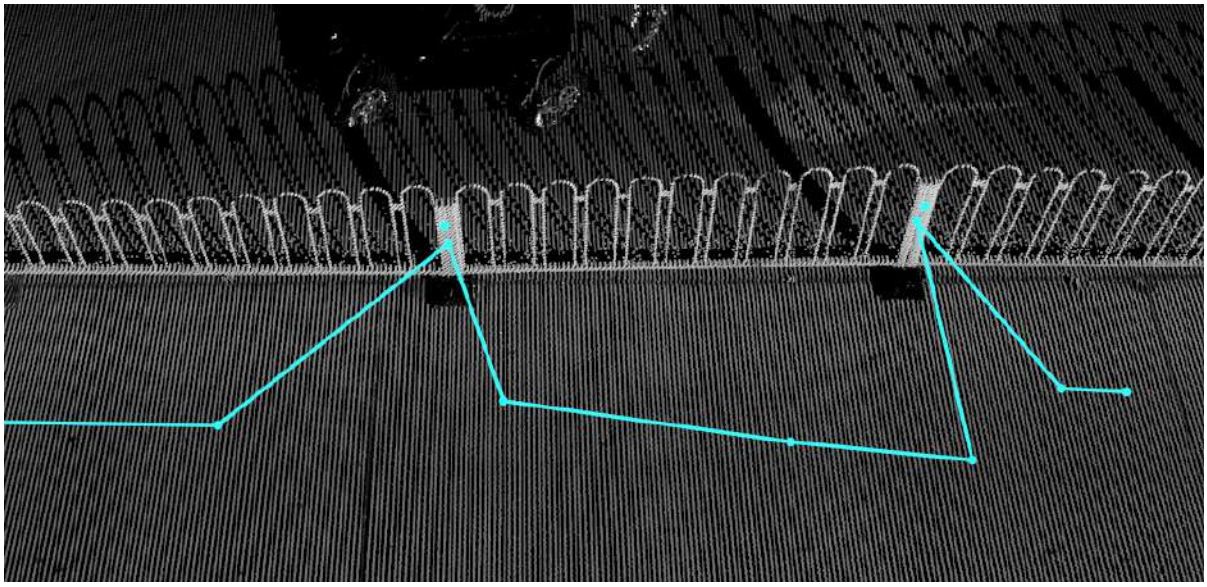
2.After the box is selected, double-click the left mouse button and the program will automatically calculate and ground the vector nodes in the range.



The effect of the node after grounding

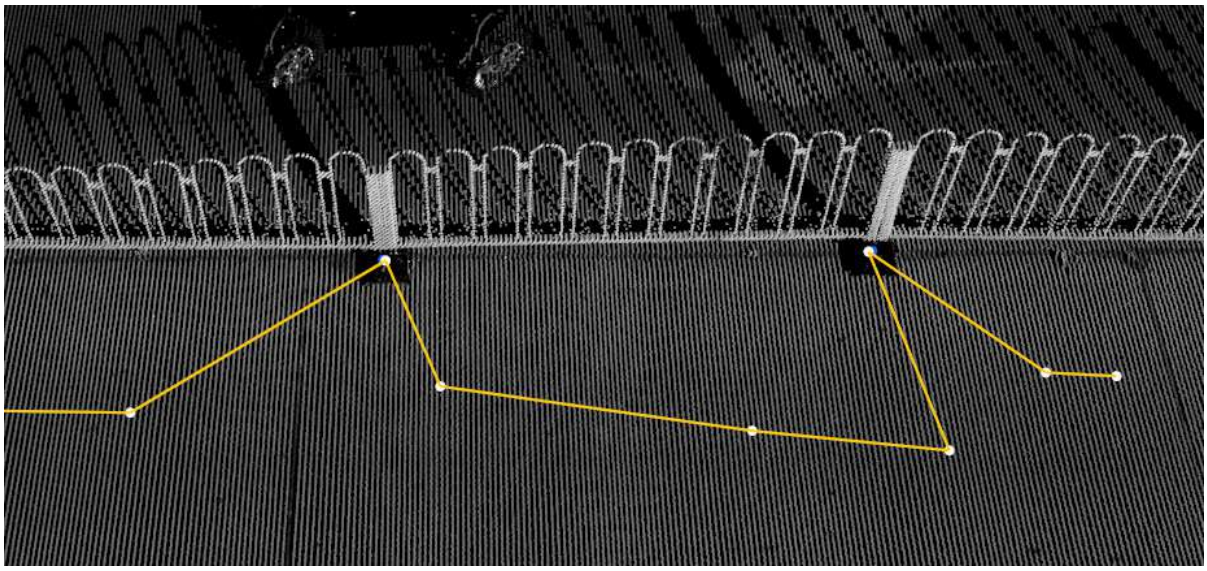
Select object method

1.Mouse click on the object to be grounded, click on the process, you can use the right mouse button, back a selection



Tap on the object to be grounded

2.Click the fit button to ground the selected object



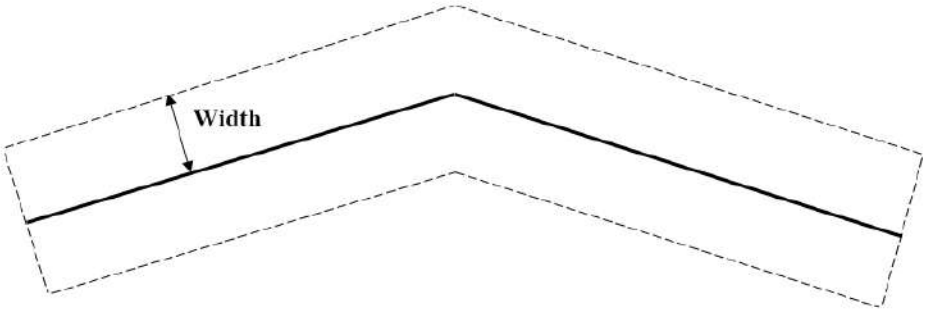
The effect of the object on the ground

Line to Polygon

Function Description: Create polygons based on the selected vector line or the selected vector line layer.

Principle

The schematic diagram is as follows, the black solid line in the middle is the reference vector line, and the outer dotted box is the generated polygon.

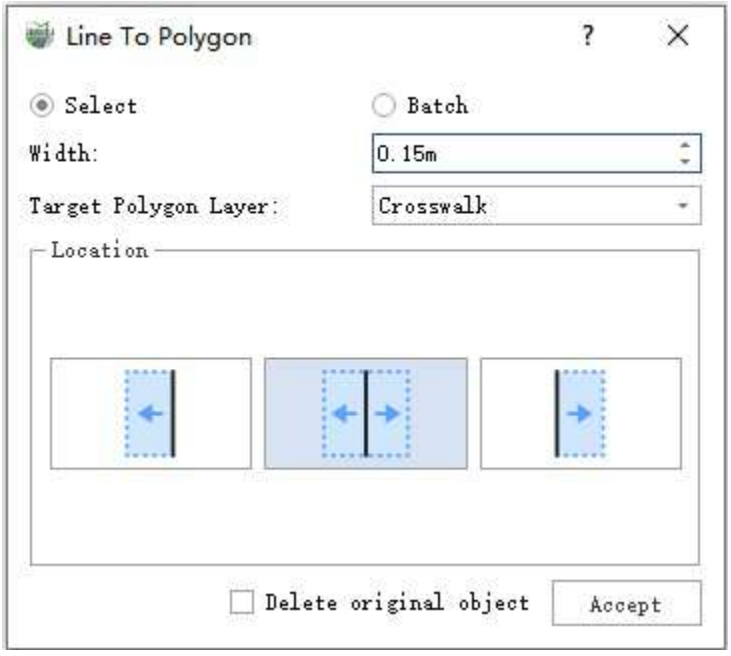


Schematic Diagram

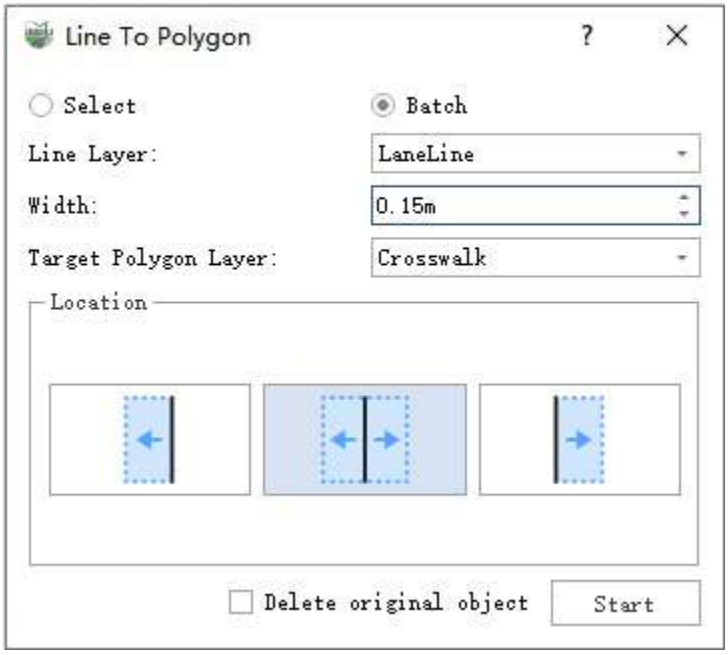
Steps

1.Click the **Line to Polygon** button in the Vector Editor.

2.The Line to **Polygon Dialog Box** pops up.



Select

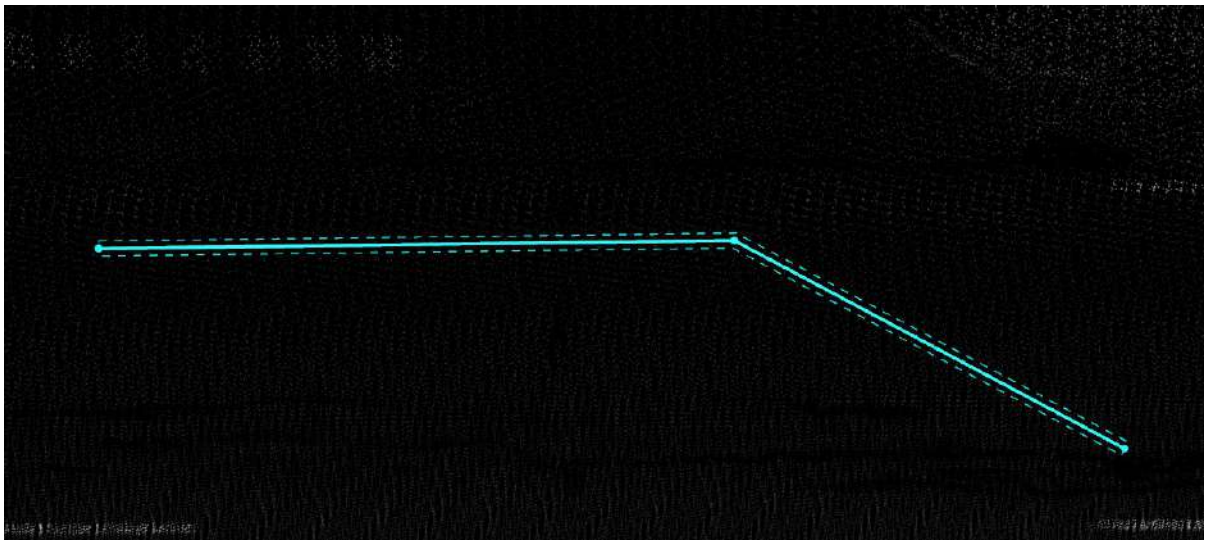


Batch Process

Parameter Description:

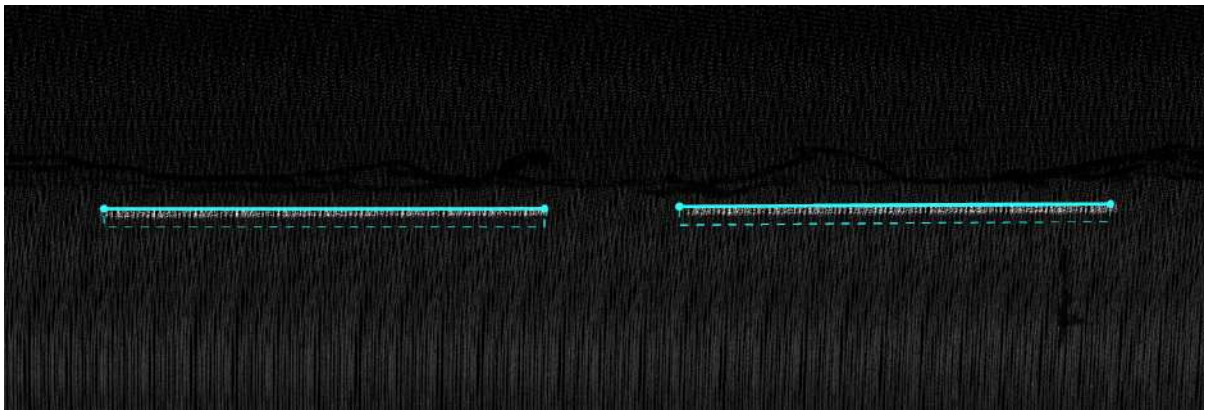
- **Select**: Single click on a vector line with the left mouse button to generate a polygon.
- **Batch**: Generate polygons for all vector line objects within the line layer.
- **Line Layer**: Only appears in batch processing mode, used to select the source layer of the vector line.
- **Width**: The width of the polygon.
- **Target Polygon Layer**: The layer in which the generated polygon will be stored.
- **Location**: Used to specify which side of the reference line the generated polygon is located on, default is middle.
    - **Left**: Generate the set width polygon on the left of the baseline.
    - **Middle**: Generate the set width polygon on both sides of the baseline, the left width is half of the width, and the right width is half of the width.
    - **Right**: Generate the set width polygon on the right of the baseline.
- **Delete original object**: Whether to delete the baseline, or default is not checked.
- **Accept/Start**: In point mode, the polygon style will be displayed in real time, and the polygon will be generated after clicking the accept button; in batch processing mode, there is no real-time display, clicking the start button will start to generate polygons in batches.

Solid Line Result



Solid Line Result

Dashed Line Result



Dashed Line Result

Polygon to Line

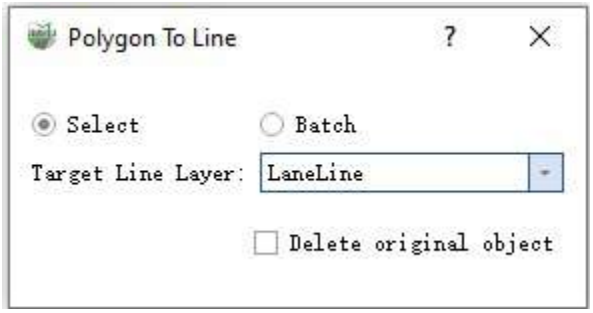
Function Description: Converts polygons into lines.

> The shape remains unchanged; only the object type is converted from polygon to line.

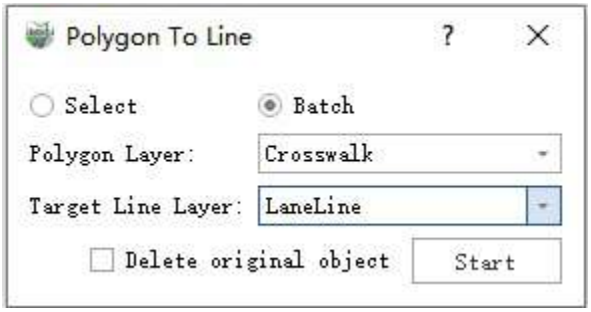
Steps

1.Click the **Polygon to Line** button in the vector editor.

2.The **Polygon to Line Dialog Box** will appear.



Selection

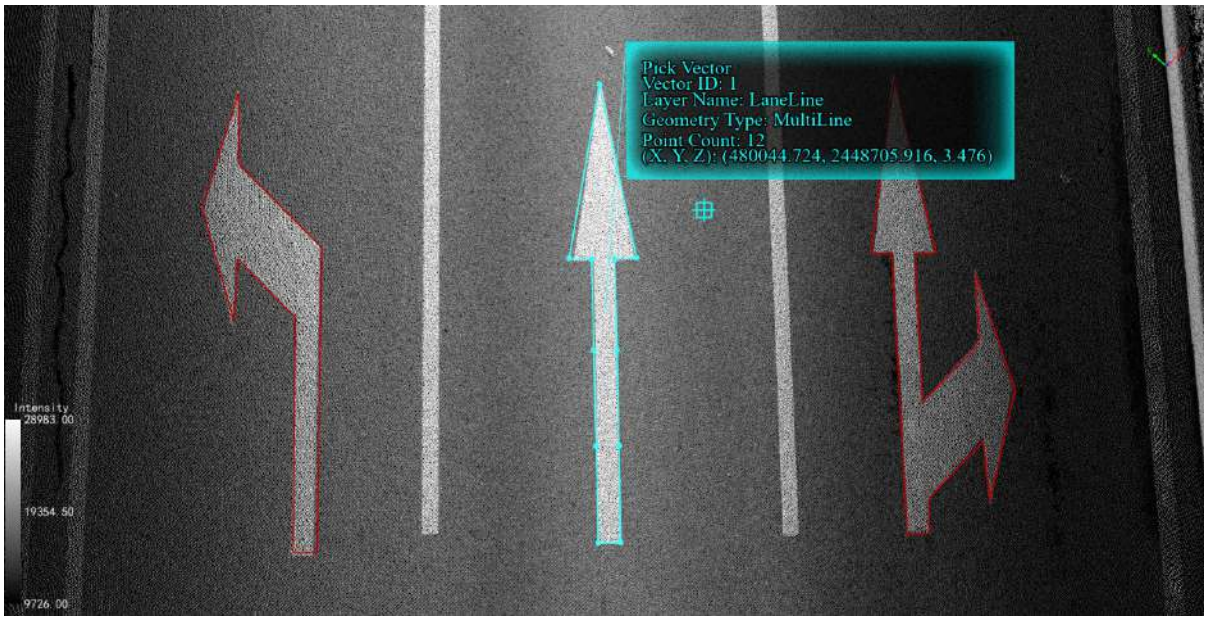


Batch

Parameter Description:

- **Selection**: Click with the left mouse button to select a polygon and convert it into a line.
- **Batch**: Converts all the polygon objects in the polygon layer.
- **Polygon Layer**: Only available in batch mode, used to select the polygon vector layer as the source.
- **Target Line Layer**: Specifies the layer to store the converted line vectors.
- **Delete Original Objects**: Whether to delete the original polygons. It is not selected by default.
- **Start**: Only available in batch mode, click to start the batch conversion.

3.Result



Solid line result

Draw By Intersection

Function Description: This function allows the user to draw a polyline or polygon with the intersection of two adjacent lines as the basic logic.

Steps

1.This function requires that all points must be on a plane, so the user will need to determine a reference plane first, refer to the locking function of begin quick horizon section:

2.Select the line layer or polygon layer, activate **Draw By Intersection** function.

3.The collection method takes 4 points as a unit, first collect 2 points to determine the reference line, collect the third point as the anchor point, and obtain the intersection point between the connection line between the fourth point and the anchor point and the reference line, at the same time, the line between the intersection point and the fourth point is taken as the current reference line and circulates in turn.

4.If it is currently a surface layer, the connection between the last 2 points of the current point sequence will intersect with the connection between the first 2 points to form a closure. If the angle is less than 15 degrees or greater than 165 degrees, it will be directly connected.

Draw By Intersection

Draw By Perpendicular

Function Description: This function allows the user to draw vertical polylines or polygons with adjacent sides according to the layer type.

Steps

1.Select the line layer or polygon layer, activate **Draw By Perpendicular** function.

2.First collect 2 points to determine the current baseline, and then add a point to determine a line vertical to the current baseline, and the newly added line will become the current baseline.

3.If the current layer is a surface layer, the last point will try to find a point on the first line, making the point perpendicular to the line connecting the last point and the first line, and closing based on this point.

Draw By Perpendicular

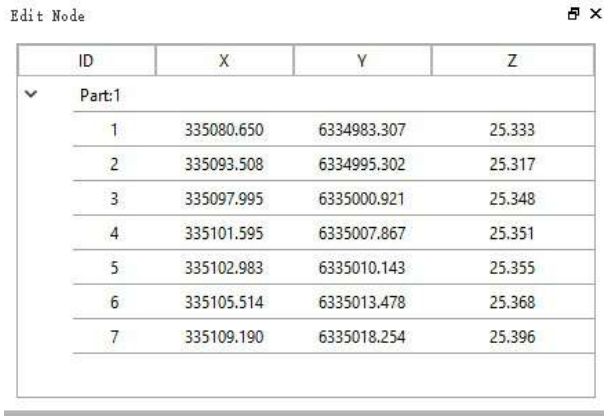
Edit Vertices

Function Description：Editing nodes in vectors supports not only optimizing adjustments to individual vector objects but also supports node optimization adjustments for multiple vector objects with collinear topological relationships.

> **Note**: After entering node editing mode, the left mouse button will only respond to vector selection, node selection, adding nodes, and similar operations. It will no longer simultaneously respond to rotating the point cloud view. If you need to rotate the point cloud view, you should press and hold the V key on the keyboard while using the left mouse button.

Steps

1.Click the **Edit Vertices** button in the node editor, Then click to select the vector object of the node to be edited. The dialog box function for vector node editing will be activated. As shown in the figure.

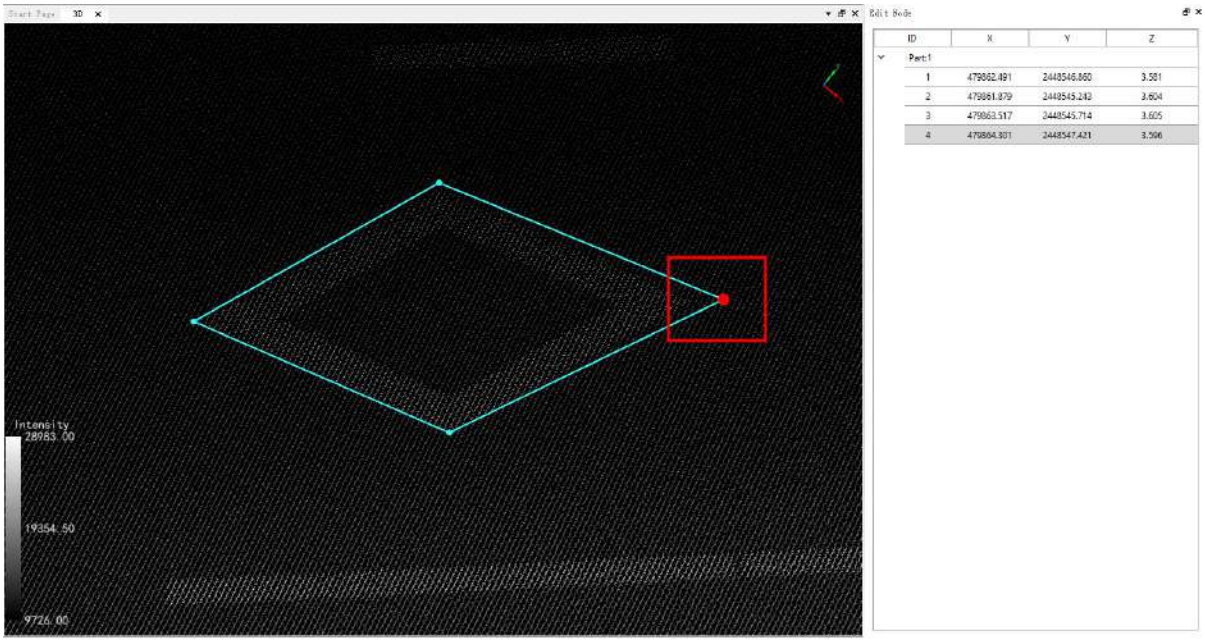


ID	X	Y	Z
Part:1			
1	335080.650	6334983.307	25.333
2	335093.508	6334995.302	25.317
3	335097.995	6335000.921	25.348
4	335101.595	6335007.867	25.351
5	335102.983	6335010.143	25.355
6	335105.514	6335013.478	25.368
7	335109.190	6335018.254	25.396

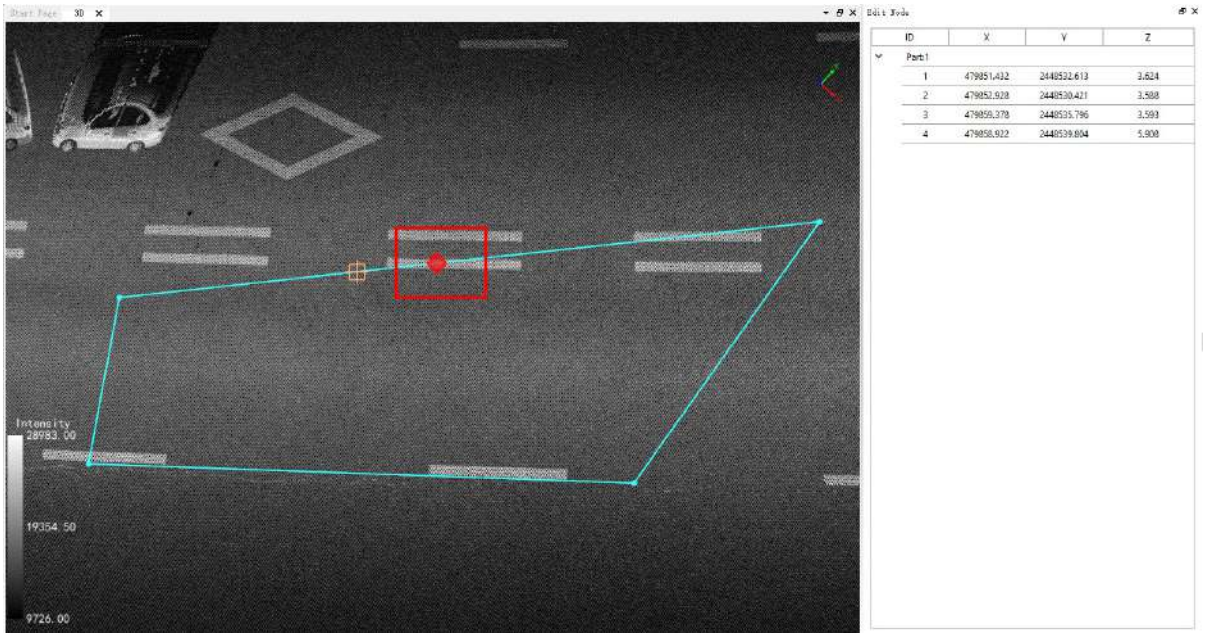
Node editing interface

2.The main functions of the node editor are as follows.

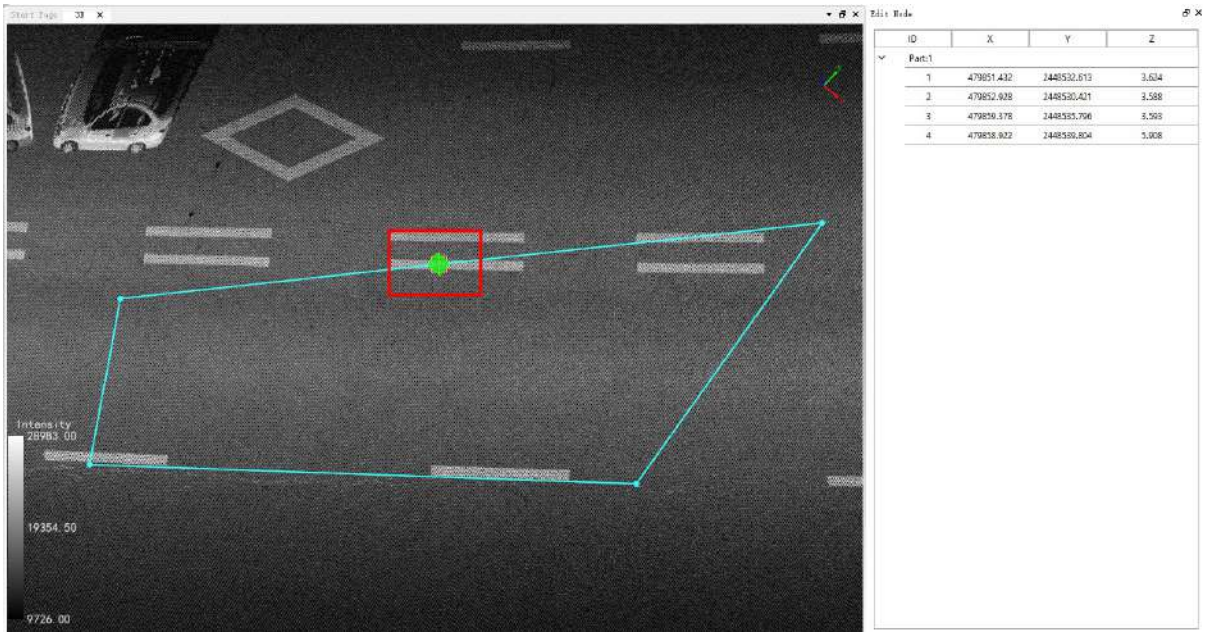
2.1 **Selected Highlight** : Mouse click on a selected node to edit one or more rows of the table will highlight the selected node of the vector in the 3D scene.



2.2 **Center Position**: Move the mouse to the line segment, and a red cube will be displayed, indicating the center position of the line segment.



- When the mouse moves to the center position, the red cube turns green, indicating that the center position of the line segment has been captured.

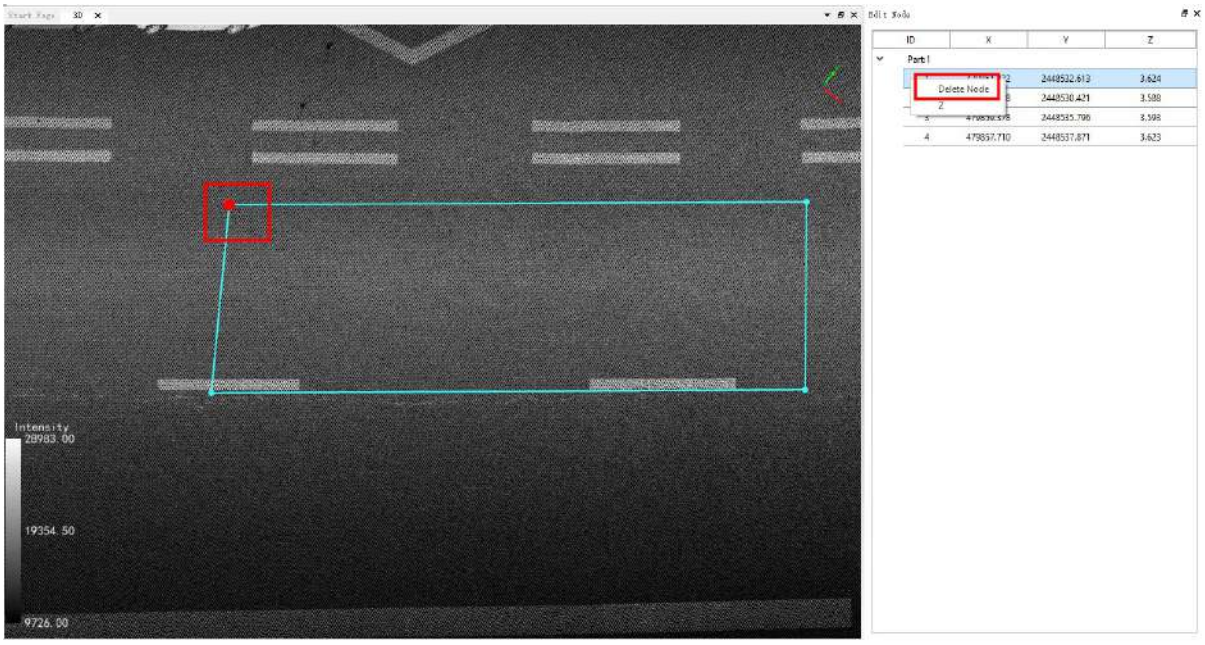


2.3 **Double-click to jump, highlight** : Double-clicking on a row of the node edit form with the mouse will switch the view of the 3D scene to that node and highlight the node.

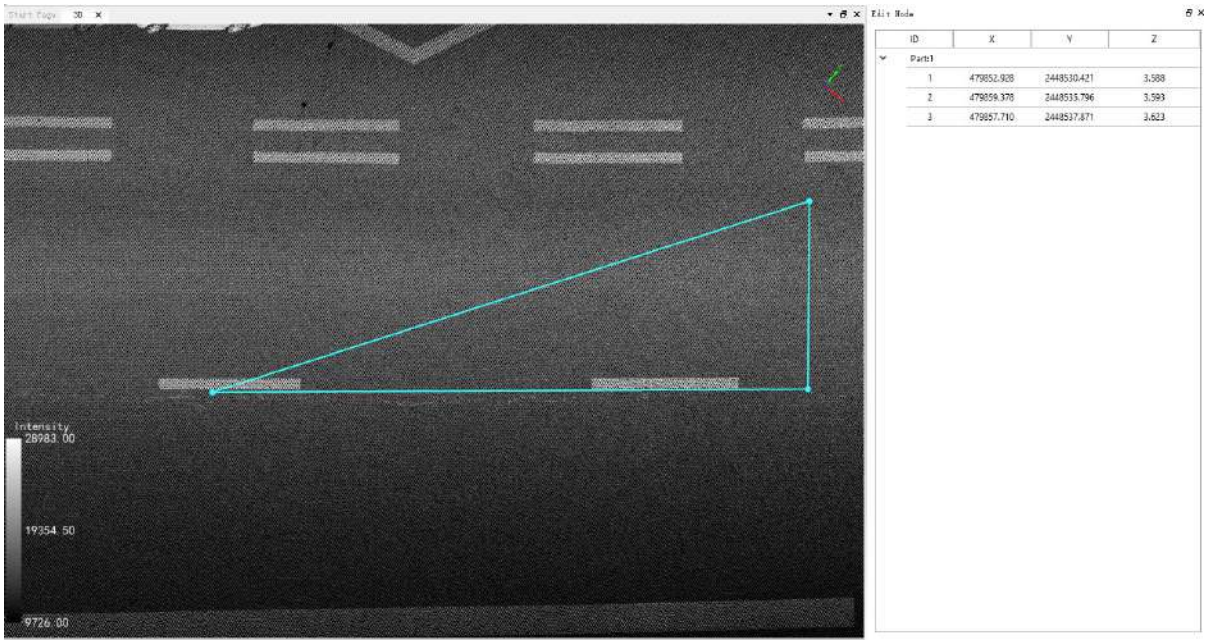
2.4 **Delete Node**: There are two ways to delete a node.

> Note: If the node to be deleted is an internal node, the nodes on both sides of the deleted node will be directly connected while the current node is deleted.

- 1.Select a row in the node edit form to delete a node. Or click on the top left cell in the node editing form, select all nodes, and delete all nodes. Take a row as an example, click the right mouse button to bring up the drop-down menu, select the function of "Delete Node", which will delete the corresponding row in the table, and at the same time will delete the selected node of the vector in the 3D scene and redraw the vector. The following figure shows.

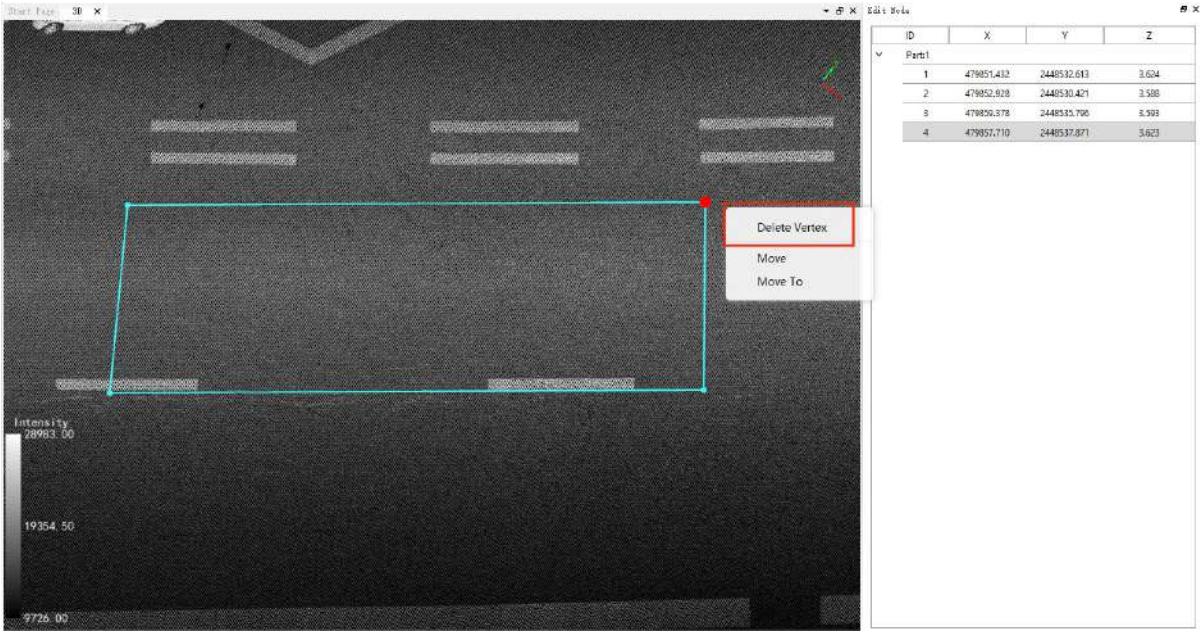


Before node deletion

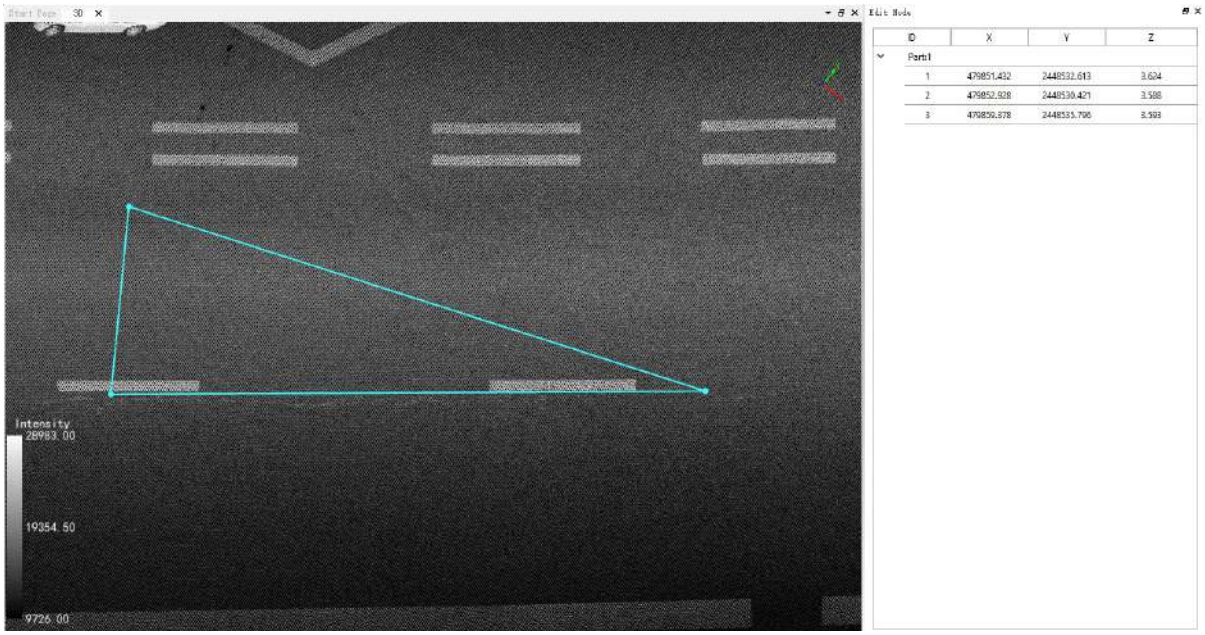


After node deletion

- 2.Move the mouse to the position of the deleted node, the cross wire turns red, indicating that the node is captured. Then click the right mouse button to pop up the drop-down menu, select the function of "Delete Node", which will delete the corresponding row in the table, and will delete the selected node of the vector in the 3D scene, and redraw the vector. The following figure shows.



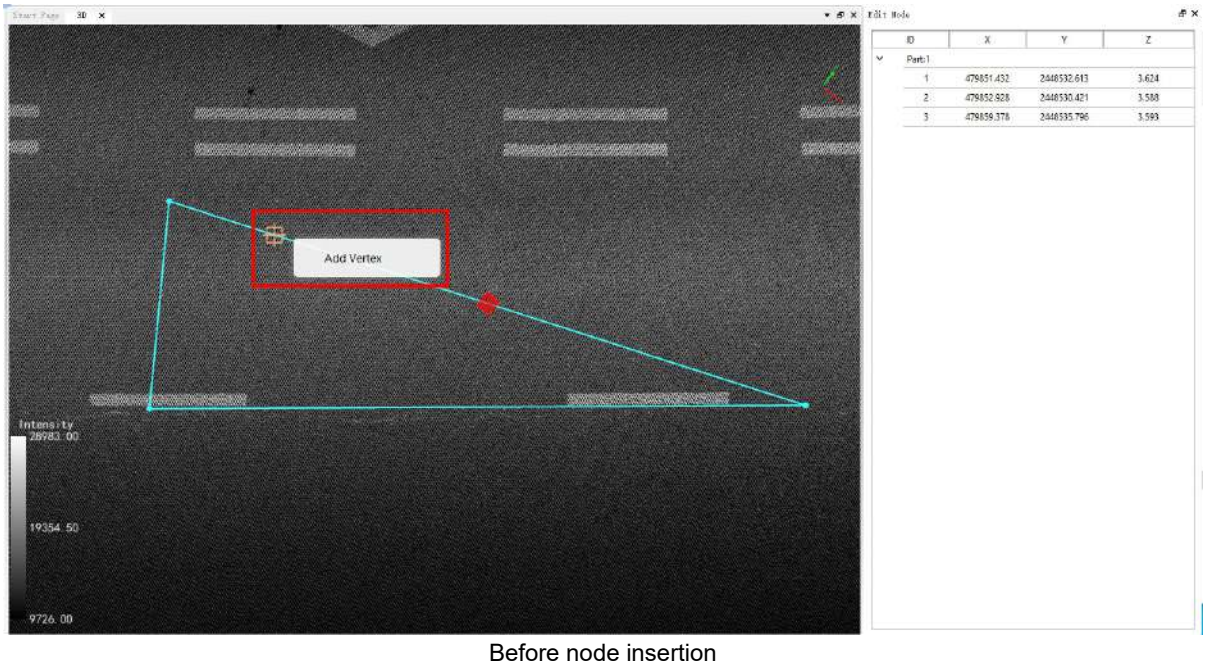
Before node deletion



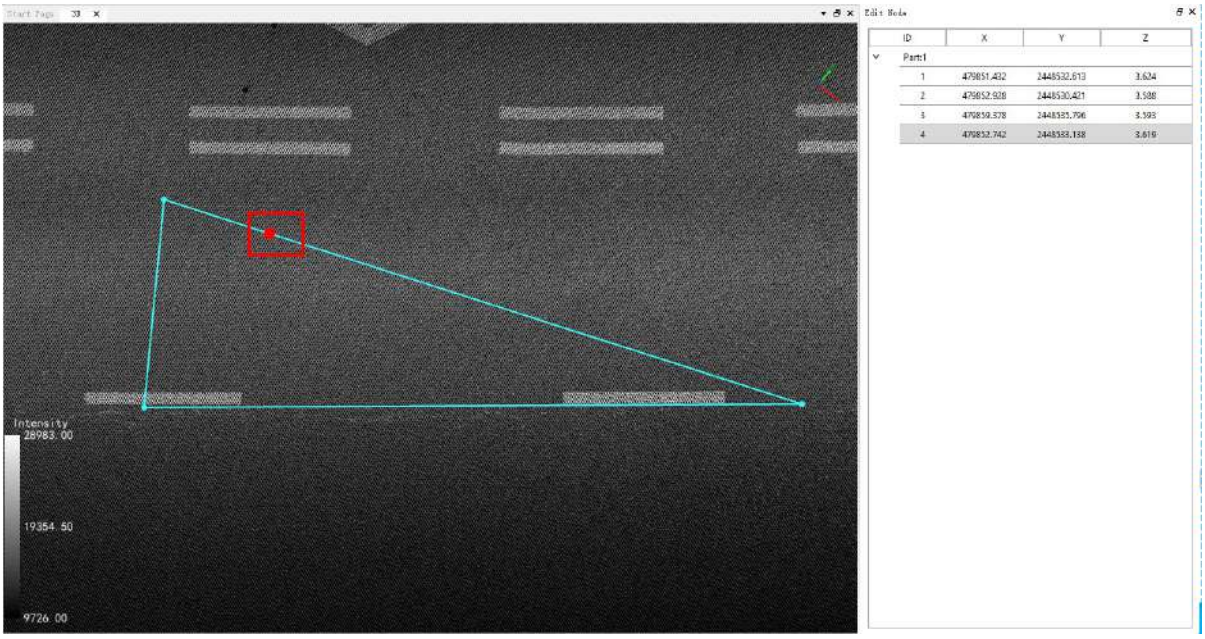
After node deletion

> Note: Keep a minimum of one node for the point layer, two nodes for the line layer, and three nodes for the surface layer.

2.5 **Add Node**: Select the location of the node you want to add in the vector object, the cross turns orange, indicating that the edge of the vector data is captured. Then click the right mouse button, pop-up drop-down menu, select the function of "Add Node", while the vector node will be added in the three-dimensional scene, and the node editing form automatically add the node information. The following figure shows.



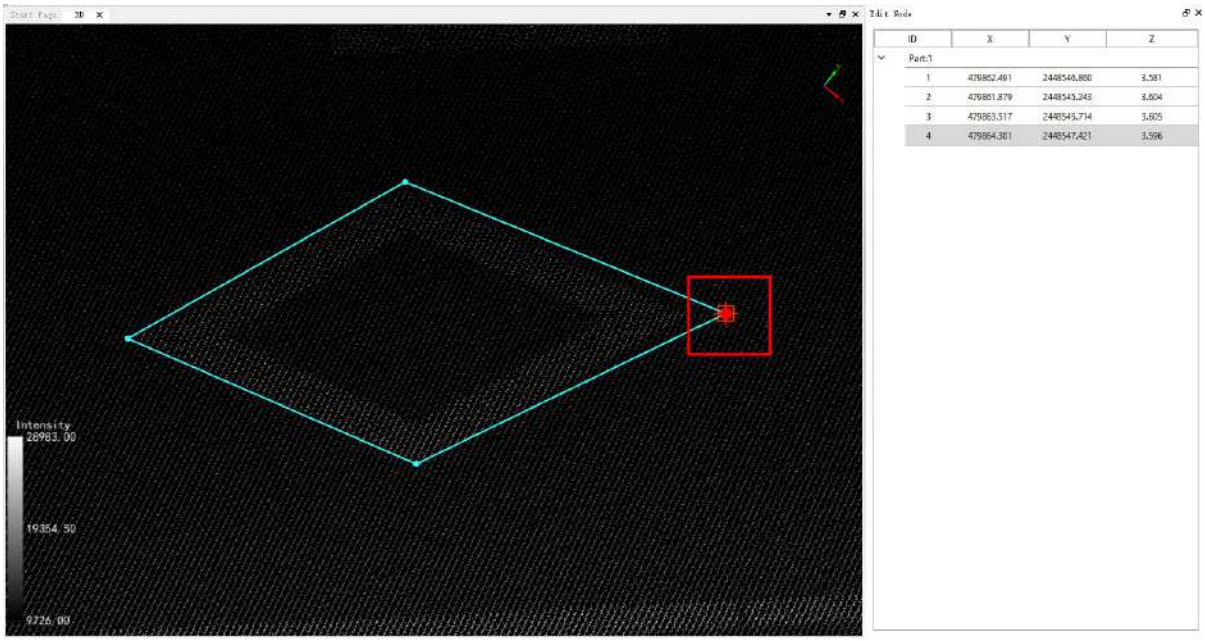
Before node insertion



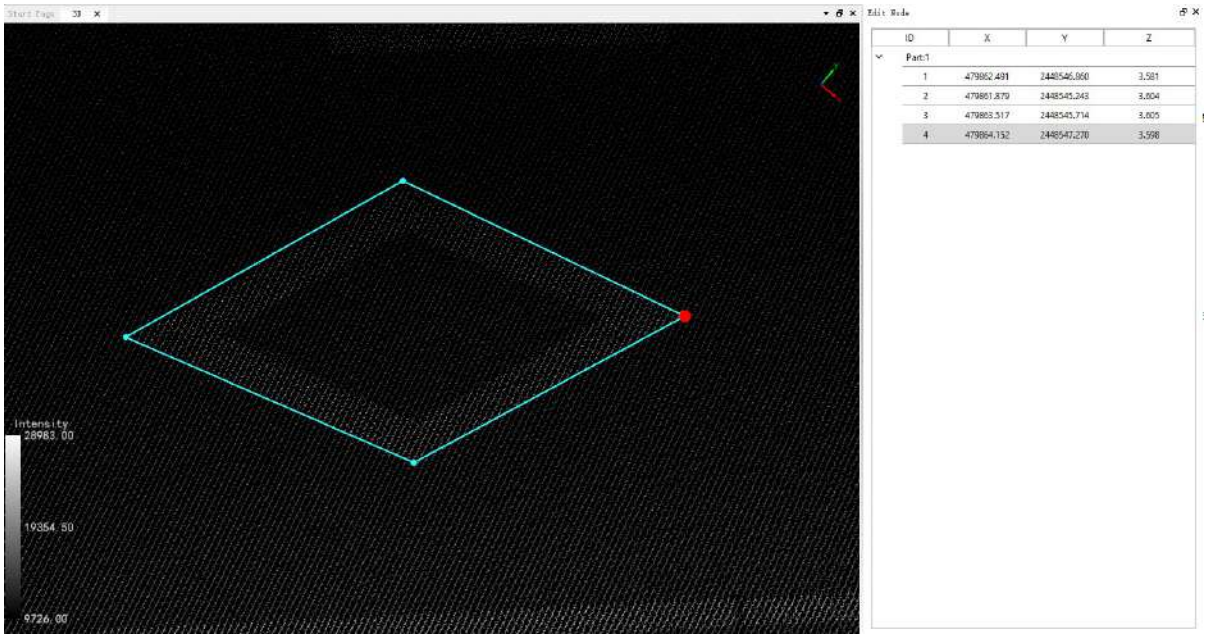
After node insertion

2.6 **Move Node**: There are two methods.

- 1.When the mouse approaches a vector node, the software automatically snaps to the nearest node, and the crosshair turns red to indicate that it has captured the node. At this point, you can press the left mouse button and drag the node to the desired position. You can also select an edge of the vector to perform a drag and move operation, as shown in the diagram below..

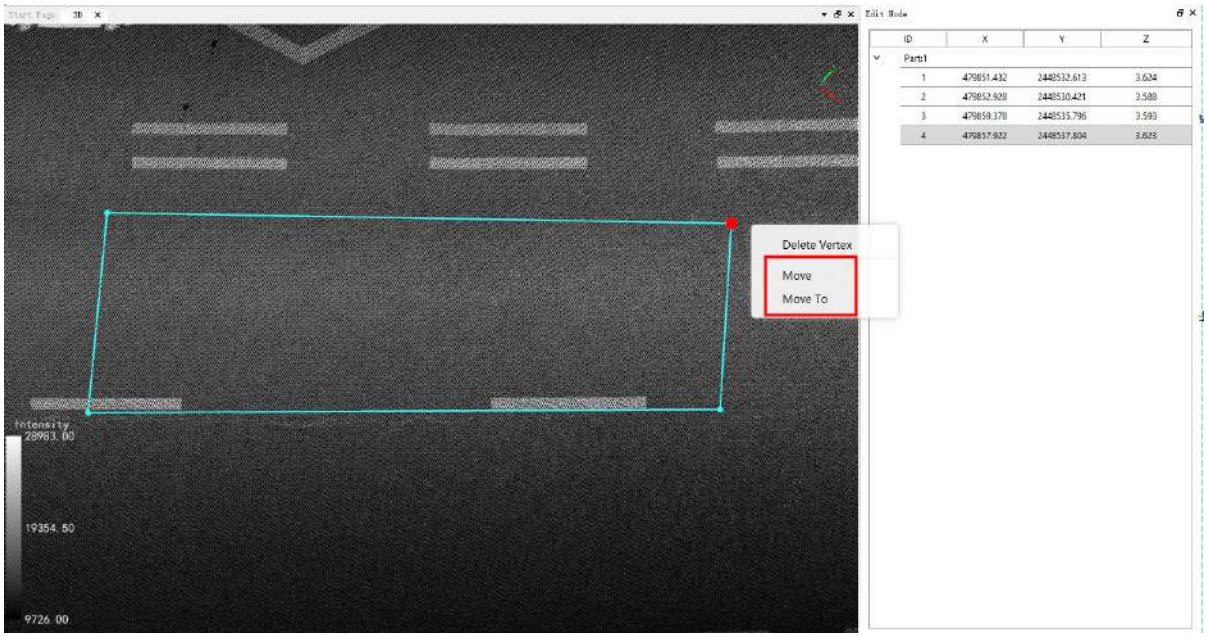


Before moving nodes

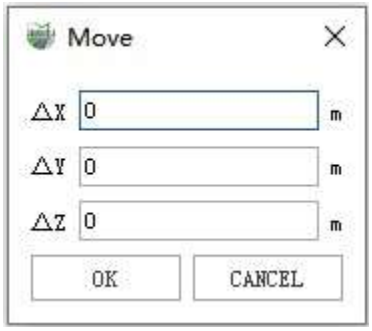


After moving nodes

- 2.When the mouse is close to the vector node, the software automatically captures the nearest node and the cross wire turns red, indicating that the node is captured, then click the right mouse button to bring up the drop-down box.

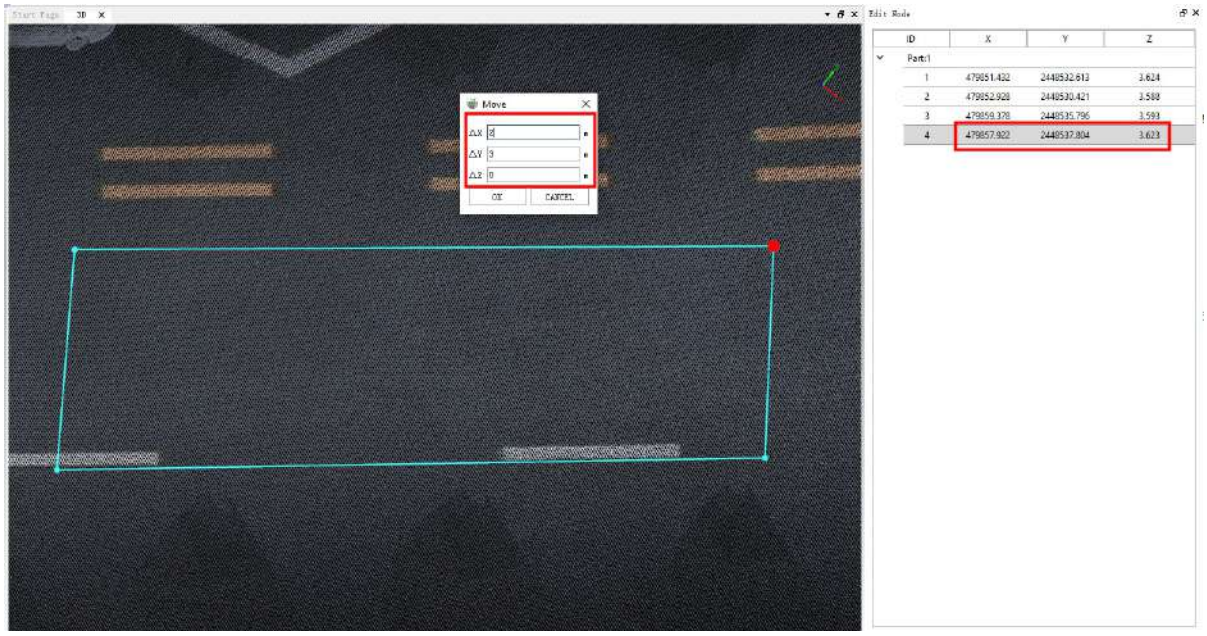


2.1 Check the "Move" function. As shown in the figure below.

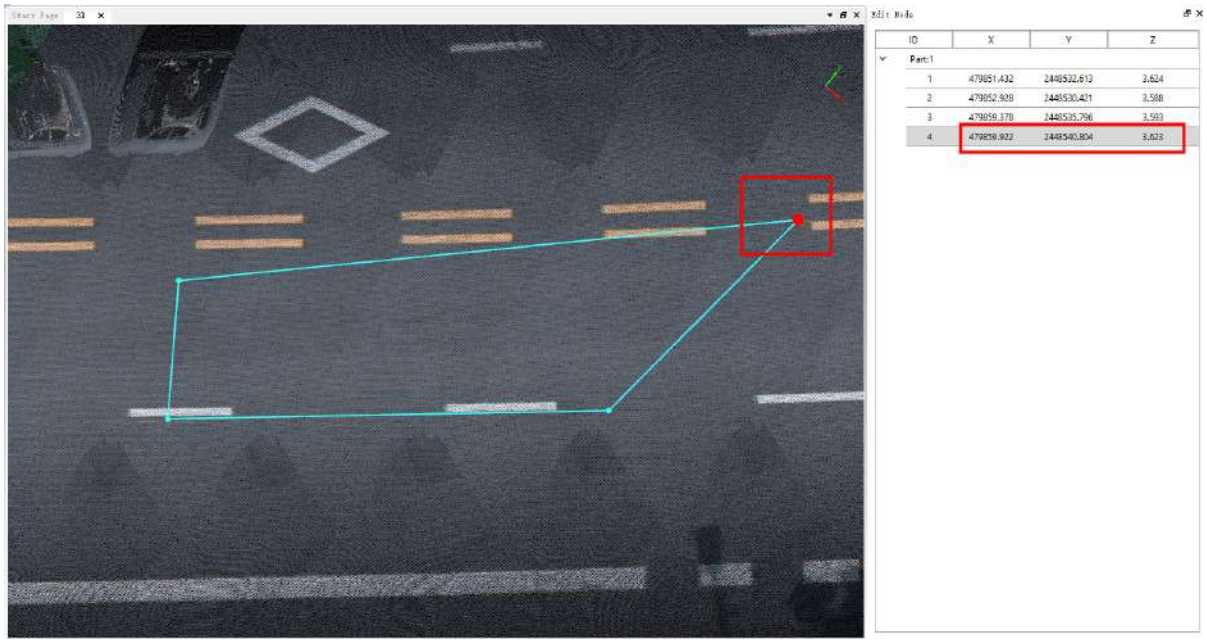


Parameter Setting

- **X**: The input value is the relative displacement of the node on the X-axis
- **Y**: The input value is the relative displacement of the node on the Y-axis
- **Z**: The input value is the relative displacement of the node on the Z-axis



Before moving nodes

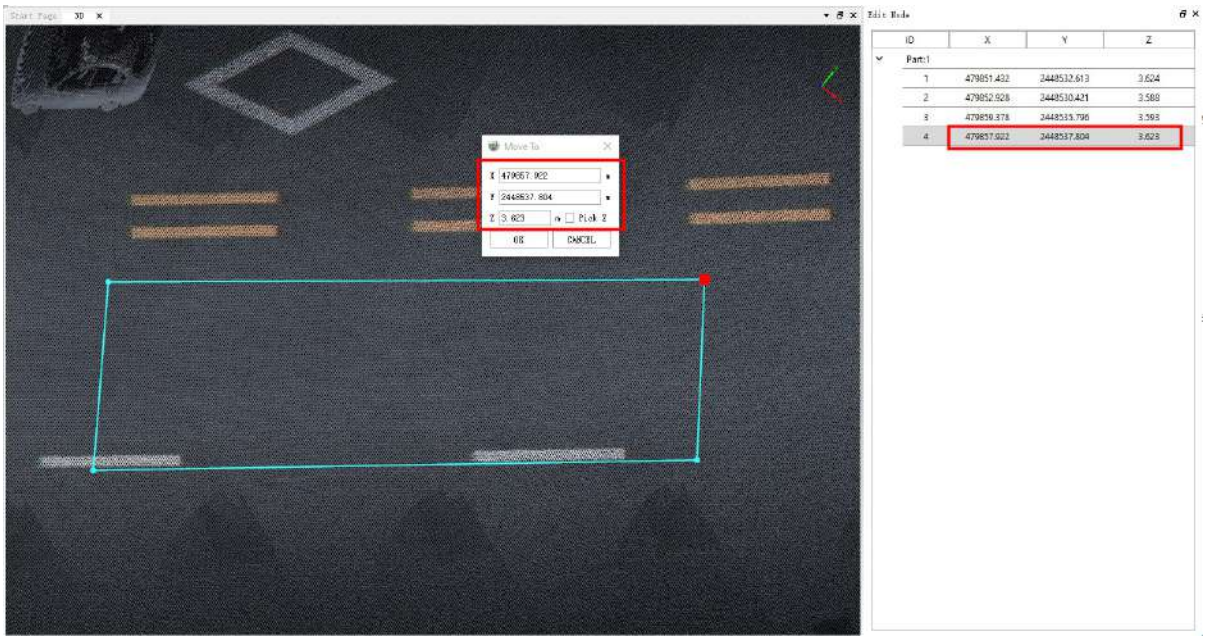


After moving the node

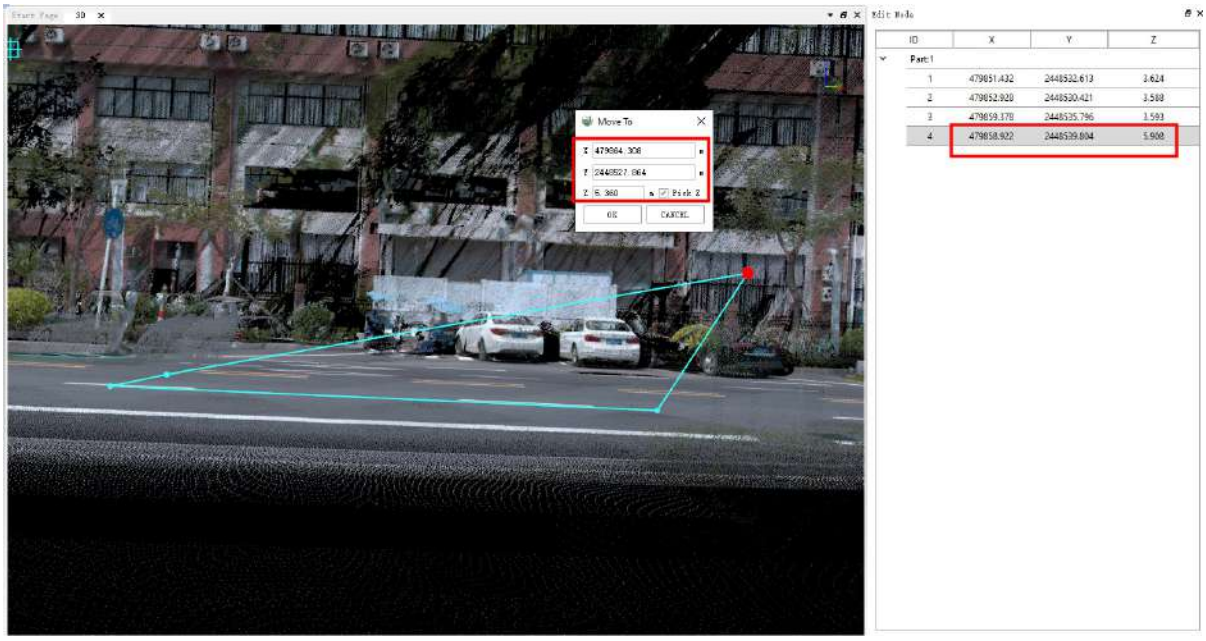
2.2 Check the "Move To" function. The following figure shows.

Parameter Setting

- **X**: The input value is the absolute position of the node on the X-axis
- **Y**: The input value is the absolute position of the node on the Y-axis
- **Z**: The input value is the absolute position of the node on the Z-axis
- **Pick Z**: With the checkbox checked, you can pick any position directly on the point cloud as a Z value

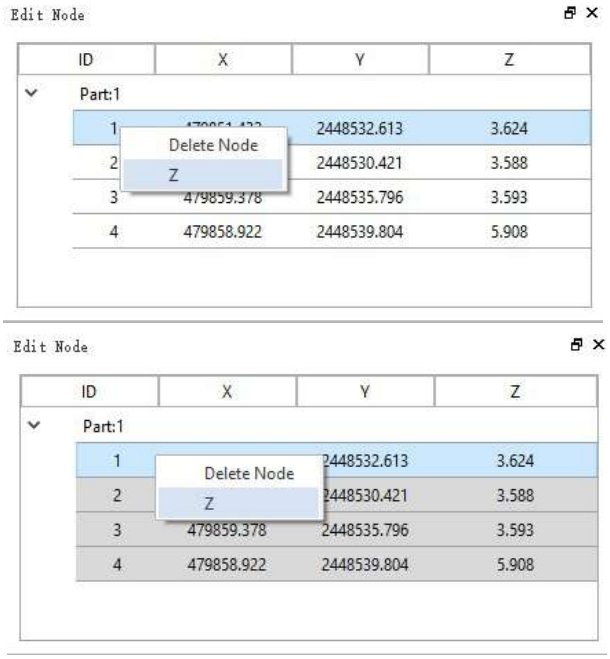


Before moving nodes



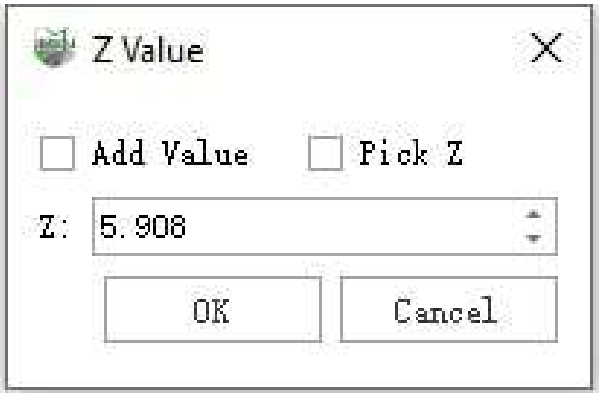
After moving nodes

2.7 **Modify Z value**: Select a row in the node editing table, then click the right mouse button to bring up the drop-down menu, select "Z" to modify a row. Or click the top left cell in the node editing table, select all nodes, and edit all nodes. Then click the right mouse button to bring up the drop-down menu, select "Z" to modify all nodes.



The dialog box of Modify Z Value will pop up:

- **Add Value**:Unchecked "Add Value" will modify the Z value of the selected node to the value in the input box and redraw the vector. If "Add Value" is checked, a new value will be added to the original Z value of the node, and the vector will be redrawn in the 3D scene.
- **Pick Z**:When add Value is unchecked, the checkbox for Select Z can be checked directly, and any position on the point cloud can then be picked as a Z value



3.Shortcut:

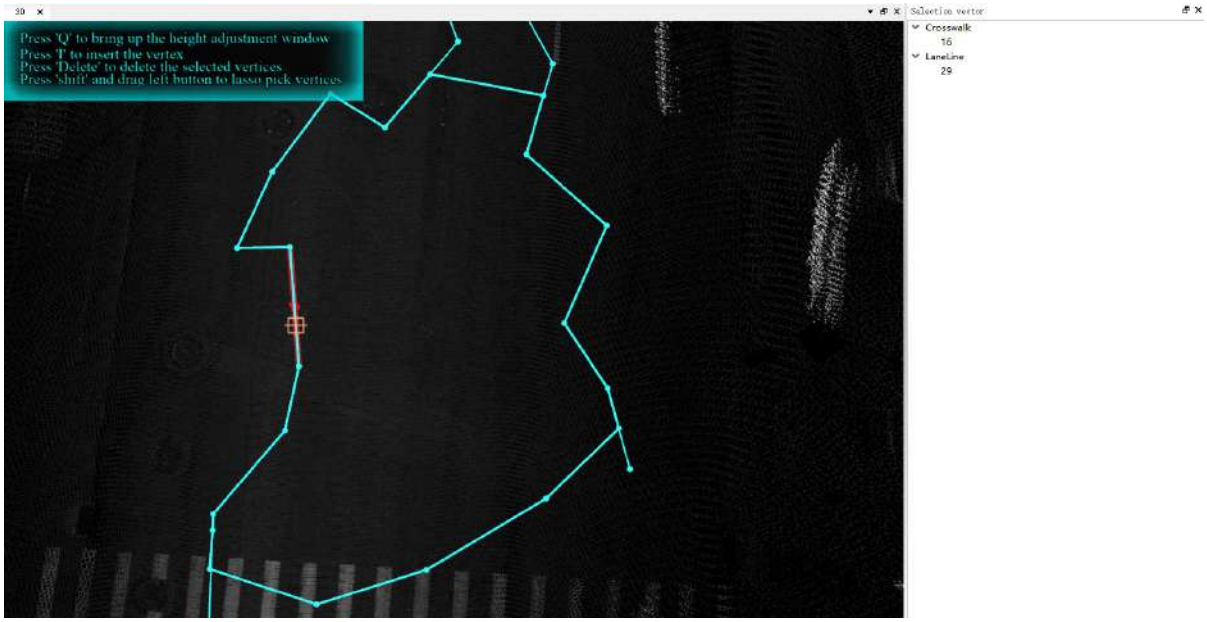
Function	Shortcut	Description
Z Value	Q/q	While adjusting vector nodes, you can use the Q key to bring up the elevation adjustment window and adjust the elevation value of the nodes.
Insert the Vertex	I/i	While editing vector nodes, you can use the I key to add a node at the desired location.
Delete the Vertices	Delete	While editing vector nodes, you can use the Delete key to delete unwanted nodes.
Lasso Pick Vertices	Shift + drag the left mouse button	While editing vector nodes, you can use Shift + drag the left mouse button to lasso-select the nodes you want to edit.
Rotate Point Clound	V/v + drag the left mouse button	While editing vector nodes, you can use V/v + drag the left mouse button to move the point cloud.

Vector editing with topological edge structure.

In node editing, if you check "With Topology," you can modify overlapping vector edges and nodes. When you move, add, or delete shared nodes, it will simultaneously apply to the nodes or edges shared by line features or polygon features.

1.Click the **Node Editing** Button, check the "With Topology" checkbox, and then use the mouse to click on the vector object you want to edit. You will see that if the vector has topological edges, when the mouse captures an edge, that edge will be highlighted in red, indicating that topological edge editing can be performed, as shown in the image.

At this point, when you click on the topological edge you want to edit, a page will pop up on the right side, listing the multiple layers where the current edge is located.



2.The main functions of topological edge and node editing, which affect multiple layers where the currently selected edge resides, are as follows:

(1)**Edge Movement**: You can directly drag the currently selected edge to the desired position with the mouse to achieve movement.

(2)**Delete Node**: Deleting nodes is the same as normal node editing. You can select a node and then right-click to delete it, or you can use the Delete key.

(3)**Add Node**: Adding nodes is similar to normal node editing. You can right-click to add a node or use the shortcut key "I" to add one. When a red box appears, indicating the center of the current edge, you can capture it and click to add a node.

(4)**Move Node**: Moving nodes is the same as normal node editing. You can directly drag the node to move it, or when you capture a node, you can right-click and use "Move" or "Move To" to perform the movement.

Compared to regular node editing, node editing on shared edges cannot be performed in the node editing list, and you cannot use the shortcut key "Q" to bring up the modification dialog. All other operations are the same.

Auxiliary Line

- Add H/V Lines
- Add Gridlines
- Add Lines
- Pick Lines

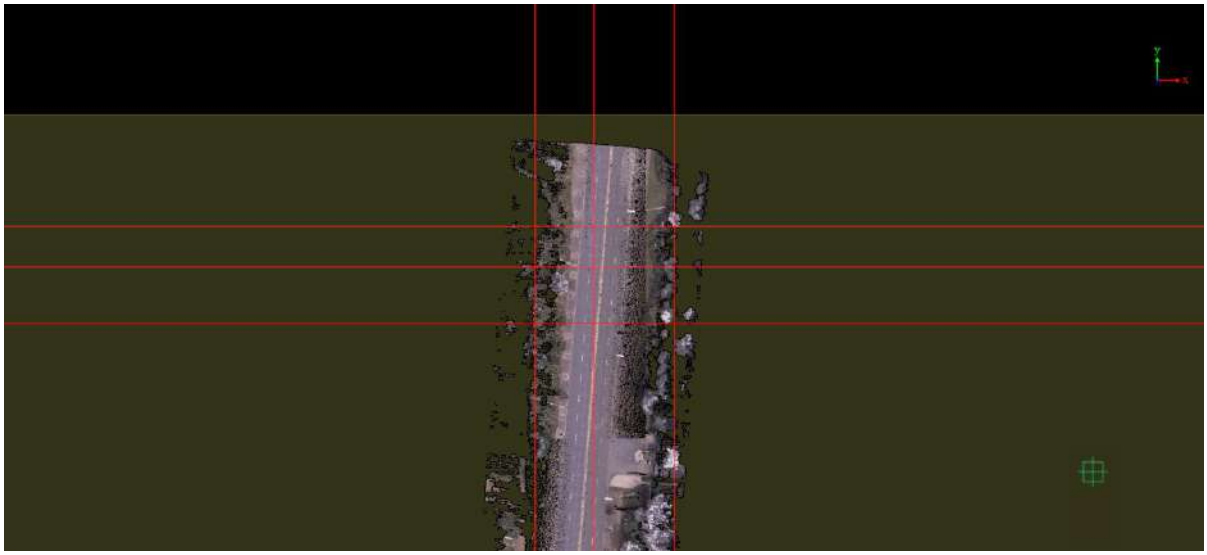
Add Horizontal or Vertical Auxiliary Line

Function description: Horizontal or vertical auxiliary lines (when moving the vector, the object will be attracted to the auxiliary line and move along the direction of the auxiliary line, ensuring that the object is arranged accurately and neatly).

Steps

1.Before entering this function, you need to switch the 3D window to 2D mode and set the reference plane.

2.Click on the **Add H/V Lines** button in the Vector Editor, then use the left mouse button to click on the position where you need to draw the horizontal auxiliary line to generate the horizontal auxiliary line. Holding Ctrl and clicking the left mouse button will generate a vertical auxiliary line. As shown below:



Drawing horizontal or vertical auxiliary lines

> Note：In the process of moving and arranging vectors using auxiliary lines, you can set the buffer distance for the object to be attracted by the auxiliary line and consider the elevation Z value when calculating the buffer distance. As shown below:

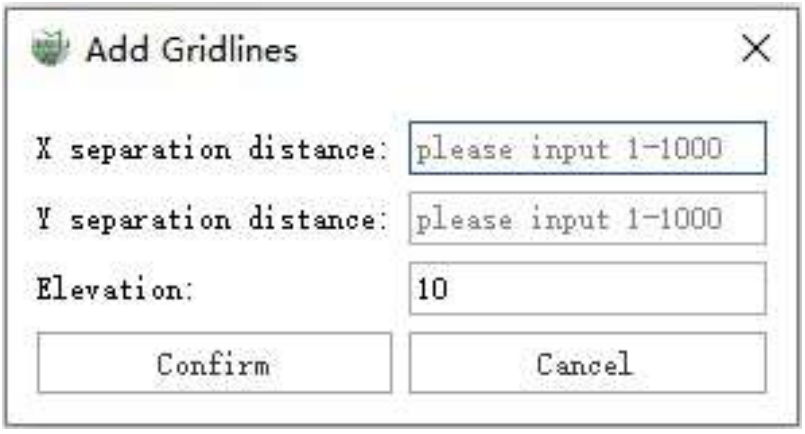
Setting the Snap Buffer Value

Add Gridlines

Functionality: Generate grid lines using the point cloud bounding box, specified XY interval distance, and elevation. Objects can snap to these lines and move along their direction, ensuring the precise and neat arrangement of objects.

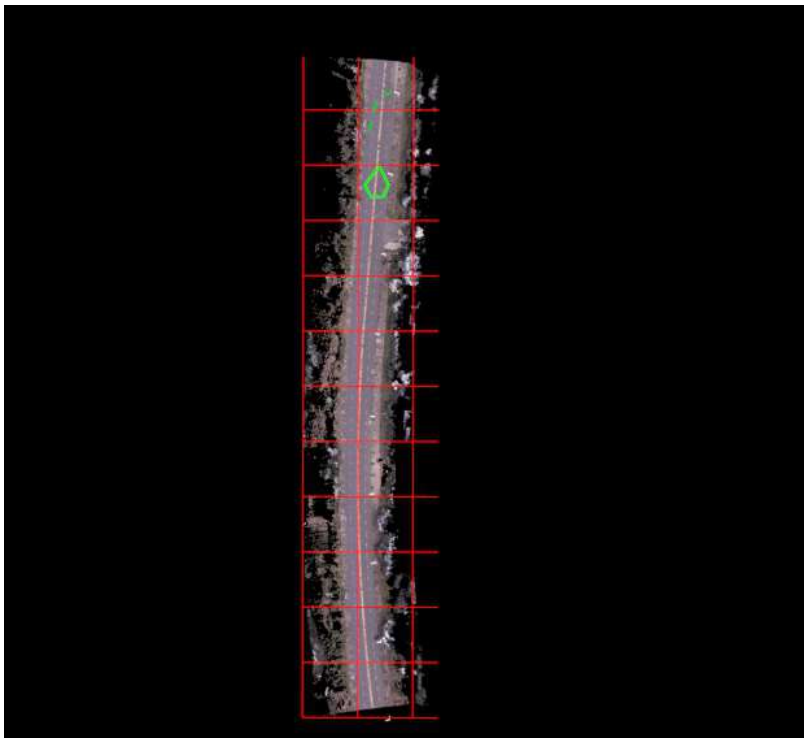
Steps

1.Click the **Add Gridlines** button in the Vector Editing toolbar. This will open a dialog box for parameter settings. See the illustration below:



Parameter Settings

2.Set the interval distance between every two grid lines in the X and Y directions, as well as the elevation value for the lines. Click "OK" to generate the grid lines. See the illustration below:



Add Grid Lines

> Note:While moving and arranging vectors using grid lines, you can set the snap buffer distance for objects to be snapped to the lines and consider the Z value (elevation) when calculating the buffer distance. See the illustration below:

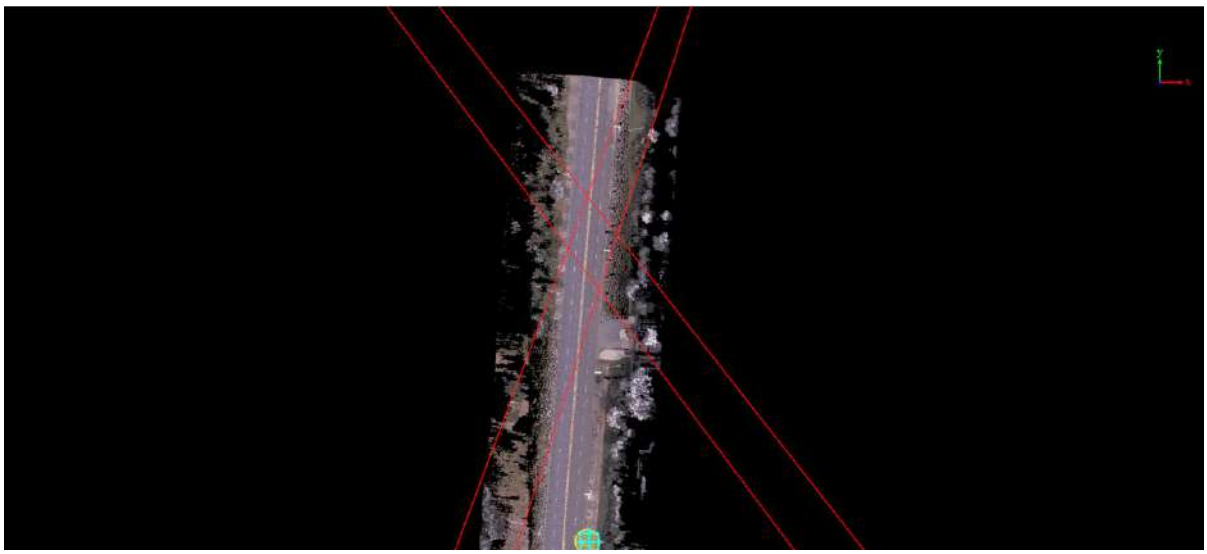
Set Snap Buffer Value

Add Lines

Functionality: Add auxiliary lines that allow objects to snap to them and move along their direction, ensuring the precise and neat arrangement of objects.

Steps

1.Click the **Add Lines** button in the **Vector Editing** toolbar. Use the left mouse button to click two points and draw an auxiliary line in any direction. See the illustration below:



Draw Auxiliary Lines

> Note: While moving and arranging vectors using auxiliary lines, you can set the snap buffer distance for objects to be snapped to the lines and onsider the Z value (elevation) when calculating the buffer distance. See the illustration below:

Set Snap Buffer Value

Pick Lines

Functionality: Using a segment of a line object or an edge of a polygon object to generate auxiliary lines (objects will be snapped to the auxiliary lines during vector manipulation, ensuring precise and orderly arrangement).

Steps

1.Click on the **Pick Lines** button in the Vector Editor. Use the left mouse button to click on a segment of a line object or an edge of a polygon object to generate an auxiliary line. See the example below:

Selecting Auxiliary Lines

> Note:
>
> - During vector manipulation using auxiliary lines, you can set the buffer distance for snapping objects to the auxiliary lines and consider the Z value in the buffer distance calculation. See the example below:

Setting Snap Buffer

Quality Check

Function Description: Provides quality checks for both attributes and geometry. Attribute checks are supported for both vector and table layers, while geometry checks are only supported for vector layers.

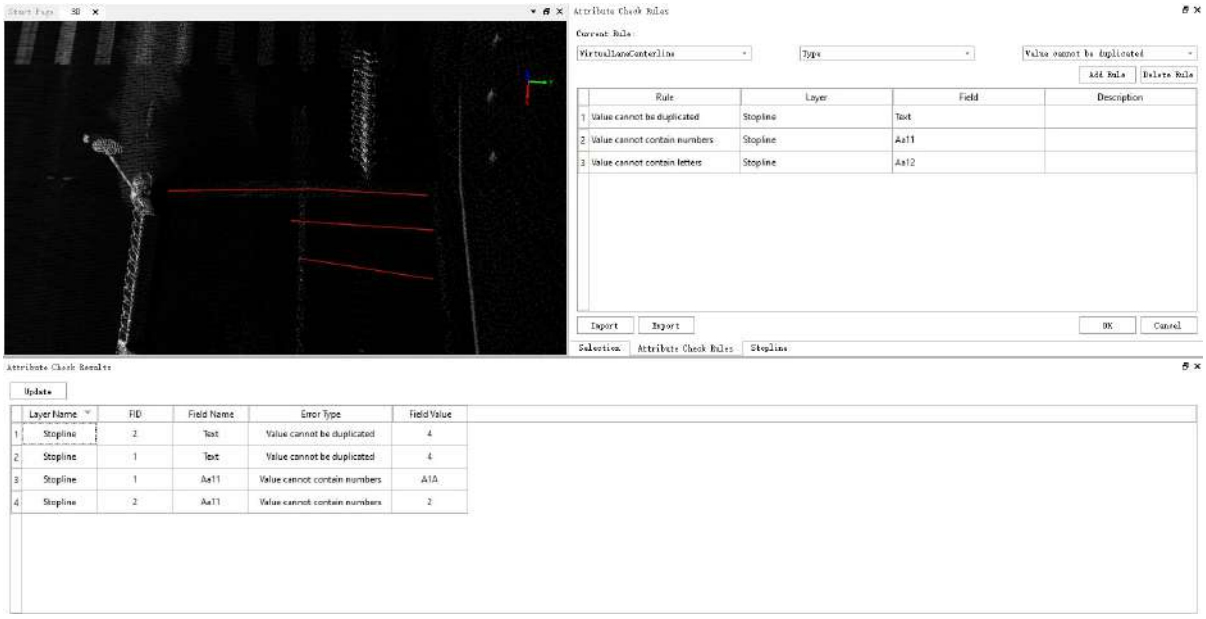
- Attribute Check
- Geometry Quality Check

Attribute Quality Check

Function Description：Check if the specified field in the selected layer meets the defined criteria.

Steps

1.Click the "Attribute Quality Control" button in the toolbar to open the Attribute Quality Control dialog box, as shown in the following image:



2.In the rules window, you can establish a series of inspection rules.

（1）In the current rule set, the first dropdown list represents selecting a layer that you want to check.

（2）Once the layer is selected, the second dropdown list will automatically display the fields associated with the chosen layer. You can then select a field based on your quality control requirements.

（3）The third dropdown list represents quality control rules, and it will provide recommendations based on the field type. These rules may include the following:

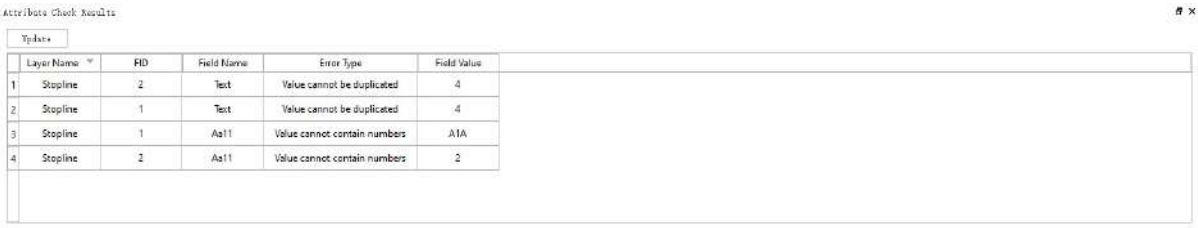
- a. **Value Should Not Start with 0**: This condition is only displayed when you select a text-type field. It is used to check if the content in the field starts with the digit 0.

- b. **Value Should Not Be Duplicated**: This rule applies to fields of any data type and checks if there are duplicate values in the field.
- c. **Value Should Not Exceed Range**: This rule is displayed when you select a numeric field, such as integer or float. It checks if the content in the field falls within the specified range. You can input a maximum and minimum value as the corresponding range.
- d. **Value Should Not Be Empty**: This rule applies to fields of any data type and checks if there are any empty or null values.
- e. **Value Should Not Contain Numbers**: This rule is for text-type fields and checks if the content contains Arabic numerals.
- f. **Value Should Not Contain Letters**: This rule is for text-type fields and checks if the content contains English letters.
- g. **Custom Rule**: Custom rules allow you to perform quality control checks on the relationship between the values filled in two fields of any layer. Currently, you can check conditions like "When Field A = XX, Field B = XX."

After the rule is set, click the Add Rule button to add the rule to the list below, and you can set multiple rules to add

3.If a rule is not needed or set incorrectly, you can delete the rule after adding it by selecting the rule and clicking the "Delete Rule" button.

4.After setting all the rules, clicking the "OK" button will initiate the automatic inspection process. The results of items that do not meet the specified conditions will be listed in the "Attribute Quality Check Results" section below for your review.



- a. In the Attribute Quality Check Results table, double-clicking any row will allow you to navigate to the current geometric object associated with that row.
- b. In the table, for the field values that do not meet the specified conditions, you can directly edit the values. After making changes, the font will turn purple. You can then synchronize these changes to the attribute table by clicking the "Update" button.

5.The quality control conditions can be exported using the "Export" button. After exporting, you can send the file to others, and they can import it using the "Import" button to directly use the conditions. The exported file will typically be in an encrypted format to maintain security and preserve the integrity of the conditions during transfer.

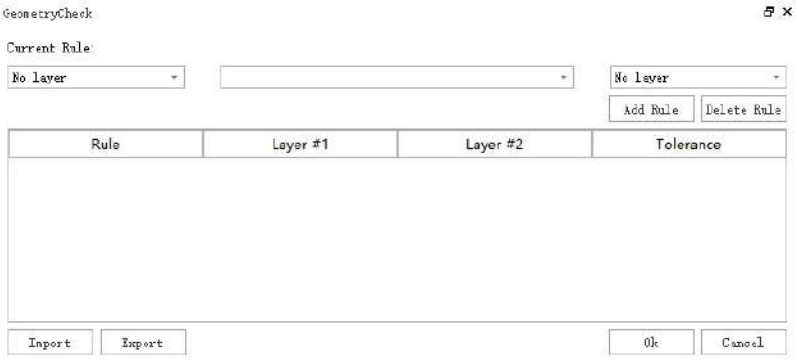
- Note: Quality control rule import will match with the current project layers based on their IDs. If the project template used to define the quality control rules is different from the project template used to import the quality control rules, the IDs of the layers with the same name may not necessarily match. This could result in import failures. Therefore, the sharing of quality control rules is limited to projects with the same template.

Geometry Quality Check

Function Description: This feature checks the vector output globally based on the selected quality check rules.

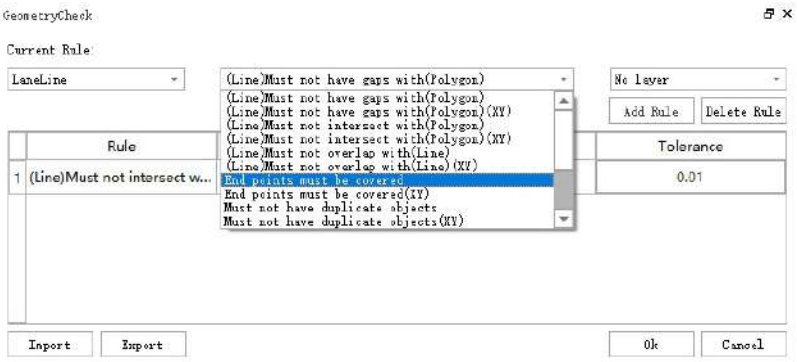
Steps

1.Click on the Geometry Quality Check button in the quick access toolbar. The geometry quality check rules setting interface will dock on the right side of the software, as shown in the figure:



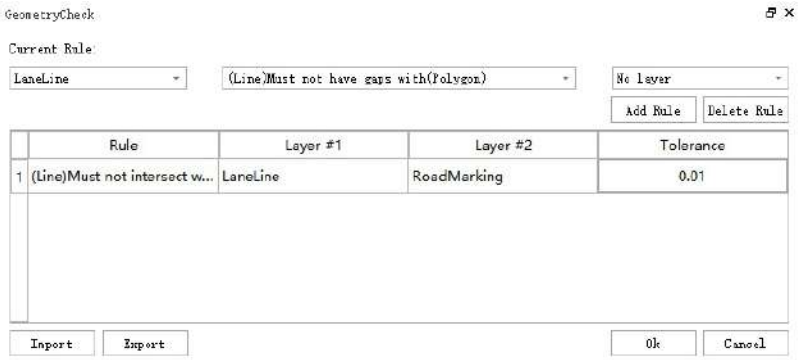
Geometry quality check rule settings interface

2.Select the layer and the quality check rules you need. Once a layer is selected, the program will automatically adapt the available quality check rules based on the geometry type of the selected layer for selection. After the quality check rules are selected, the program will filter out the available second layers for selection based on the first layer and quality check rules. As shown in the figure:



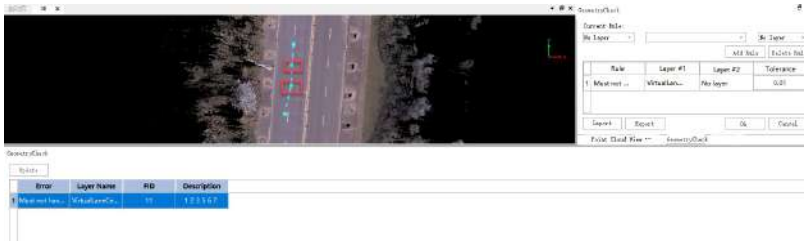
Select the layer and quality check rules

3.After the selection, click Add Rule, and the layer information and rules will be added to the to-be-checked rule list. This function supports checking multiple rules at once. If there is an error in the selection, you can select a row in the rule list and click Delete Rule to delete it. Click the tolerance value in the list to set the quality check tolerance. As shown in the figure:



Rule list display after selection

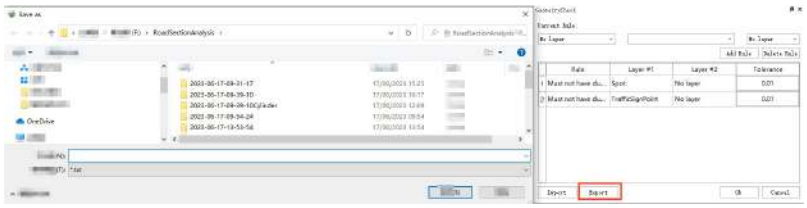
4.After the rules are selected, click OK. The function starts to check all the vector outputs of the selected layers. If there are many vectors, it may take some time, please wait patiently. After the check is completed, the check result list will dock at the bottom of the software interface, as shown in the figure:



Display of results after quality check

5.The result list will display the error information of the problematic geometry object, the layer it belongs to, the Fid of the object, and some check rules also have descriptions. For example, when checking whether the line object has duplicate nodes, the sequence number of the duplicate nodes will be displayed in the description. Double-click a row in the result list, and the 3D window can directly jump to the location of the object.

6.Click the Export button in the quality check rule settings interface, and a path selection window will pop up. You can save the multiple quality check rules set this time as a txt format file, for reuse next time to avoid setting again and wasting time. As shown in the figure:



Export quality check rules

Save quality check rules as a file

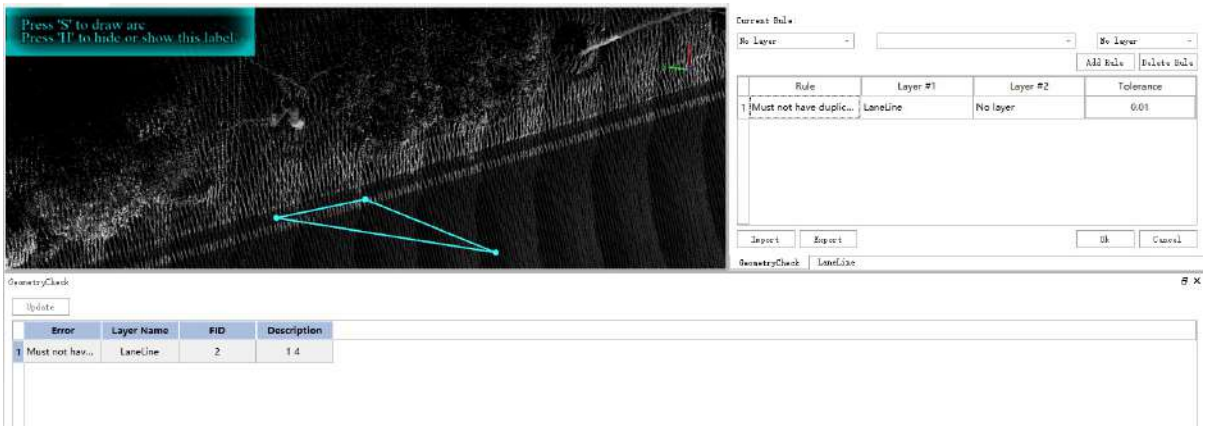
7.Click the Import button in the quality check rule settings interface, and select the previously saved quality check rule file, the previously saved quality check rules will be displayed in the quality check rule list, click OK to quickly check the quality. As shown in the figure:

Import quality check rules

Function

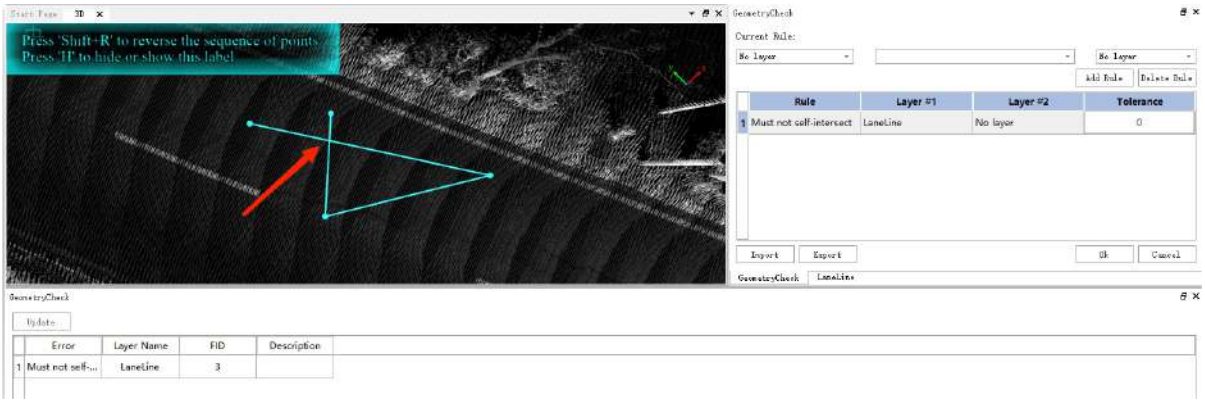
1.The quality check rules currently supported by geometry quality check are as follows:

1.1 **Vector objects cannot have duplicate nodes**: Supports line objects and polygon objects.



Self-intersect check

1.2 **Vector objects cannot self-intersect**: Supports line objects and polygon objects.

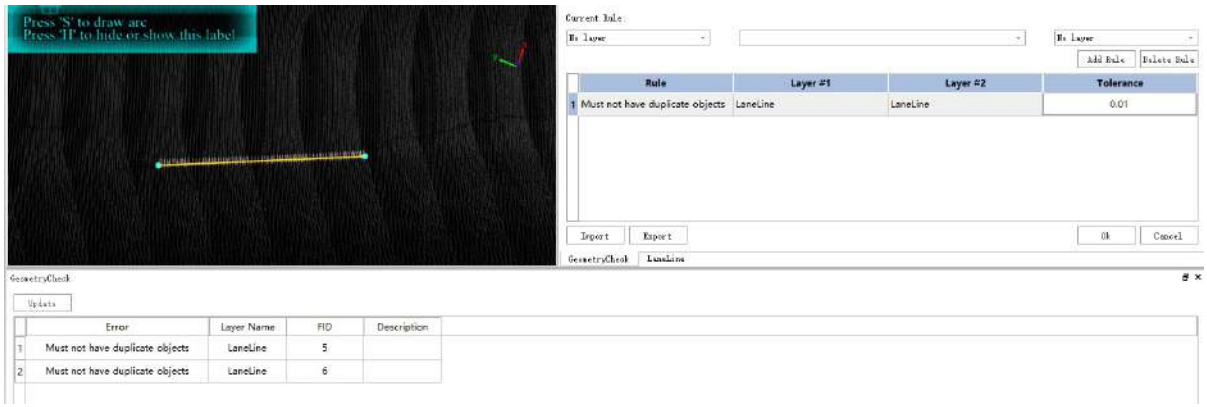


Self-intersect check

1.3 **Vector objects cannot have inflection nodes**: Supports line objects and polygon objects.

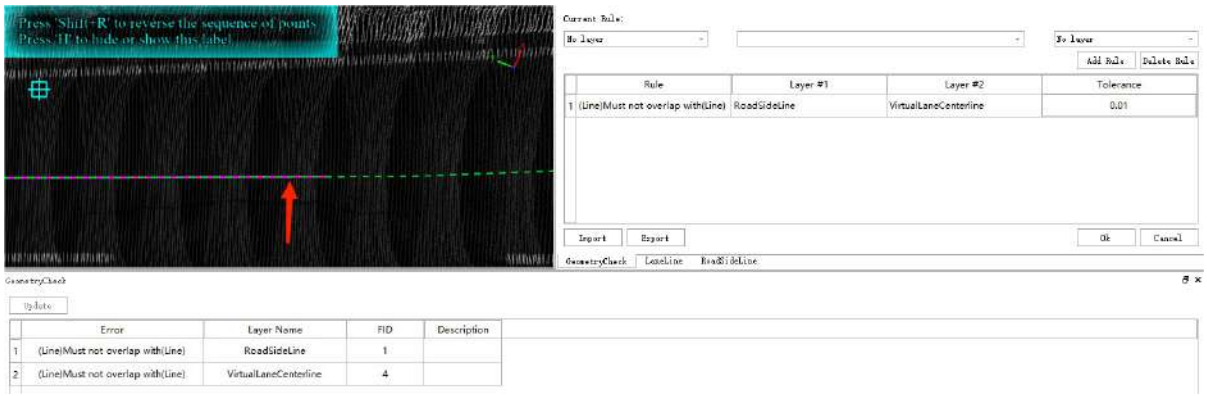
Inflexion point check

1.4 **Vector objects cannot have duplicate geometry**: Supports point objects, line objects, and polygon objects.



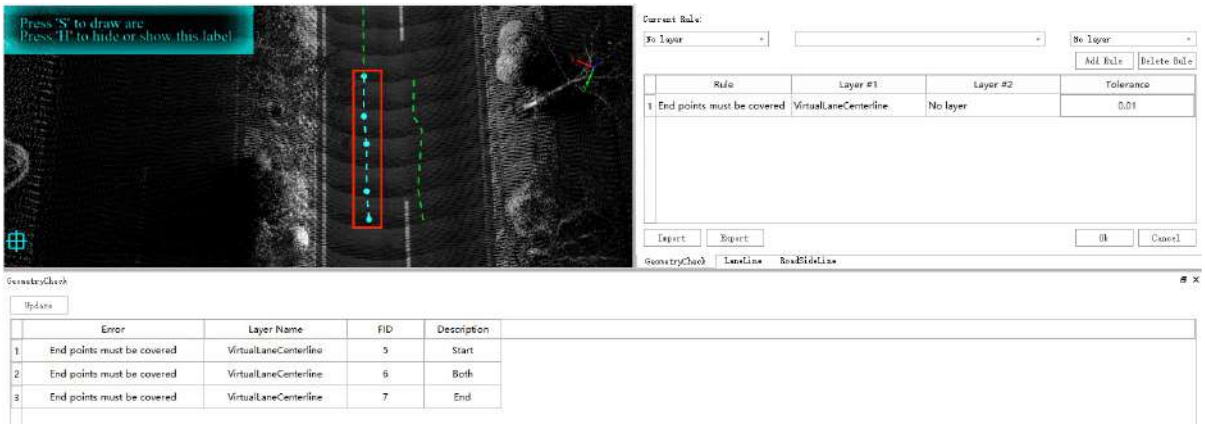
Geometry duplication check

1.5 **Vector line objects cannot partially overlap with each other**: Supports checks between line layers.



Line overlap check

1.6 **The start and end points of vector line objects must be covered**: Supports checks of line layers.

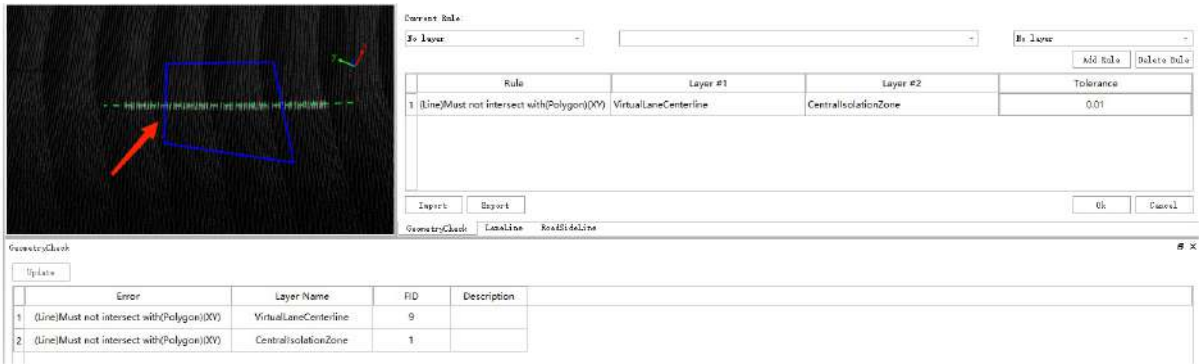


Line start and end overlap check

1.7 **Vector line objects and polygon objects cannot be collinear**: Supports checks between line layers and polygon layers.

Line and polygon collinearity check

1.8 **Vector line objects and polygon objects cannot intersect**: Supports checks between line layers and polygon layers.

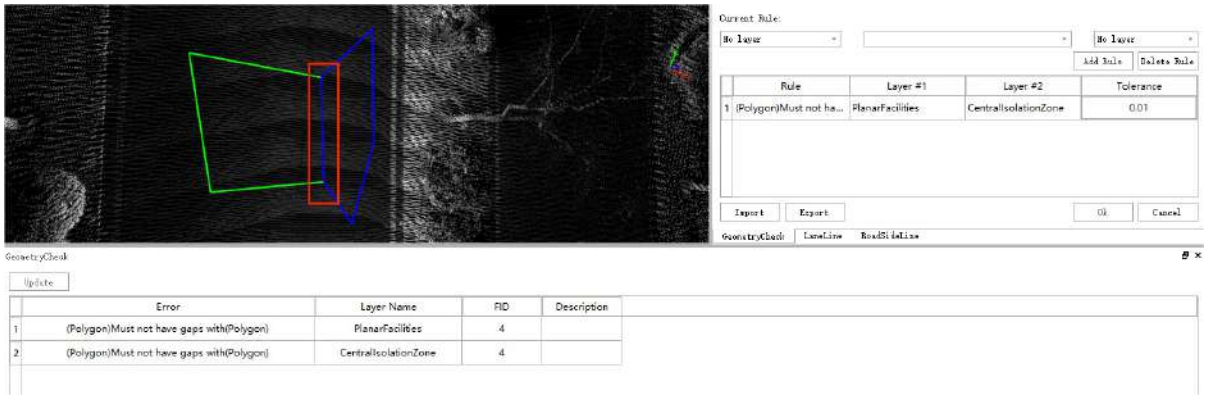


Line and polygon intersection check

1.9 **Vector polygon objects cannot intersect with each other**: Supports checks between polygon layers.

Polygon overlap check

1.10 **Vector polygon objects cannot be collinear with each other**: Supports checks between polygon layers.



Polygon collinearity check

There are two-dimensional and three-dimensional checks for each of the above quality check rules. The two-dimensional checks are based on the XY plane, and the conditions for three-dimensional checks are more strict. The two vector objects that must be checked need to be coplanar in three-dimensional space.

Floor Plan

The Floor Plan module contains the following features:

Classify by Deep Learning

Auto Vectorization

Length Measurement

Area Measurement

Angle Measurement

Line

Draw By Intersection

Draw By Perpendicular

Break Lines

Merge Lines

Line Intersection

Move Vector

Node Editor

Close Line

Export DXF

Blue Print

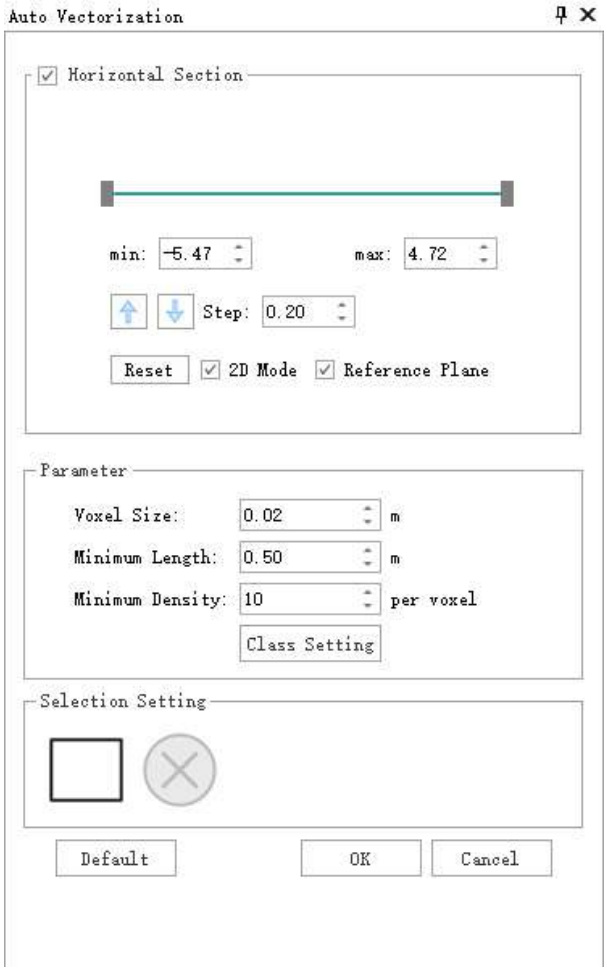
Auto Vectorization

Function Description：According to the horizontal cross-section, the point cloud data is intercepted, and the planar line vectors are automatically extracted from the intercepted point cloud data.

> Note: This function is applicable to indoor house plans and other needs, line elements obvious data

Steps

1.Click the **Automatic Vectorization** button to bring up a pop-up box：



Parameter Dialogue

Parameter Description：

- **Horizontal Section Tool**：For operations such as operating the position and thickness of horizontal sections.
  - **Minimum, Maximum**: The upper and lower positions of the horizontal section.
  - **Steps**: Setting the step length of the movement, together with the move up or move down buttons, can move the cross-section with the set step length.
  - **Reset**：Restore all parameters to their default positions.
  - **2D Mode**：Tick the box to observe and browse the point cloud data in 2D view; uncheck the box to observe and browse the point cloud data in 3D view.
  - **Reference Plane**：Whether to display the reference plane where the horizontal section is located

> In order to ensure the quality of the data, the horizontal sections intercepted need to remove as much as possible the horizontal surfaces such as floors, ceilings, etc., and keep only the vertical wall parts. Also to ensure point density, the recommended thickness (maximum - minimum) is 0.5 metres or more.

- **Parameters**:
  - **Voxel Size**:Sample resolution size of point cloud data.

    > The smaller the voxel, the higher the resolution and the higher the accuracy, but the longer the calculation time. Recommended default values
- **Minimum Length**: the shortest length of the generated vector lines, vector lines smaller than this threshold will be filtered out, used for filtering fine short lines, can be adjusted according to the actual situation
- **Minimum Density**:The minimum value of the number of points within a single voxel, voxels smaller than this threshold are not involved in vector line generation and are used to filter noise. It must not be set too large, **too large** threshold will cause a sharp reduction in the vector lines generated.
- **Class Setting**:You can set the category information of the point cloud to be filtered, and select only the category in which the wall is located
- **Selection Setting**:
  - **Rectangular Box Selection**:Rectangularly boxes a point cloud within a certain range, generating only the vector data that lies within the boxed range. The default is to use all point cloud data.
  - **Cancel**:Unchecks the range of the box.
- **Default**:Restores the parameter section to its default value.
- **OK**:After setting the parameters, click the OK button to start the automatic detection.
- **Cancel**:Exit function

2.Click OK to detect automatically

Horizontal section point cloud

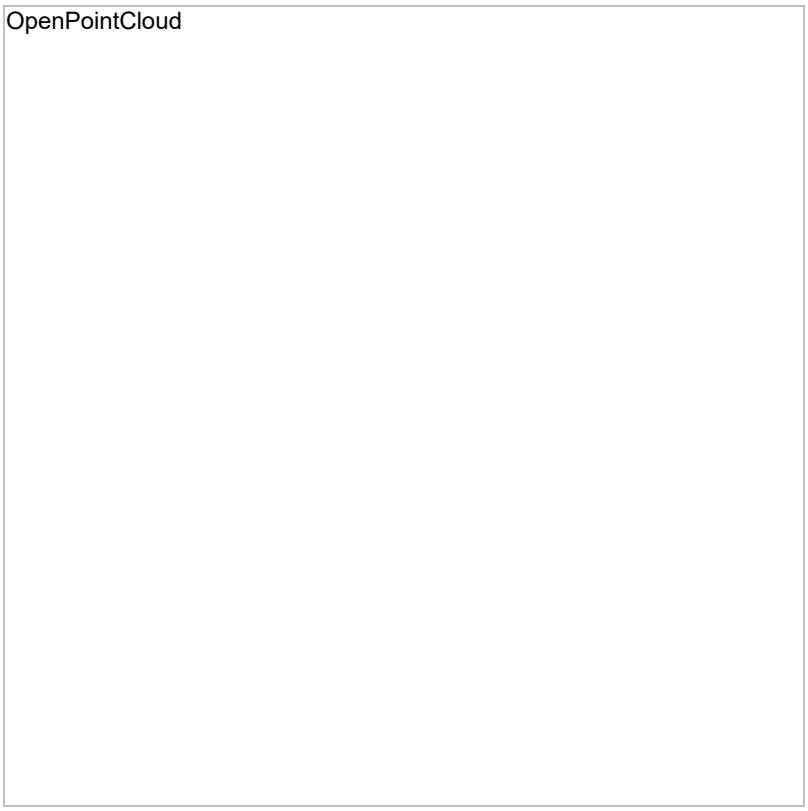
Detection results

Close Line

Functional Description：This function allows you to quickly close lines that need to be closed.

Steps

1.Click the **Close Line** button to bring up the pop-up box:



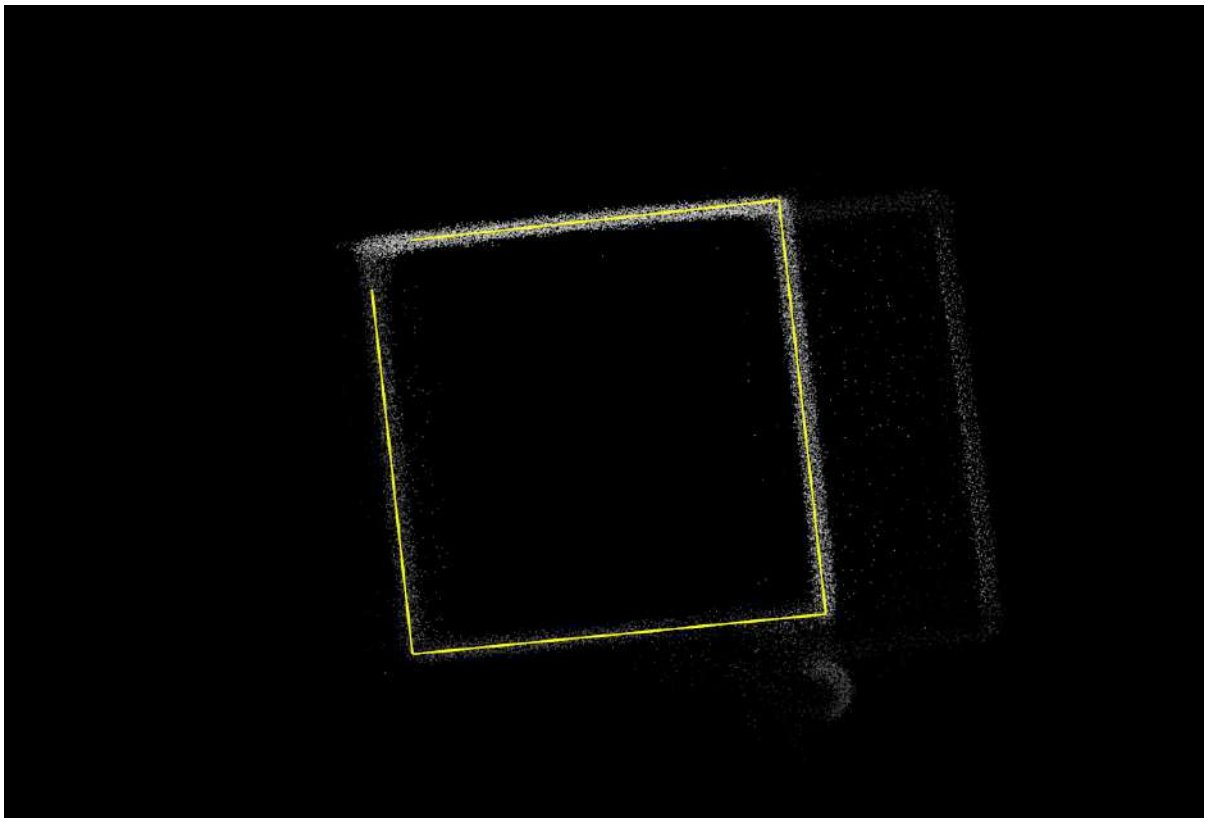
Parameter Dialog

Parameter description：

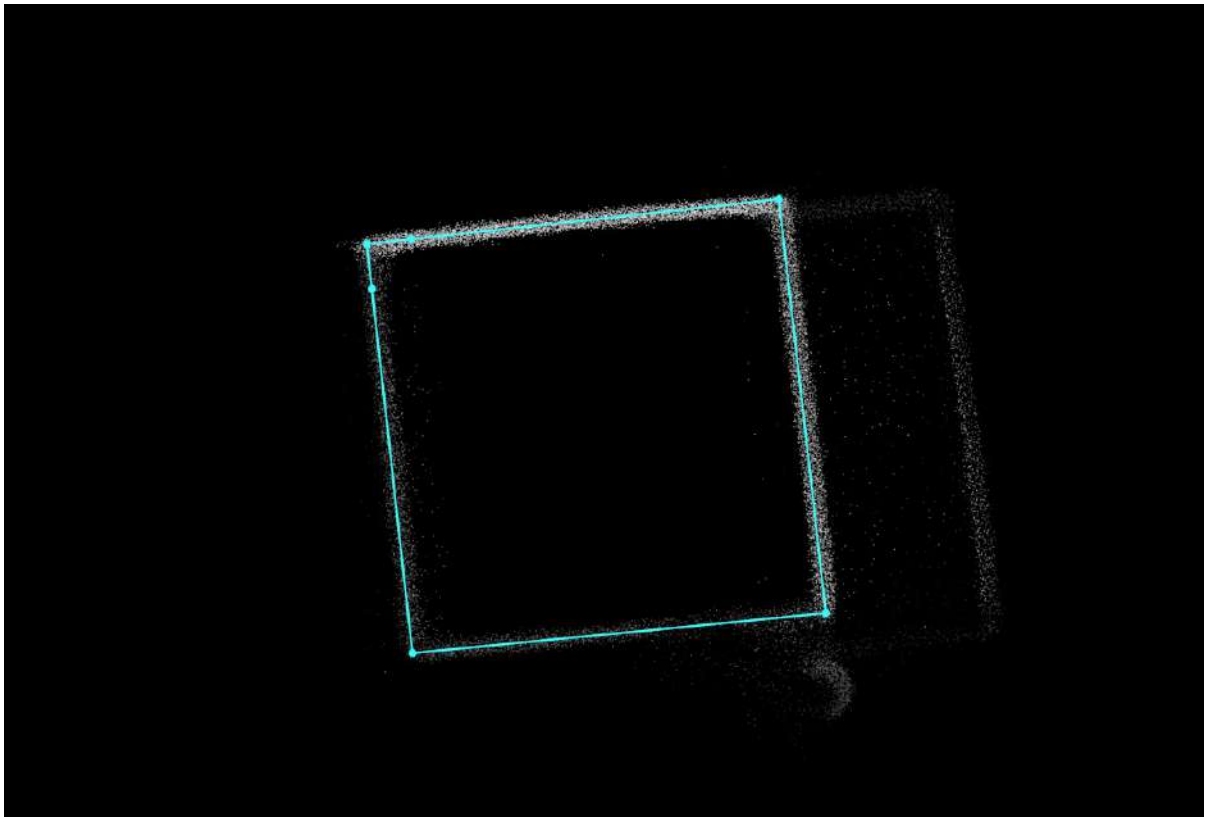
- **Adaptive**: The adaptive mode software will automatically calculate the position of the line closure for connection.
  - **Parallel Angle**: If the angle between the first and last line segments in the line to be connected is less than a given threshold, it will be considered parallel and used to directly connect the first and last, and non-parallel lines will be connected by intersection.
- **Connect first and last**: this mode is to force the first and last points of the line to be connected.
- **Intersect**: This mode is to force to calculate the intersection point of the rays of the line segments of the line to be connected to be connected, if it is not possible to calculate the intersection then it will not be connected.

2.Click with the left mouse button on the line to be connected

Scene 1：

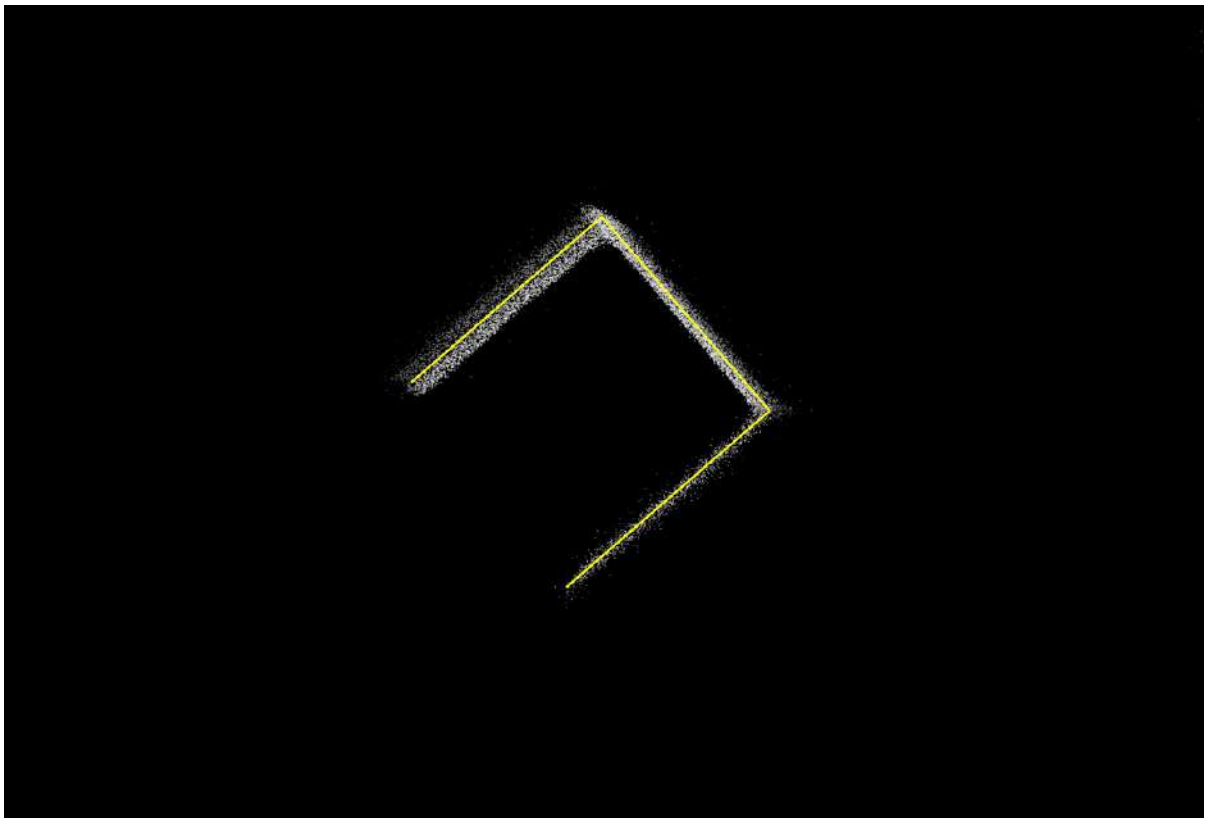


Before

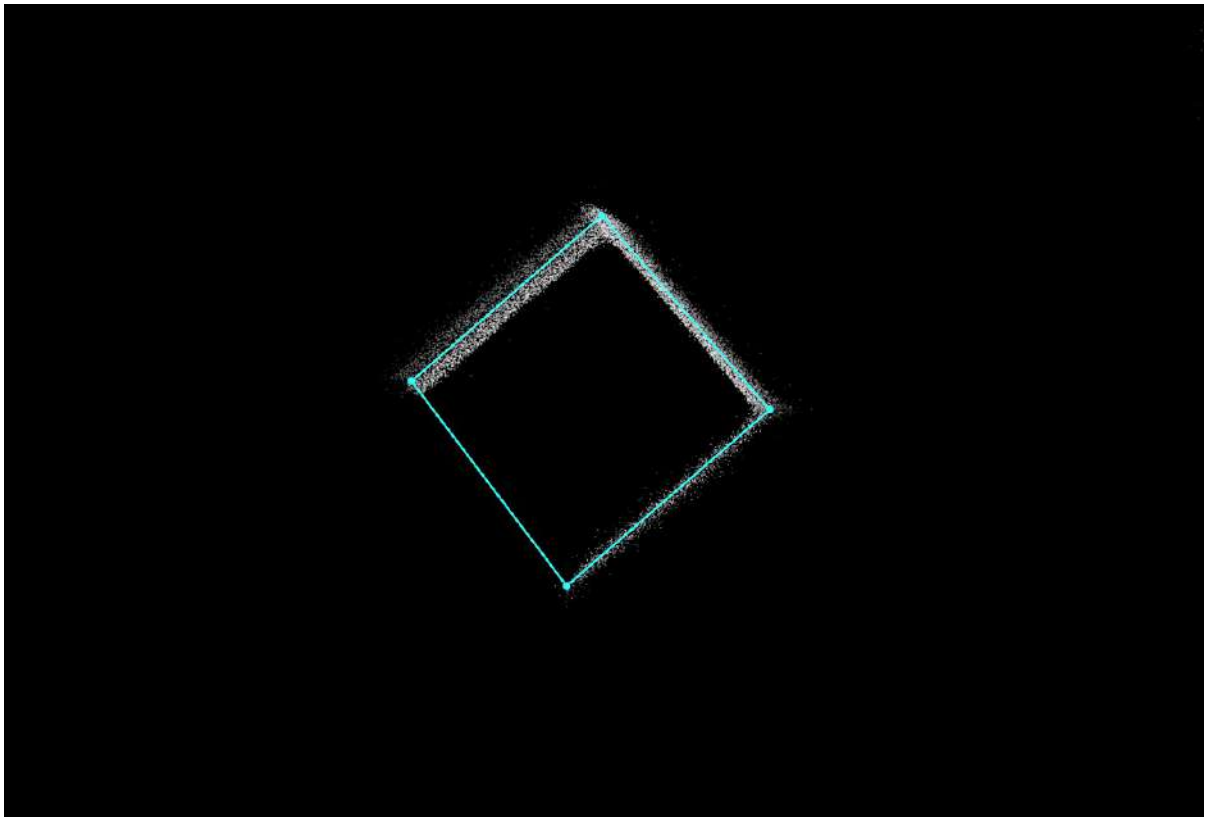


After

Scene 2:



Before



After

Export DXF

Function Description：Export the vector lines of the floor plan in *.dxf* format.

Steps

1.Click the **Export DXF** button, select the location where the .dxf file is to be saved, and the software will automatically export all the floor plan layers.

> Note: If there are no geometric objects in the floor plan layer, the layer will not be exported

Blue Print

Function Description：Export blue print of the point cloud as image data.

Steps

1.Click the **Export Blue Print** button and select the location and file name where the image file will be saved.

> Choose between .tif and .png formats.

Blue Print

Facade Survey

As shown in the following figure, the functions included in the facade survey page are:

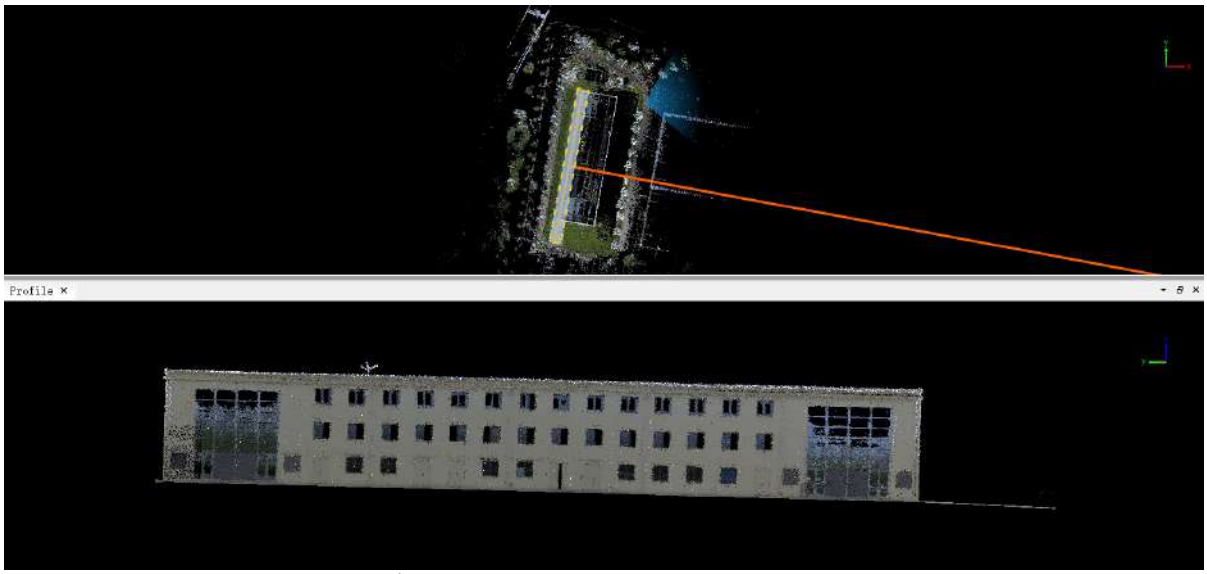
- Set Vertical
- BaseLine
- Boundary Line
- Profile
- Drawing
- Tectonic Collect
- Array
- Edit
- Export

Set Vertical

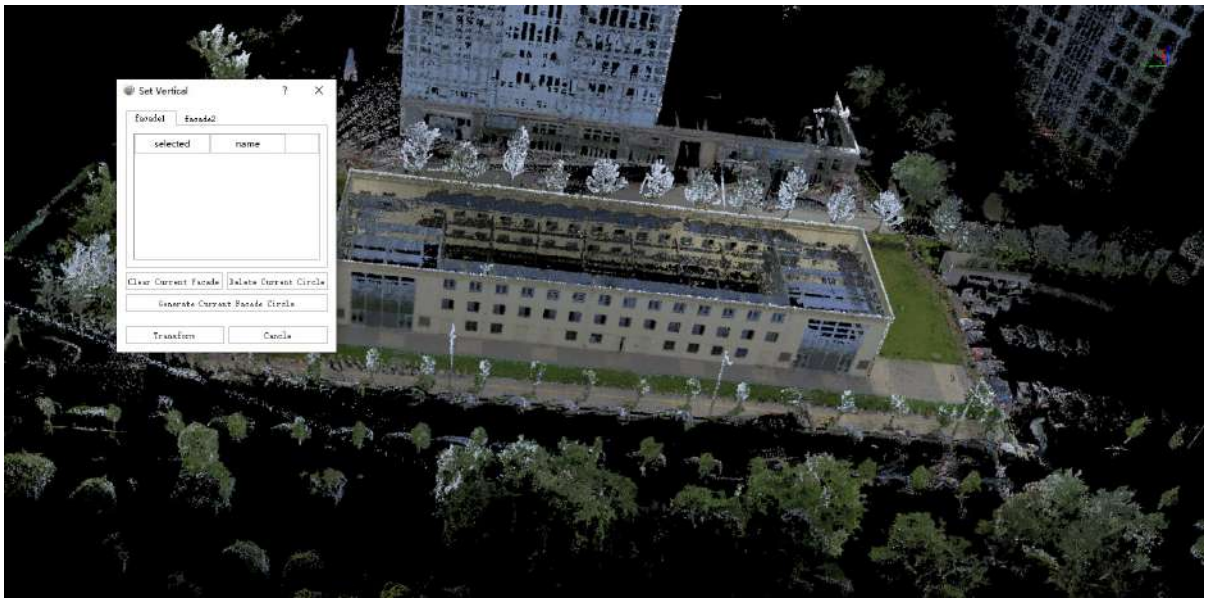
Function Description：This tool allows the user to apply vertical correction of the side facades of a building, and is performed based on two adjacent side facades of a building.

Steps

1.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.

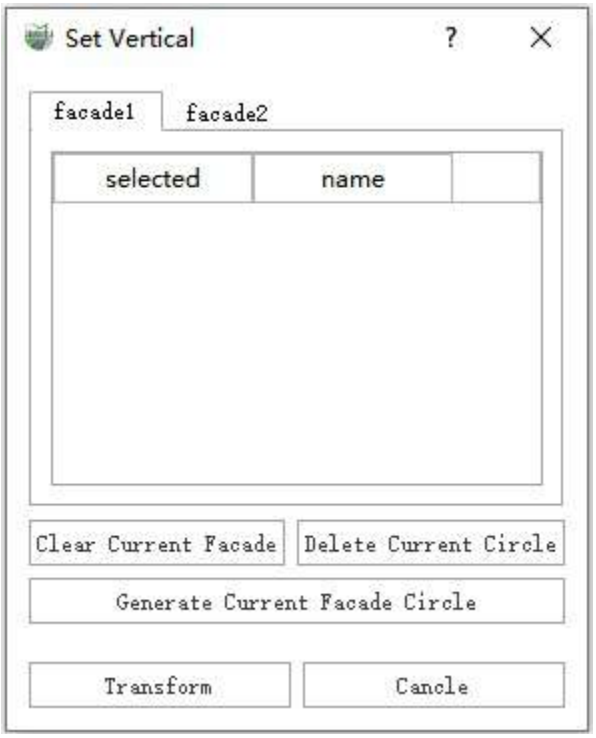


2.You can activate the "Set Vertical" function in "Facade Survey" to make the side elevation vertical:



（1）Open the point cloud fitting capture tool in the software.

（2）Click the Set Vertical button to pop up the set vertical operation pop-up box.



（3）Under the "facade1" tab page, on one side elevation of the building, evenly select several (at least three) fitting planes under the same vertical plane from top to bottom (Figure 1 below), and then click "Generate Current Facade Circle" to generate the fitted vertical plane of the current side elevation (Figure 2 below).

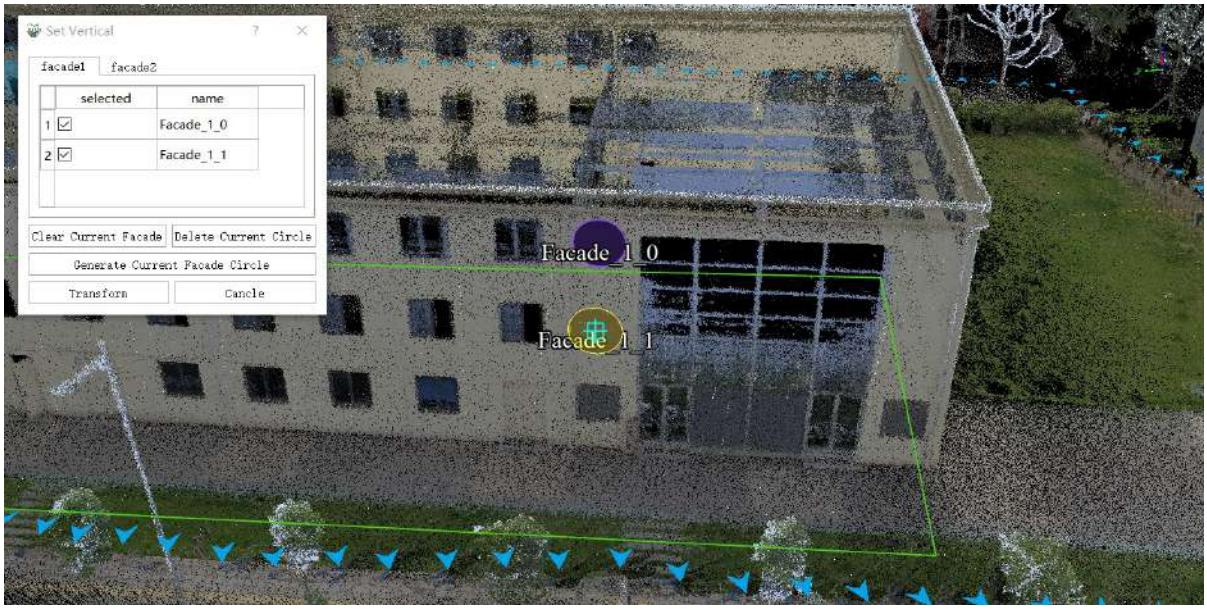


Figure 1 Selecting the Fitting Plane

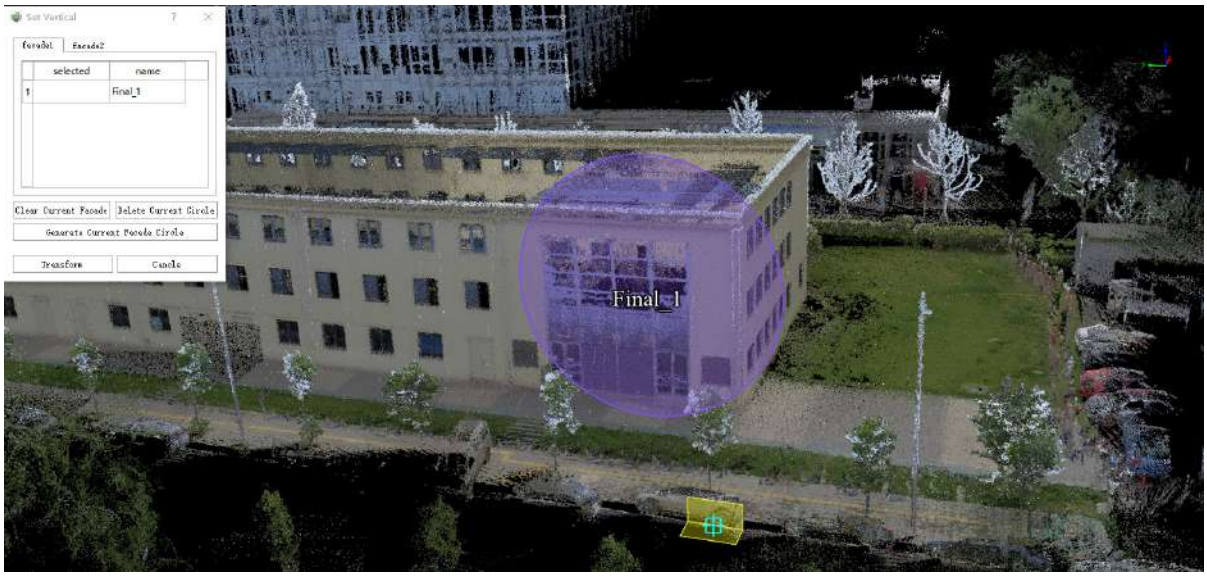
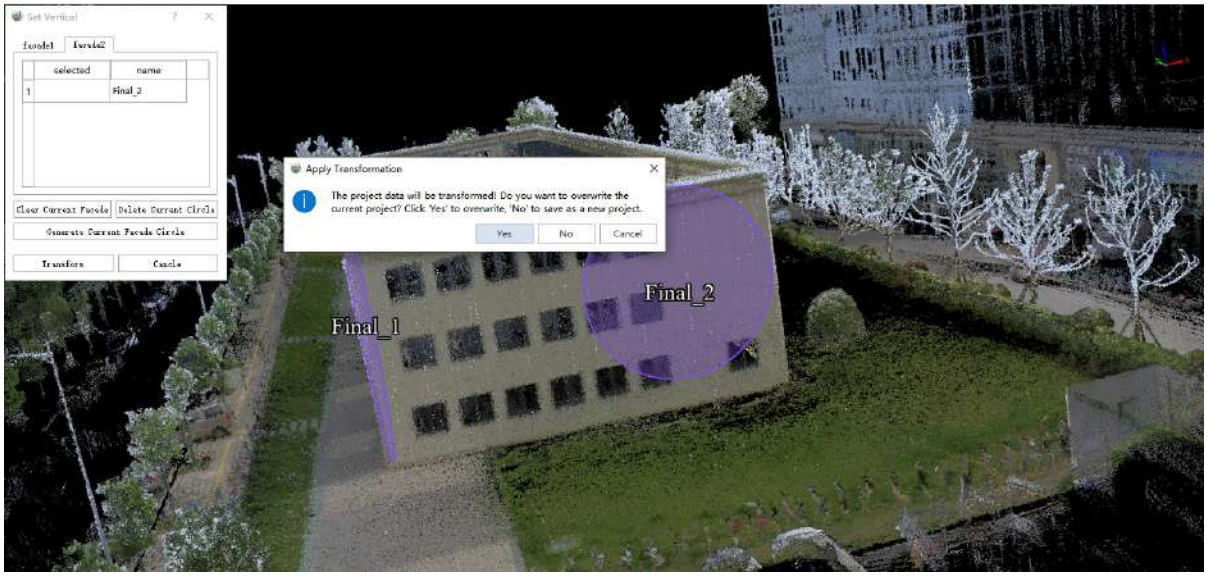


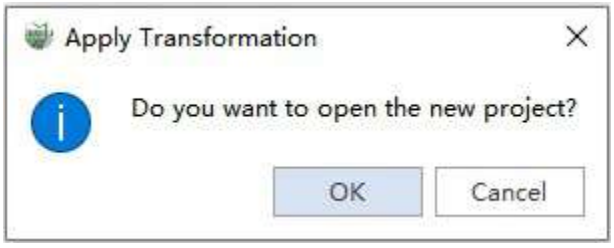
Figure 2 Generate a facade circle

Under the "facade2" tab page, select the side elevation that is connected to the edge of facade 1, and then repeat the operation of facade 1 to generate a second facade circle.

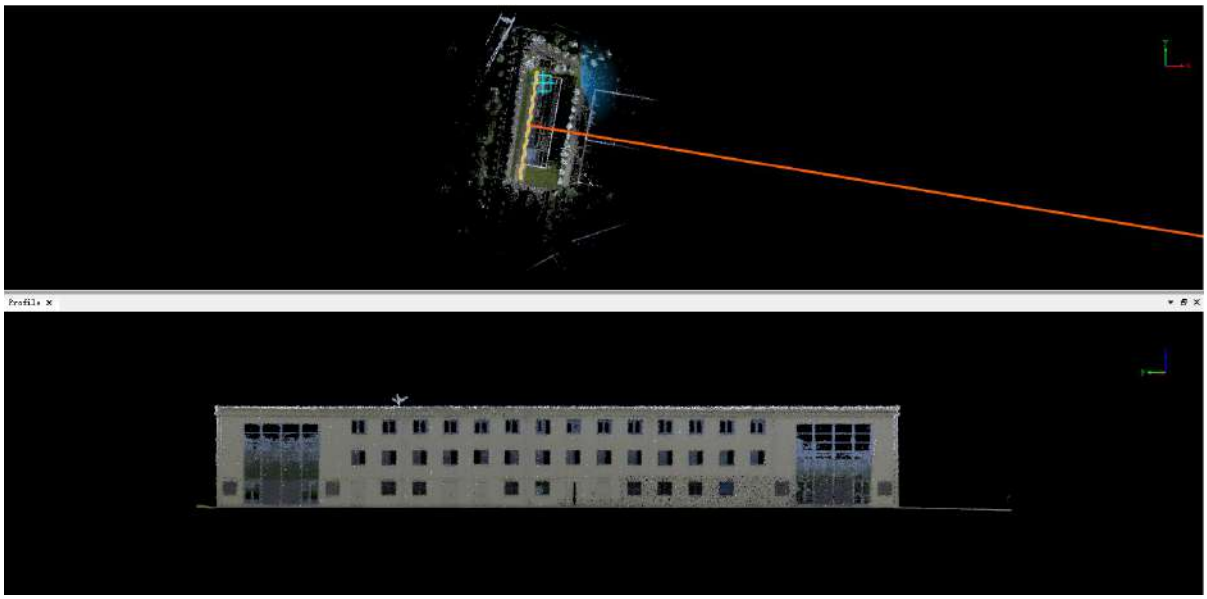
（4）After the two side facade circles are generated, confirm that they are correct, and click the "Transform" button to apply the transformation. Select "Yes", the point cloud, panorama and trajectory (if it is set) loaded in the current project will be rewritten, which is equivalent to rewriting the original data.



If you select "No", a new project with a native project name suffix “_Vertical.LiMMP“ will be regenerated according to the converted point cloud, panorama and trajectory (if it is set), and you will be prompted to open a new project after the conversion is completed.



（5）The point cloud after resetting the vertical is shown in the figure below.



BaseLine

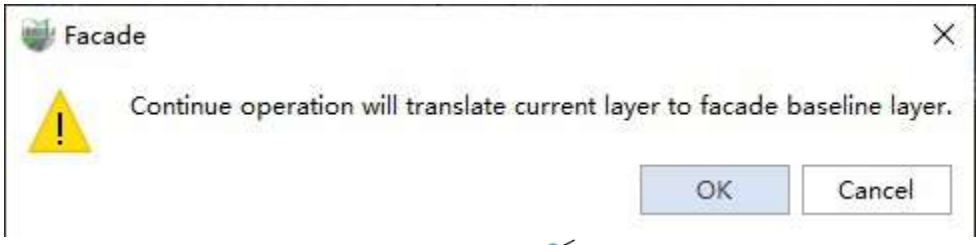
Function Description：This function allows the user to create a baseline from a horizontal structure diagram.

Steps

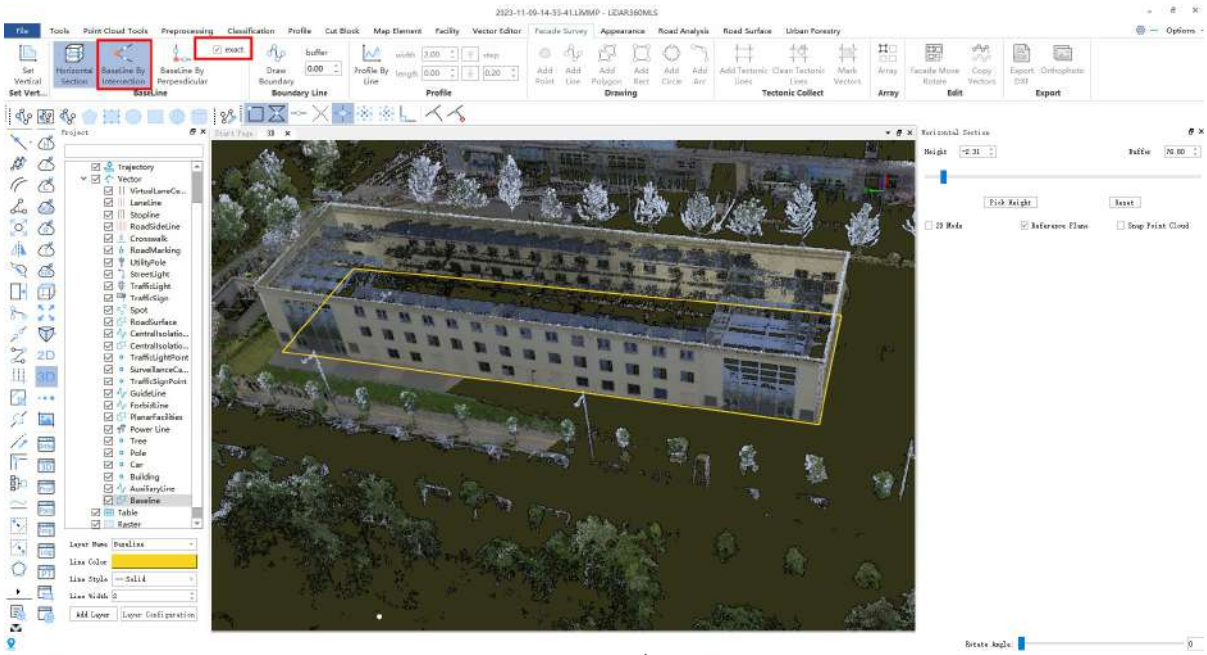
> Attention: You must have a horizontal structure diagram before draw the side facade elements.

1.First, you need to create a new horizontal structure map layer, and then select the layer to activate the horizontal section function. For details on how to use it, please refer to Tool-HorizonSection.

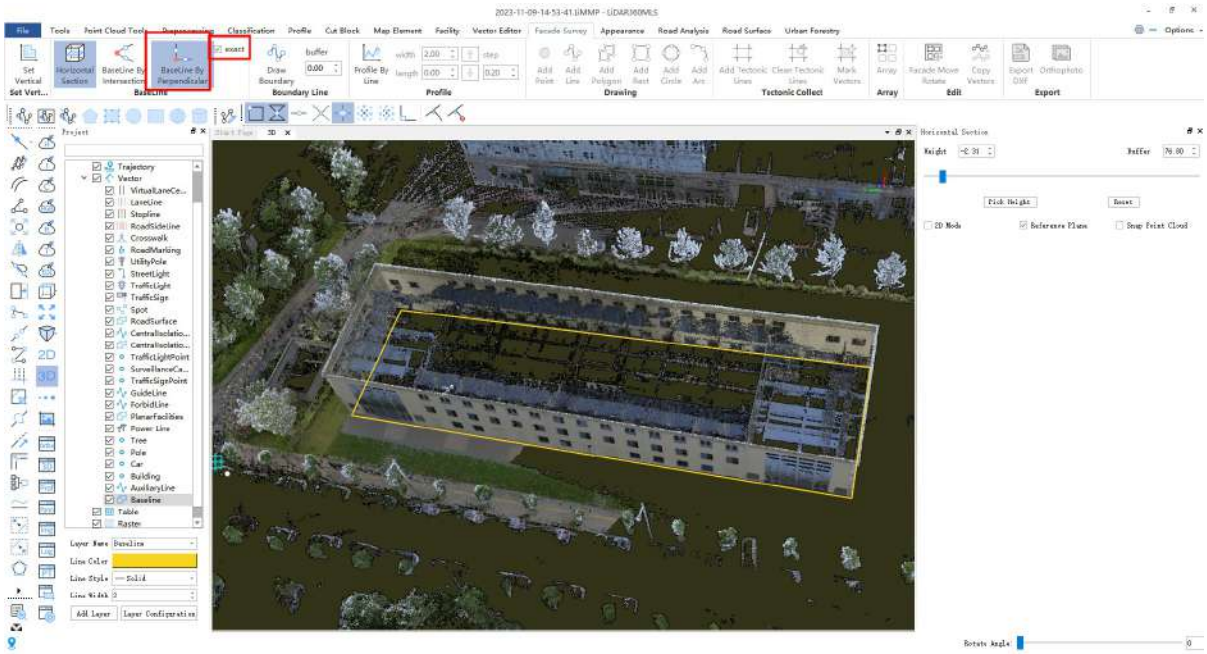
2.Select the tool in "**Baseline Collection**" under the "**Façade Survey**" menu bar, and "**Exact**" is checked by default. If you select any tool again, you will be prompted to convert the selected layer into a baseline layer, because the later selection of the edge facade controls the layer selection. Only the baseline layer and the range line layer can be selected to intercept the section.



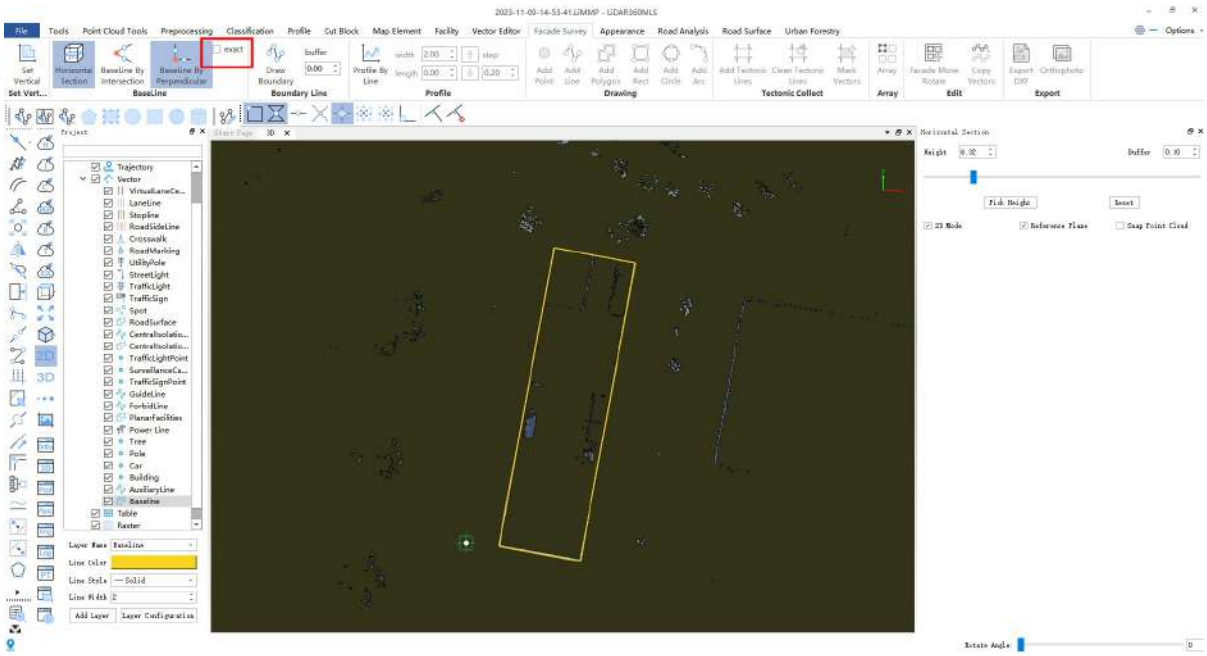
- In exact mode, clicking the **BaseLine By Intersection** button, you need to click on two points of point cloud on each side facade, and double-click on the last point to complete the closure.



- In exact mode, using the **BaseLine By Perpendicular** tool, you need to click two points on the first side elevation. Click the point on the previous point cloud on the other side facades, and double-click the last point to complete the closure.



- Unchecked "exact", the two tools will turn the 3D window into top view + orthogonal projection. At this time, it is recommended to change the buffer in **"horizon section**" to 0.1 to reduce the thickness of the point cloud, which can increase the precision of the drawing.



Boundary Line

Function Description：This function allows the user to create a boundary line based on the baseline layer, that can be expanded outward to a certain extent.

Steps

1.According to the facade drawing specifications, if you need to draw a range line, you should activate **Draw Boundary** tool in **Facade Survey**. Select a base line and you can choose to drag it or draw it out at the length of the boundary line. Enter a fixed value to define the boundary line, which will be drawn in an inline range graph layer.

Profile

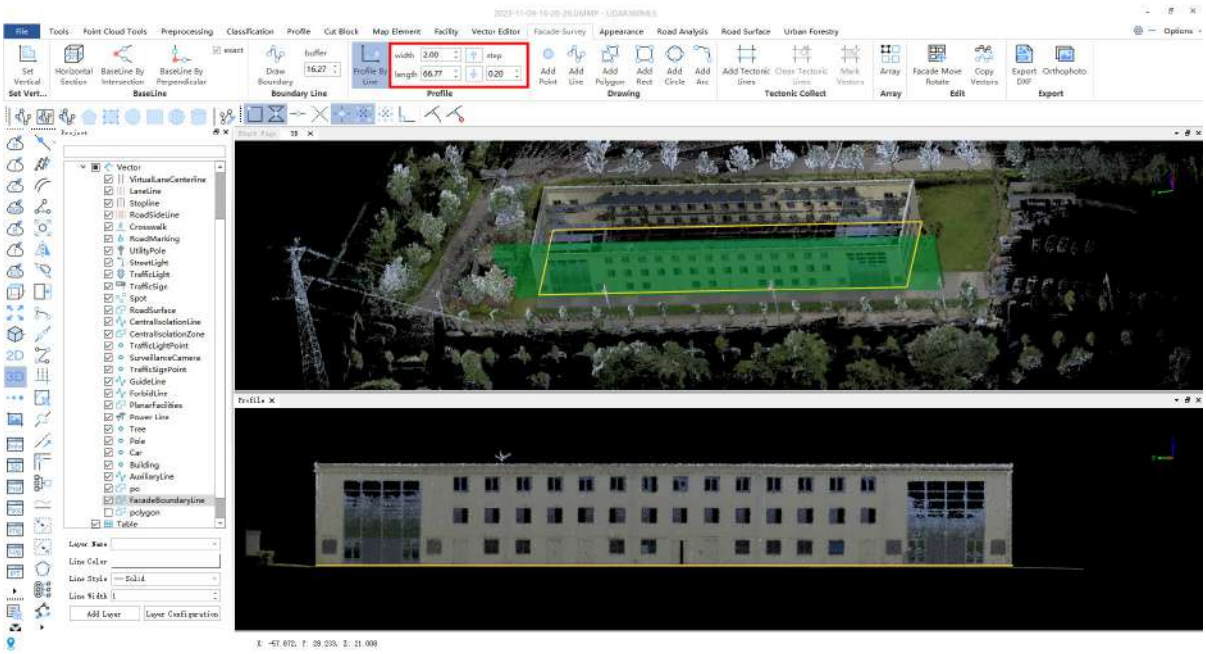
Function Description：This function allows the user to select an edge of the baseline or boundary line, and intercept the point cloud with the default thickness, and generate an profile view.

Steps

1.Click the **Profile By Line** button to activate the function of generating profile.

2.Click a vector edge of the datum plane drawn in advance, and in the section window, the side profile corresponding to the current edge can be automatically generated.

3.You can adjust the thickness of the point cloud display through "width", and adjust the left and right display range of the point cloud through "length". Adjust the position of the point cloud display "forward" or "backward":



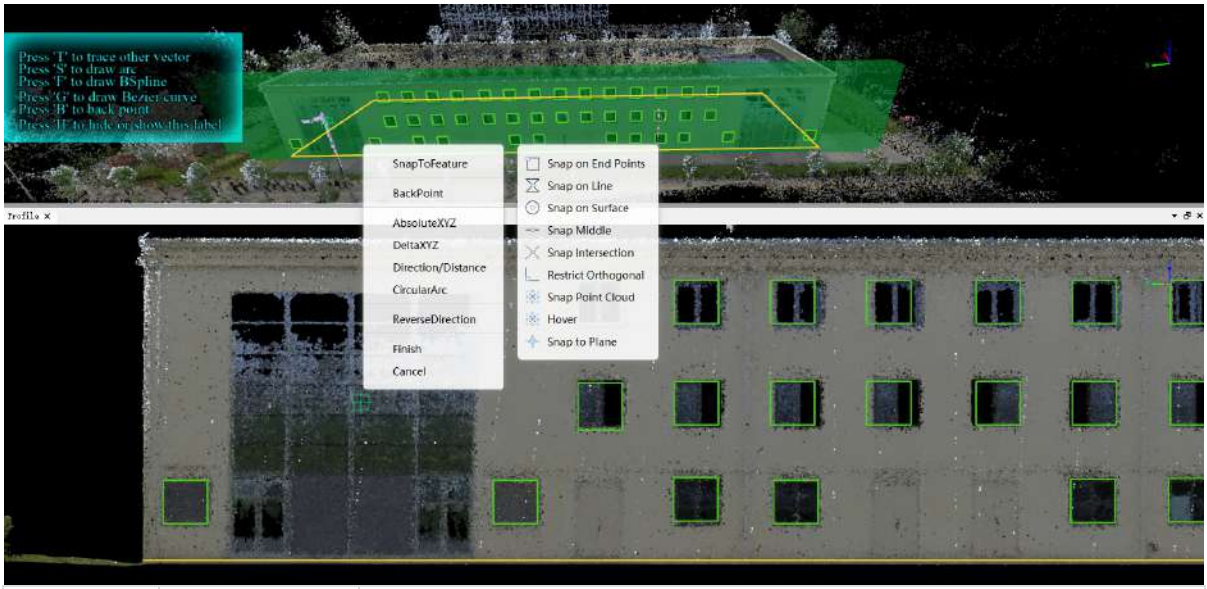
Drawing

Function Description：This function allows the user to draw features within an open dataset and is the same as the drawing function under the vector editing menu page.

Steps

1.Point element drawing: Select a point layer. Click on the **point** button and select a point in the point cloud under the profile window with the mouse. And then a circle will be drawn at this position according to the current layer.

2.Line element drawing: Select a line layer. Click on the **line** button. Using the mouse to draw line elements in the point cloud under the profile window, where line features need to be drawn, click the left mouse button continuously to determine multiple nodes of the line. Right click during drawing to activate the right-click menu, as shown in the following figure:



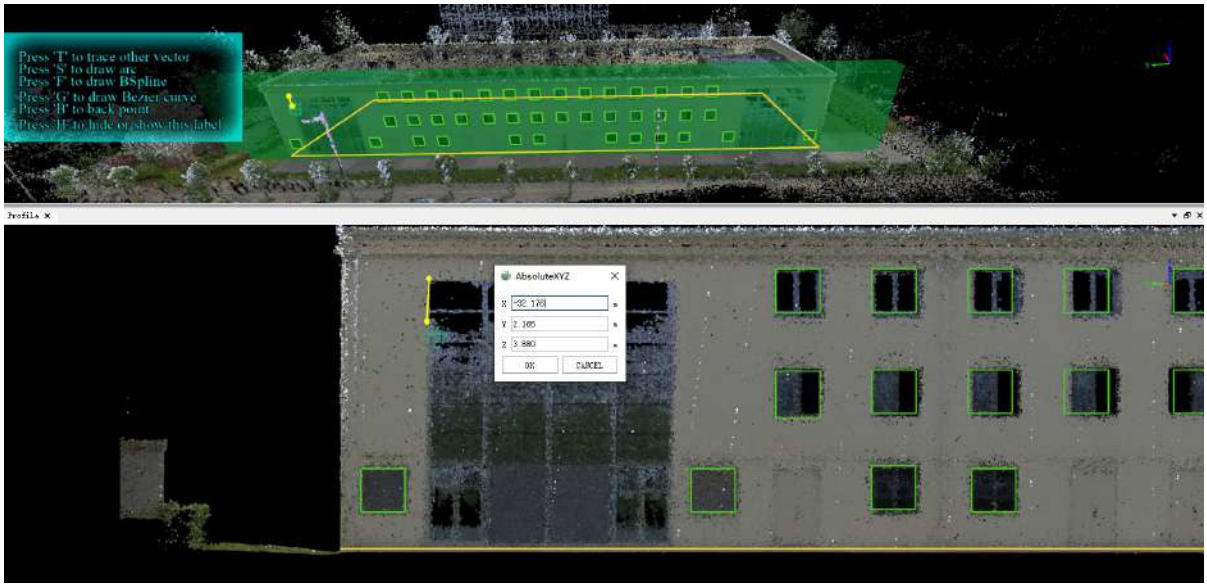
Function	Shortcut	Descripción
Drawing Arc	S/s	Short press the s key to switch the drawing mode from drawing a straight line to drawing an arc. The arc adopts the three-point mode. The first point is the last point before short press s, the second point is the end point of the arc, and the third point controls the arc
Backing off a point	B/b	Press b briefly to back out of the drawn node
Tracking Mode	T/t	In the area where vector tracking needs to be activated, press the "T" key briefly to switch to tracking mode. Once in tracking mode, click on a starting point on the object to be tracked. While moving the mouse along the object to be tracked, vector nodes will be automatically collected without the need for manual node placement. During the tracking process, you can cancel the current tracking by right-clicking the mouse. Note that after cancellation, not only will the tracked nodes be cleared, but the tracking mode will also be exited by default. If you only want to cancel part of the tracking, you can reverse the mouse movement while tracking to undo the tracked nodes. When you need to stop tracking, simply click the

		left mouse button once at the ending position, and it will automatically switch back to normal drawing mode.
Plot Spline Curve	F/f	During the drawing process, if you need to create a Spline curve, you can press the "F" key to switch to Spline curve drawing mode. In this mode, the added nodes will be calculated and added according to the Spline curve.
Drawing Bezier Curves	G/g	During the drawing process, if you need to create a Bézier curve, you can press the "G" key to switch to Bézier curve drawing mode. In this mode, the added nodes will be calculated and added according to the Bézier curve.

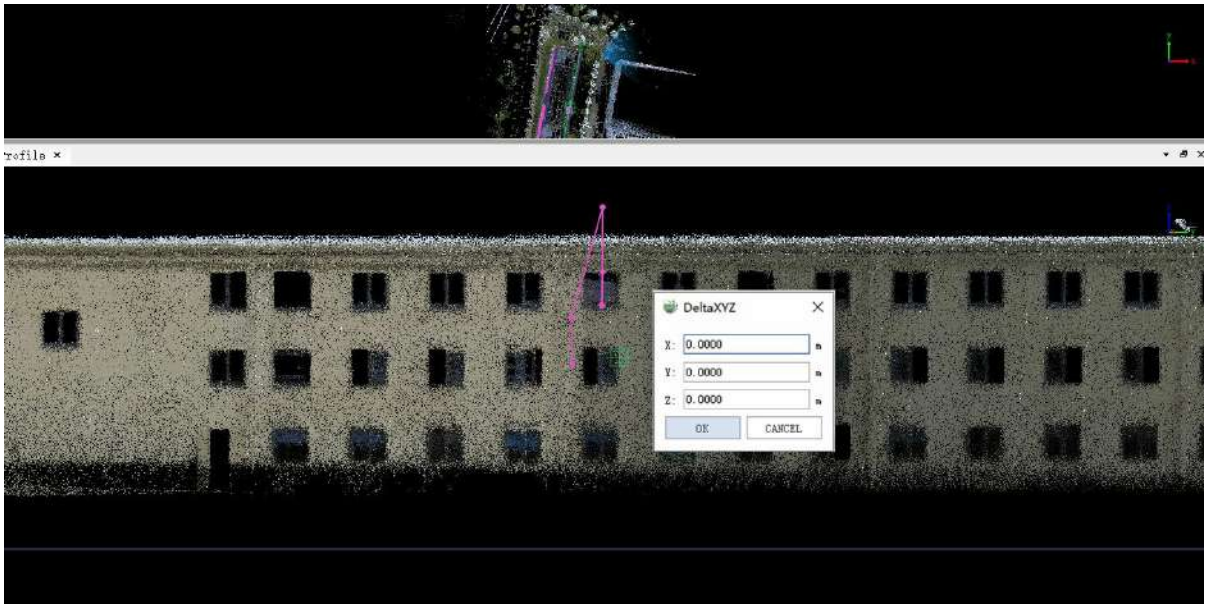
(1)**SnapToFeature**: You can switch various capture modes at any time in the SnapToFeature menu.

(2)**BackPoint**: Click BackPoint to fallback nodes in order. (the first node cannot be rolled back). During drawing, the shortcut key of BackPoint is b/B.

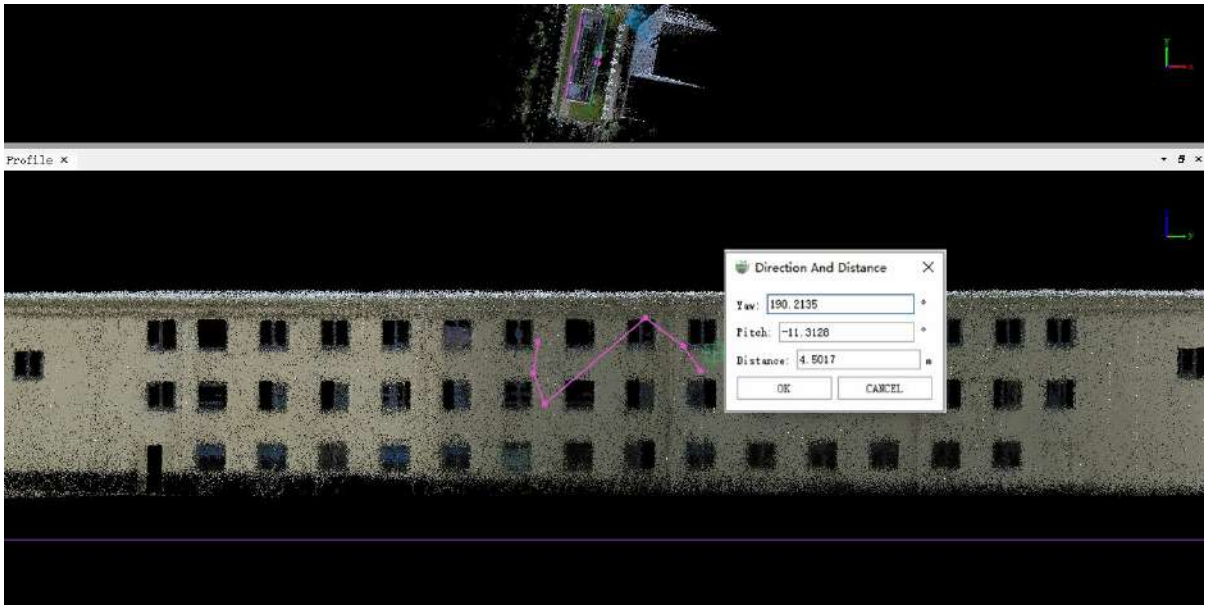
(3)**AbsoluteXYZ**: With the AbsoluteXYZ function, you can specify the coordinates of AbsoluteXYZ during the drawing process. The coordinate value entered in the pop-up box is the position drawn by the node.



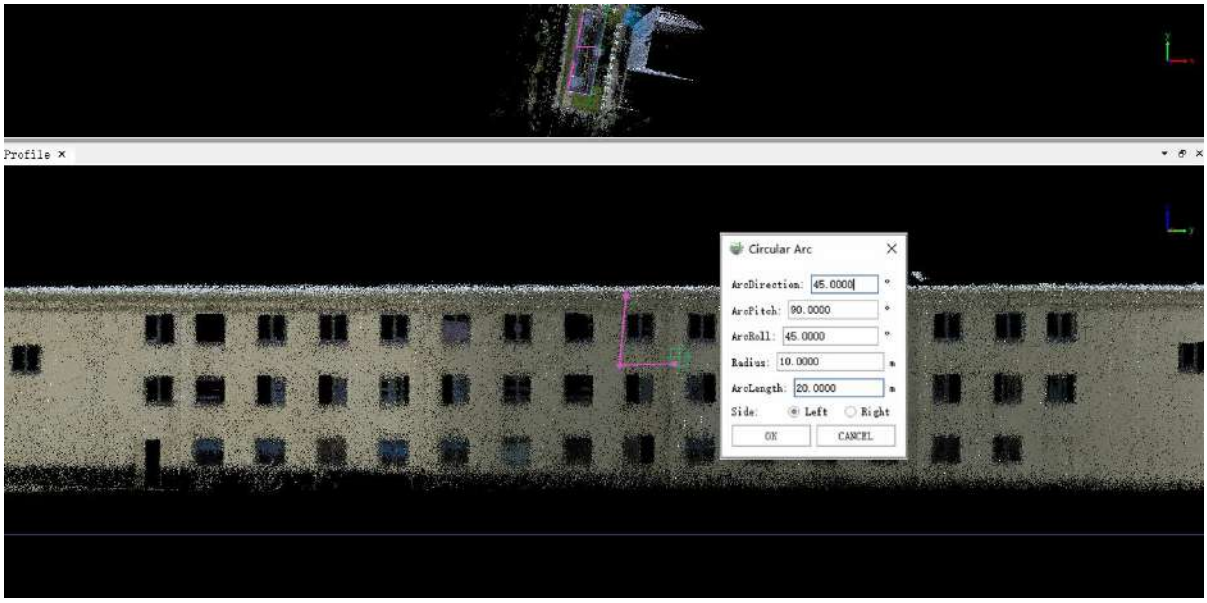
(4)**DeltaXYZ**: With the DeltaXYZ function, you can specify the coordinate position of the next node relative to the previous node in the drawing process. The coordinate value entered in the pop-up box is the position relationship between the next node and the previous node.



(5)**Direction/Distance**: With the Direction/Distance function, you can specify the direction and distance of the next node relative to the previous node in the drawing process. The yaw angle is the included angle with the Y axis of the coordinate system, the pitch angle is the included angle with the XY plane of the coordinate system, and the distance is the distance between the next point and the next point in the above angular direction. In the profile window, there is no need to modify the yaw angle.



(6)**Circular Arc**: With the circular arc function, an arc with a specified direction, size and position can be generated during the drawing process. As shown in the following figure, the ArcDirection refers to the angle of the arc relative to the Y axis of the coordinate system, ArcPitch refers to the direction relative to the XY plane of the coordinate axis, and ArcRoll refers to the rotation angle around the XY plane of the coordinate axis. Radius refers to the length of the two ends of the generated arc, ArcLength refers to the side length of the generated arc, and Side controls the current plane position to the left or right. (Note: the profile window is not applicable to arc calculation, and the generated arc will not be displayed completely in the profile window)



(7)**ReverseDirection**: During the line drawing process, the direction of the drawn vector line can be reversed with the ReverseDirection function, and the drawing can continue based on the new direction.

(8)**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.

(9)**Cancel**: During the drawing process, you can click the right mouse button to pop up the Cancel button to exit the current drawing.

3.Polygon element drawing : Select a surface layer, use the mouse in the point cloud under the profile window, where the surface element needs to be drawn, and click the left mouse button continuously to determine multiple nodes of the surface. Click the right mouse button during the drawing process to activate the right-click menu. The function is the same as right click function of line feature drawing.

4.Rectangular element drawing : Select a surface layer and draw a rectangular element by drawing.

5.Circle element drawing : Select a surface layer and draw a circle element by three points.

6.Arc element drawing : Select a line layer and draw arc line elements by three points.

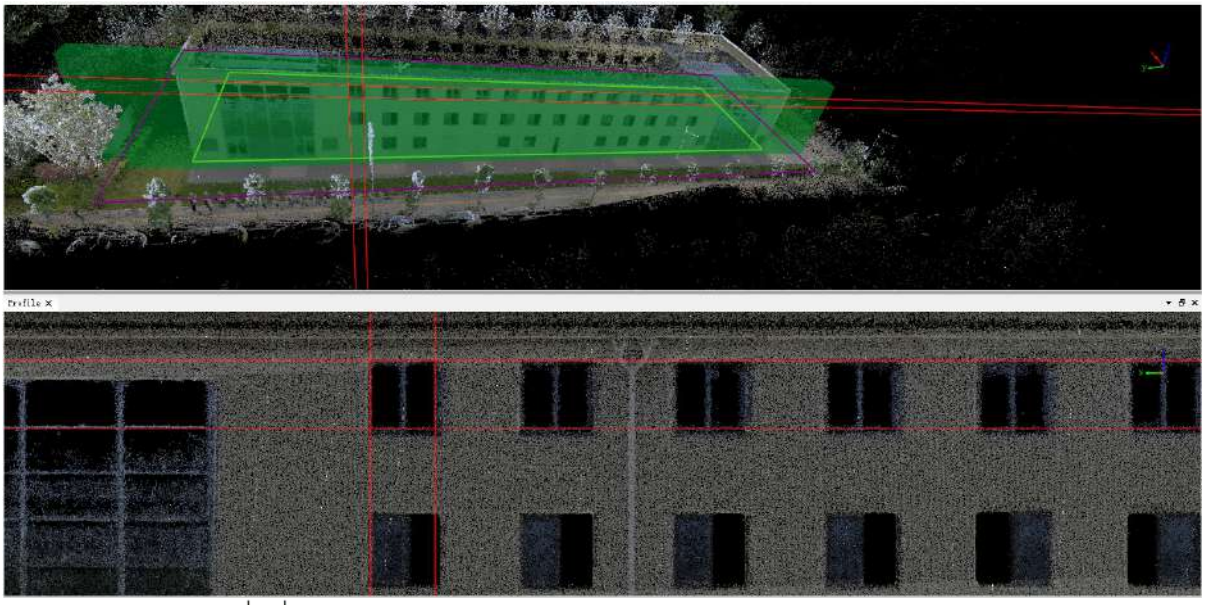
> Note: the use of basic feature drawing tools in facades is the same as that of basic feature drawing tools in vector editing module.

Tectonic Collect

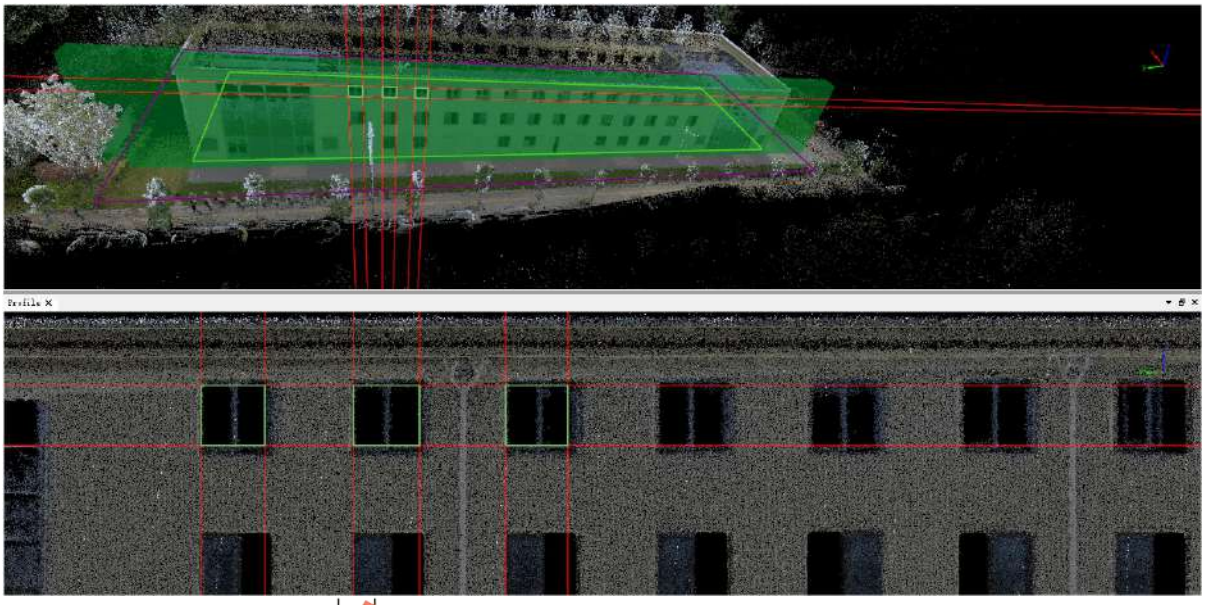
Function Description：This function allows the user to use features to draw side elevations that are regularly parallel.

Steps

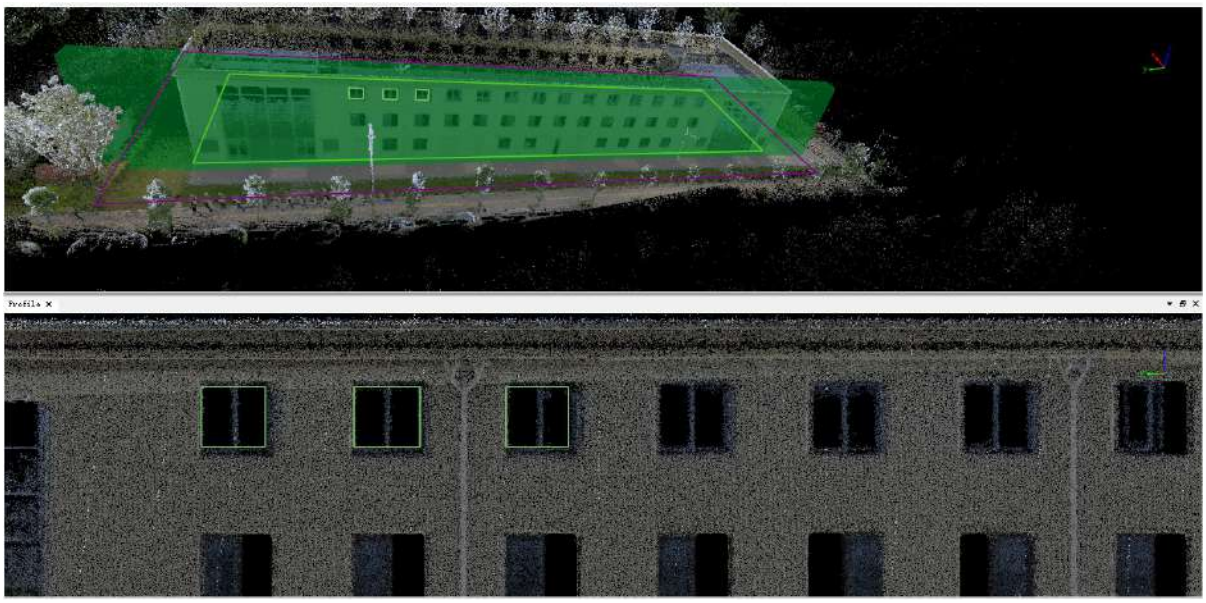
1.Activate **Add Tectonic Lines** , left click to add horizontal tectonic line, ctrl + left click to add vertical tectonic line, and select the surface layer to be drawn.



2.Click **Mark Vectors** 2.to automatically generate a rectangle in the middle of the four tectonic lines, which can mark multiple vectors continuously. (When adding a tectonic line, the right mouse button is used to roll back the previous tectonic line, and you can roll back multiple times)



3.Click **Clean Tectonic Lines** to display only the generated vector.


Profile

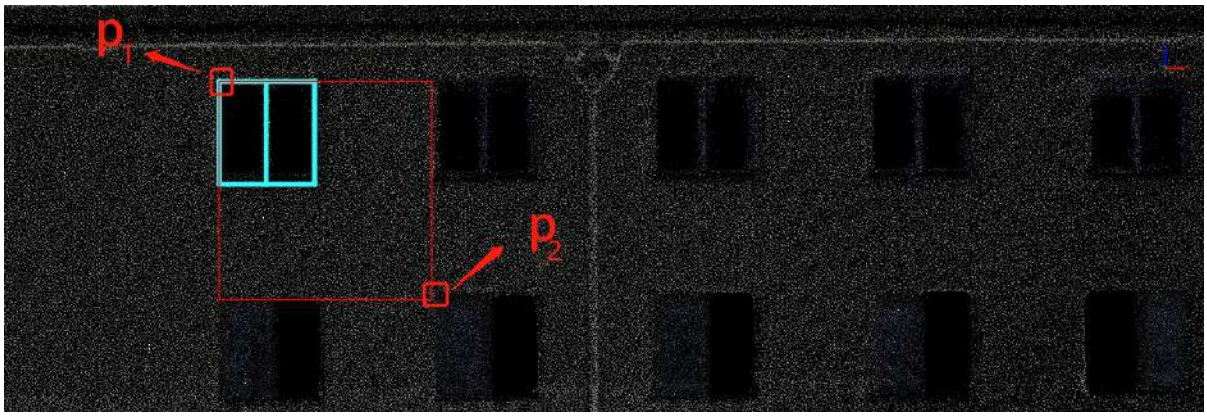
Array

Function Description: This tool allows the user to quickly draw elements with the same structure arranged at equal intervals.

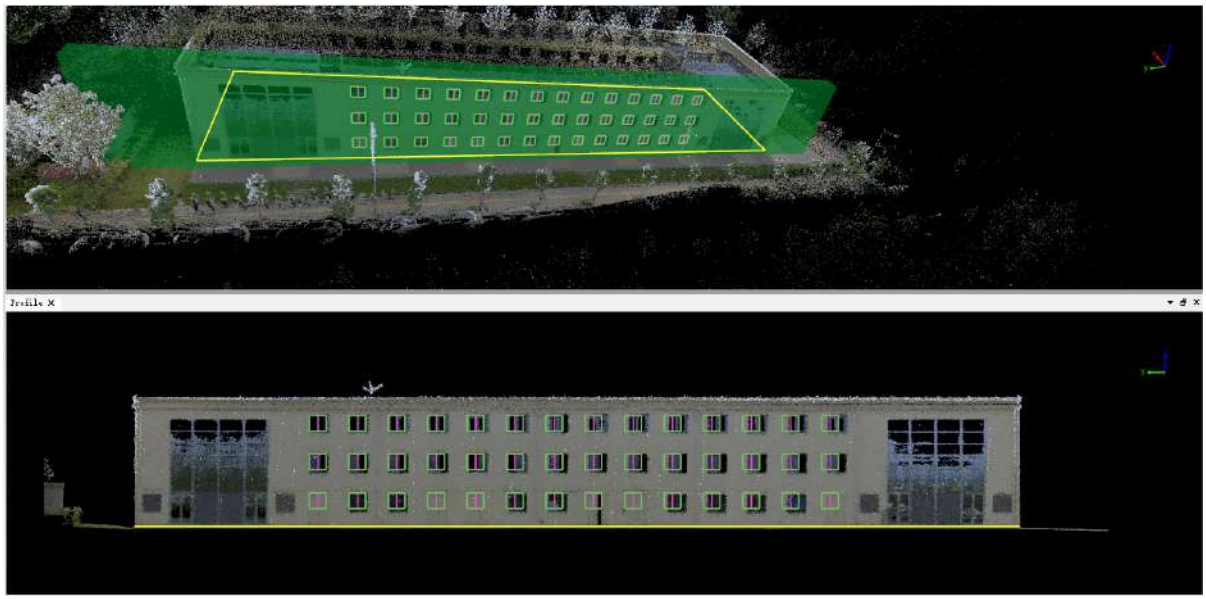
Steps

1.As shown in the figure below, the side fa?ade windows are arranged nearly equally spaced and have the same window structure.

2.At this point, you can use the "**Array**" button. Activating the tool, box select the objects that need to be arrayed with the left click, and then click "Enter" to combine them. As the figure shown below, click left mouse button to drag a box. The first point is the starting point, and the second point is the first point that needs to be copied. You can re-pull the box several times to select the position. After determining the length and width of the box, click "Enter" to lock it.



3.Drag the mouse in the direction of the box or in the opposite direction until all the arrays are completed. And double-click the mouse to complete the array. This enables you to draw a whole wall of windows and their interior elements very quickly.

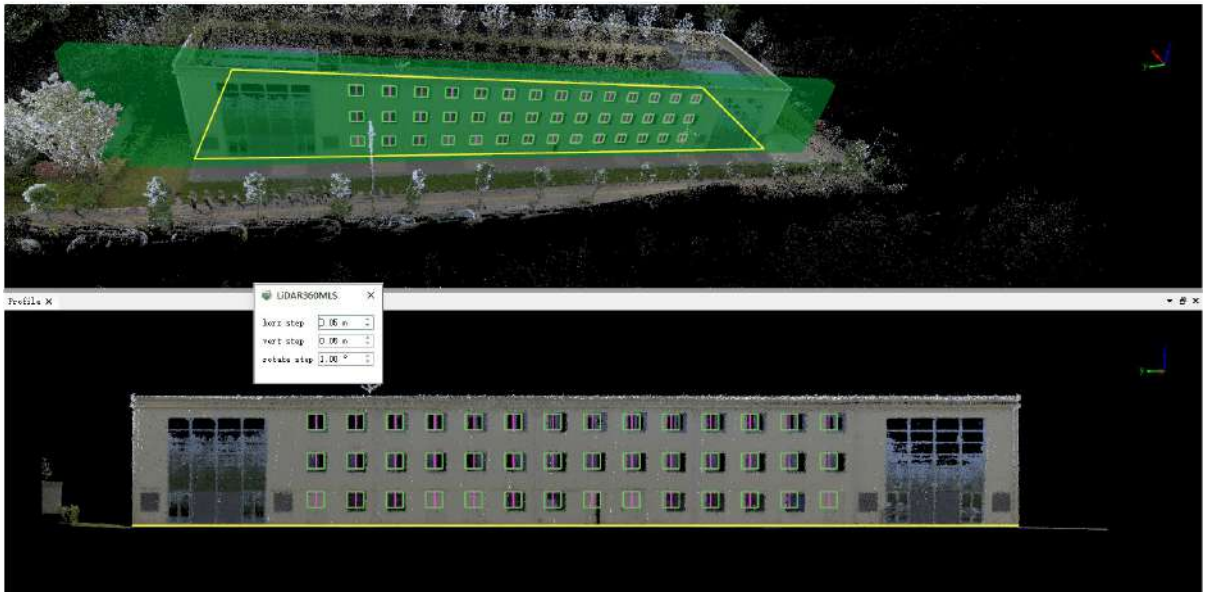


Facade Move Rotate

Function Description: If some elements are not drawn accurately, this allows the user to utilize motion and rotation to compensate and adjust the drawn elements to improve their accuracy.

Steps

1.Activate **Facade Move Rotate** , box select the elements that need to be fine-tuned with the left click to adjust the movement and perform rotation compensation.



2.Use the up, down, left and right keys on the keyboard to fine tune the position, Q(q) /R(r) to rotate counterclockwise / clockwise, and press "Enter" to lock after adjustment:

3.After adjustment, the changes are shown in the figure below:

Copy Vector

Function Description: This tool allows the user to draw elements to any position of the current profile.

Steps

1.Click "Copy Vector" button in Editing activate the copy vector function.

2.Press and hold the left mouse button, box select one or several objects to be copied, and then click “Enter”.

3.You will see that the selected objects have been grouped together and moved with the mouse.

4.Then move the mouse to the location where you want to copy, and click the left mouse button to complete the copy process. The left mouse button can always be used to click and place.

5.After all copies are completed, right-click to exit copying.

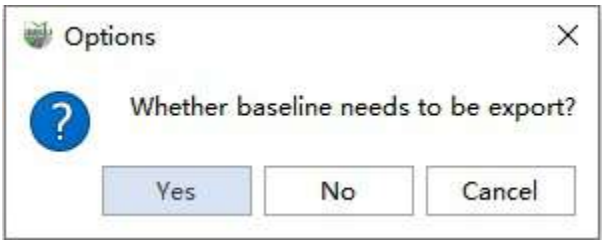
Export

Function Description: The export tool can export project data in various formats as input for third-party software, which can be used for data display or further data processing.

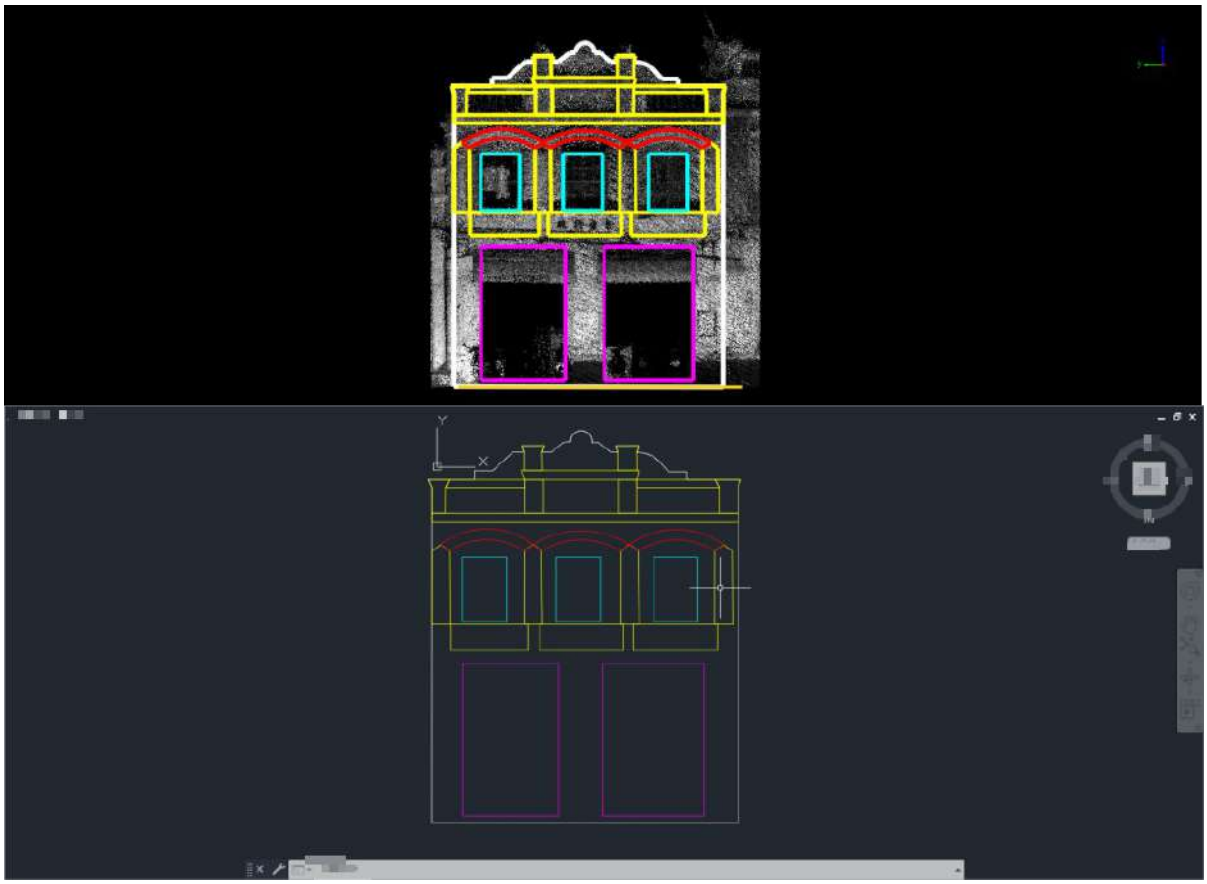
Steps

Export DXF format

1.Click the **Export DXF** function, and you will be prompted whether to save the baseline.

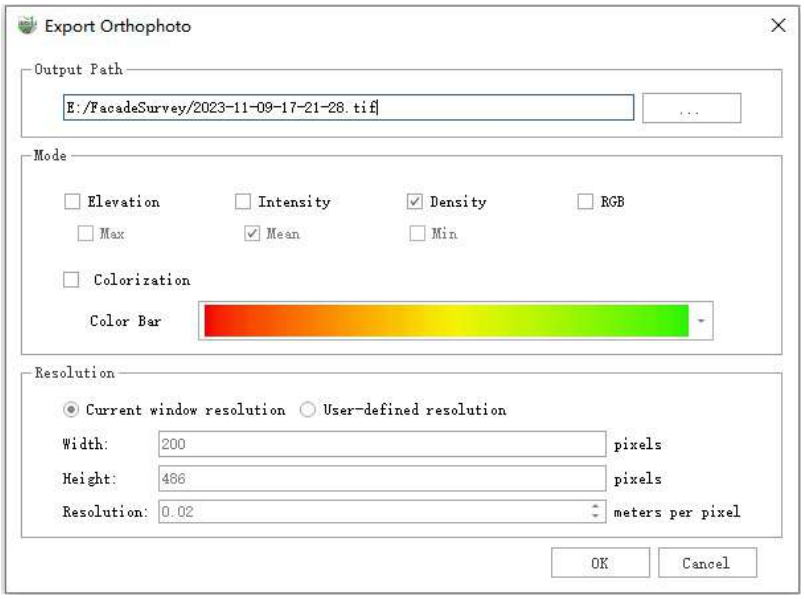


2.If you choose "Yes", the corresponding baseline and range line of the facade (if any) will be saved together. The exported DXF can be opened with CAD software.



Export Orthophoto

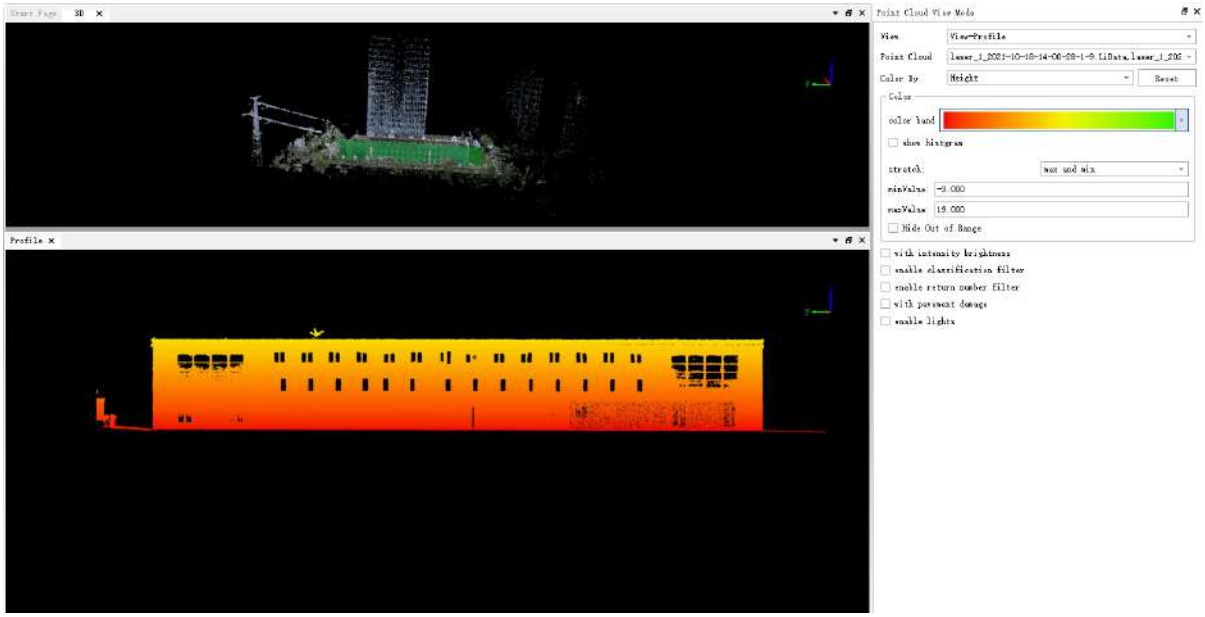
1.Click on the **Orthophoto** feature, and the Export Orthophoto dialog box will pop up as shown. The Export Orthophoto can use the Horizontal Section tool and can export the elevation, intensity, density, and RGB image of the point cloud. The exported image supports the TIF and PNG formats, and the exported image can be opened and measured in other GIS software.



2.Colorization: Whether to set the rendering color of the exported orthophoto as an option, you can customize the color bar according to the needs for export rendering.

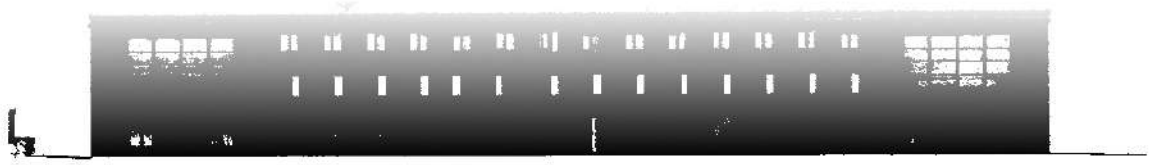
Export by Elevation

1.Select **Elevation** in the **Orthophoto Setting Window - Mode**, and the point cloud in the 3D window will be displayed **according to elevation**. At the same time, the Point Cloud Color Mode Setting window on the right side of the software will pop up, and you can choose the current color bar and other settings for displaying according to elevation through the Point Cloud Color Mode Setting window.



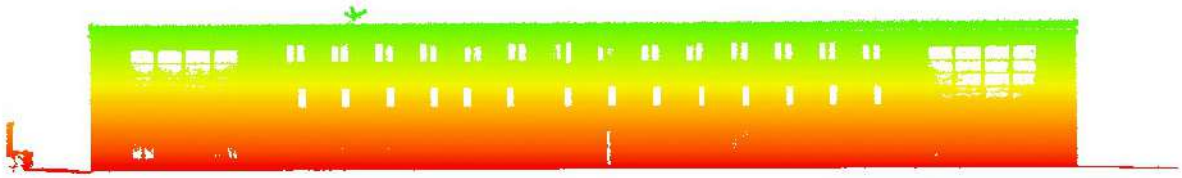
Display by Elevation

2.After the point cloud display settings are completed, you can choose whether the exported image is colorized. If colorization is not selected, a binary image will be output.



Export Binary Image by Elevation

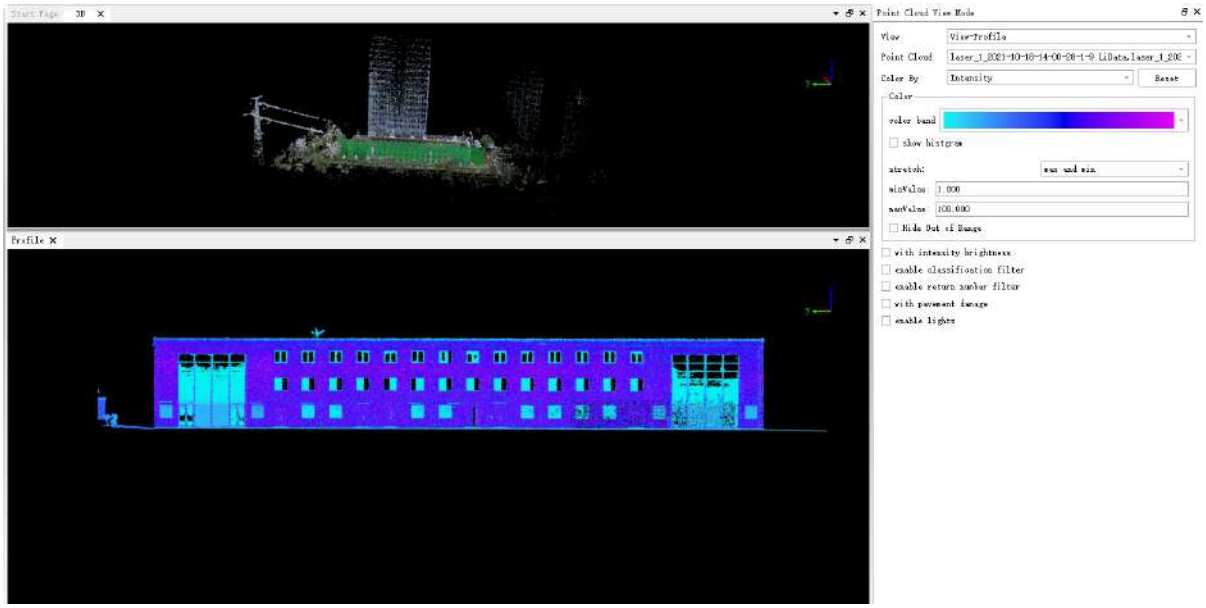
3.If colorization is selected, a colorized image will be output.



Export Color Image by Elevation

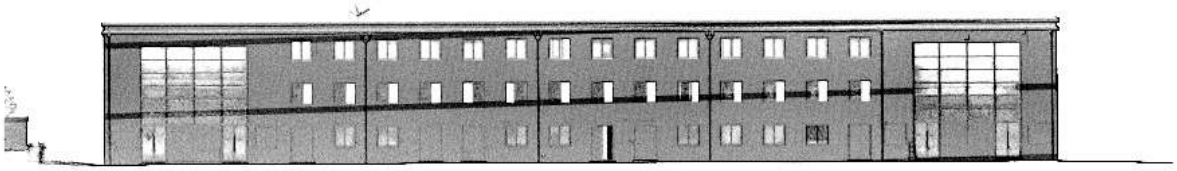
Export by Intensity

1.Select **Intensity** in the **Orthophoto Setting Window - Mode**, and the point cloud in the 3D window will be displayed **according to intensity**. At the same time, the Point Cloud Color Mode Setting window on the right side of the software will pop up, and you can choose the current color bar and other settings for displaying according to intensity through the Point Cloud Color Mode Setting window.



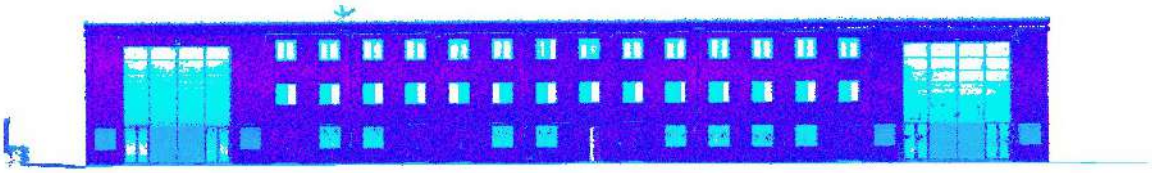
Display by Intensity

2.After the point cloud display settings are completed, you can choose whether the exported image is colorized. If colorization is not selected, a binary image will be output.



Export Binary Image by Intensity

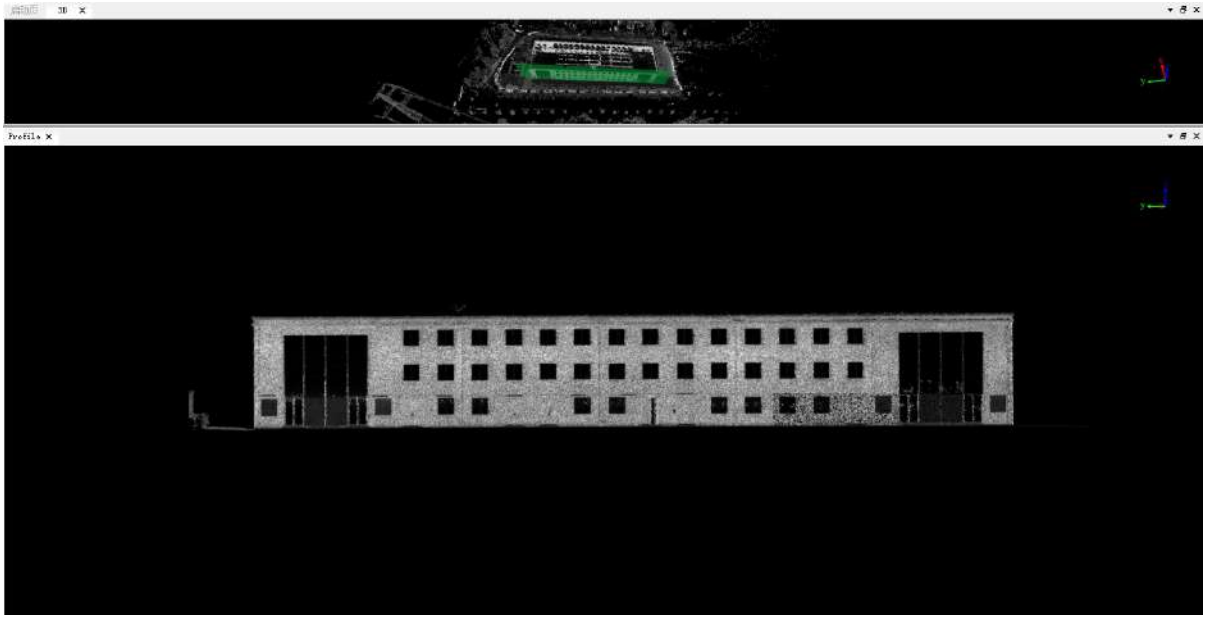
3.If colorization is selected, a colorized image will be output.



Export Color Image by Intensity

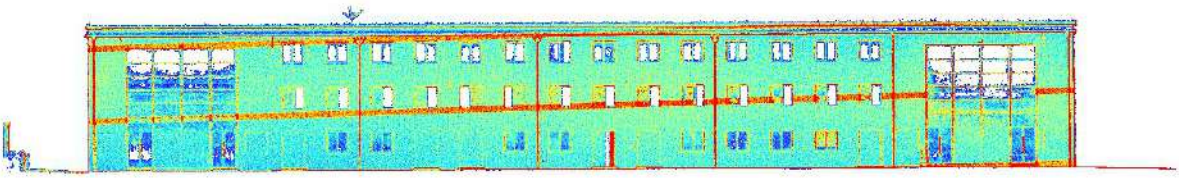
Export by Point Cloud Density

1.Select **Density** in the **Orthophoto Setting Window - Mode**. Since the point cloud display mode does not have the option to display **according to density**, the 3D window will not change. The color bar for exporting the image according to point cloud density will be set in the Orthophoto Setting Window. If colorization is not selected, a binary image will be output.



Export Binary Image by Density

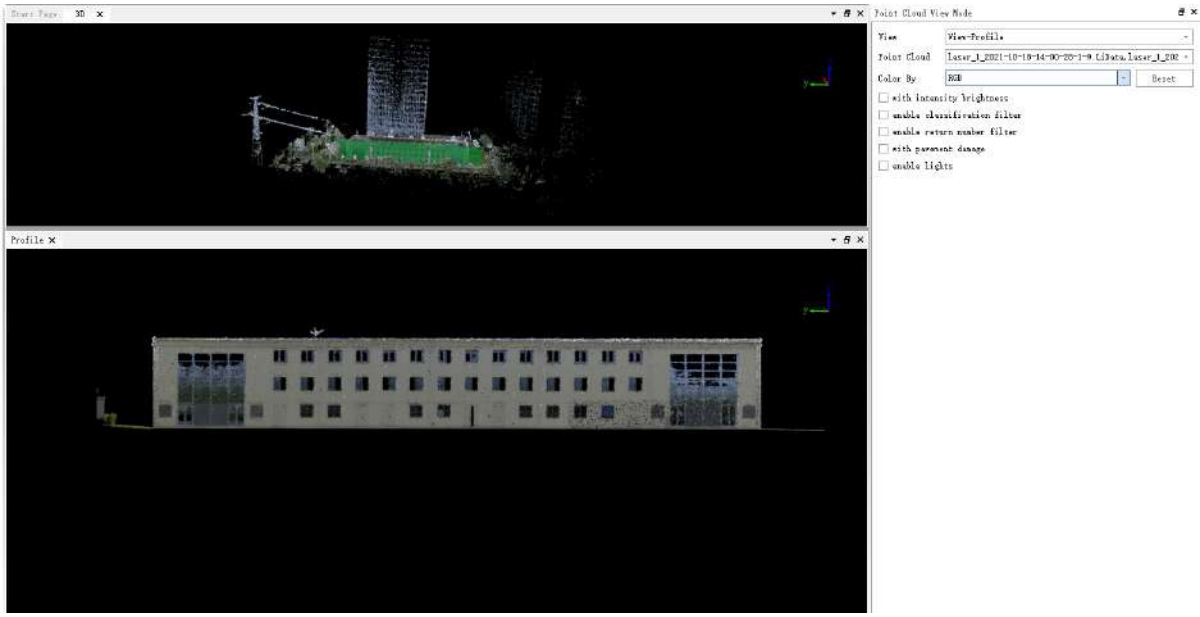
2.If colorization is selected, a colorized image will be output.



Export Color Image by Density

Export by Point Cloud RGB

1.Select **RGB** in the **Orthophoto Setting Window** - Mode. If the point cloud contains **RGB** information, the point cloud in the 3D window will be displayed **according to RGB**. If the point cloud does not contain RBG information, the software output window will give a warning prompt. After selecting RGB, the colorization option setting will be invalid, and only a colorized image containing the point cloud RBG information can be exported.



Display by RGB

Export Color Image by RGB

> Note: In each mode of export, there are options to select the maximum, minimum, and average values. This setting takes effect in the mode settings for exporting by point cloud elevation and intensity.
>
> - Maximum: When calculating the pixel value of each pixel in the image, it is calculated according to the maximum of all point cloud elevations or intensities mapped to the current pixel.
> - Minimum: When calculating the pixel value of each pixel in the image, it is calculated according to the minimum of all point cloud elevations or intensities mapped to the current pixel.
> - Average: When calculating the pixel value of each pixel in the image, it is calculated according to the average of all point cloud elevations or intensities mapped to the current pixel.

3.Set the resolution：

- **Current Window Resolution**：The aspect ratio and resolution of the output image will be exactly the same as the 3D window, and no setting is supported.
- **User-defined Resolution**：The aspect ratio of the output image will be consistent with the 3D window, and the resolution of the image supports users to set the world coordinate accuracy mapped by each pixel.

Appearance

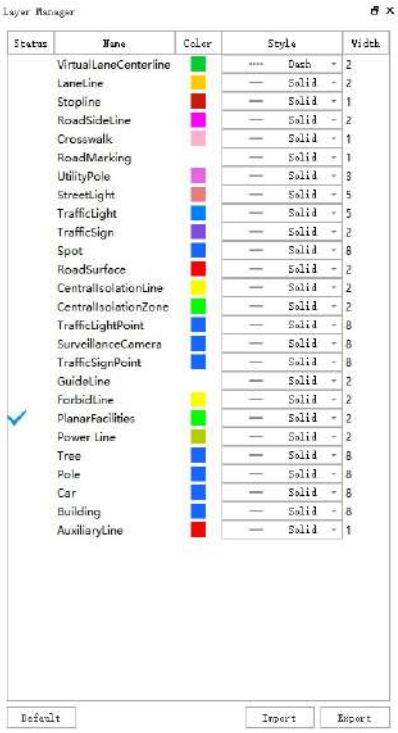
- Layer Manager
- Symbol
- Annotation
- Label Layers

Layer Manager

Function Description: This feature allows for modifying layer colors, line types, etc., and supports importing, exporting, and other operations on previously set layers.

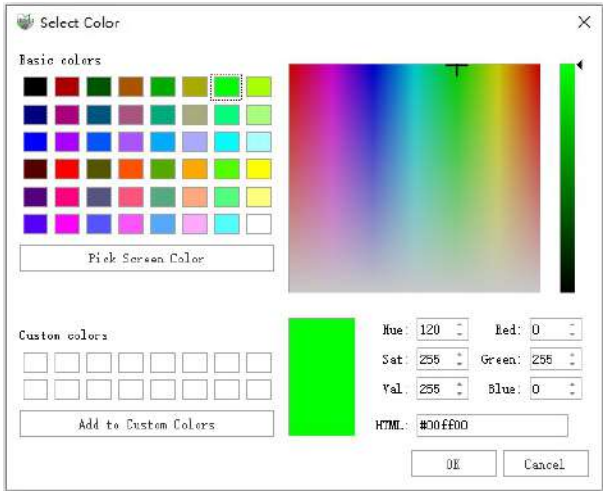
Steps

1.Click on **Layer Style Settings** button to pop up the **Layer Manager dialog box**



Parameter Description:

- **Status**: The selected vector object shows a checked state.
- **Name**: Display the name of each vector.
- **Color**: You can modify the color of the layer.
  - Double-click to pop up the settings dialog box.



- **Line Type**: You can modify the line type of the layer.
- **Line Width**: You can set the width of the line.
- **Import**: Import configuration files with the **.ini** extension.
- **Export**: Export the set layers and save them as configuration files with an **.ini** suffix.
- **Default**: Click this button to restore all parameters to their default values.

Symbol

Show Symbols

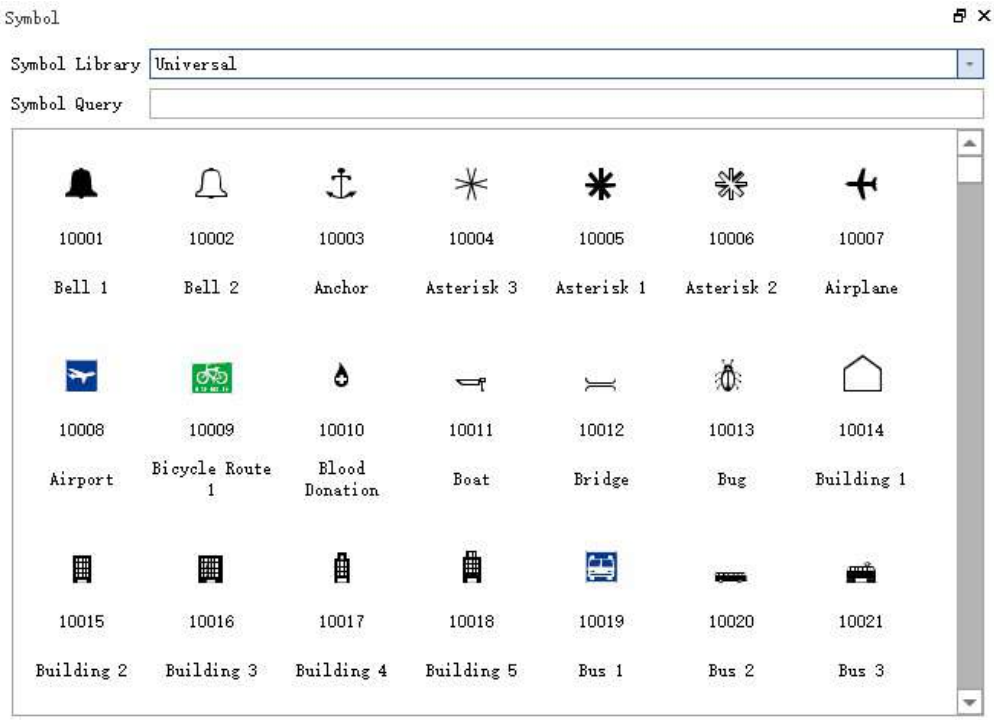
Feature Description: This tool allows the user to control the display and visibility of symbols, if point features in the project symbols have been set.

Code Table

Feature Description: This tool allows the user to open the existing built-in symbol library of the software.

Steps

1.Click **Code Table** button and the **Symbol dialog box** will pop up.



Parameter description:

- **Symbol Library**: Select Standard or Universal Mode
- **Symbol Query**: Enter via the text box to search for the symbol you want.

Add Feature

Feature Description: This tool allows the user to collect point features directly with symbols.

Steps

1.Click **Add Feature** button, **Code table** pops up.

2.Once you've determined which point feature layer to draw, select the appropriate symbol for your needs and collect it at the corresponding location on the point cloud.

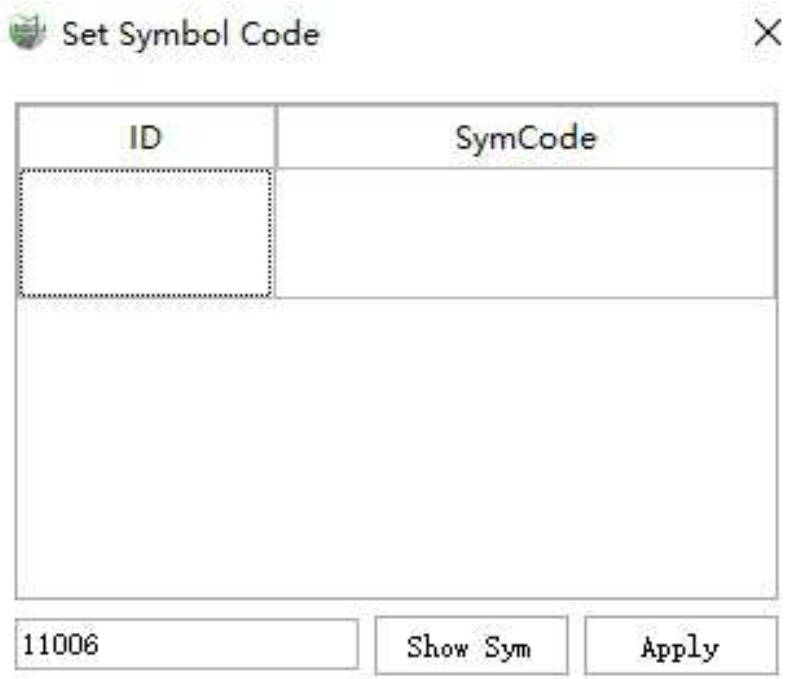
> Click **Show Symbol** button to display the icon of the corresponding symbol.

Set Code

Feature Description：This tool allows the user to modify existing symbols or set corresponding symbols for point features without symbols.

Steps

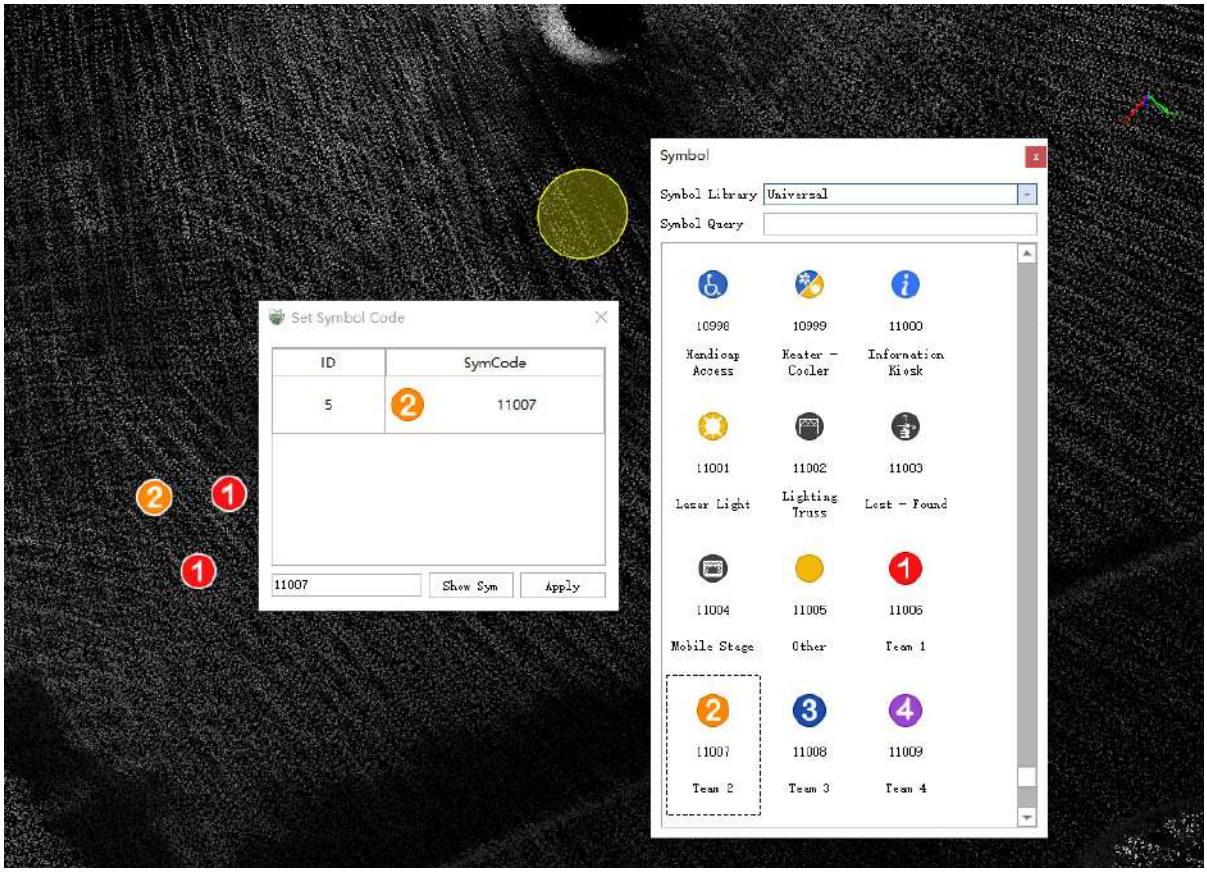
1.Click **Set Code** button and the **Set Symbol Code dialog box** will pop up.



Parameter description:

- **Show Sym**: Displays the symbol table and modifies the current symbol.
- **Apply**: Modifies the symbol.

2.Click the point feature on the point cloud to modify/set the symbol, the Set Symbol Code dialog box displays the feature ID and symbol code. You can re-select a new symbol in the symbol library or enter the code of a new symbol in the text box below the Set Symbol Code dialog box. Click Apply to complete the symbol modification or add new settings.


Set Symbol Code
ID
SymCode
5
11007
11007
Show Sym
Apply
Symbol
Symbol Library
Universal
Symbol Query
10998
Handicap Access
10999
Heater - Cooler
11000
Information Kiosk
11001
Laser Light
11002
Lighting Truss
11003
Lost - Found
11004
Mobile Stage
11005
Other
11006
Team 1
11007
Team 2
11008
Team 3
11009
Team 4

Annotation

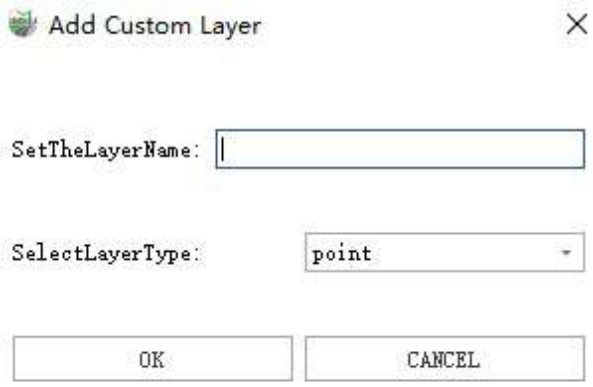
Add text annotation

Function Description: This tool allows the user to annotate the nodes/edges of point clouds, vectors, or vector objects.

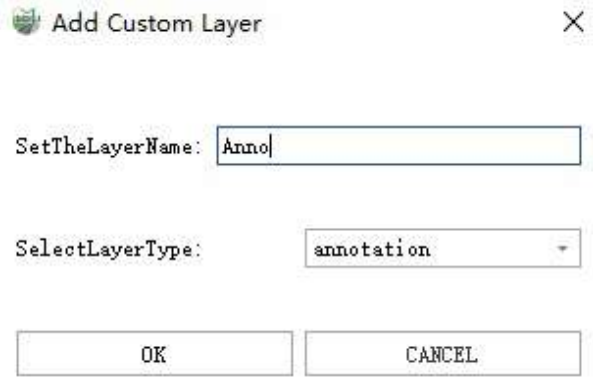
> This function will create a new annotation layer.

Steps

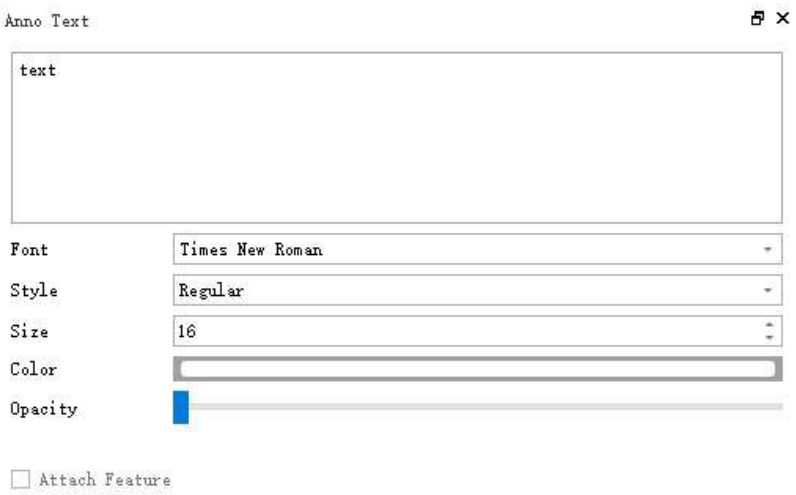
1.Custom vector > Right-click to add a layer, **Add Custom Vector** pops up.



2.Set the layer name as required, select the layer geometry type **Anno**, and click OK.

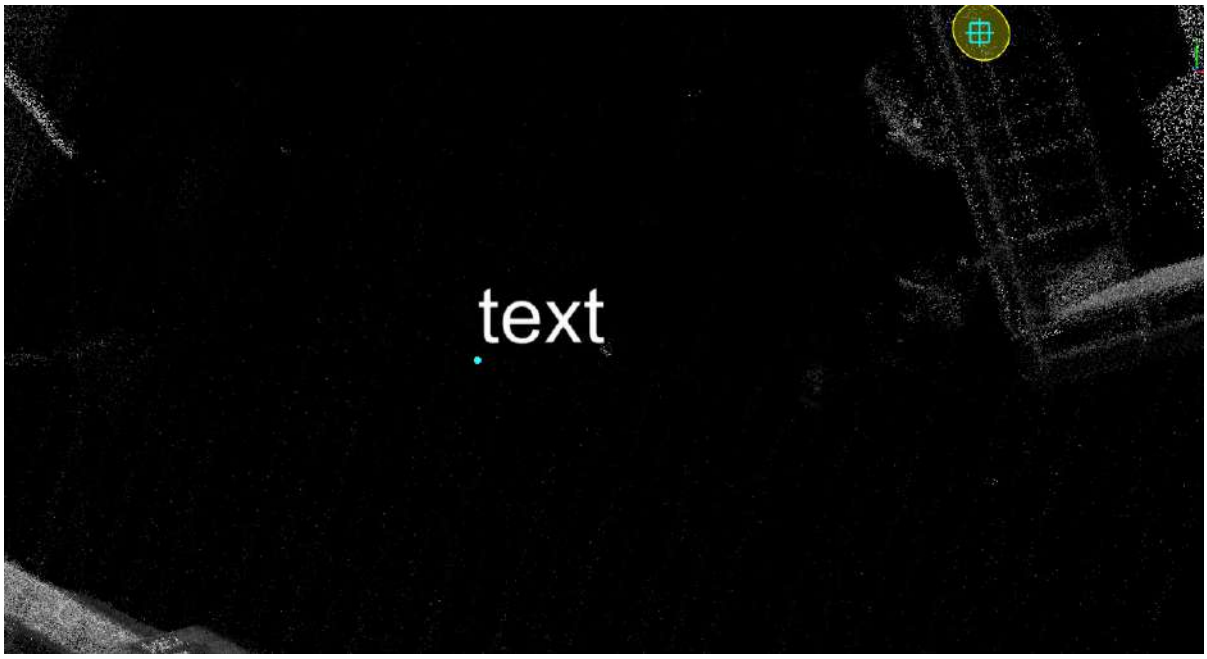


3.click **Add text annotation** button and the **Anno Text dialog box** will pop up.



Parameter description:

- **Font**: Sets current font.
- **Style**: Current font available styles.
- **Color**: Current font color.
- **Opacity**: Sets text transparency.
- **Attach Feature**: The current text is annotation for a feature or feature edge.
- **Apply**: Click to save the current settings.

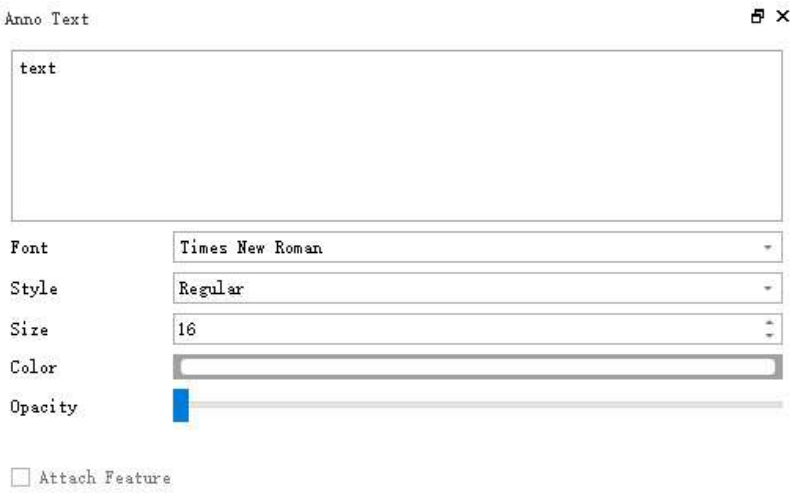


Modify Annotation

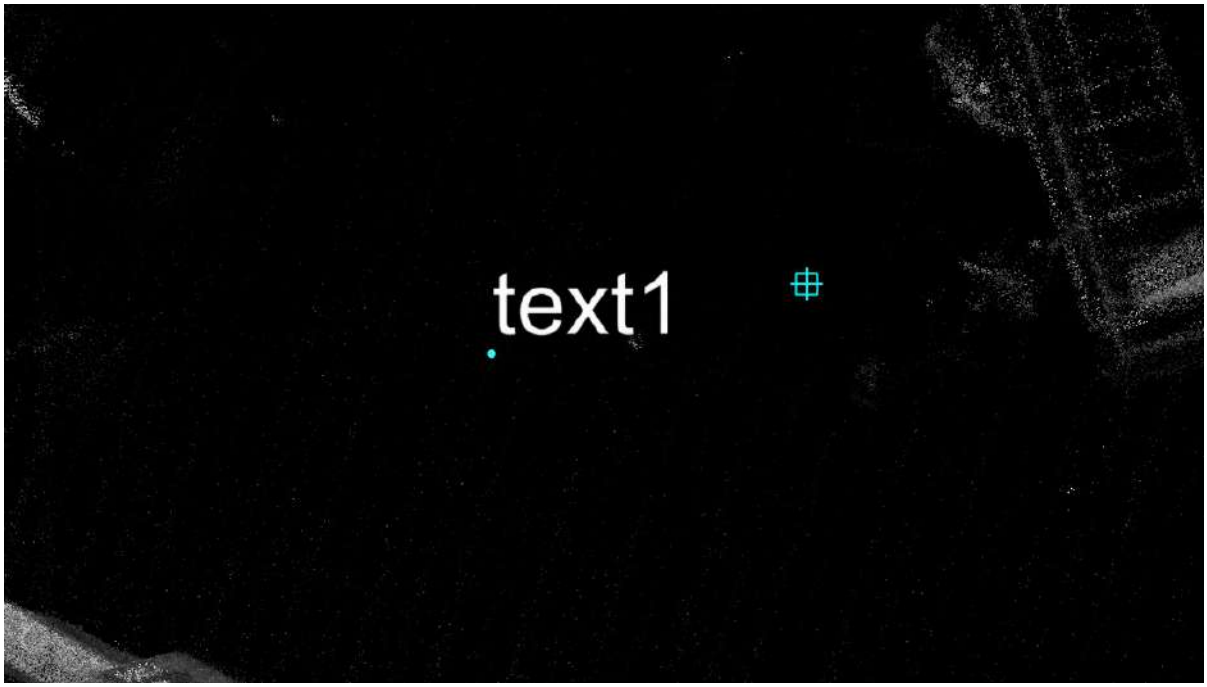
Function Description: This tool allows the user to modify existing annotations.

Steps

1.Click **Modify Annotation** button, click the annotation you want to modify, and the **Anno Text** dialog box will pop up.



2.Re-edit in the text editing box, set the font, color, Opacity, etc, and click Apply, you to "save" the current settings. Additional element information cannot be modified.



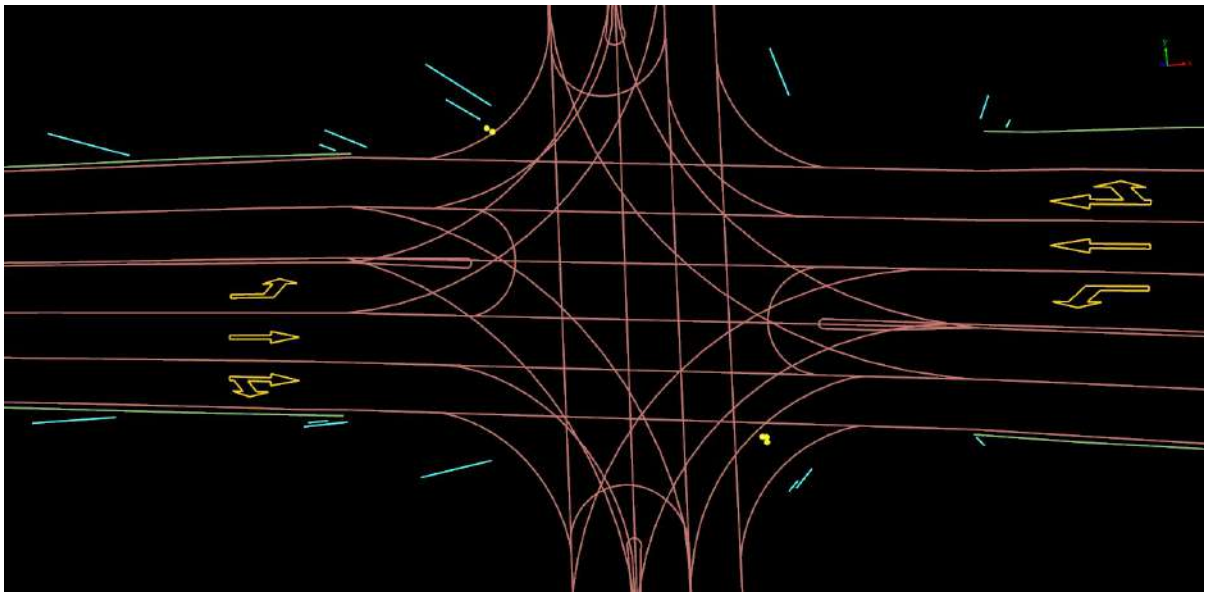
Label Layers

Function Description：This tool allows the user to set the font, size, color and other styles of the layer label, as well as the direction of the label and other operations.

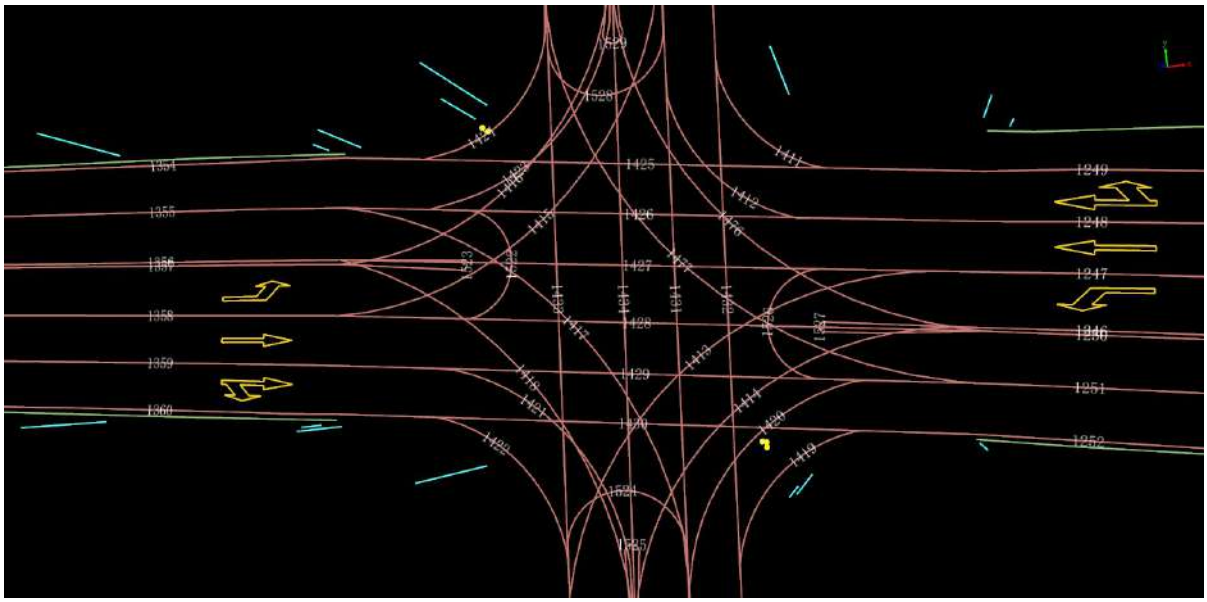
> Note: This function is used to ensure that there is vector result data in the project and that the attribute table field already has a value.

Steps

1.Right-click on the layer where the vector has been drawn in the directory tree and check the "Label" option to activate the display of the layer label.



Before the layer label is checked



After the layer label is checked

2.In the layer label menu bar, you can modify the layers and fields of the display label, and you can modify the style of the display label.

- **Layer**：Select the layer on which you want to display labels.
- **Field**：Select the corresponding field based on the layer.
- **Label Feature**：Shows or hides labels.
- **Label Direction**：Choose a view or scene.
- **Font**：Choose the style of the font.
- **Size**：Sets the size of the font.
- **Color**：Sets the color of the font.
- **Bold**：Sets the font to be bold.

> Note: Currently, label styles cannot be set by layer

Road Analysis

Road analysis is primarily used for various analytical tasks related to roads.

The main components of road analysis include:

- Road Condition Analysis
- Pavement Damage Analysis
- Viewshed Analysis
- Road Cross-Section Analysis

Road Condition Analysis

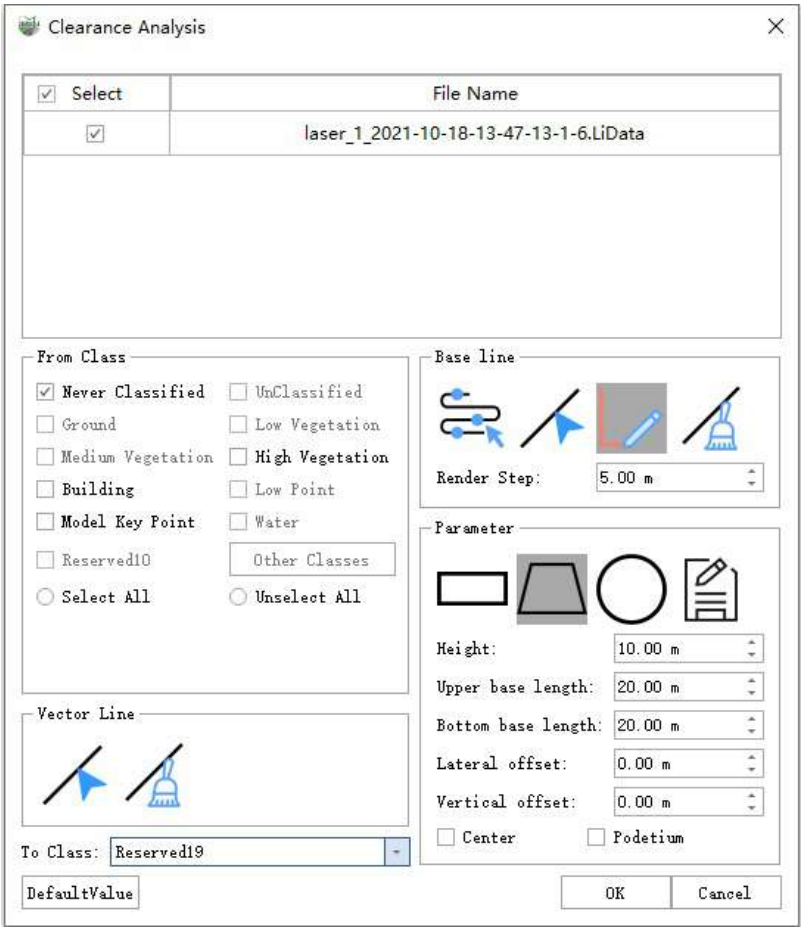
- Clearance Analysis
- Clearance Analysis Data Management
- Export Report
- Bridge Clearance
- Export Report
- Height Limit Analysis
- Export Report

Clearance Analysis

Function Description: This feature is used to calculate whether there are obstructions on the selected path. After selecting the template and analysis path, the analysis will mark the points within the clearance area as target classes, and the vector line will be marked in red. It supports the output of reports. A detailed introduction will be provided below.

Steps

1.Click the **Clearance Analysis** button, and the **Clearance Analysis Dialog Box** pops up.



2.Select the input file and point cloud class.

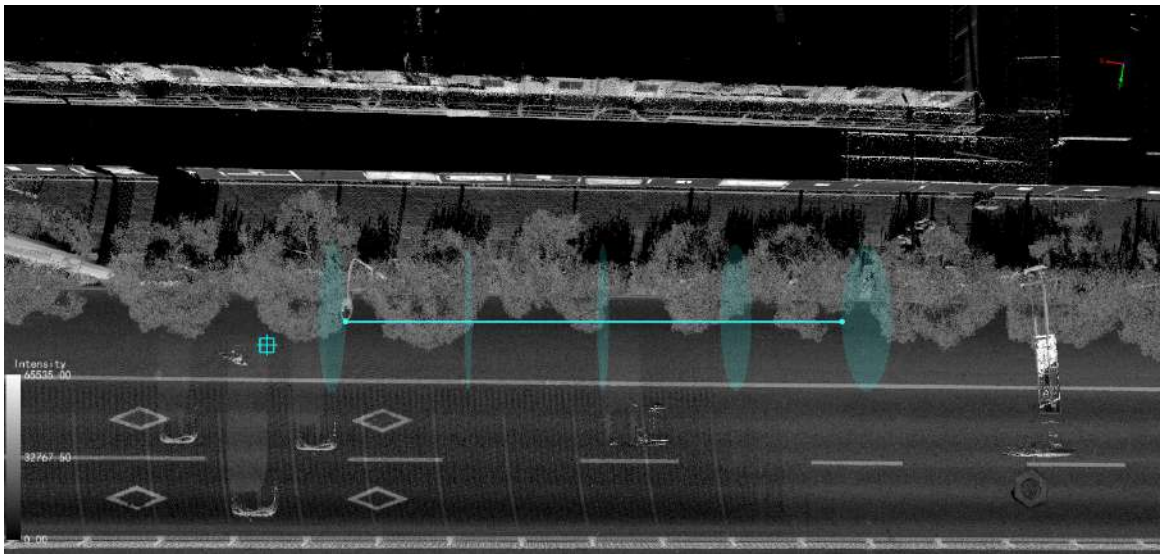
3.Select the vector lines involved in the calculation.

Parameter Description:

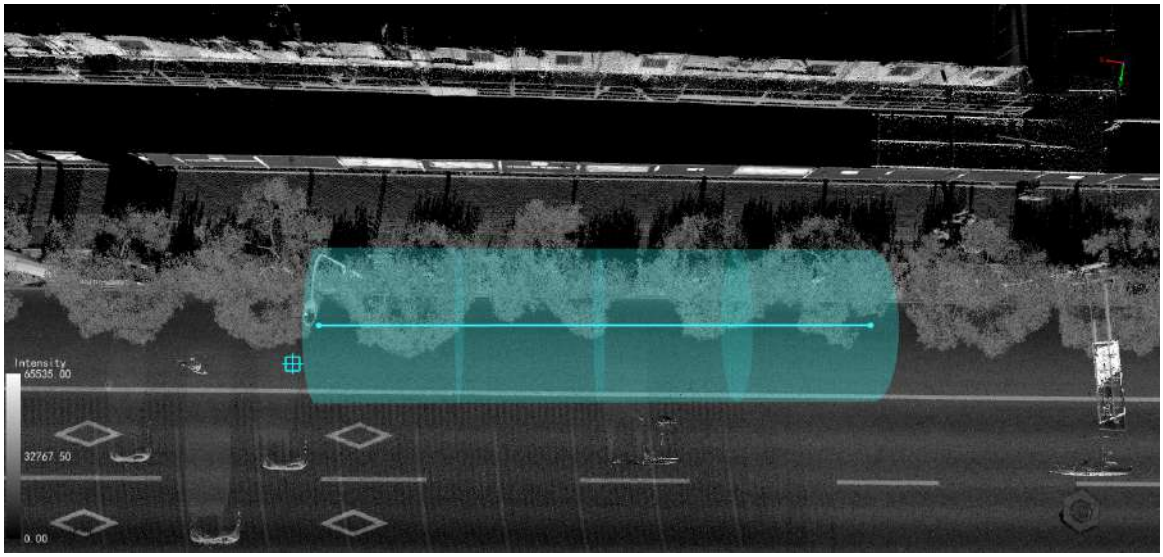
- **Select**: Select the point cloud file to participate in the calculation.
- **From Class**: The class involved in obstruction calculation.
- **To Class**: If a point is obstructed, its class is set to the target class.
- **Vector Line**: Select the vector line to participate in the calculation.
- **Base line**: It includes three modes: trajectory segment selection, existing vector line selection, and custom drawing line.
- **Parameter**: The distance between two display slices.

> Note: The slice position is not the actual vector node position.It is only used for display.

- **Unselected state**



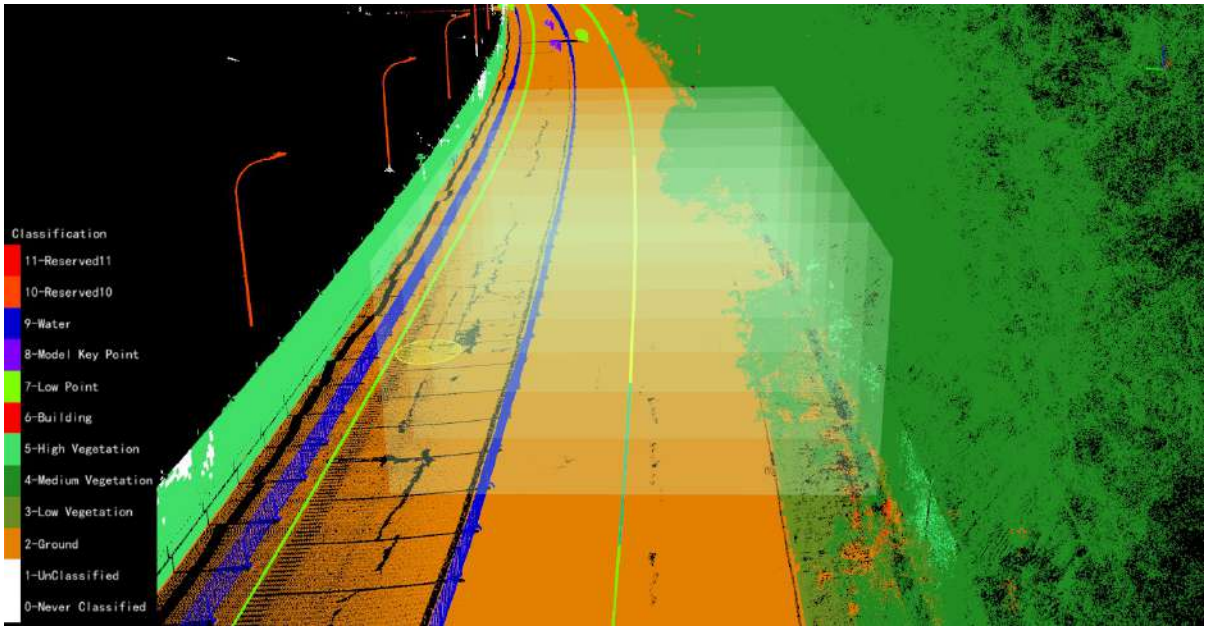
- **Selected state**



- **Default Value**：Click this button to restore all parameter defaults.

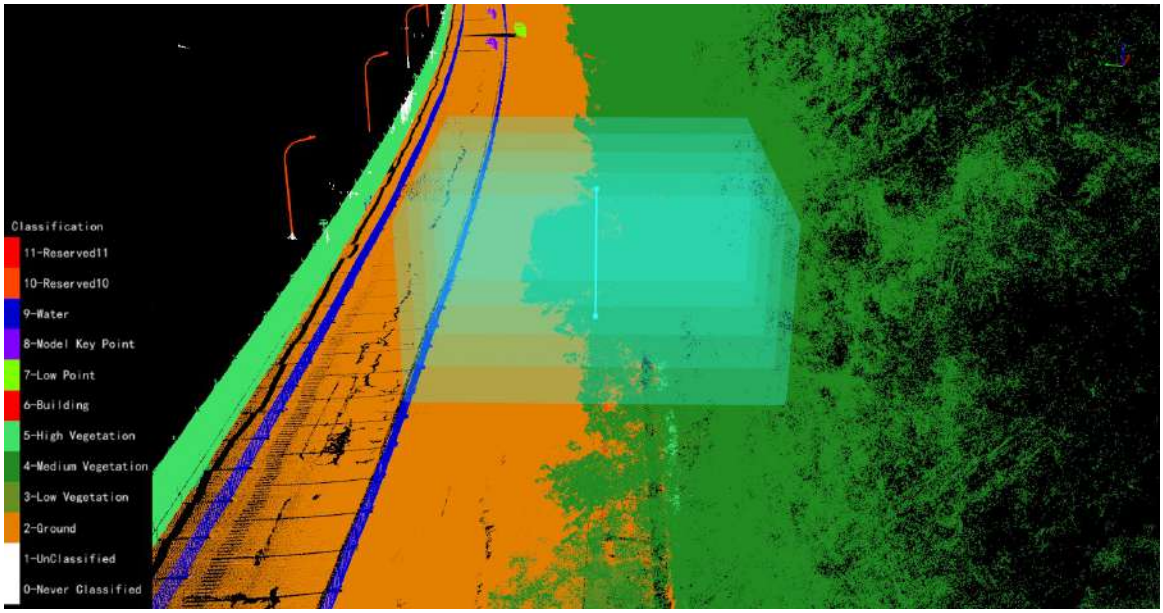
4.Choose the baseline (trajectory, vector line, custom line).

- **Select on Trajectory**：A trajectory file must be loaded when you creat a new project, and then click at the start position of the trajectory segment to be selected. Move the mouse and the trajectory will be selected in real-time. Click at the position where you want to end the selection, and then you can select a segment of the trajectory as the baseline. Multiple segments can be selected.



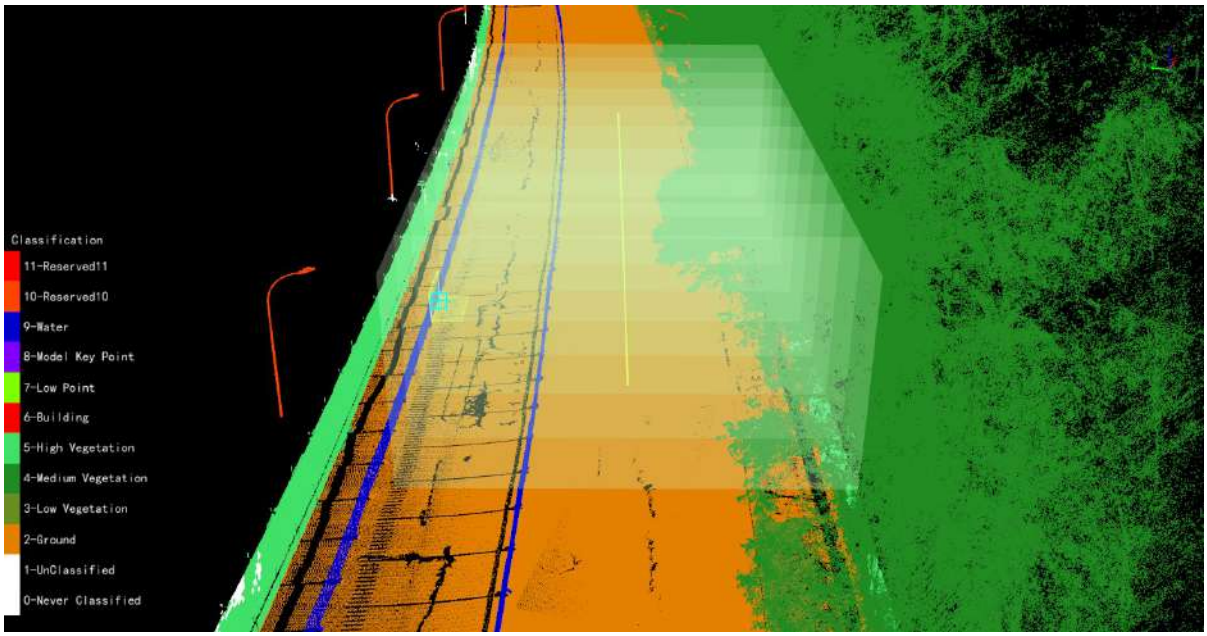
Observation point selection trajectory

- **Pick Base Line**：Choose one or more existing vector lines as the baseline.



Observation point selection vector line

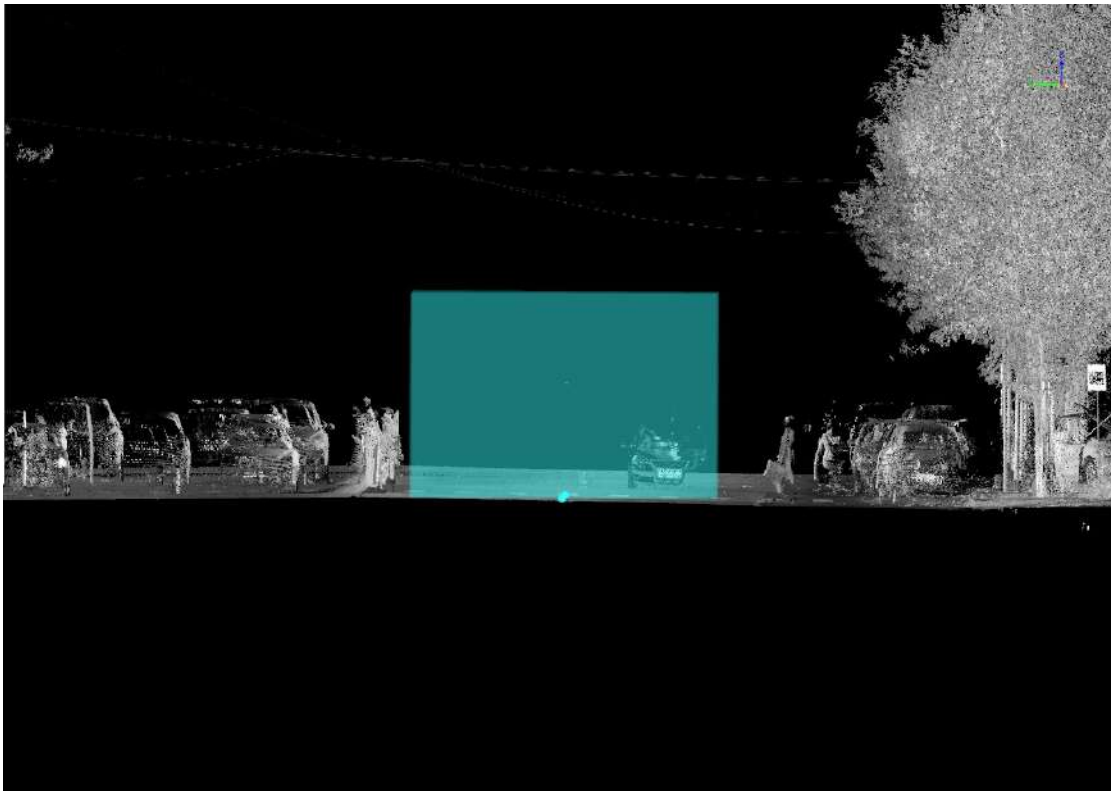
- **Draw Base Line**：After selecting the custom line, you can draw a temporary vector line at the position to be analyzed as a reference line.



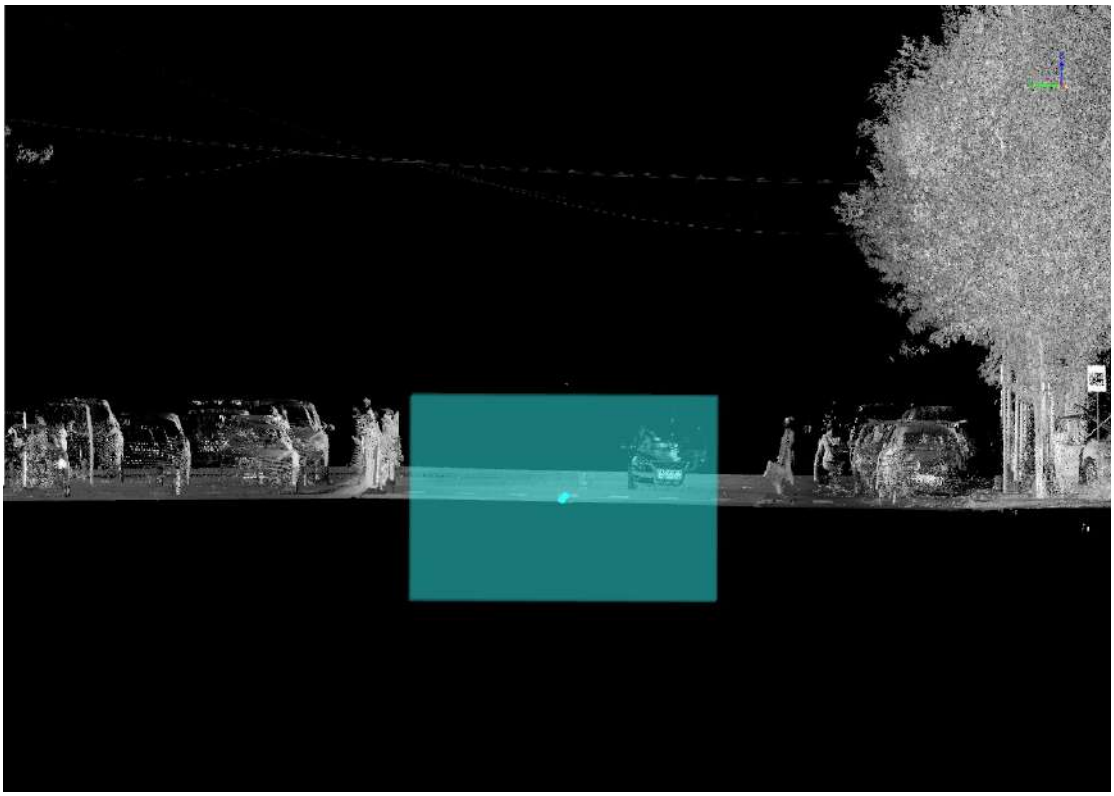
Observation point selection custom line

5.Choose a template (rectangle, trapezoid, circle, custom) and set parameters. Then click OK.

- **Rectangle**:
  - **If the center is not checked**: the middle point of the bottom edge of the rectangle is the specified center position.

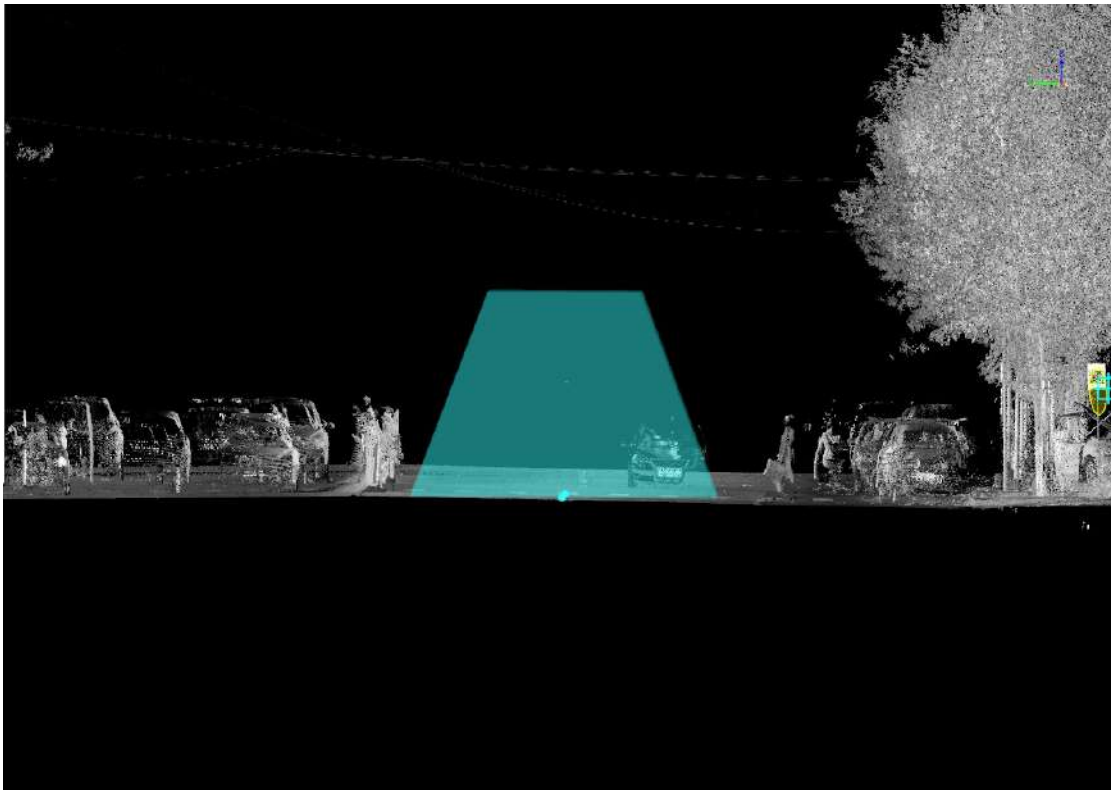


  - **If the center is checked**: the geometric center of the rectangle is the specified center

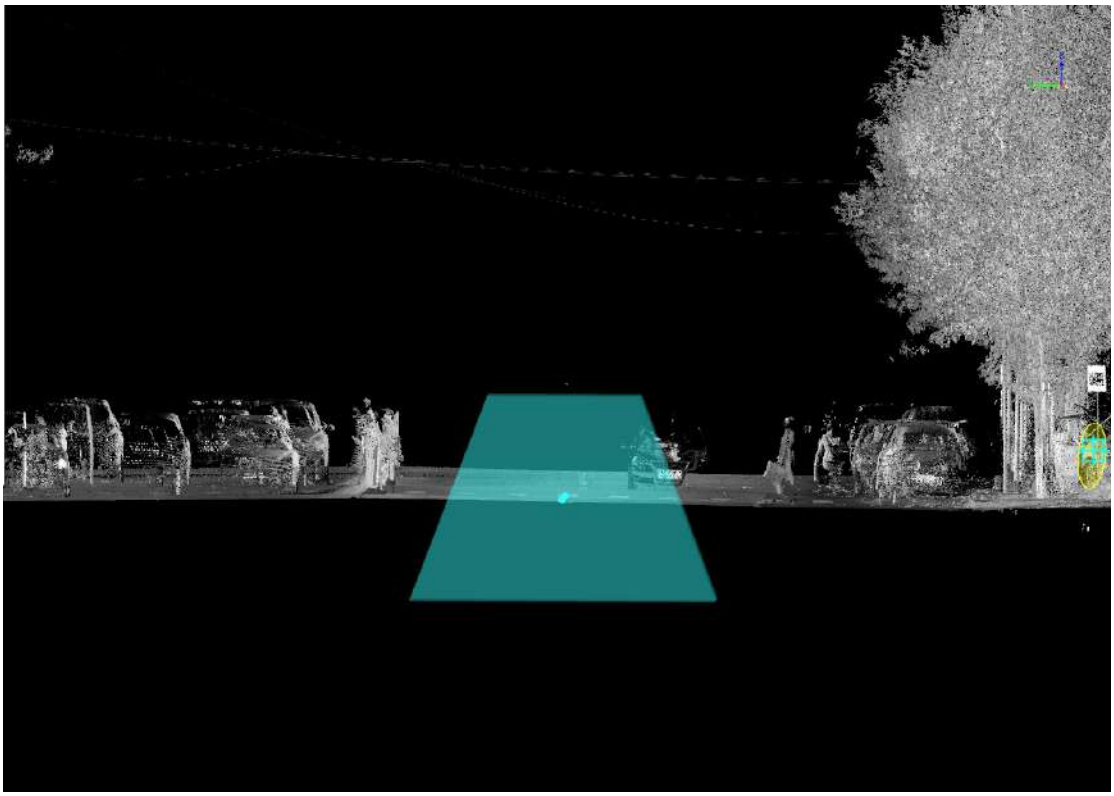


- **Height**: It is used to set the height of the analysis area rectangle.
- **Width**: It is used to set the width of the analysis area rectangle.
- **Lateral offset**: With the selected baseline as the reference line, it means the horizontal distance between the specified center position of the rectangle and the reference line. The left side of the forward direction of the reference line is a negative value, and the right side is a positive value.
- **Vertical offset**: With the selected baseline as the reference line, it means the vertical distance between the specified center position of the rectangle and the reference line. Above the reference line (Z-axis) is positive and below is negative.

- **Trapezoid**:
  - **If the center is not checked**: the middle point of the bottom edge of the trapezoid is the specified center position.



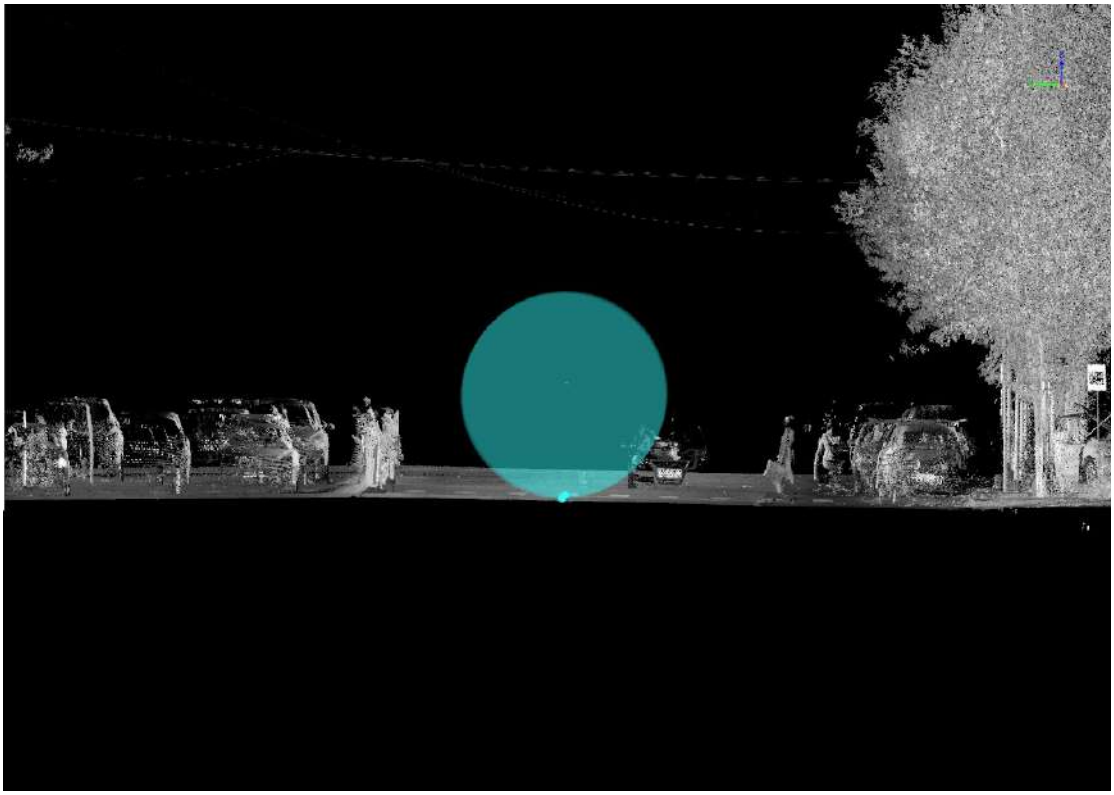
- **If the center is checked**: the geometric center of the trapezoid is the specified center position.



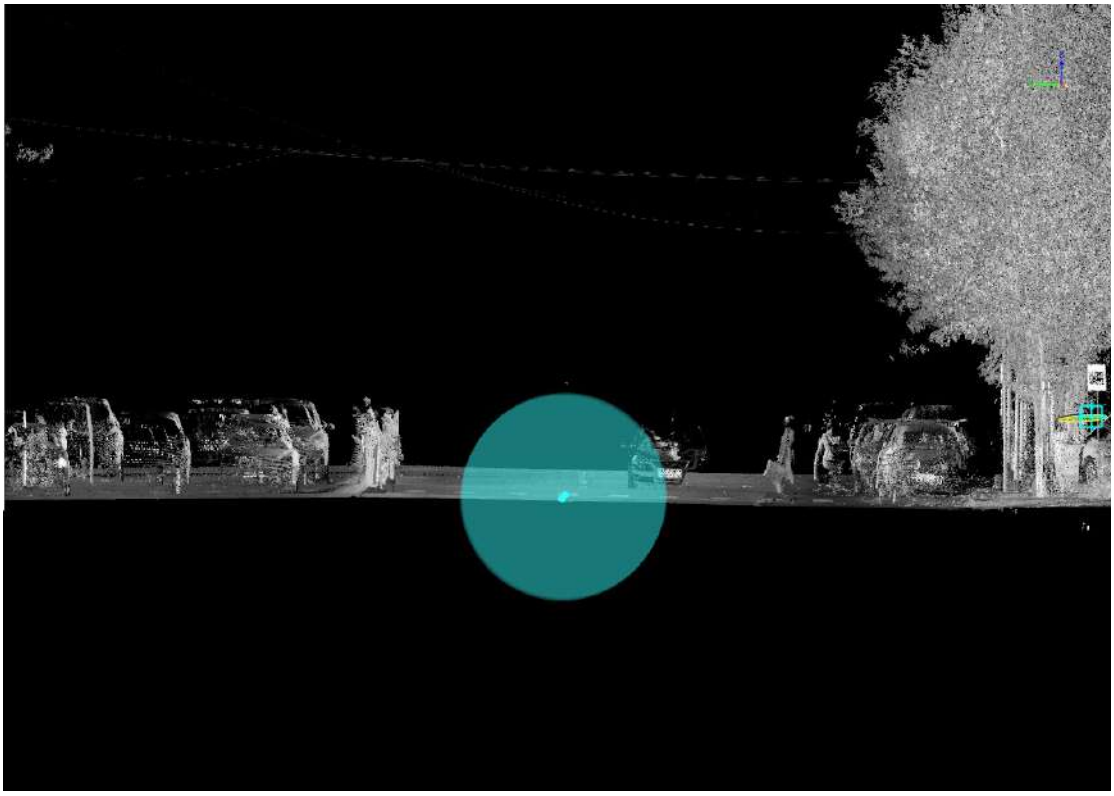
- **Height**: Similar to **Rectangle Mode**.
- **Upper Base Length**: Used to set the width of the analysis area trapezoid.
- **Bottom base leneth**: Used to set the width of the analysis area trapezoid.
- **Lateral offset**: Same as rectangle.

- **Vertical offset**：Same as rectangle.

- **Circle**：
  - **If the center is not checked**: the lowest position of the circle is the specified center position.



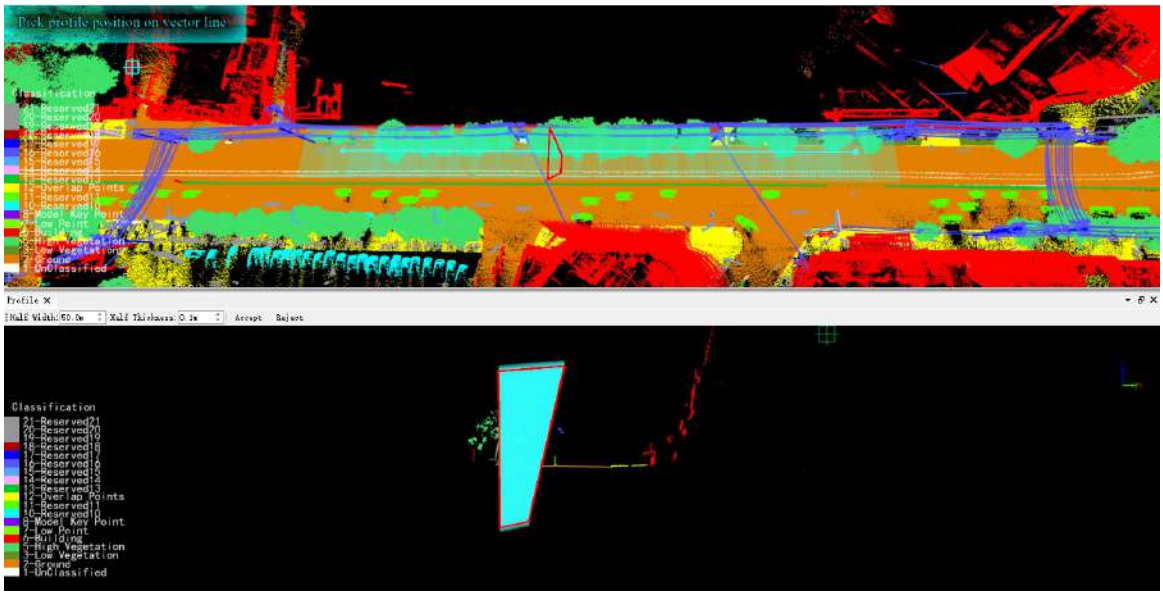
  - **If the center is checked**: the center of the circle is the specified center position.



  - **Radius**：The radius of the circle mode.

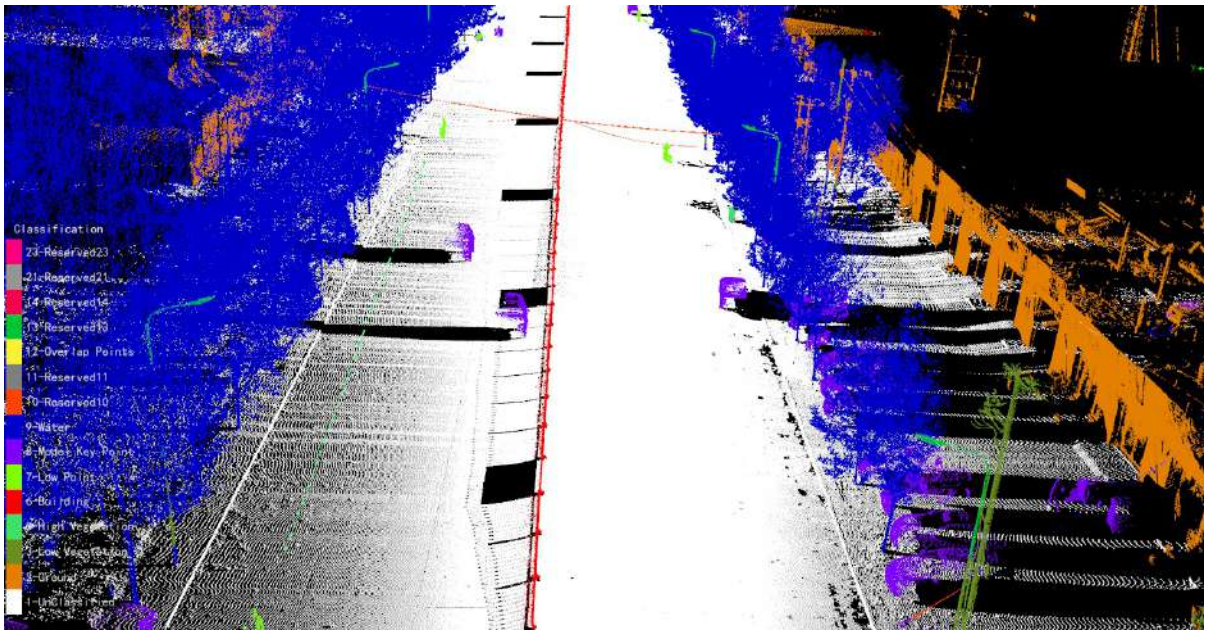
- **Lateral offset**：Same as rectangle.
- **Vertical offset**：Same as rectangle.

- **Custom**：Select a point on any vector line (generally select the current baseline in business). The profile window will display the profile along the tangent direction, and you can draw a template polygon in this window.

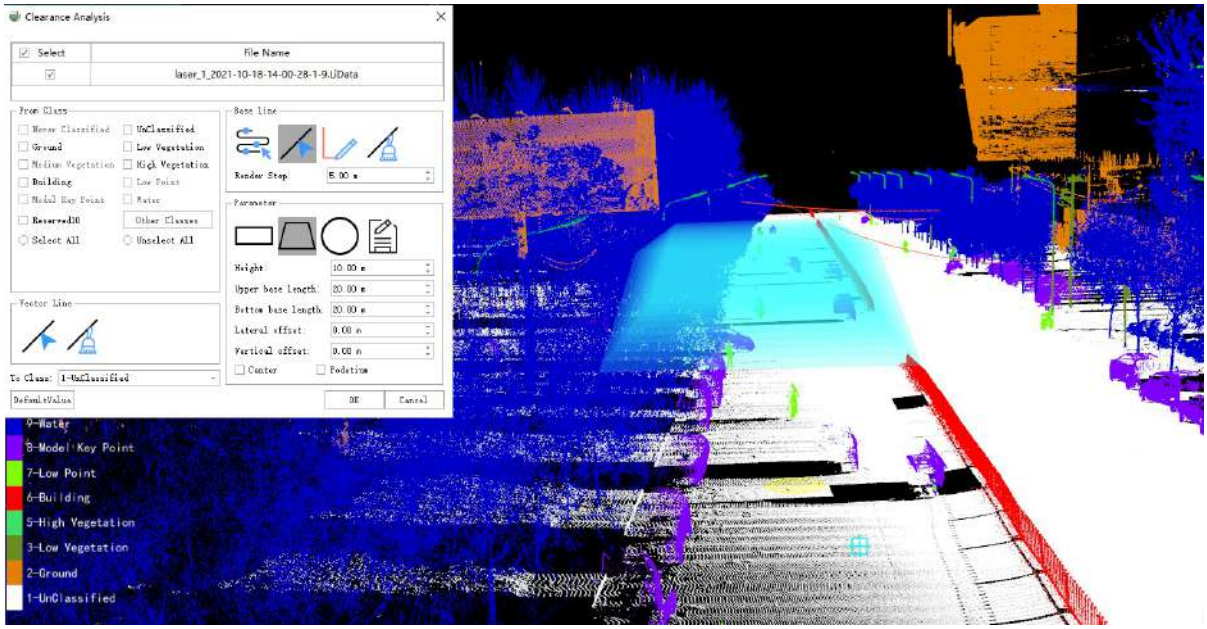


Window parameters are as follows：

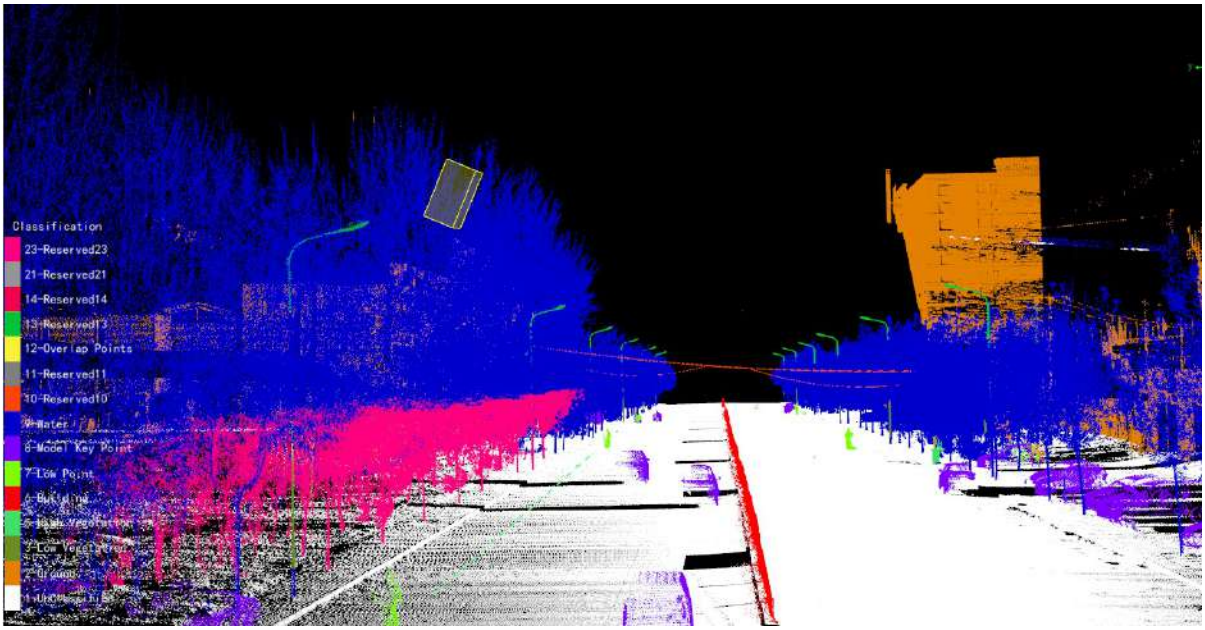
- **Half Width**: The half width of the point cloud profile display.
- **Half Thickness**: The half thickness of the point cloud profile display.
- **Length**: The upper left corner of the window will display the distance between the current point and the last point in real time.
- **Accept**: Accept the drawn result.
- **Reject**: Reject the drawn result.
- **Center**: Refer to the three modes above.
- **Podetium**: If it is not selected, the clearance area will be displayed as a slice. If selected, it will be displayed as a cylinder.



Before enabling clearance analysis

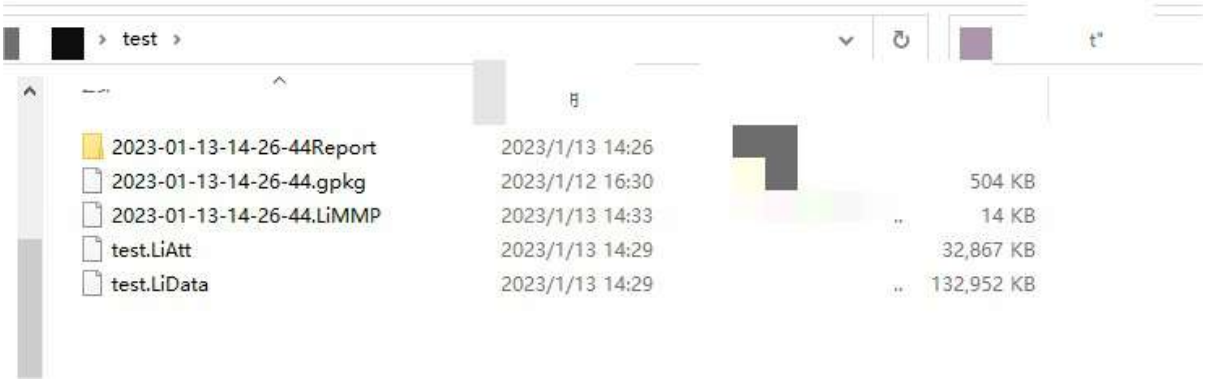


After selecting the template and setting parameters



Result after analysis

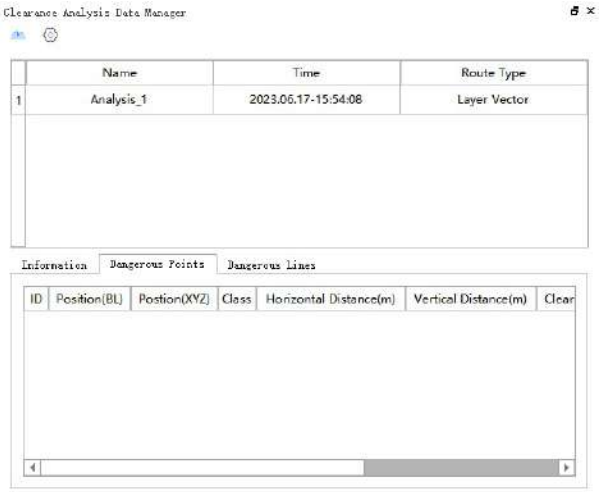
6.After the analysis is over, the view will automatically jump to the top view of the selected area, and a **report folder** is automatically generated. The report folder is in the same directory as the current project file, and the naming rule is: project name + Report.



Report Folder

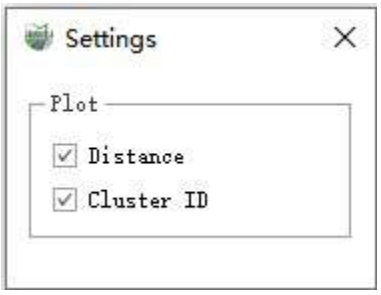
Clearance Analysis Data Manager

Function Description: Used for the display and management of clearance analysis results, it is divided into a result list and a message information table for each result.



Toolbar

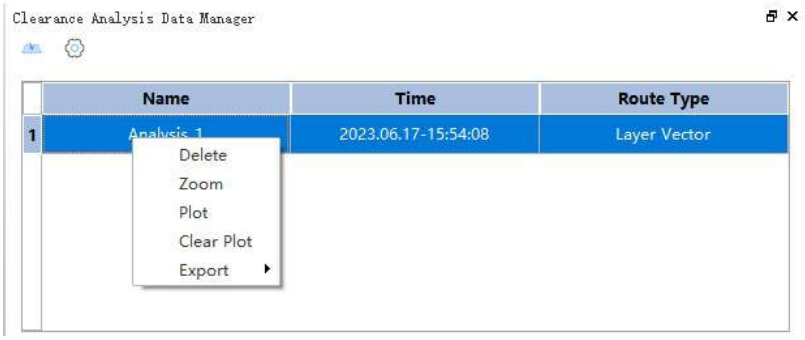
- **Import**: Import the clearance analysis results.
- **Settings**: Provide settings related to drawing.



Results List

Each clearance analysis result is displayed as a record in the list, showing the number, time, and baseline type information.

Right-click on any analysis record to open the menu.

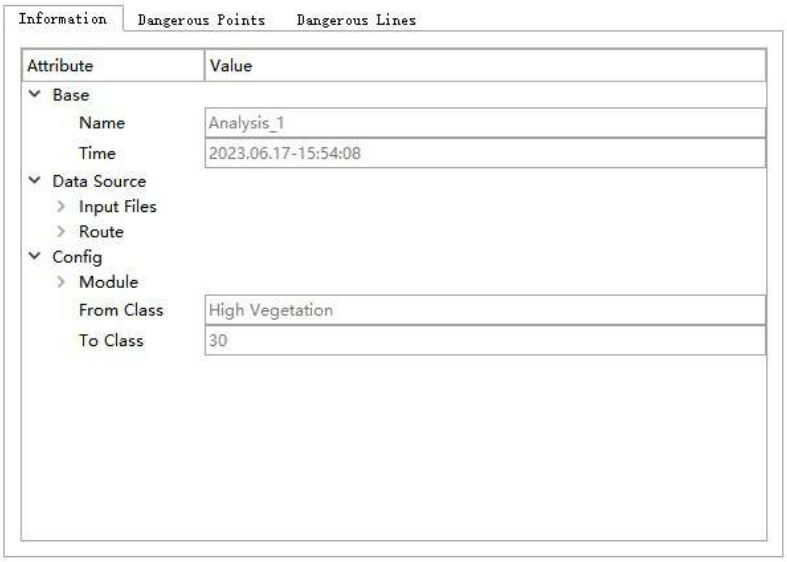


- **Delete**: Delete this record.
- **Zoom**: Zoom to the analysis path.
- **Plot**: Draw analysis results, including each dangerous cluster ID and the shortest distance from the cluster to the path.
- **Clear Plot**: Clear all drawing results.
- **Export Analysis**: Export the current record to a text file.
- **Export Report**: Export the current record to a report.

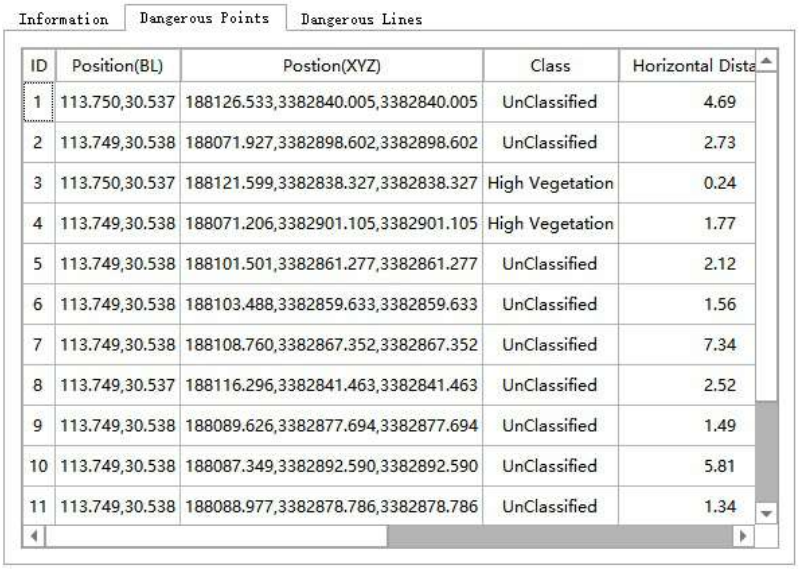
Detailed Information

Provide a display of path basic information, dangerous points and dangerous vector lines.

- **Information**: Including analysis time, analysis settings and other basic parameters.

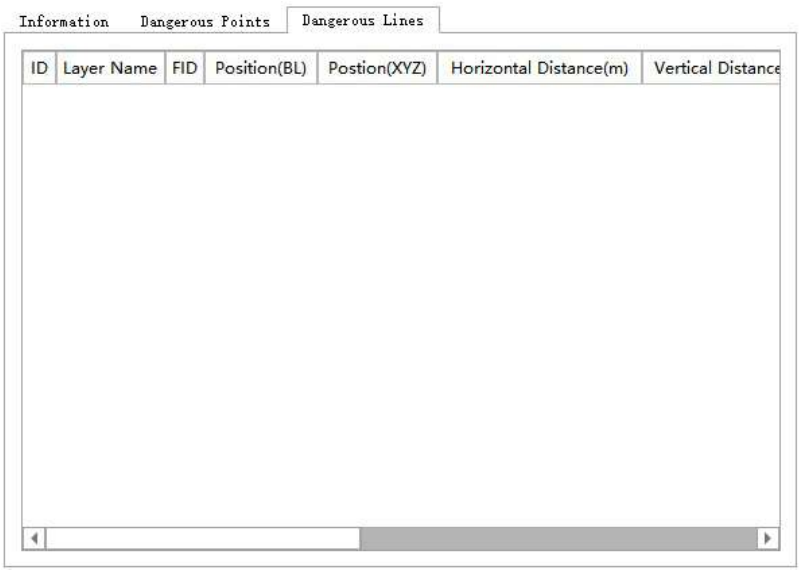


- **Dangerous Points**: Dangerous point clusters, including location, category, distance and other information.



ID	Position(BL)	Postion(XYZ)	Class	Horizontal Dista
1	113.750,30.537	188126.533,3382840.005,3382840.005	UnClassified	4.69
2	113.749,30.538	188071.927,3382898.602,3382898.602	UnClassified	2.73
3	113.750,30.537	188121.599,3382838.327,3382838.327	High Vegetation	0.24
4	113.749,30.538	188071.206,3382901.105,3382901.105	High Vegetation	1.77
5	113.749,30.538	188101.501,3382861.277,3382861.277	UnClassified	2.12
6	113.749,30.538	188103.488,3382859.633,3382859.633	UnClassified	1.56
7	113.749,30.538	188108.760,3382867.352,3382867.352	UnClassified	7.34
8	113.749,30.537	188116.296,3382841.463,3382841.463	UnClassified	2.52
9	113.749,30.538	188089.626,3382877.694,3382877.694	UnClassified	1.49
10	113.749,30.538	188087.349,3382892.590,3382892.590	UnClassified	5.81
11	113.749,30.538	188088.977,3382878.786,3382878.786	UnClassified	1.34

- **Dangerous Vector Lines**: Including location, class, distance and other information.

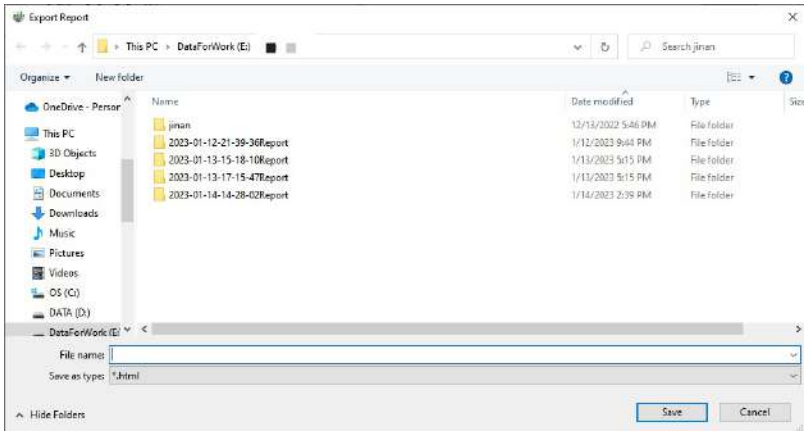

Information
Dangerous Points
Dangerous Lines
ID
Layer Name
FID
Position(BL)
Postion(XYZ)
Horizontal Distance(m)
Vertical Distance

Export Report

Function Description: For clearance analysis, the clearance analysis data management window can be used to export the report of one or more analysis results or the overall report of all analysis results.

Steps

1.Method 1: Click the dropdown box next to **Clearance Analysis** and select **Export Report** button. A save file dialog box will pop up. Then choose the report file name and the location to save.



Save file dialog box

2.Method 2: Click the dropdown box next to **Clearance Analysis**, and select **Clearance Analysis Data Manager** button. In the manager, select one or more data rows and then right-click **Export->Export Report** button so that the save file dialog box will pop up. You can choose the report file name and the location to save.

Save file dialog box

3.The report content is divided into the header, table of contents and specific content.

(1)Header and Table of Contents

Pavement Damge Report

GreenValley International

Generated in2023.01.13

Catalogue:

Region 1

Region 2

Header and Table of Contents

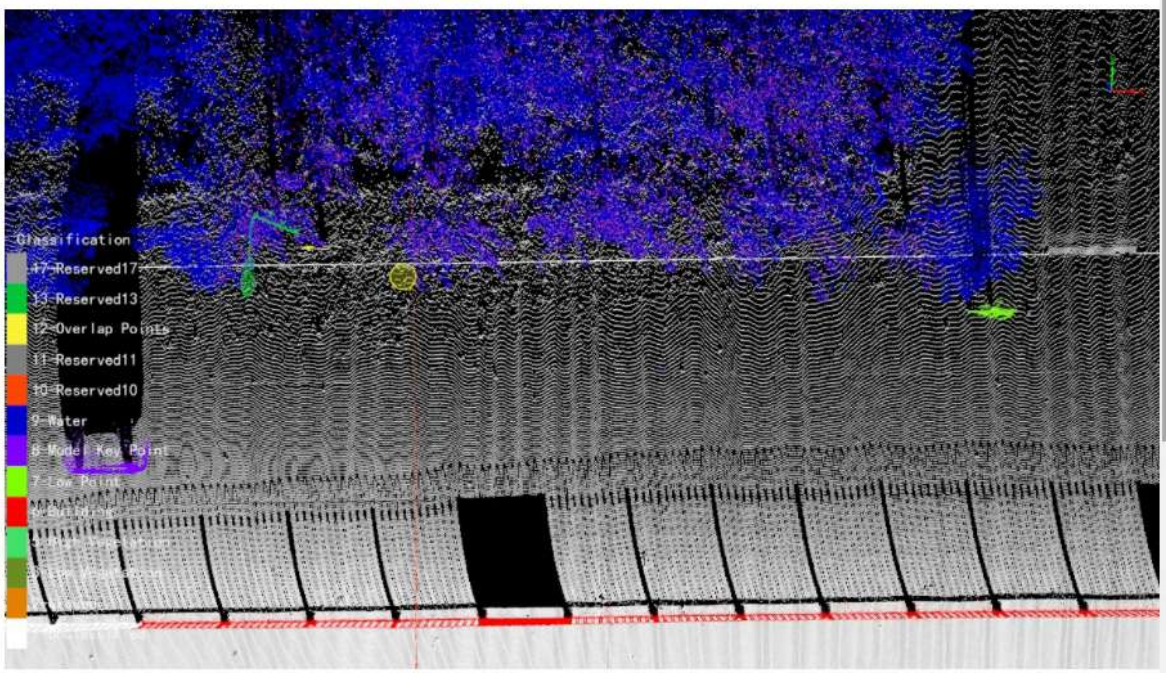
Click on the Region in the table of contents to jump to the corresponding location.

(2)Specific Content

Region 1

Input Parameters:

Generated Time	Generated in2023.01.13 18:36:47
Files	2018-12-03-10-18-01-6.LiData
From Class	9、
Base Line Type	Custom Line
Render Step	5.000000
Base Line Position	Bottom Middle
Pattern	Rect
Height	10.000000
Width	20.000000
Lateral Offset	0.000000
Vertical Offset	0.000000



Output:

Total Points	2522025
Danger Points	119407
Danger Points Proportion	4.734568%
Danger Cluster Center	593605.875000 , 4211417.000000 , 32.273766 593603.562500 , 4211417.500000 , 27.642365 593597.187500 , 4211419.500000 , 27.797459 593601.312500 , 4211414.000000 , 34.121826 593608.562500 , 4211410.500000 , 29.096058 593613.812500 , 4211421.500000 , 35.059727 593604.437500 , 4211416.500000 , 27.483723 593608.125000 , 4211416.500000 , 27.471399 593601.312500 , 4211412.000000 , 33.613480 593611.562500 , 4211419.500000 , 37.170547

Overall image of specific content

Content Explanation：Each analysis result is stored in levels, and one analysis result content includes input parameters, all dangerous point cluster list information, and detailed information about each dangerous point cluster.

- **Input Parameters**：The parameters used in this detection
  - **Cloud Files**: The LiData file name used in this detection
  - **From Class**: The source category used in this detection
  - **To Class**: The target category used for detection
  - **Base Line Type**: Baseline type, including Trajectory, Vector Line, and Custom Line three types
  - **Base Line Length(m)**: Baseline length
  - **Layer Name**: The model types include: rectangle, trapezoid, circle, and custom.
  - **Module Type**: A template type
  - **Radius, Height, Width, Module Center, Horizontal Offset, Vertical Offset**: Represent height, width, horizontal offset, and vertical offset, respectively. For details, refer to Clearance Analysis
- **All Dangerous Point Cluster List Information**: Include each point cluster's number, location, category, horizontal projection plane, closest horizontal distance to the baseline, vertical distance and clearance distance.
- **Detailed Information of Each Dangerous Point Cluster**: Include each point cluster's number, location, category, horizontal projection plane, closest horizontal distance to the baseline, vertical distance and clearance distance.

Bridge Clearance

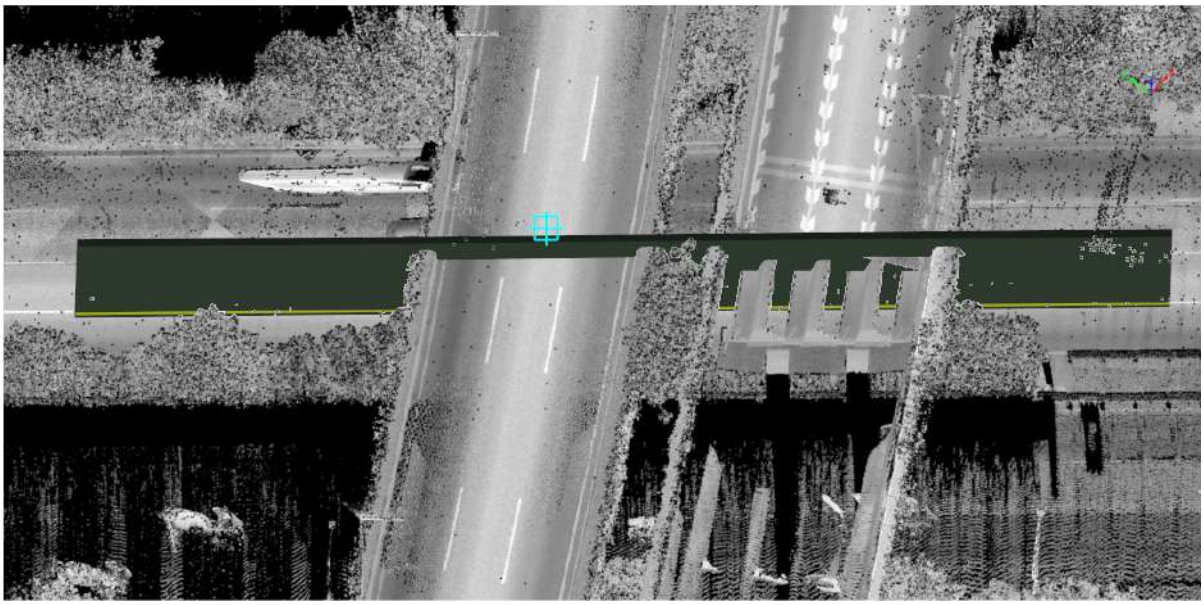
Function Description: This feature is used for the height analysis of bridges and can guide vehicle passage.

Function Description

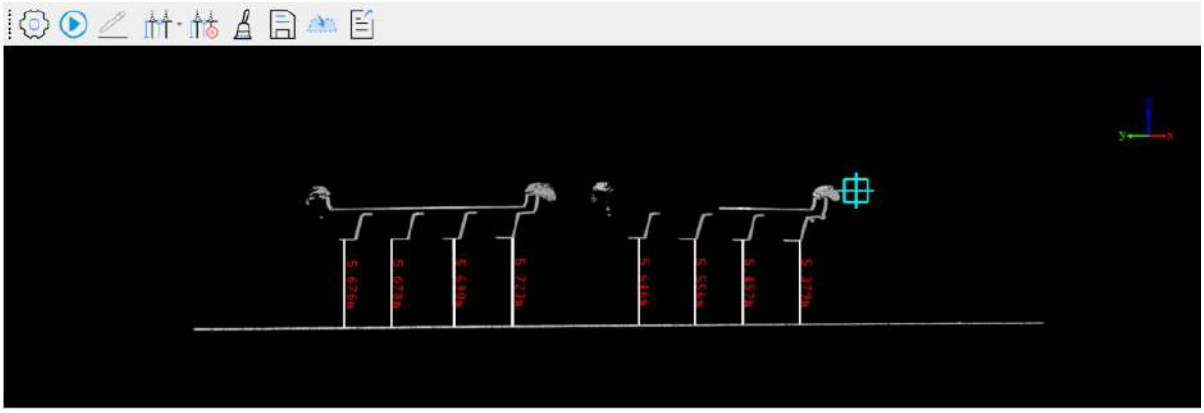
It provides the measurement function from the bridge to the ground, including manual measurement and automatic measurement, and supports exporting measurement results and output reports.

Steps

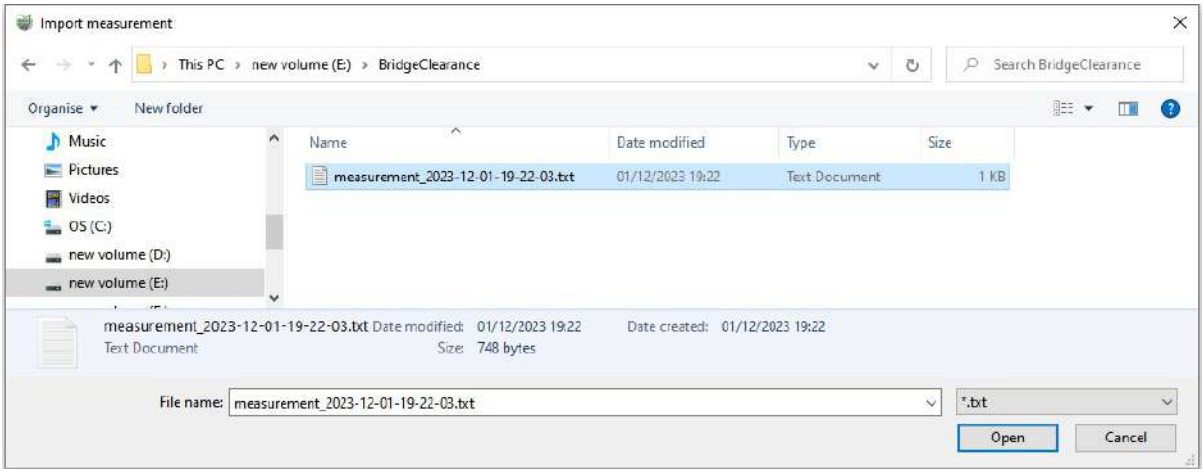
Click on **Bridge Clearance** button, Firstly, click two points on the ground in the 3D window to determine the bridge section, and then you can use the measurement tool to measure:



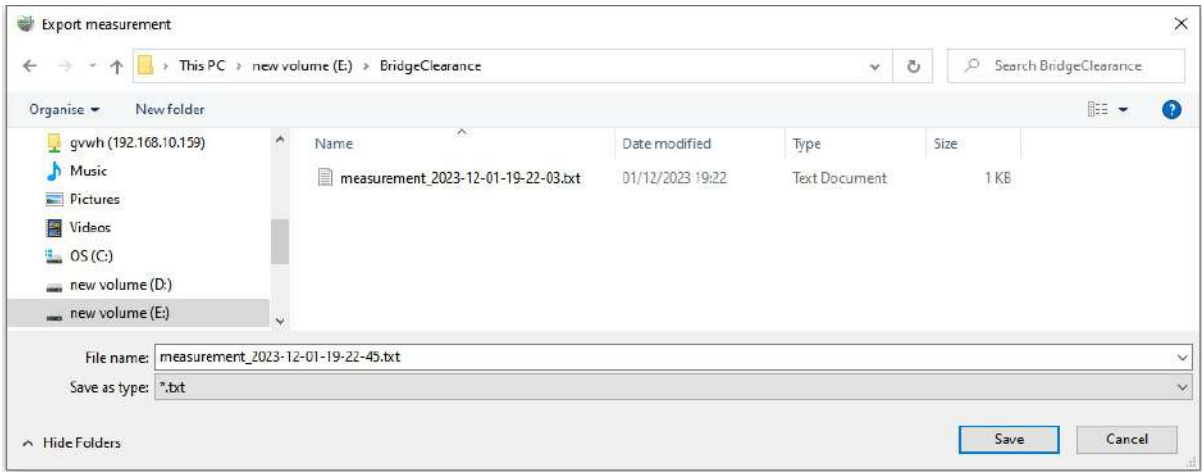
- **Auto Measure**：The program automatically recognizes bridge points and ground points and obtains measurement results automatically based on measurement parameters.



- **Start Edit**: After clicking, you can manually measure (you can also manually measure after clicking 'Automatic Measurement'). Note that if the program fails to recognize the bridge and ground, only two-point measurements can be performed.
- **Manual Measurement**: There are three modes: Single Measurement, Double Measurement and Interval measurement.
    - **Single Measurement**: Click on any point on the view bridge or ground to get the measurement result.
    - **Double Measurement**: Firstly click a point on the view bridge or ground, and then click the second point on the opposite side of the height to get the measurement result.
    - **Interval Measurement**: Firstly click a point on the view bridge or ground as the starting point, and then click the second point to confirm the endpoint horizontally. The measurement result will be generated at equal intervals between these two points.
- **Delete Measurement**: Delete a single measurement result.
- **Clear Measurement**: Clear all measurement results.
- **Save**: Save the measurement results to the project directory.
- **Import**: Import measurement results.

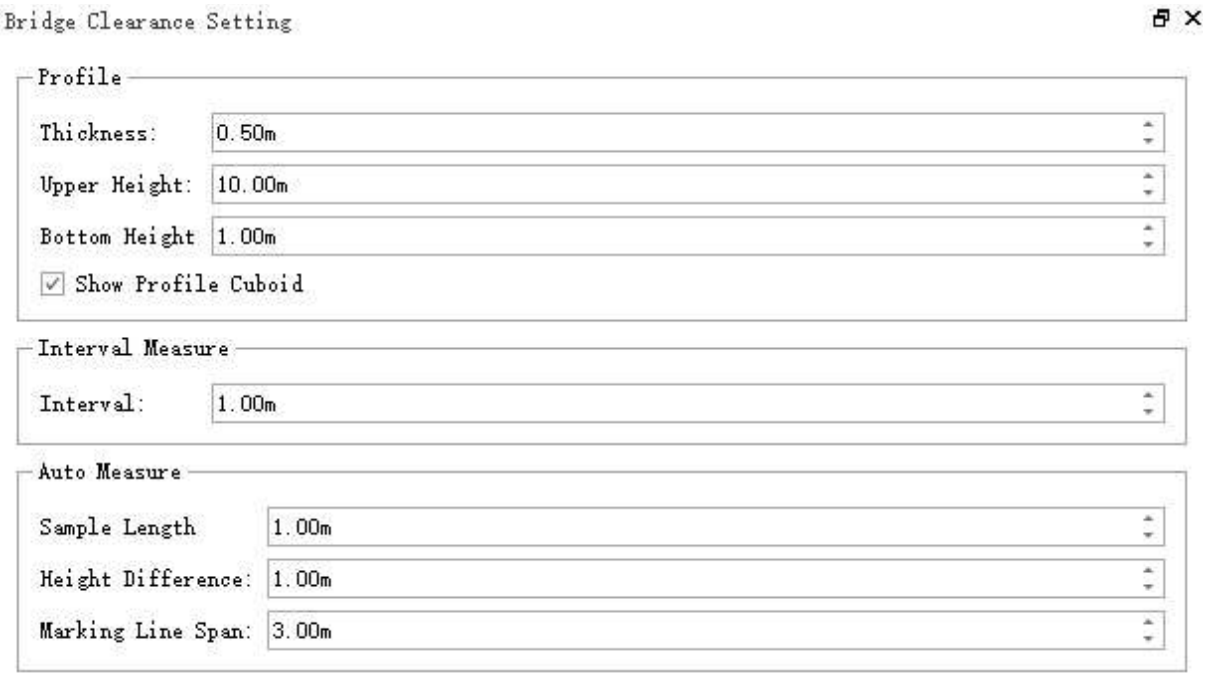


- **Export**: Export measurement results.



Function Parameters

Click the **Settings** button, as shown below



- **Profile**：Parameters related to the bridge section. Only the point cloud in the section will be involved in the measurement.
  - **Thickness**: Section thickness.
  - **Upper Height**: Upper height of the section.
  - **Bottom Height**: Lower height of the section.
  - **Show Frofile Cuboid**: Check to display the section body in the 3D window, and uncheck to hide it.
- **Interval Measure**：Parameters related to equal interval measurement.
  - **Interval**: Interval value.
- **Auto Measure**：Parameters related to automatic measurement.
  - **Sample Length**: The point cloud of this length is intercepted at the measurement position for bridge point extraction.
  - **Height Difference**: Points which are larger than the height difference are regarded as road surfaces. The parameter is used to identify bridge points.
  - **Marking Line Span**: Automatic measurement spacing.

Export Report

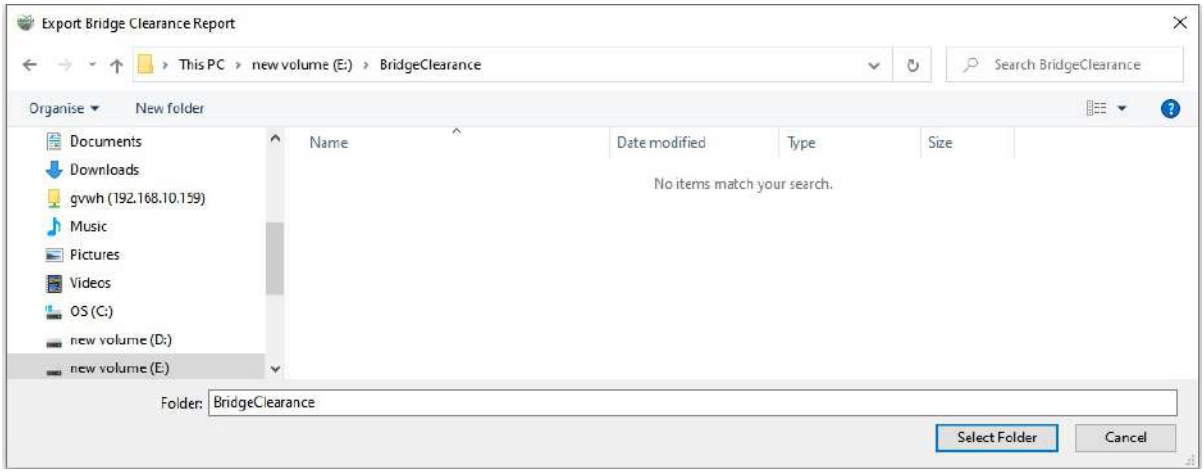
Function Description: Generate an overall report file based on the results of each previous analysis.

Function Description

Related data is stored in the 'BrigeClearance' folder in the project directory. Before generating the report, make sure the data has not been deleted.

Steps

1.Click the drop-down box next to **Bridge Clearance** and select the **Export Report** button. The save file dialog box pops up to select the report file name and the location to save it.



Save File Dialog

2.The report content is divided into three parts: header, content, and specific content

Header and Table of Contents

Header and Table of Contents

Click on BridgeClearance_ID under the directory to jump to the corresponding location.

Specific Content

Measurement Parameter

Profile	Thickness(m)	0.500
	Upper height(m)	10.000
	Bottom height(m)	1.000

Clearance Screen Shot

Clearance Height Statistical Table

Number	Measurement Mode	Parameter(m)				Node Coordinate(m)						Height(m)	Statistical Height(m)	
		Interval	Sample Length	Height Difference	Marking Line Span	P1_X	P1_Y	P1_Z	P2_X	P2_Y	P2_Z		Min	Max
1	AUTO	--	1.000	1.000	3.000	792710.9570	2504605.8000	9.4680	792710.9500	2504605.8090	3.7940	5.674	5.377	5.723
2	AUTO	--	1.000	1.000	3.000	792712.9840	2504603.4230	9.5060	792712.9770	2504603.4310	3.8130	5.693		
3	AUTO	--	1.000	1.000	3.000	792715.4970	2504600.4770	9.5480	792715.5060	2504600.4660	3.8580	5.690		
4	AUTO	--	1.000	1.000	3.000	792717.9390	2504597.6130	9.6030	792717.9300	2504597.6240	3.8800	5.723		
5	AUTO	--	1.000	1.000	3.000	792723.1220	2504591.6360	9.5820	792723.1200	2504591.6380	3.9330	5.649		
6	AUTO	--	1.000	1.000	3.000	792725.5560	2504588.6810	9.5140	792725.5590	2504588.6780	3.9590	5.555		
7	AUTO	--	1.000	1.000	3.000	792727.5590	2504586.3430	9.4460	792727.5480	2504586.3450	3.9890	5.457		
8	AUTO	--	1.000	1.000	3.000	792729.9710	2504583.6040	9.3950	792729.9720	2504583.6030	4.0180	5.377		

Overall Image of Specific Content

Content Explanation：Each analysis result is stored in tiers, and the content of an analysis result includes input parameters, result images, and result information.

- **Input Parameters**：The parameters used in this measurement
  - Thickness
  - Upper Height(m)
  - Bottom height(m)

  Refer to Bridge Clearance for specifics
- **Result Images**
- **Analysis Result List**: Include the number of each measurement line, the measurement mode, the parameters corresponding to the mode, the coordinates of the two points, the height, and the minimum and maximum heights.

Height Limit Analysis

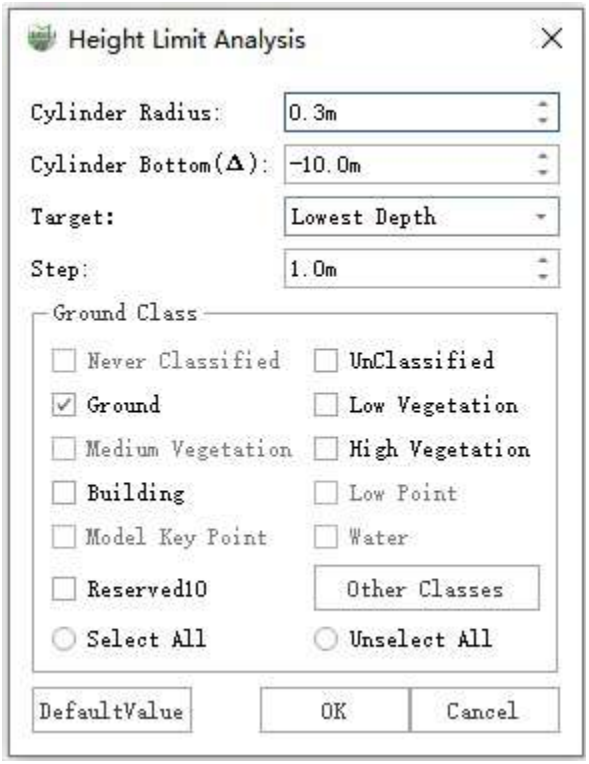
Function Description: This function is used for analyzing the height of power lines.

Function Description

It provides the capability to measure the height of power lines above the ground. It supports both automatic measurement with regular intervals and report generation.

Steps

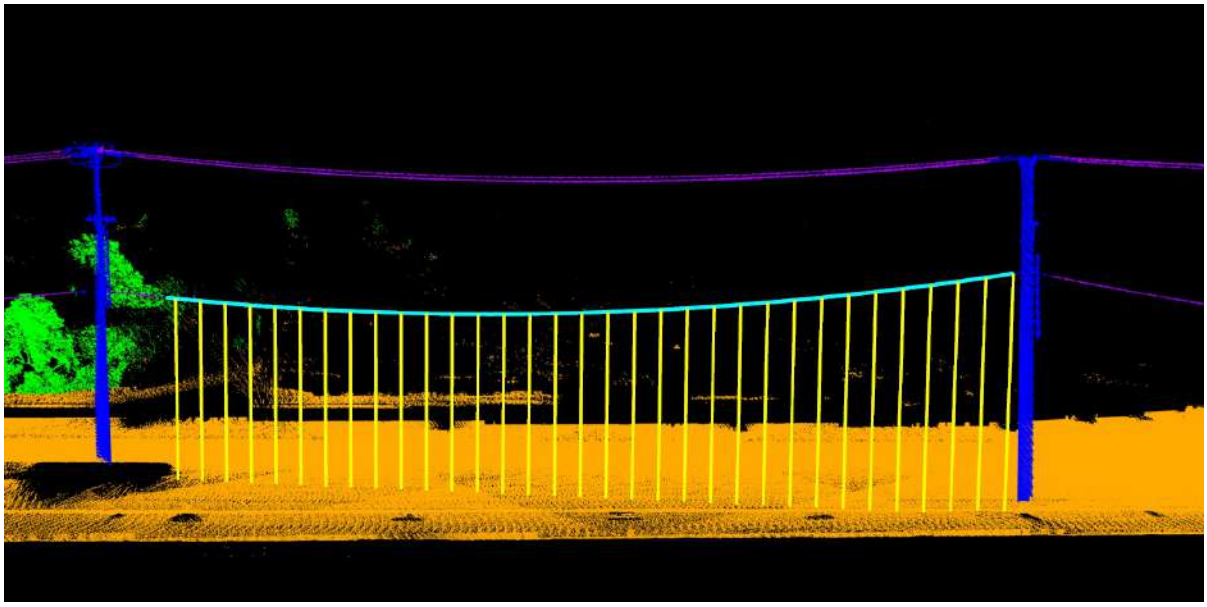
1.Click on **Height Limit Analysis** button. A pop-up dialog box appears.



Parameter Description

- **Cylinder Radius**: Radius of the cylinder used for sampling at measurement points.
- **Cylinder Bottom(A)**: Height below the sampling cylinder at the measurement points.
- **Target**: Minimum Height (the lowest point within the cylinder is considered as ground). Maximum Height (the highest point within the cylinder is considered as ground).
- **Step**: Interval measurement.
- **Ground Class**: Class of the ground points. If not classified, all classes can be selected.

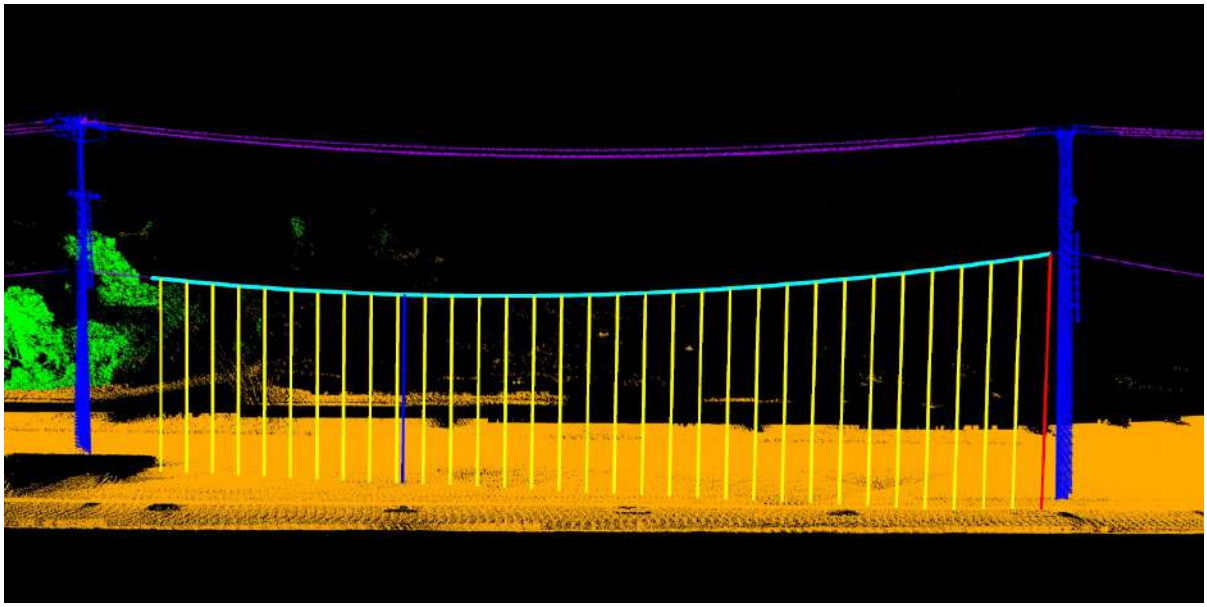
2.Select the power line vector. Then set the parameters and ground point categories (setting appropriate categories can improve the accuracy of automatic measurement). Click "OK" to start the measurement.



3.Report Generation.

Export all measurement results to a report. For the results, please refer to the Power Line Height Limit Analysis Report.

4.After exporting the report, you can see in the 3D window that the minimum height vector line is blue and the highest height vector line is red.



Export Report

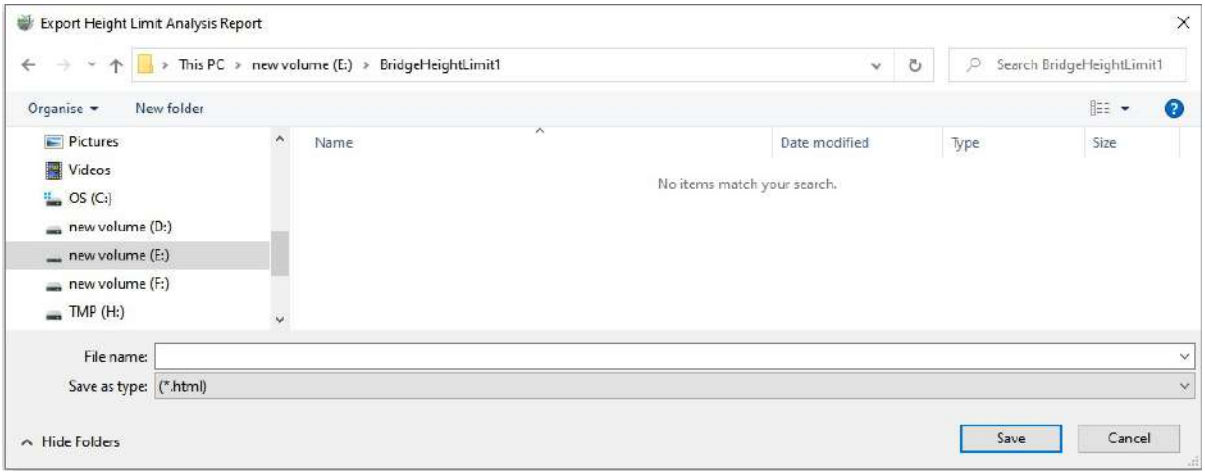
Function Description: Generate an overall report file based on previous analysis results.

Function Instructions

The related data is stored in the 'Power Line Clearance' and 'Power Line Clearance Settings' layers. Before generating the report, make sure the data in the table has not been deleted.

Steps

1.Click the dropdown box next to **Height Limit Analysis** and select **Export Report** button. A save file dialog box will pop up. Then choose the report file name and the location to save.



Save file dialog box

2.is divided into the header, table of contents, and specific content.

(1)**Header and Table of Contents**

1.Header and Table of Contents

Click on the PowerLine_number in the table of contents to jump to the corresponding location.

(2)**Specific Content**

1.1. Input Parameter

Input Parameter

From Class	Ground
Cylinder Radius(m)	0.3
Cylinder Bottom(m)	-10.0
Step(m)	1.0
Target Type	Lowest Depth

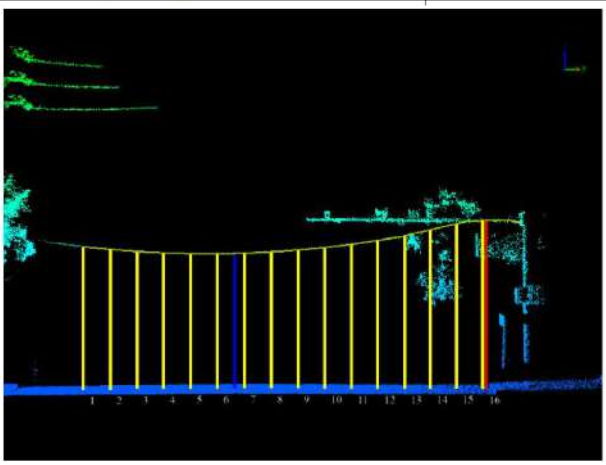
1.2. Hight Clearance Analysis Result

Analysis Route Information

Layer Name	Power Line
FID	1
Start Position	No Coordinate
End Position	No Coordinate
Min Height(m)	5.019
Min Height Position	No Coordinate
Max Height(m)	6.279
Max Height Position	No Coordinate

Analysis Result List

No	Position	Height(m)
1	No Coordinate	5.092
2	No Coordinate	4.900
3	No Coordinate	4.716
4	No Coordinate	4.565
5	No Coordinate	4.466
6	No Coordinate	4.421
7	No Coordinate	4.376
8	No Coordinate	4.388
9	No Coordinate	4.437
10	No Coordinate	4.501
11	No Coordinate	4.626
12	No Coordinate	4.792



Overall image of specific content

(3)**Content Explanation**: Each analysis result is stored in levels, and one analysis result content includes input parameters, analyzed power line information, and an analysis result list.

- **Input Parameters**: The parameters used in this detection
  - From Class
  - Cylinder Radius
  - Cylinder Bottom
  - Step
  - Target Type

> For details, refer to Power Line Analysis

- **Analyzed Power Line Information**
  - **Layer Name**: Layer name
  - **FID**: Power line object FID
  - **Start Position**: Start position coordinates
  - **End Position**: End position coordinates
  - **Min Height(m)**: Minimum height
  - **Min Height Position**: Minimum height position coordinates
  - **Max Height(m)**: Maximum height
  - **Max Height Position**: Maximum height position coordinates
- **Analysis Result List**: About all measured numbers, position coordinates, and height.

Road Damage Analysis

Road Damage Analysis is a set of tools and functions used for analyzing road damage-related issues.

Mainly includes：

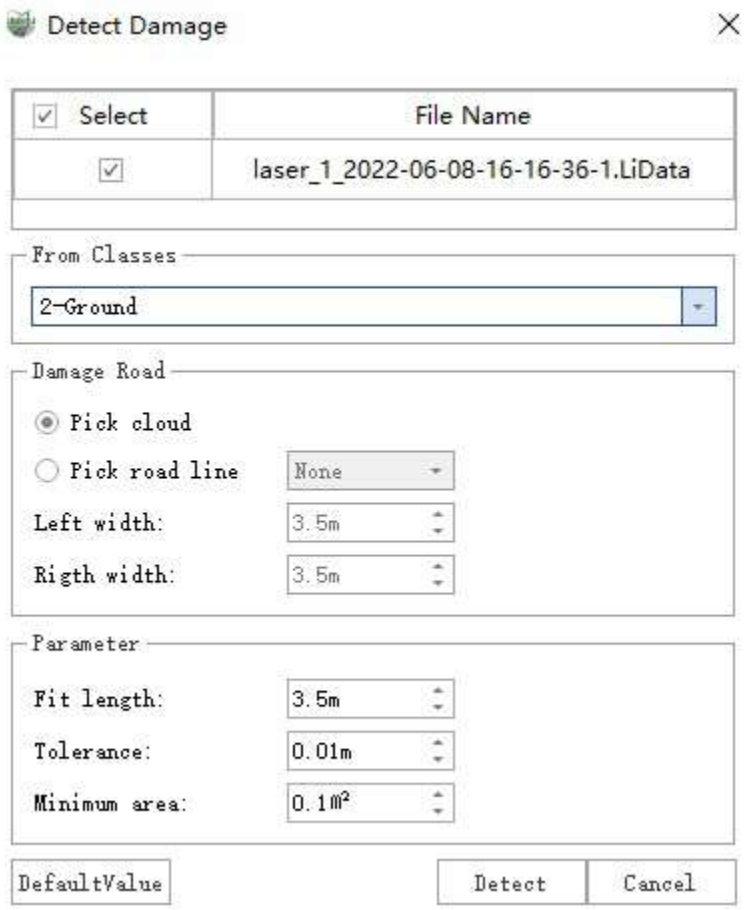
- Damage Detection
- Pavement Condition Index
- Export Report

Damage Detection

Function Description: This function is used to detect road damage areas. It provides two modes: point cloud and road line. It supports six types of damage detection: ruts, undulations, cracks, potholes, shallow pits and bulges. The point cloud mode is used for small-scale detection of areas of interest, while the detection results cannot be used for road damage index calculation and report export. The road line mode is for overall detection on the road scale while the results can be used for subsequent road damage index calculation and report export, providing a reference for road repair work.

Steps

1.Click the **Damage Detection** button to open the following dialog box:



Damage Detection Dialog Box

Parameter Description

- **Point Cloud File**: Select the point cloud file to be detected.
- **From Class**: The class to which the ground points belong. If not classified, you can select 'All'.
- **Pick cloud**: Polygonally select the point cloud to be detected.
- **Pick road line**: Selection of road lines.

- **Left Width**: The width on the left side of the road line.
- **Right Width**: The width on the right side of the road line.
- **Road Block Layer**: The block information of the road line will be saved in this layer, and the layer must be a polygon layer.
- **Damage region layer**: Each damaged area will be saved as a polygon in this layer, and the layer must be a polygon layer.
- **New Road Block Layer**: Create a new Road Block layer. The layer created here will contain some special business identifiers.
- **New Damage region layer**: Create a new Damage Area layer. The layer created here will contain some special business identifiers.

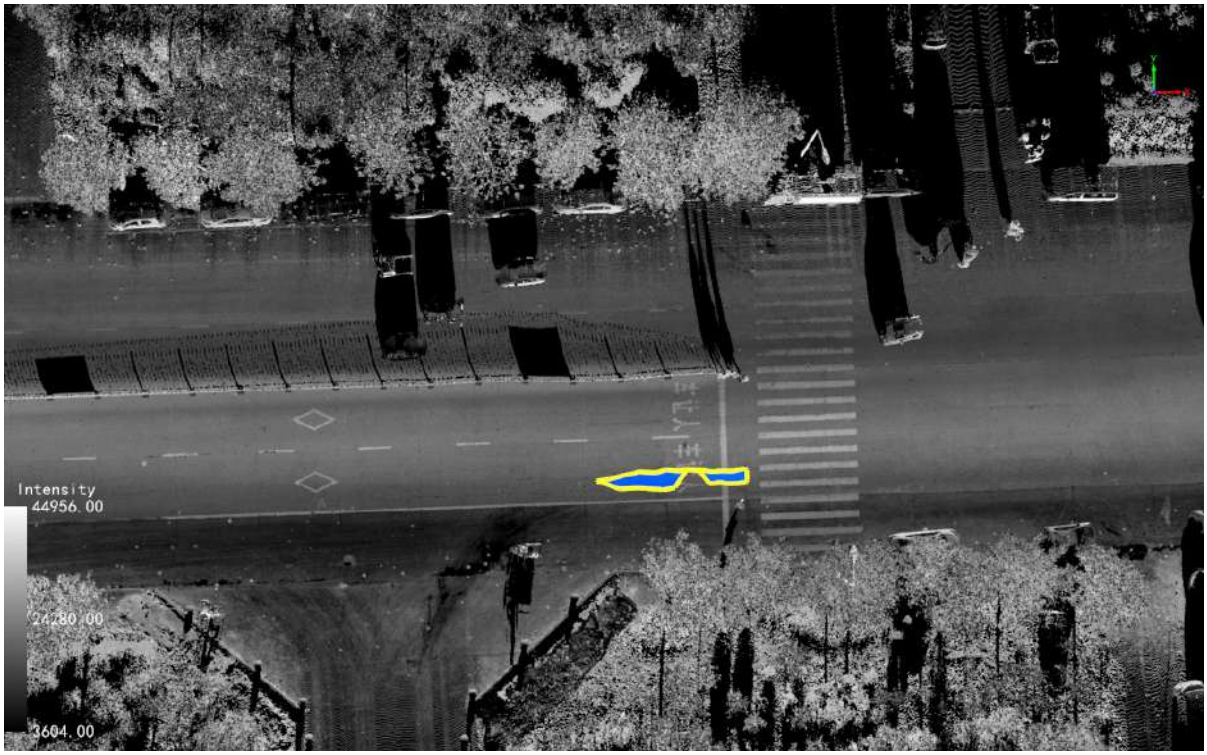
2.Firstly, choose the point cloud file and ground point type. For the point cloud and road line modes, the procedures are as follows:

(1) Point cloud: Select one or more point clouds in the 3D window and choose the damage area layer. If there is no existing layer, you can create a new one.

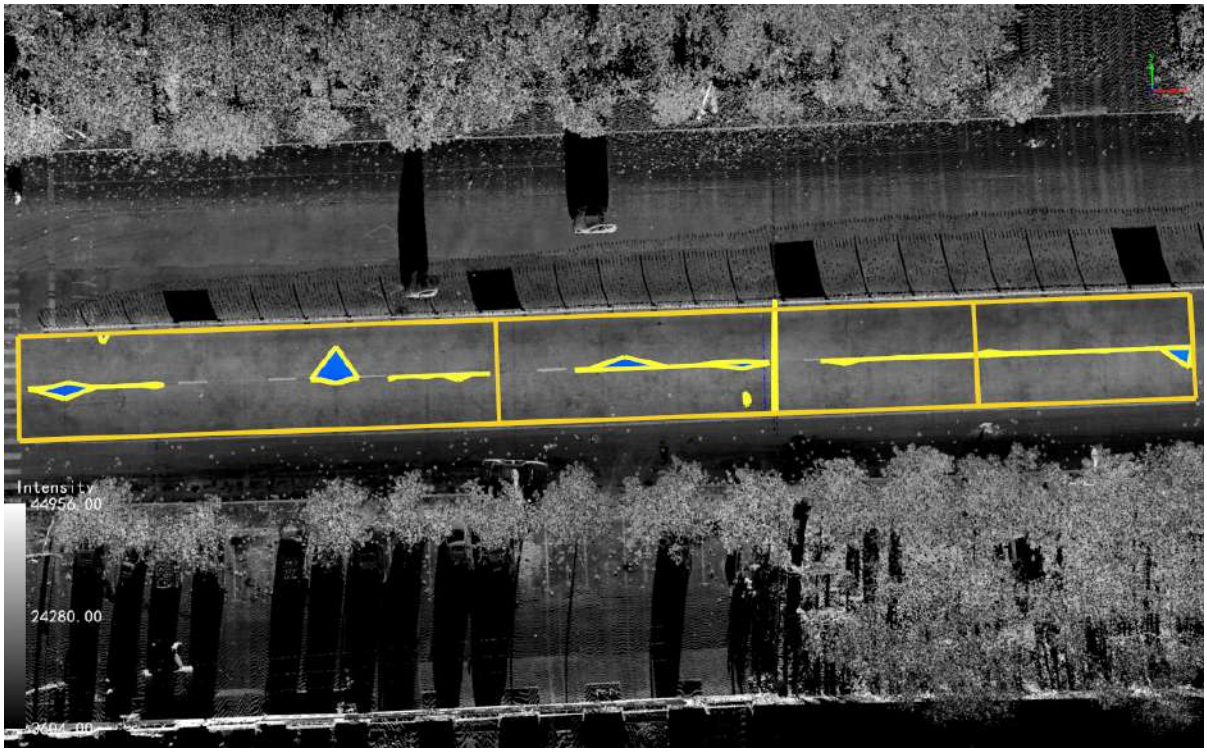
(2) Road line: In the 3D window, select one or more road lines or directly specify the layer to select all vector lines within the layer. Adjust the width to the left and right to ensure the detection area covers all roads to be detected. Select the Road Block layer and the damage area layer. If there is no existing layer, you can create a new one.

> For the road line mode, the damage detection will divide the analysis area into blocks along the direction of the road centerline based on the set road width. Each block serves as a damage detection unit and its size is 225 square meters.

3.Click **Detect** to get the detection results.



Point cloud Damage Detection



Road line Damage Detection

Pavement Condition Index

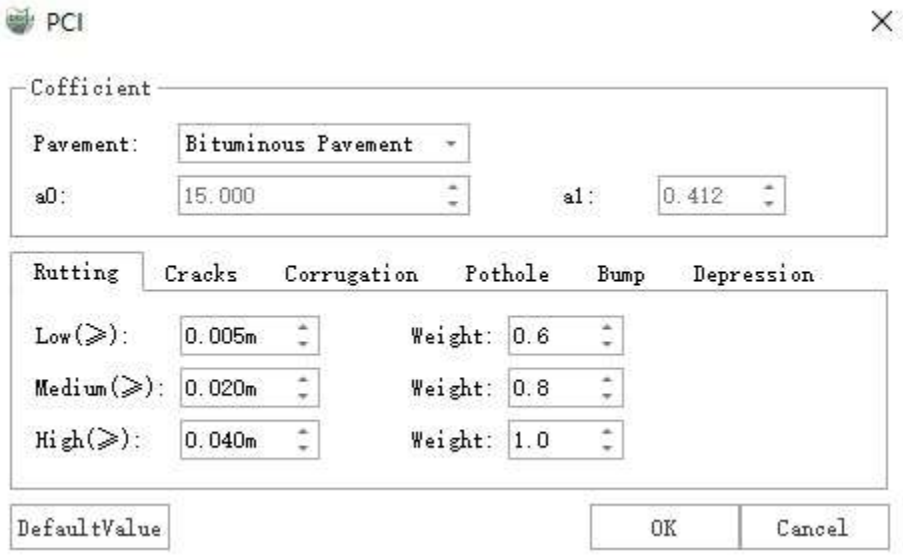
Function Description: This function is used to calculate the PCI (Pavement Condition Index) parameter.

Function Explanation

It calculates the PCI parameter for all roadblocks based on the results of the Damage Detection.

Steps

1.Click on the **PCI** button. This will open the following dialog box:



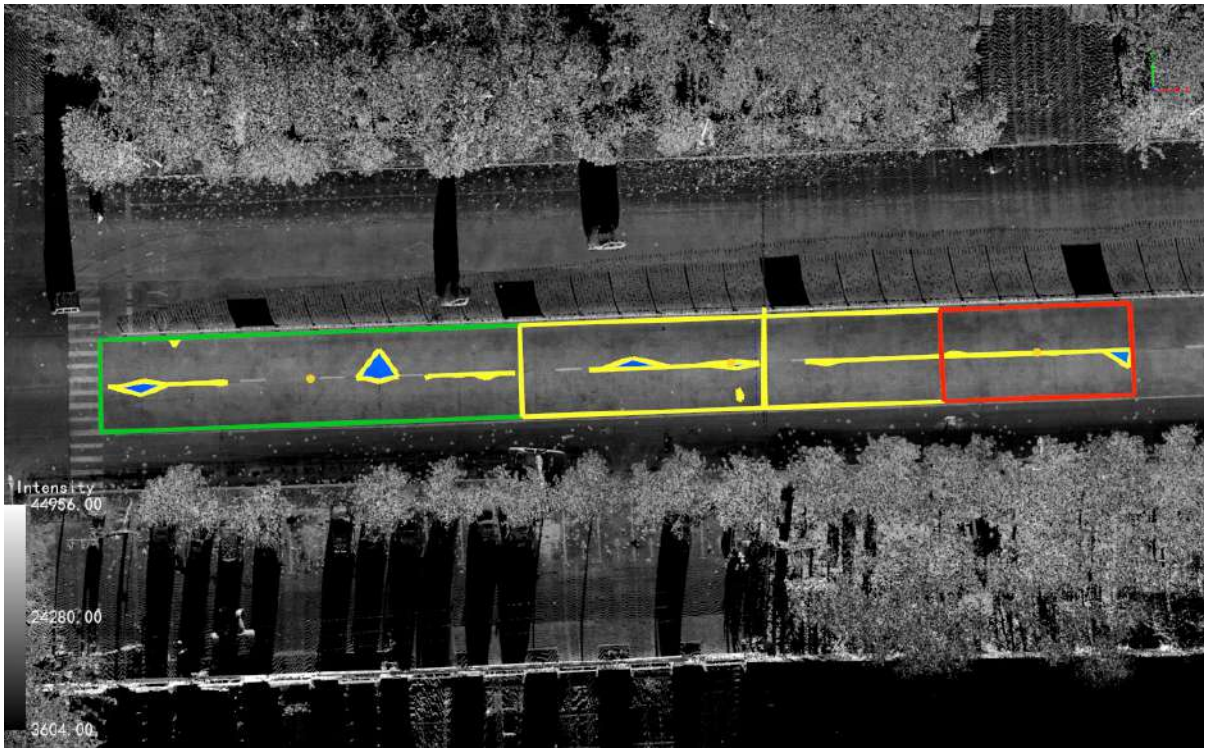
PCI

Parameter Description

- **Cofficient**: There are three built-in pavement types: asphalt, concrete, and gravel. Users can also define their own pavement parameters.
- **Weights**: The weights assigned to different damage types.
- **Block node layer**: The layer where each roadblock region will be saved as points. The layer must be a point layer.
- **New Block node layer**: Create a new layer to store the roadblock nodes. This layer will include some specific information for further analysis.

2.Select the pavement type or manually specify the a0 and a1 parameters.

3.Set the weights for different damage types and choose the result layer to save the computed PCI values. Click "OK" to proceed.



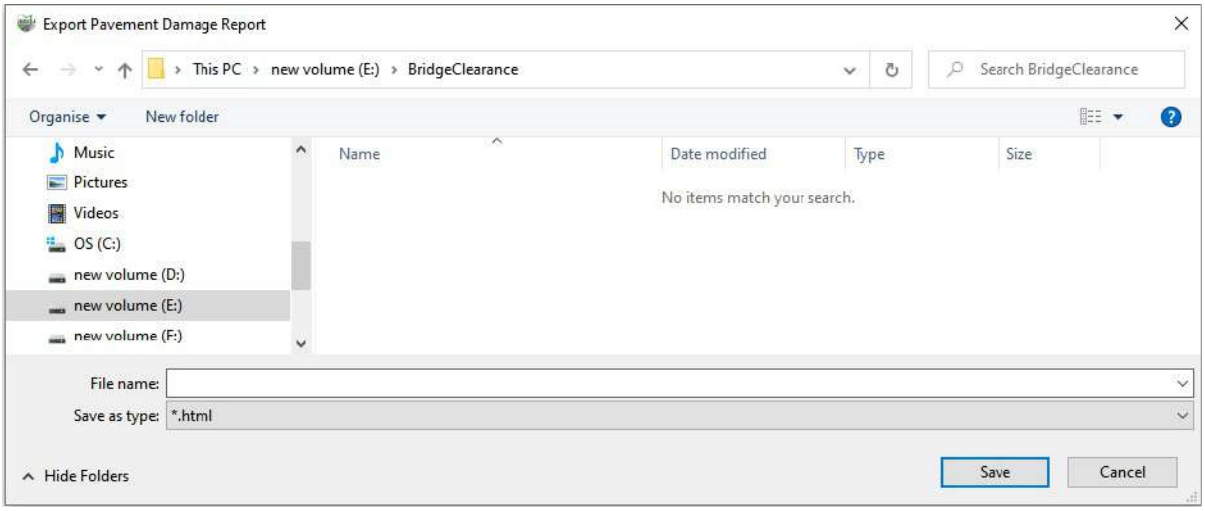
Pavement Condition Index

Export Report

Function Description: The road damage analysis module allows you to generate reports for individual or multiple pavement damage analysis results, as well as a comprehensive report for all analysis results.

Steps

1.Click on the **Export Report** button. This will open a dialog box to save the report. Choose a file name and location to save the report.



Save File Dialog

2.The report consists of three main parts: header, table of contents, and specific content.

（1）Header and Table of Contents

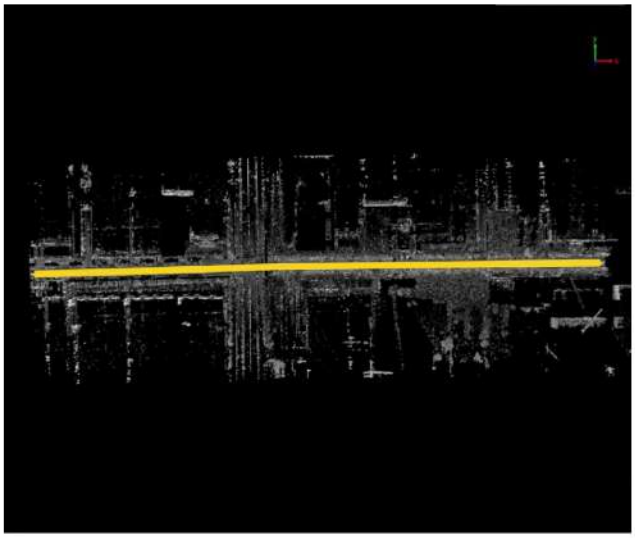
Header and Table of Contents

Click on the "Pavement" in the table of contents will jump to the corresponding section.

（2）**Specific Content**：It includes the summary information of all damaged pavements and the detailed information of each individual roadblock.

Summary Statistical Table

Pavement Name	Position(m)	Area(㎡)	Block Number	PCI Average	PCI Level
000854	593568.997,4211399.949,27.295	4010	18	100	Good(1)



Summary PCI View

Pavement Statistical Table

Block ID	Position(m)	Area(㎡)	PCI	PCI Level
1	593568.997,4211399.949,27.295	225	100	Good(1)
2	593629.803,4211400.690,27.409	225	100	Good(1)
3	593690.610,4211401.397,27.405	225	100	Good(1)
4	593746.100,4211401.865,27.205	185	100	Good(1)
5	593508.191,4211399.236,27.294	225	100	Good(1)
6	592900.239,4211386.929,26.928	225	100	Good(1)
7	592839.460,4211384.991,27.011	225	100	Good(1)
8	592778.670,4211383.379,27.113	225	100	Good(1)
9	592717.890,4211381.465,27.236	225	100	Good(1)
10	593264.964,4211396.414,27.546	225	100	Good(1)
11	593325.770,4211397.152,27.566	225	100	Good(1)
12	593021.801,4211390.681,27.175	225	100	Good(1)
13	593143.360,4211394.501,27.607	225	91	Good(1)
14	593386.577,4211397.863,27.520	225	100	Good(1)
15	593204.158,4211395.677,27.525	225	100	Good(1)
16	592961.016,4211388.919,26.939	225	100	Good(1)
17	593082.579,4211392.638,27.534	225	100	Good(1)
18	593447.384,4211398.550,27.407	225	100	Good(1)

Overall Content

Content Explanation:

- **Summary Statistical Table**: Contain all the damaged pavements. Include the road name, location, area, number of blocks, average PCI value and PCI grade.
- **Road Statistical Table**: Include the location, area, PCI value, and PCI grade of each roadblock.

Viewshed Analysis

The main components of the viewshed analysis include：

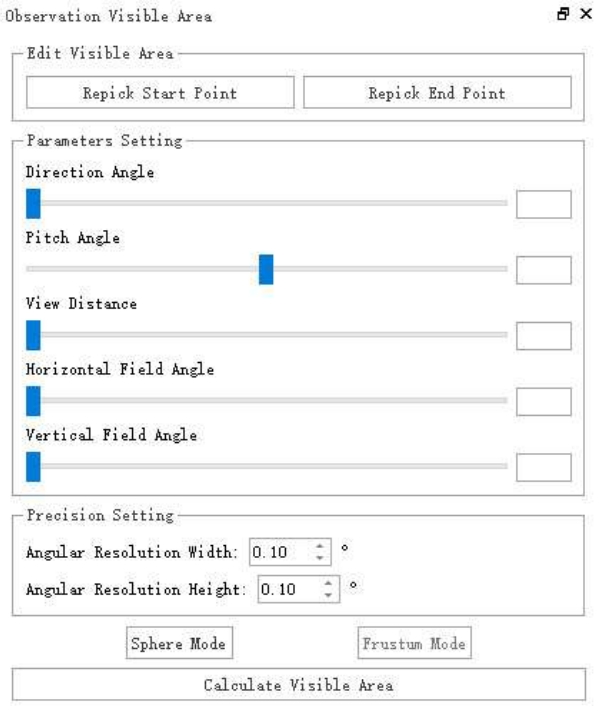
- Observation Observation Analysis
- Target Observation Analysis

Observation point visual field analysis

Function Description: This tool can be used to establish an observation point and analyze the occlusion of the target point from the observation point.

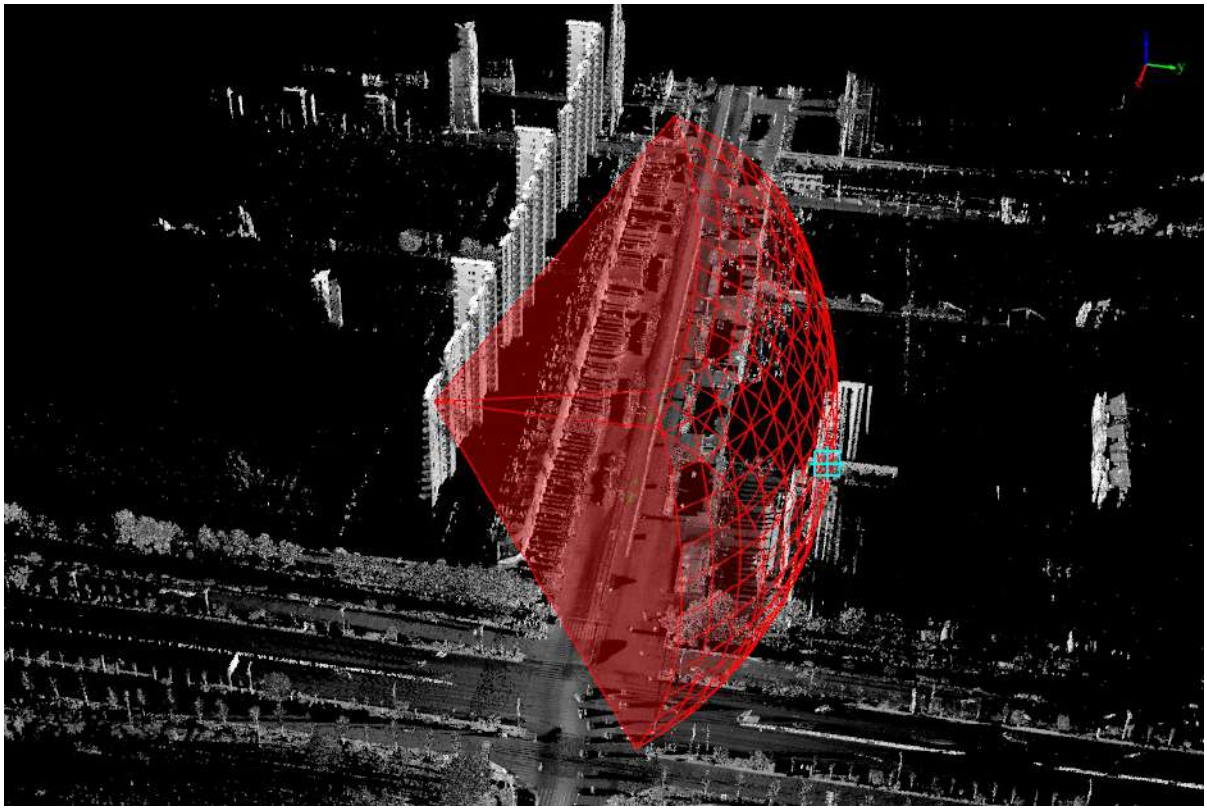
Steps

1.In the road analysis menu bar, click **Observation point visual area analysis** button, the visual field editing settings page pops up.



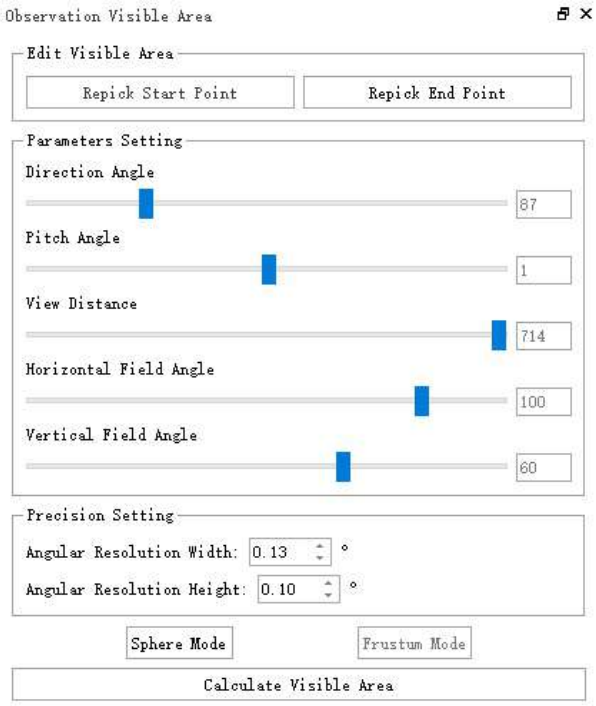
Visual field editing

2.Use the left mouse button to select the position of the observation point in the 3D viewport, and then slide the mouse. The 3D viewport will render the observation range from the selected point to the position of the mouse movement in real time. The vertebral body：



Observation range Visual vertebral body

3.Double-click the left mouse button at the target point to determine the end point of the vertebral body. At this time, the optic vertebrae will no longer change with the mouse.



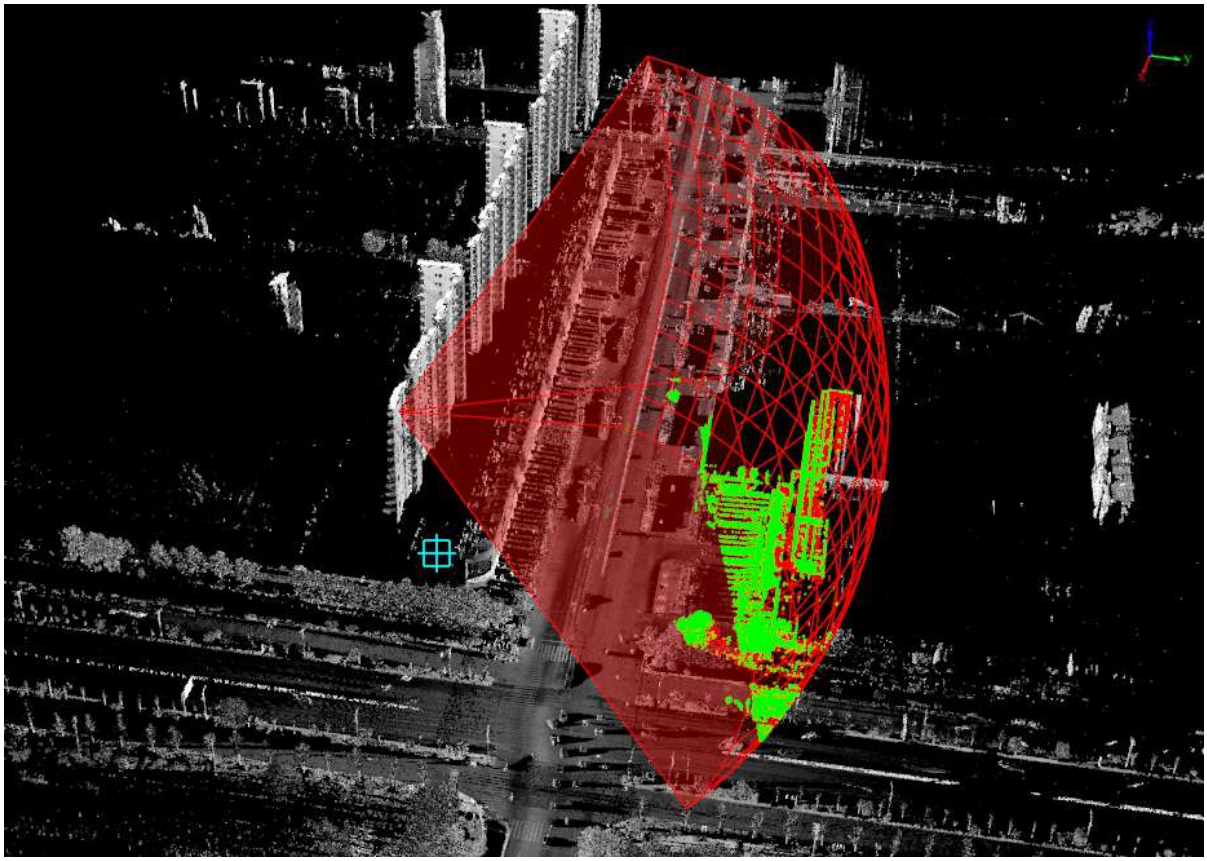
After the starting point is selected, the angle is displayed

4.In the pop-up settings window, the various angles of the current visual vertebral body will be displayed. If you want to adjust the starting position of the observation and the viewing angle range of the observation, you can set it in the parameter adjustment pop-up window, click the **RePick start point** button, you can reselect the starting point of the observation in the 3D viewport. Click the **RePick end point** button to reselect the end point position of the optic vertebrae.

5.If you need to adjust the viewing angle, you can set it by dragging and dropping the slider in the parameter settings.

- **Parameter setting**:
  - Slide the **Direction Angle** slider to adjust the horizontal orientation of the visual vertebral body.
  - Slide the **Pitch Angle** slider to adjust the vertical orientation of the visual vertebral body.
  - Slide the **Visual Distance** slider to adjust the distance of the observation range.
  - Slide the **Horizontal Field Angle** slider to adjust the observation range in the horizontal direction.
  - Slide the **Vertical Field Angle** slider to adjust the observation range in the vertical direction.

6.When all adjustments are completed, click the **Calculate Visible Area** button, and wait for the calculation progress bar at the bottom of the software to end, you can display the calculation results of the visual field in the 3D viewport.



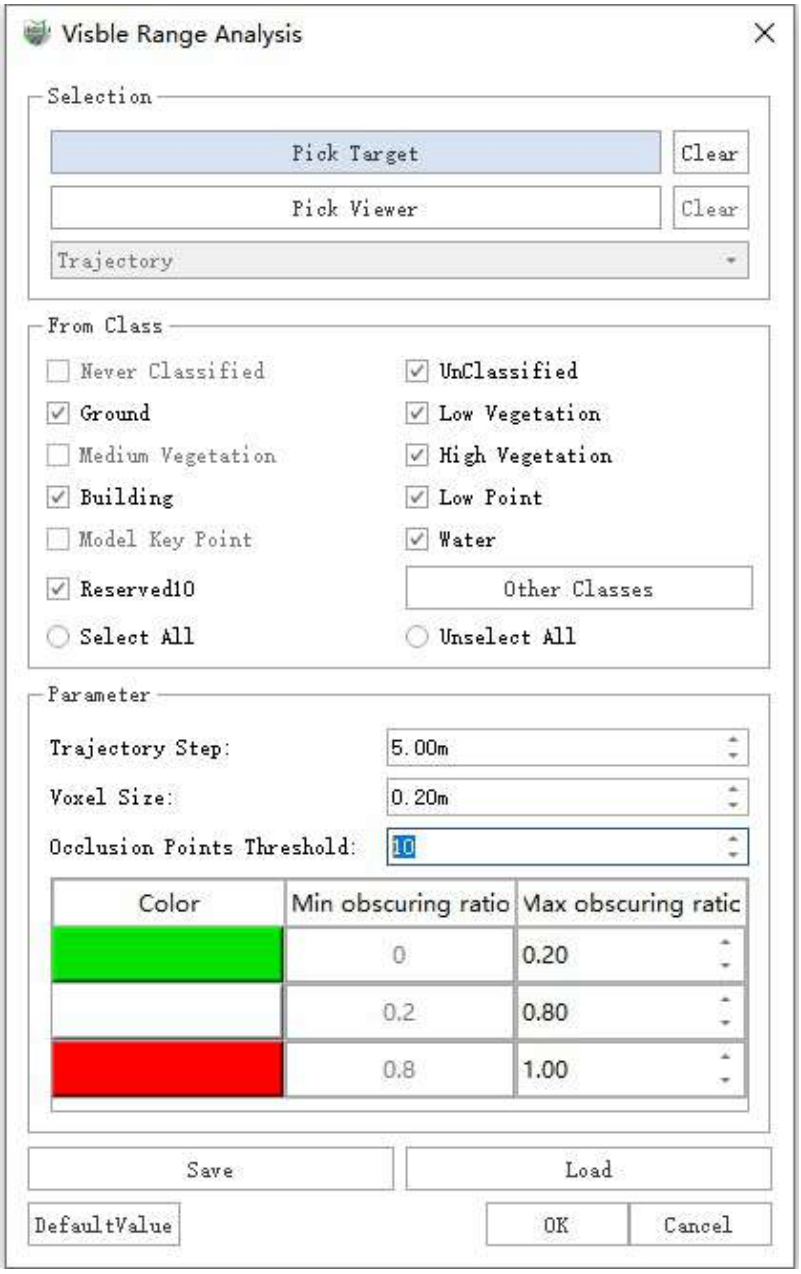
Observation point visual field analysis and calculation results

Target

Function Description: This tool can be used to calculate the visibility between the target point cloud and the observation point, which can be used to detect whether the location of the traffic facility is reasonable and whether it meets the requirements of traffic safety.

Steps

1.Click on the road analysis page **Target** button in the road analysis page, the settings page will pop up on the software page at this time:

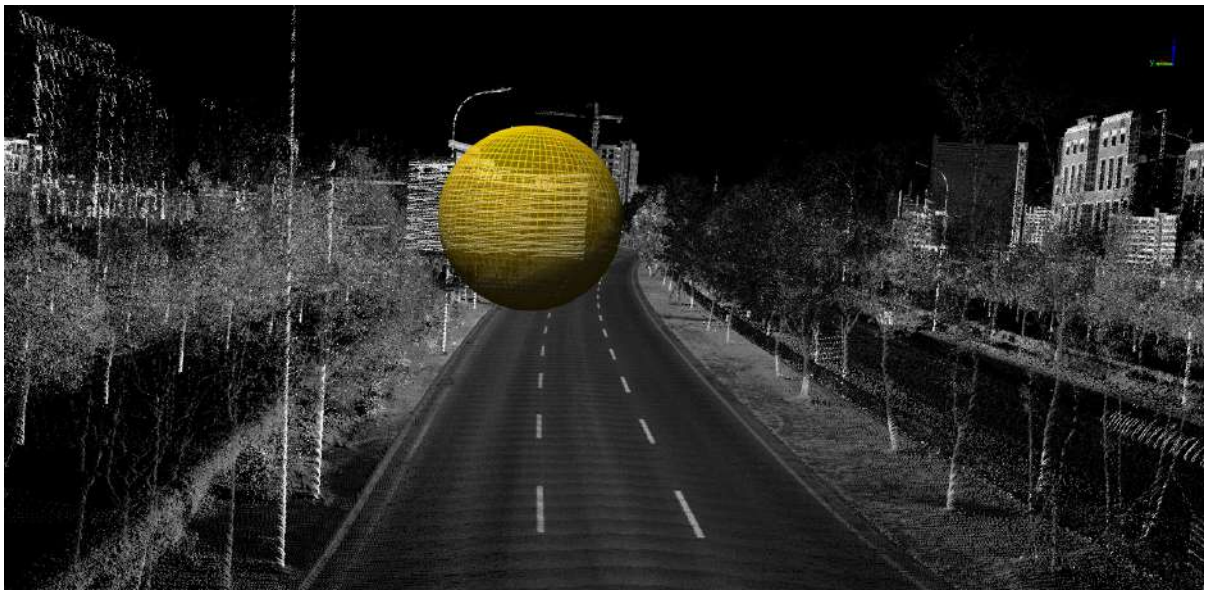


Parameter adjustment pop-up window

Parameter description:

- **Select the target point**: Select the cluster of points to be observed through the ball selection.
- **Select observation point**: Select observation point data, including three modes: trajectory segment selection, existing vector line selection, and custom line drawing.
- **From Class**: Select the point cloud category for observing point data.
- **Visual parameters**: Adjust the visual parameters to obtain different visual effects.
  - **Trajectory Step size**: When the track segmentation mode is selected, set the observation point to take the point step size on the track. since the trajectory points are too dense, the trajectory points are thinned at a step interval, and the thinned trajectory points are used for visual calculation.
  - **Voxel Size**: he smaller the voxel, the finer the analysis result and the longer the calculation time. The default is recommended.
  - **Occlusion Points Threshold**: When the number of occlusion points between the target point and the observation point is less than the threshold value, it is considered that there is no unobstructed
  - **Obscuring ratio range**: The obscuring rate is divided into three levels, which are represented by different colors (customizable).After the analysis is over, by adjusting the level range, the display effect will be refreshed in real time.
- **Save and Load**: Read and write visual domain analysis results.
  - **Save**: Save the visual field analysis results, including the center of the target point, the observation point, and the masking rate.
  - **Load**: Import the saved visual domain analysis results and display them in the view (they need to be in the same coordinate system).

2.Click the **Select target point** button, and in the point cloud, use the ball selection method to select the point cluster of the observed point.You can click Empty to re-select.

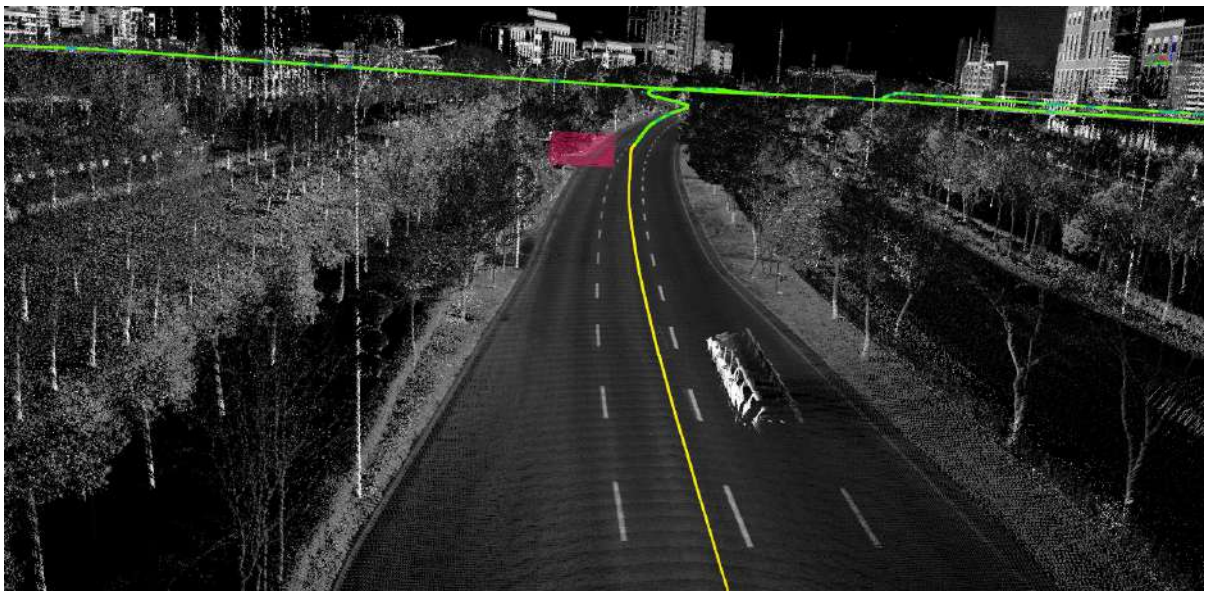


Before the target point is selected

After the target point is selected

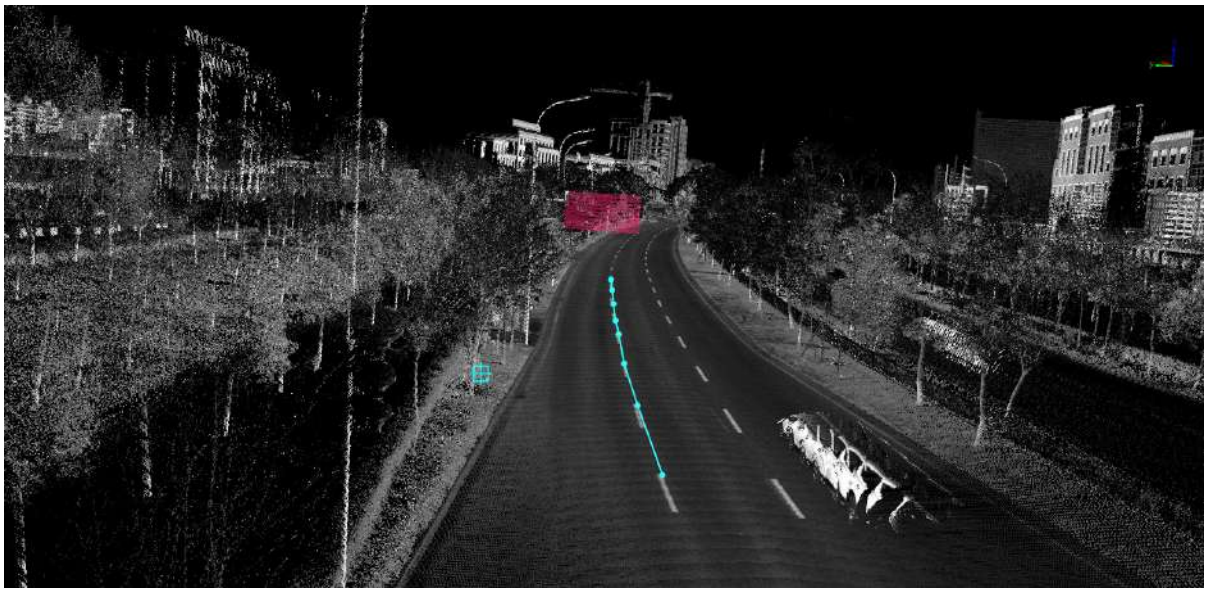
3.Click the **Select observation point** button to activate the observation point type drop-down box.

- **Track**: You must load the track file when you create a new project, then click at the starting position of the track segment you want to select, push the mouse, and the track will be selected in real time. Double-click at the location you want to end the selection and you can select a certain segment of the track as the observation point.



Select trajectory at the observation point

- **Vector line**: You must draw a line element in advance, and use the selection vector line mode to select the vector line as the observation point.

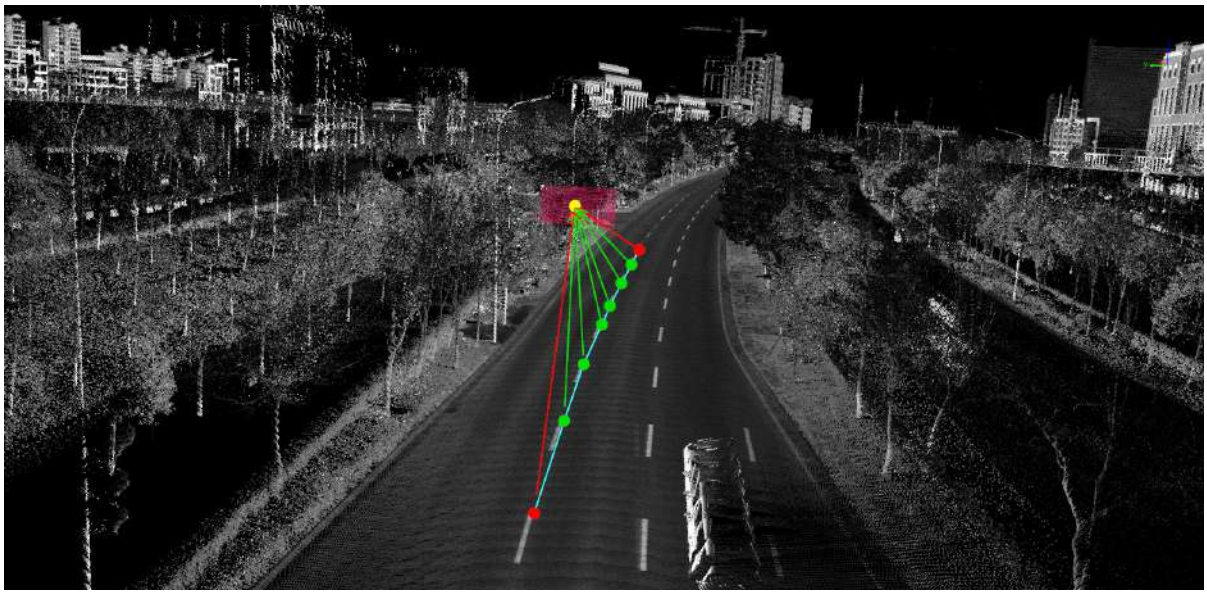


Select vector line at the observation point

- **Custom line**: After selecting the custom line, you can draw a temporary vector line at the location that needs to be analyzed as an observation point

Select custom line at the observation point

4.After clicking OK, perform visual domain analysis.



Analysis results

5.The analysis results can be exported a *.vis file by clicking the save button, you can also reload the file to display and view in the future.

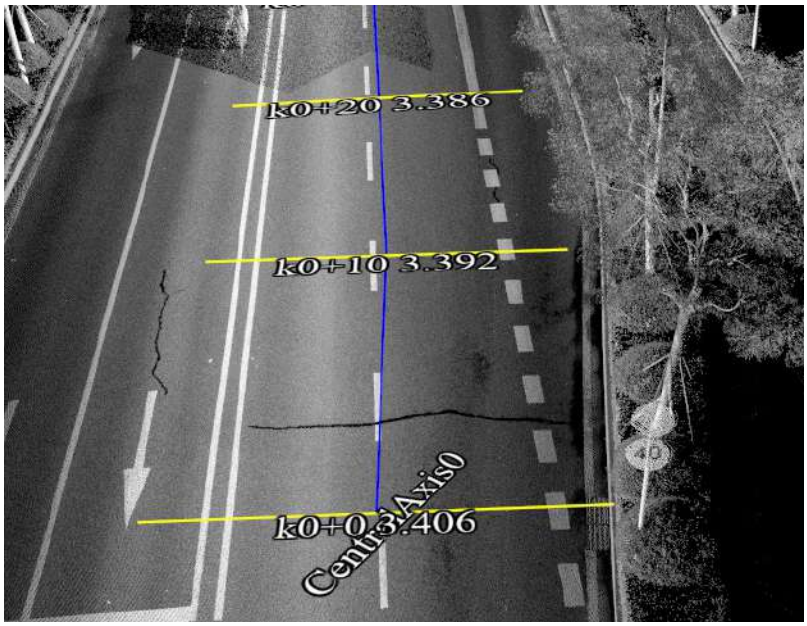
Road Cross-Section Analysis

- Road Reference Line Setup
- Generate Section Views
- Show/Hide
- Clear Sections
- Section 3D Window Control

Road Reference Line Setting

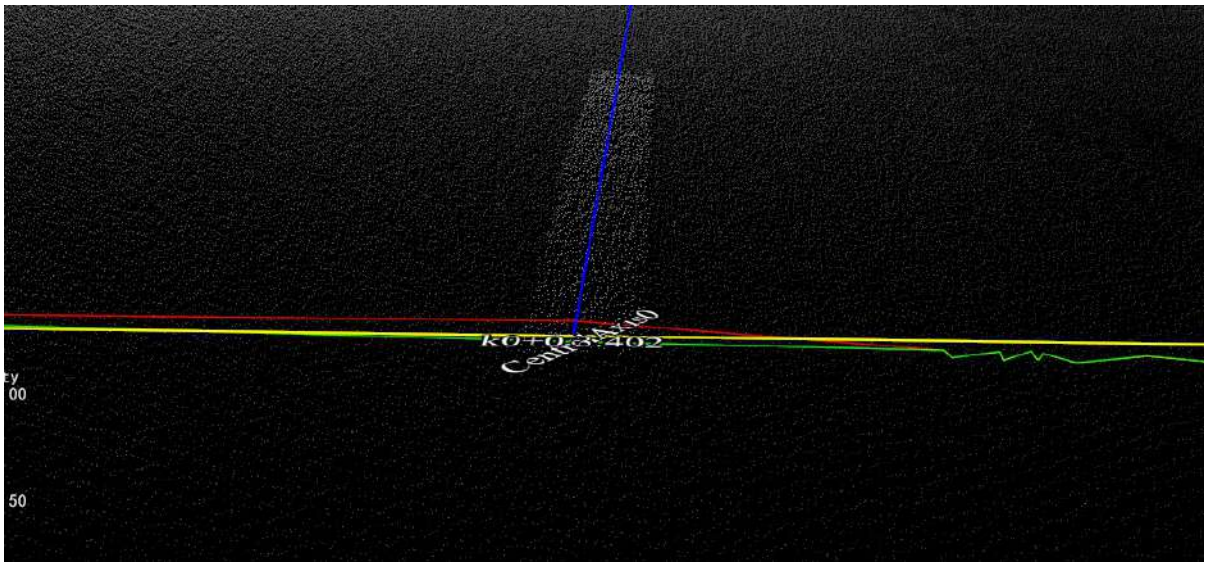
Definition of Terms

- **Longitudinal Reference Line**: Represent the direction of the road. It's a virtual reference line, as shown by the blue line in the image below.
- **Cross-sectional Reference Line**: Perpendicular to the direction of the road. It's a virtual reference line and is the basic reference line for cross-sectional analysis, as shown by the yellow line in the image below.



- **Design Section**: Section data generated from design data. It mainly represents the data of the current stage, i.e., the data of different periods at the same location, used for cross-sectional comparison analysis and fill-excavation analysis. As shown by the green line in the image below.
- **Measurement Section**: Section data generated from current data, as shown by the red line in the image below.

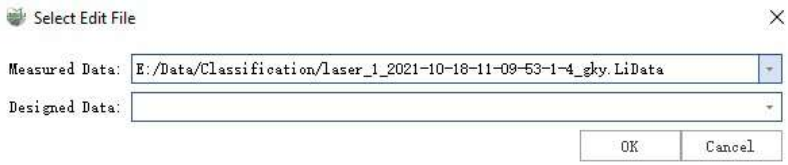
> To distinguish effectively, the data has been manually exaggerated for display.



Editor

Function Description: Enable/disable the road cross-sectional analysis function.

Start Editing



Parameter Explanation:

- **Measurement Data**: Used to generate real sections. The default color is red and measurement data is a required field. The dropdown list includes the point cloud data opened in the current project.
- **Design Data**: Used to generate the design section. The default color is green and design data is optional. If you need to use the subsequent fill-excavation analysis, the design data is required.

> Measurement data and design data are all point cloud data opened in the current project, and they must have overlapping physical ranges. Thus, the generated design section and the actually measured section of this time have practical significance for comparison analysis. The measurement data is the actual cross-sectional data used in this measurement, and the design data is the point cloud data of the **same area at different times**.

End Editing

Exit the overall function of road cross-sectional analysis and remove all types of vector data generated during the use of the function.

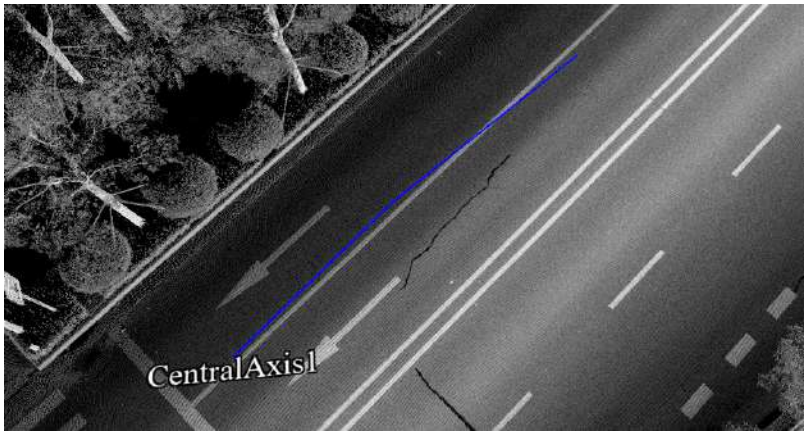
> Before clicking End Editing, please make sure that the data that needs to be saved has been saved. After exiting the function, all data will be cleared.

Create Reference Line

Function Description: Manually draw a directional line (longitudinal reference line) for the road cross-sectional analysis.

Steps

1、Left-click the mouse in sequence.



Dashed dialog box

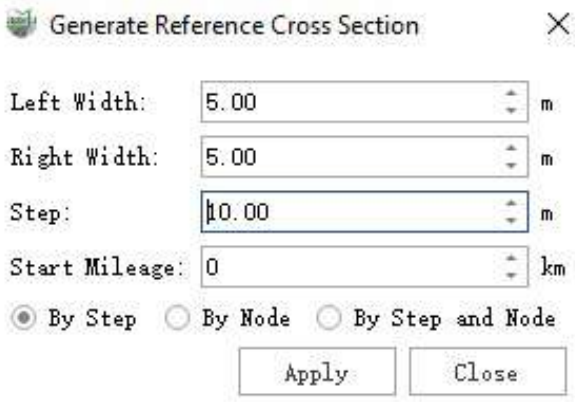
2、Double-click the left button to end creation.

Function Description: Left-click the mouse to select an existing line object as the longitudinal reference line.

Auto-generate Orthogonal Cross-section

Function Description: Auto-generate reference cross-section based on the set parameters.

Parameter Dialog Box



Parameter dialog box

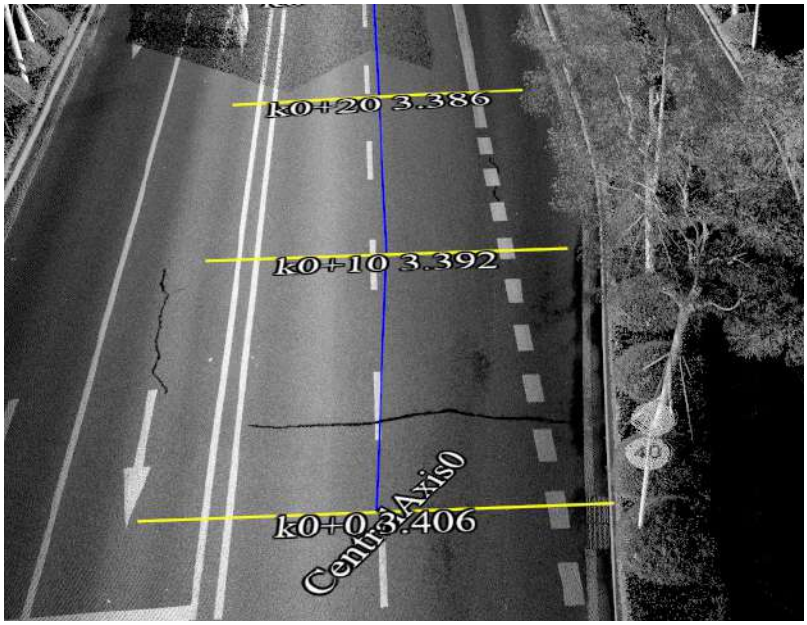
Parameter Explanation

- **Left Width**: The distance from the farthest left of the cross-sectional reference line to the longitudinal reference line that default left width is "5". If you want to use road property calculation, it should be set to half a lane width.
- **Right Width**: The distance from the farthest right of the cross-sectional reference line to the longitudinal reference line that default right width is "5". If you want to use road property calculation, it should be set to half a lane width.
- **Step Size**: The step size to generate the reference cross-section.

- **Start Mileage (default is "0")**: The starting mileage to generate the reference cross-section. The mileage of the cross-section accumulates based on this. Each segment of the road may have a different starting mileage post. The cross-section is named by default in the form of mileage post km + m and users can modify the name of the cross-section.
- **By Step Size**: Generate reference cross-sections at a specific step size.
- **By Node**: Only generate reference cross-sections at the node points of the longitudinal reference line.
- **By Step Size and Node**: Generate reference cross-sections at a specific step size and also handle nodes.

Illustration

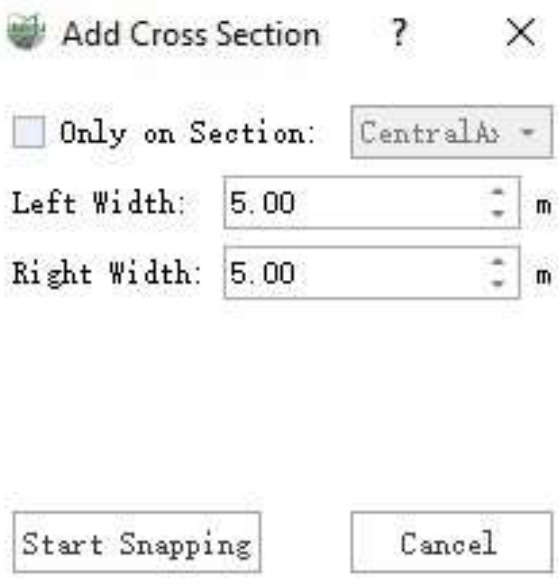
The blue vector line is the manually drawn longitudinal reference line, named CentroAxis0, where 0 is the line number. The yellow line is the reference cross-section generated according to the step size, named K0 + 0 3.406, where K0 is the pile number. + 0, +10 and etc. are the distances from the starting point. 3.406, 3.392 and etc. are the height values at the intersection of the cross-sectional reference line and the longitudinal reference line.



Parameter dialog box

Manually Generate Orthogonal Cross-section

Function Description: Auto-generate reference cross-section based on the set parameters.
Parameter Dialog Box



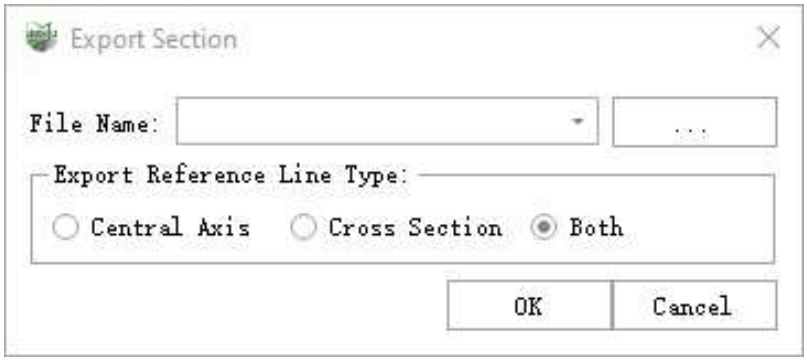
Parameter dialog box

Parameter Explanation

- **Only Add on Specific Central Line**: Only add on the specific center line (longitudinal reference line). It's ineffective when it captures other center lines (longitudinal reference lines).
- **Left Width**: The default is 5.0 meters. The distance from the farthest left of the cross-sectional reference line to the longitudinal reference line.
- **Right Width**: The default is 5.0 meters. The distance from the farthest right of the cross-sectional reference line to the longitudinal reference line.
- **Start Snapping**: Begin to add cross-sections. Move the mouse to any center line, and click the left mouse button when the center line is highlighted. Followingly, a new cross-section is added to the center line.

Save Reference Line

Function Description: Save reference lines and cross-sectional lines in a file.
Parameter Dialog Box



Parameter dialog box

Parameter Explanation

- **File Name**: Combined with the button next to the edit box, choose the location and file name for saving. The saved result supports .dxf, .shp, and .txt formats.

- **Export Reference Line Type**
  - Center Axis: Only save the longitudinal reference line.
  - Section: Only save the cross-sectional reference line.
  - Axis and Section: Save both the longitudinal reference line and the cross-sectional reference line.

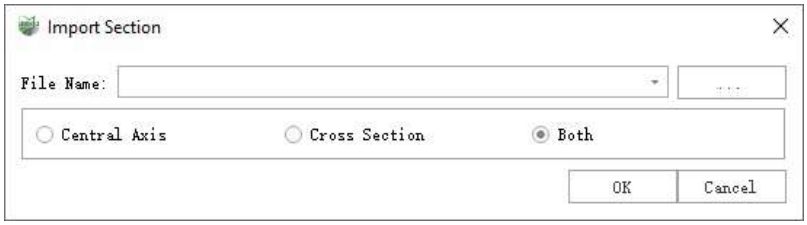
Import

Function Description: Import existing section data from external sources, including reference lines, section analysis files and design sections.

Import Reference Line

Function Description: Import reference line data saved by the "Save Reference Line" function from an external source.

Parameter Dialog Box



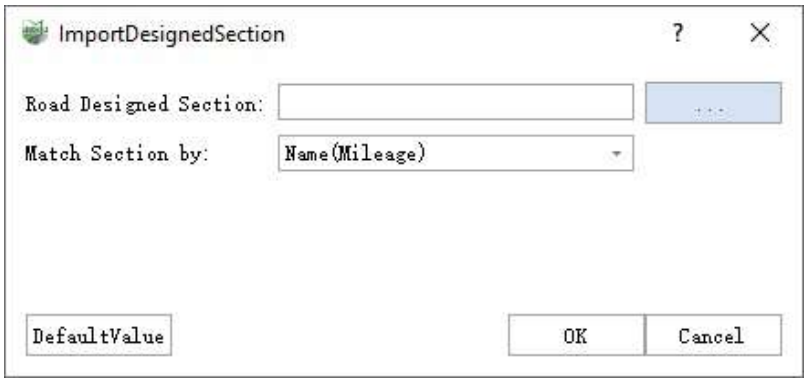
Parameter dialog box

Parameter Explanation

- **File Name**: Combined with the button next to the edit box, choose the file to be imported. Support .dxf, .shp, and .txt formats.
- **Import Reference Line Type**
  - Center Axis: Only import longitudinal reference lines.
  - Section: Only import cross-sectional reference lines.
  - Axis and Section: Import both longitudinal reference lines and cross-sectional reference lines.

Import Design Section

Function Description: Import design section data, and read in format is .csv. This file is generated by the Export Section function. A normal quantity survey does not require an import. If you need to calculate earthwork or do a section comparative analysis, you need to import the design section.



Parameter dialog box

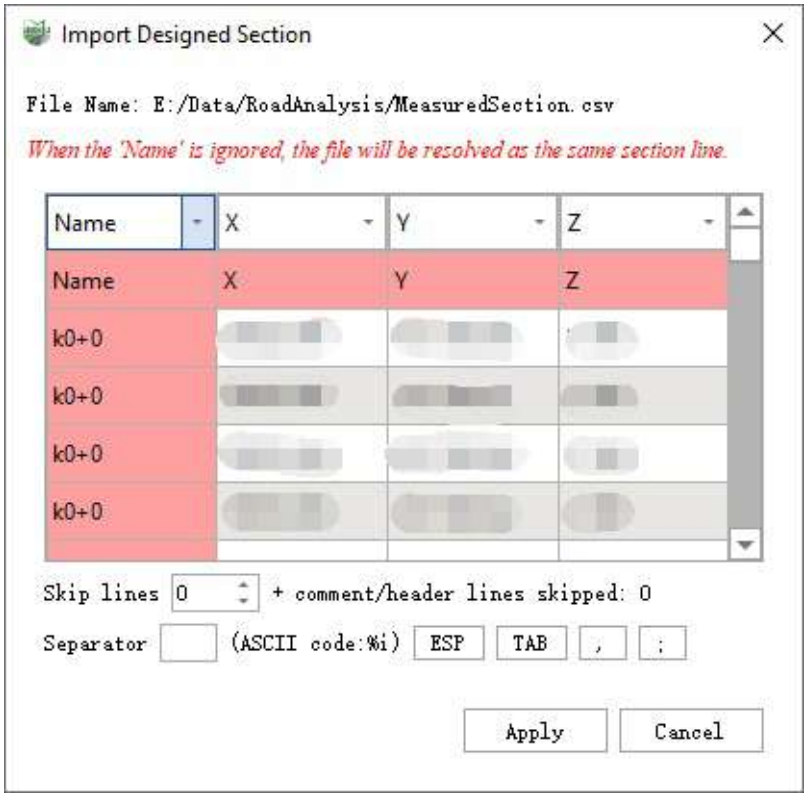
Parameter Explanation

- **Road Design Section**: Combined with the button next to the edit box, choose the .csv file of the design section to be imported.
- **Match Section**: The matching mechanism between the design section and the current measured section, line to line. Currently, it only supports name matching, i.e., only when the name of the measured section line and the design section line are the same can they be matched.

Steps

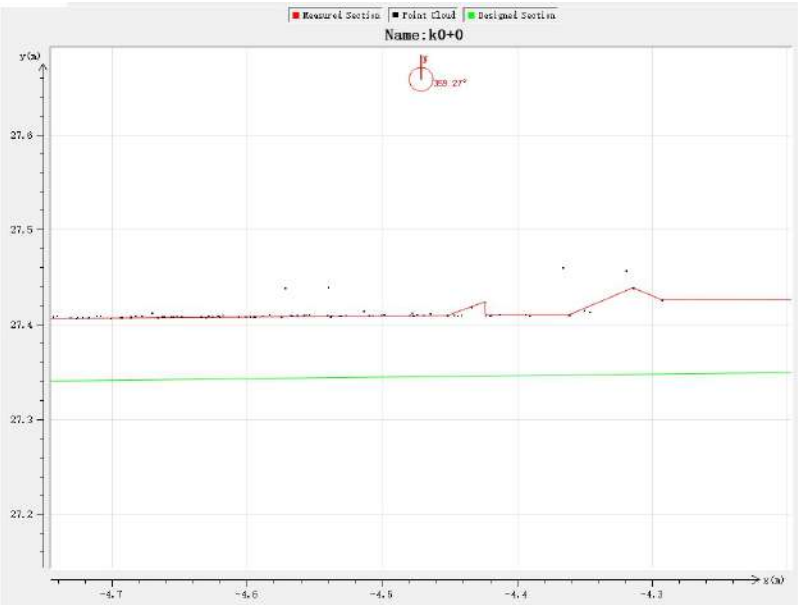
1.Select the design section to be imported.

2.Confirm and adjust the data format of the read-in data.



Parameter dialog box

3.Result View: The green line is the design section, and the red line is the current measured section.



Parameter dialog box

Generating Section Views

- **Generate Road Section Views**

Generate section views of road surfaces. The generation method can be found in the Section Analysis Window Control document.

- **Generate Tunnel Cross-Section Views**

Generate section views of tunnels. The generation method can be found in the Section Analysis Window Control document.

Hidden

Function Description: This option controls the concealment of cross-section correlated vector lines.

Show spindle

Function Description: This option controls the concealment of the vertical reference line.

Show horizontal line

Function Description: This option controls the concealment of the cross-section reference line.

Display measurement section

Function Description: This option controls the concealment of the measurement section.

Display designed section

Function Description: This option controls the concealment of the designed section.

Show all

Function Description: This option controls the display and concealment of all reference lines.

Clear Section

Function Description: These options allow the user to clear cross-section related information.

> Note: The clear function will eliminate the existing actual data, and it is **unrecoverable**, please choose carefully

Clear Central Axis and its Measured Sections

Function Description: This option clears the data related to the vertical reference line.

Clear Cross Sections and its Measured Section

Function Description: This option clears the horizontal reference line and the corresponding measurement section.

Clear Measured Sections

Function Description: This option clears measured sections.

Clear All Sections

Function Description: This option clears all data, including vertical reference lines, horizontal reference lines, measurement sections, and designed sections.

Section 3D Window Control

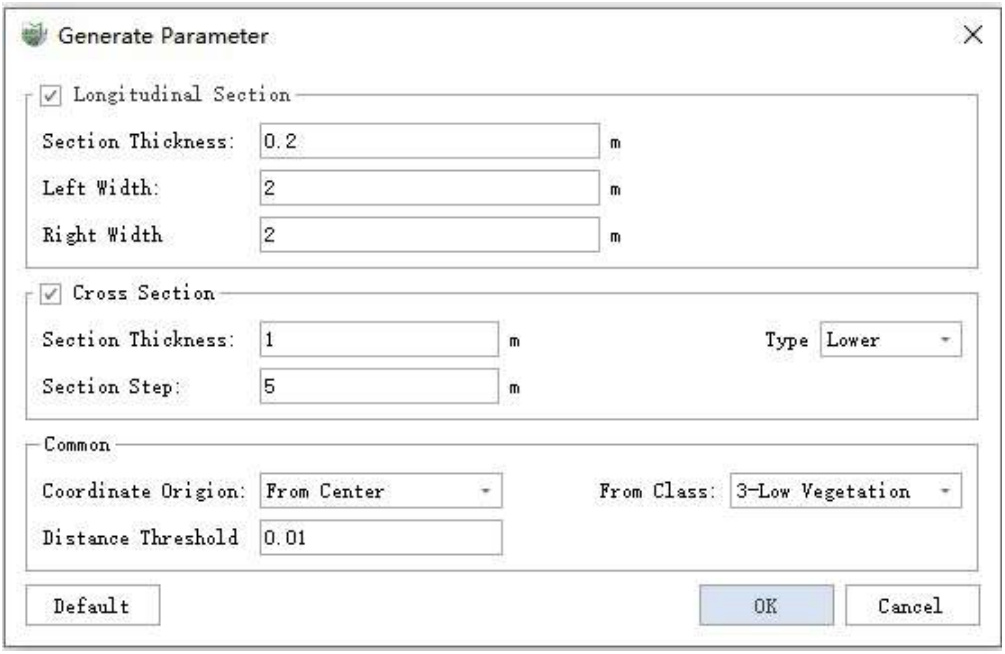
Function Description: Perform calculations, modifications, and save/export operations related to cross-sections.

> Note: This function requires the activation of **Generate Road Section Views** or **Generate Tunnel Section Views**.

Calculation

Function Description: Firstly, generate longitudinal sections based on existing longitudinal reference lines and related parameters. Secondly, generate cross sections based on existing cross reference lines and related parameters.

1.Click the **Calculate** button.



Parameter Dialog Box

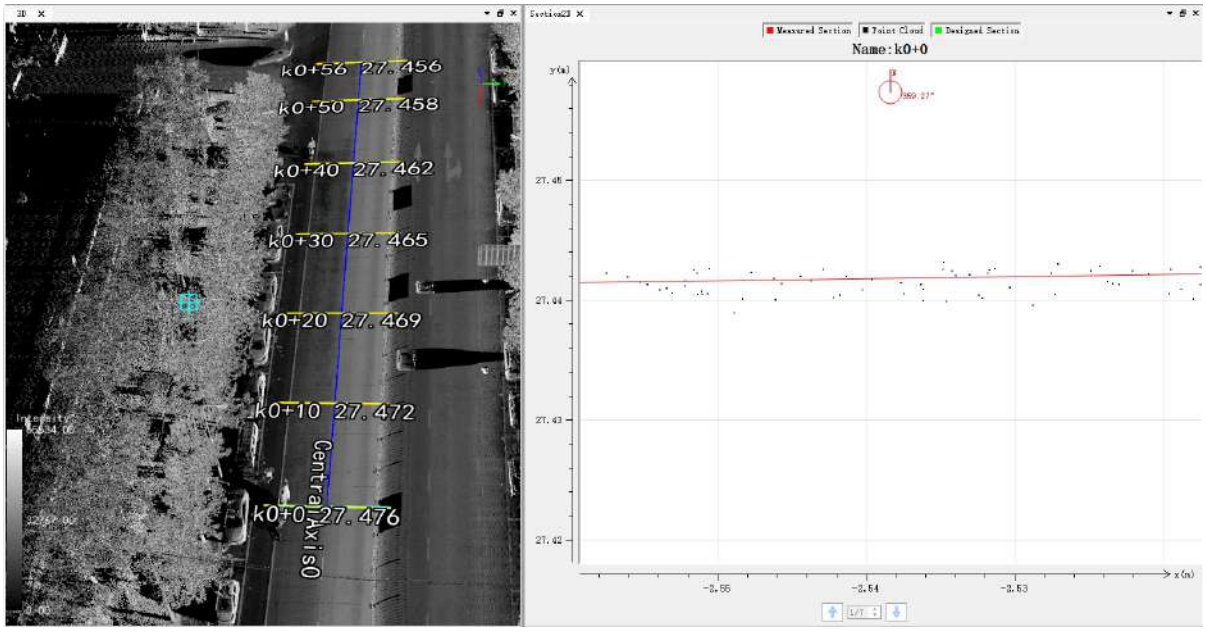
Parameter Description

- **Longitudinal Section Parameters**
  - **Section Thickness**: The width of the point cloud data along the longitudinal reference line for generating longitudinal sections.
  - **Left Width**: The width of the point cloud data on the left side of the longitudinal reference line, used for calculating the RDI parameter.
  - **Right Width**: The width of the point cloud data on the right side of the longitudinal reference line, used for calculating the RDI parameter.
- **Cross Section**
  - **Section Thickness**: The width of the point cloud data along the cross reference line for generating cross section views.
  - **Type**: Generate cross section views based on the height type of the section data.

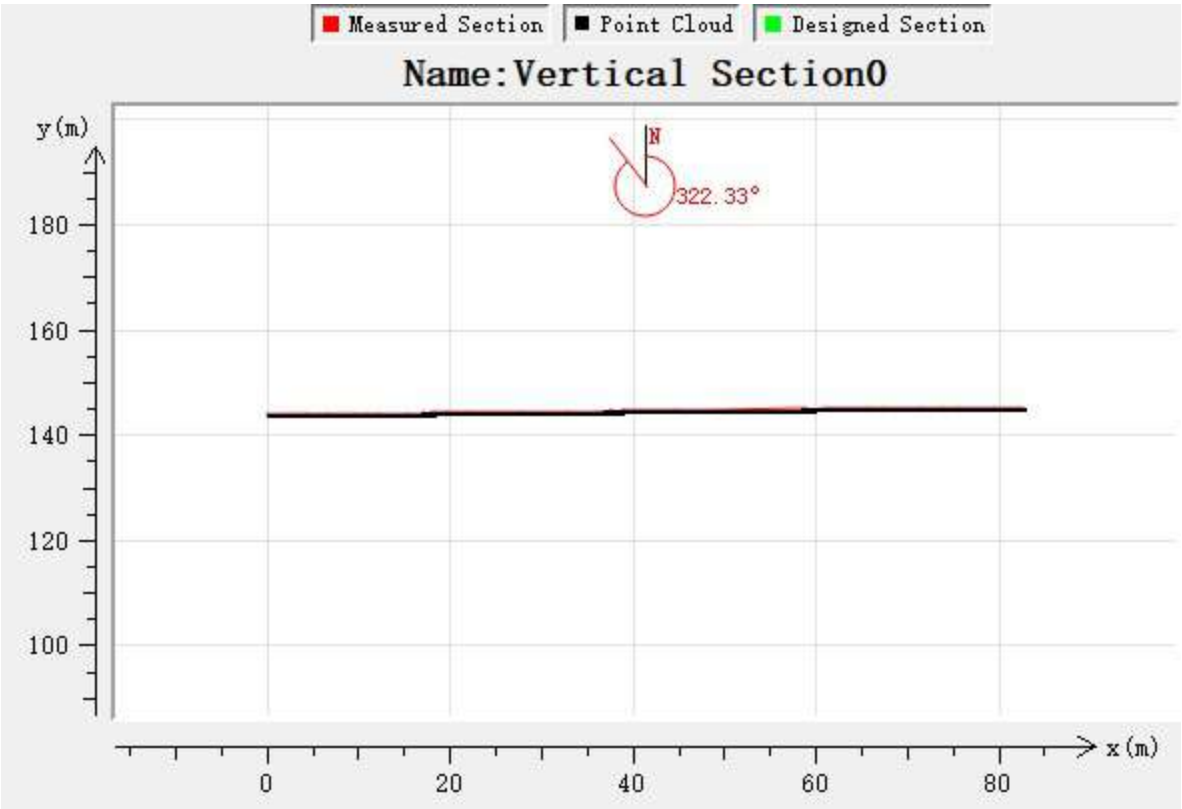
- **Lower Boundary (Default)**: Generate cross section views based on the lowest point of the section data.
- **Upper Boundary**: Generate cross section views based on the highest point of the section data.
- **Section Interval**: The measurement of the section will be simplified based on the set interval and distance threshold. The section will be interpolated using linear interpolation at positions that are multiples of the section interval. If this value is set to 0, the section will be simplified as a whole.

- **General**
  - **Source Class**: The point cloud class to be used for generating the sections. It is important to select the correct source category, as choosing an incorrect category may result in incorrect section generation.
  - **Distance Threshold**: The section will be simplified using the Douglas algorithm based on this parameter. A larger value will result in fewer points and more simplification, while a smaller value will retain more points and details.
  - **Coordinate Origin**: The center position of the section. It is recommended to choose the center. **If road parameters need to be calculated, the center mode must be selected.**

Result Illustration: The red lines represent the generated measured sections.



Cross Section

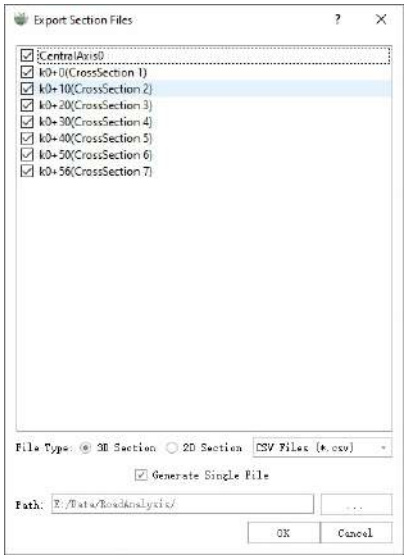


Longitudinal Section

Export Section

Function Description: Allow to export 2D or 3D section lines in various file formats. It supports saving as a single file or multiple files. Currently, formats such as ".csv", ".hdm", ".shp", and ".dxf" are supported.

1.Click on the **Export Section** button.Then the panel the parameter dialog box pops up.

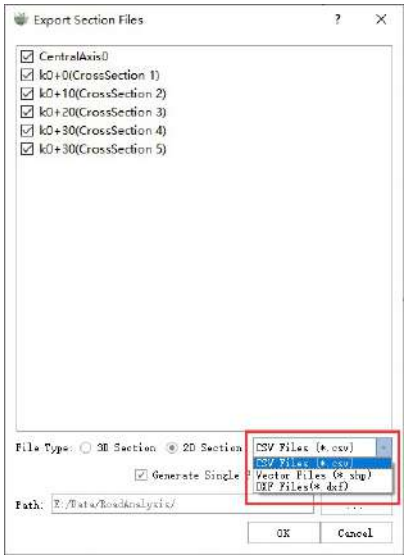


Parameter Dialog Box

Parameter Description

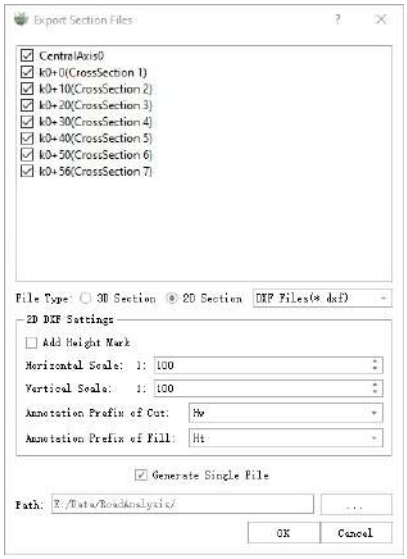
- **csv**：The ".csv" format is a general format used within the software. When importing design sections, the data is read in as a ".csv" format.

- **hdm**：The ".hdm" format supports different software formats such as WGS84 and Southern CASS. You can set the format of the ".hdm" file according to your needs. The parameter settings interface is shown in the following image:



Parameter Dialog Box

- **dxf、.shp**：The ".dxf" and ".shp" formats are vector files that support being imported into third-party software such as AutoCAD to view section results and related data. Here, we take generating a ".dxf" file as an example. The parameter settings are shown in the following image. Set the output path and click OK to proceed.

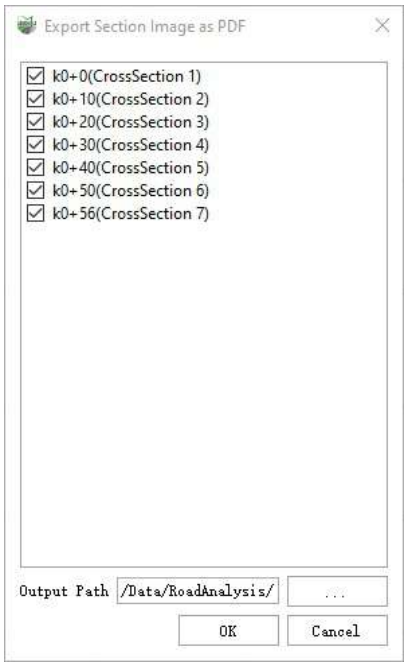


Parameter Dialog Box

Exporting PDF

1.Click on the **Export Pdf** button to open the parameter dialog box.

2.Select the export path and complete the export of section information as a PDF file.

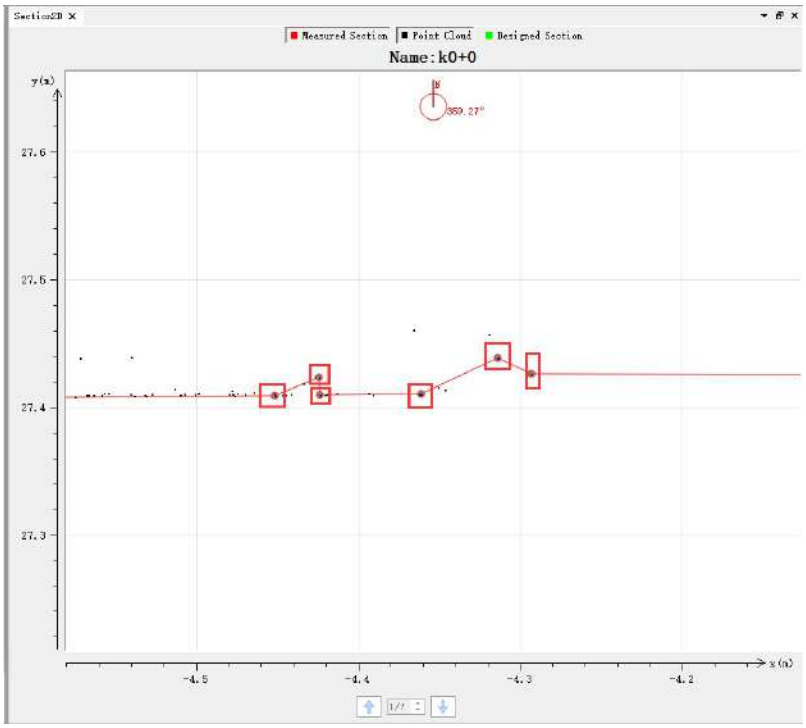


Parameter Dialog Box

Editing Section Nodes

Function Description: The editing section nodes feature allows you to drag and add nodes to edit the section. It is used to edit tunnel boundaries that may contain errors or noise.

- **Editing Section Line Nodes**: Click on the **Edit Section Line Nodes** button. The section line nodes will be highlighted and become editable. Drag the nodes to appropriate positions to edit them. You can see the editing result reflected in real-time in the point cloud view, allowing you to visually inspect the editing result.
- **Adding Section Line Nodes**: Click on the **Add Section Line Nodes** button. The section nodes will be highlighted and become editable. Click on the desired position to automatically add a node. Complete the addition and editing of nodes. Similar to editing section line nodes, the editing result will be displayed in real-time in the point cloud view.

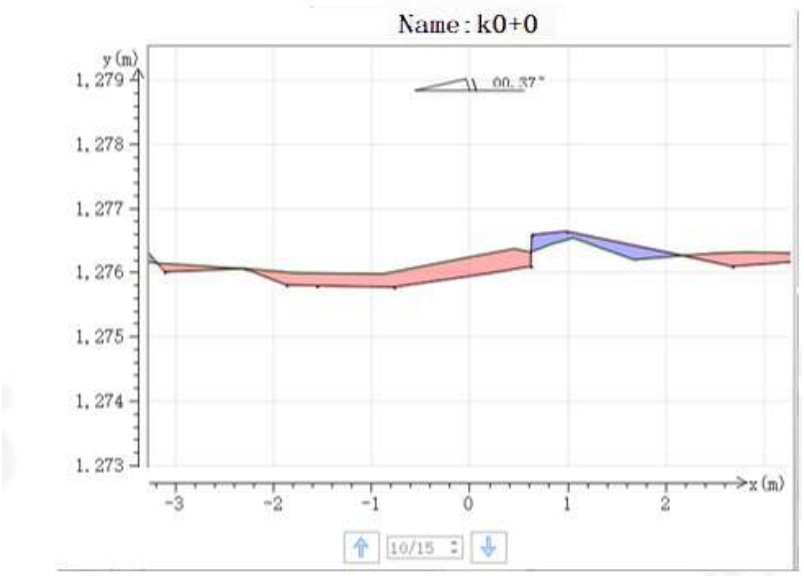


Edit Nodes

Section Comparison

Function Description: Compare the generated road reference section with the road design section and calculate the cut and fill volumes. Cut and fill areas are displayed in different colors (color can be set).

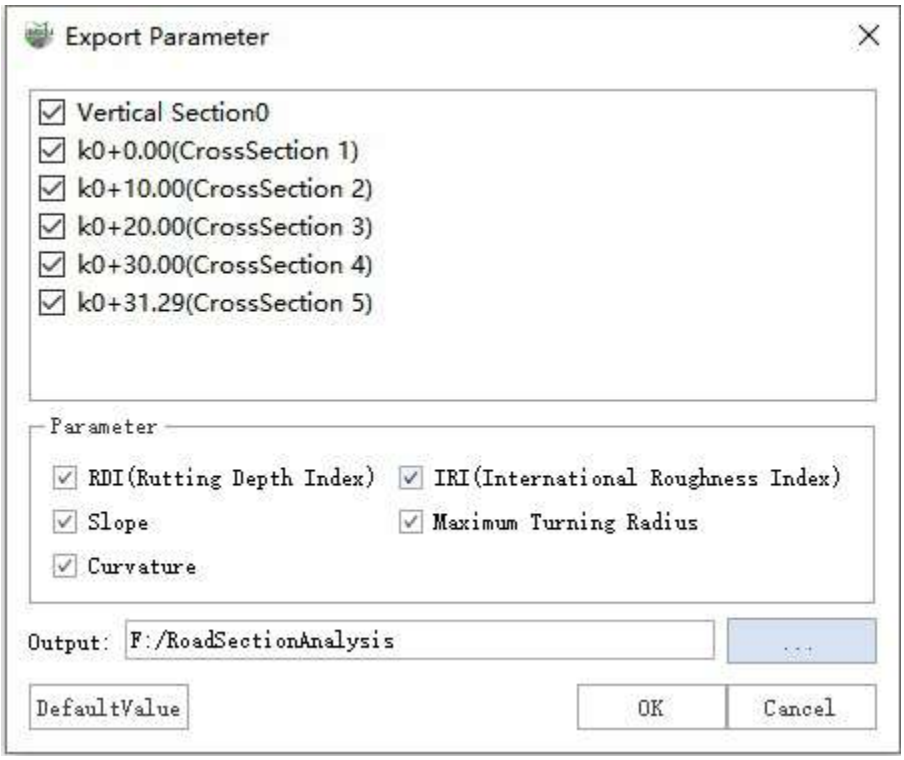
> Note: This feature is only available when design data or design section have been imported.



Section Comparison

Export Parameter

1.Used to export parameters related to longitudinal sections (cross section parameters are not supported at this time). Click on the **Export Parameter** button, and then the Save File Dialog pops up. You can check the parameters and sections to be exported and choose the location to save.



Export Parameters

2.Document content: The parameter file consists of three parts, which are the marker row, the column header row and the data row, as follows.

```
label section parameter
X,Y,Z,international roughness index,slope gradient,max turn radius,curvature
188544.471,3382391.887,142.404,0.000,-0.196,0.000,-0.000
188545.948,3382390.527,142.368,0.000,-0.832,0.000,-0.000
188547.642,3382389.452,142.334,0.000,-2.007,0.000,-0.000
188549.172,3382388.160,142.284,0.000,0.006,0.000,-0.000
188550.791,3382386.987,142.270,0.024,-0.789,0.000,-0.000
188552.339,3382385.716,142.234,0.024,-0.169,0.000,-0.000
188553.927,3382384.504,142.191,0.016,-1.071,0.000,-0.000
188555.572,3382383.362,142.168,0.016,-0.795,0.000,-0.000
188557.164,3382382.151,142.159,0.023,-0.131,0.000,-0.000
188558.740,3382380.919,142.132,0.025,-0.163,0.000,-0.000
188560.279,3382379.641,142.096,0.025,0.407,0.000,-0.000
188561.497,3382378.673,142.056,0.023,-0.638,0.000,-0.000
```

Parameter File Format

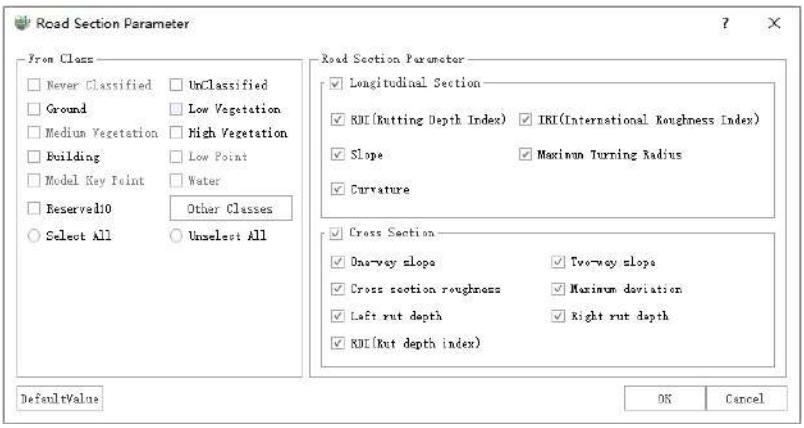
Exporting Comparison Report

Click on the button to export the current section view and information as a comparison report. And then save it to the specified folder.

Calculate Road Parameters

Calculate relevant road parameters based on the cross and longitudinal reference lines.

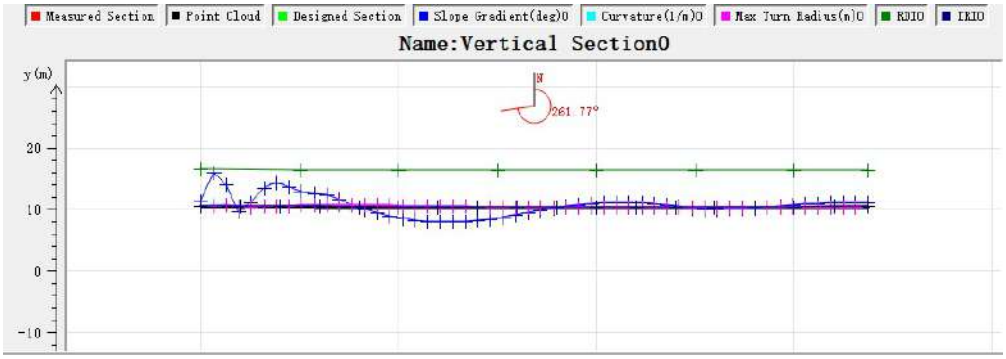
Parameter Dialog Box



Road Parameter Calculation

Parameter Description

- **Source Class**: The point cloud class involved in the computation of road parameters. Choosing the correct source category is important, as choosing the wrong source category may lead to incorrect results.
- **Longitudinal Section Parameters**: Can be displayed on the canvas. Clicking on the respective buttons controls the visibility of the result parameters.
  - **Rutting Depth Index (RDI)**: Reflect the depth of ruts on the road surface.
  - **International Roughness Index (IRI)**: Measure the quality of the road surface and reflect the level of road surface wear.
  - **Grade**: The vertical change in height along the road.
  - **Maximum Turning Radius**: The maximum turning radius of the road in the lateral direction.
  - **Curvature**: The curvature of the road in the horizontal direction.



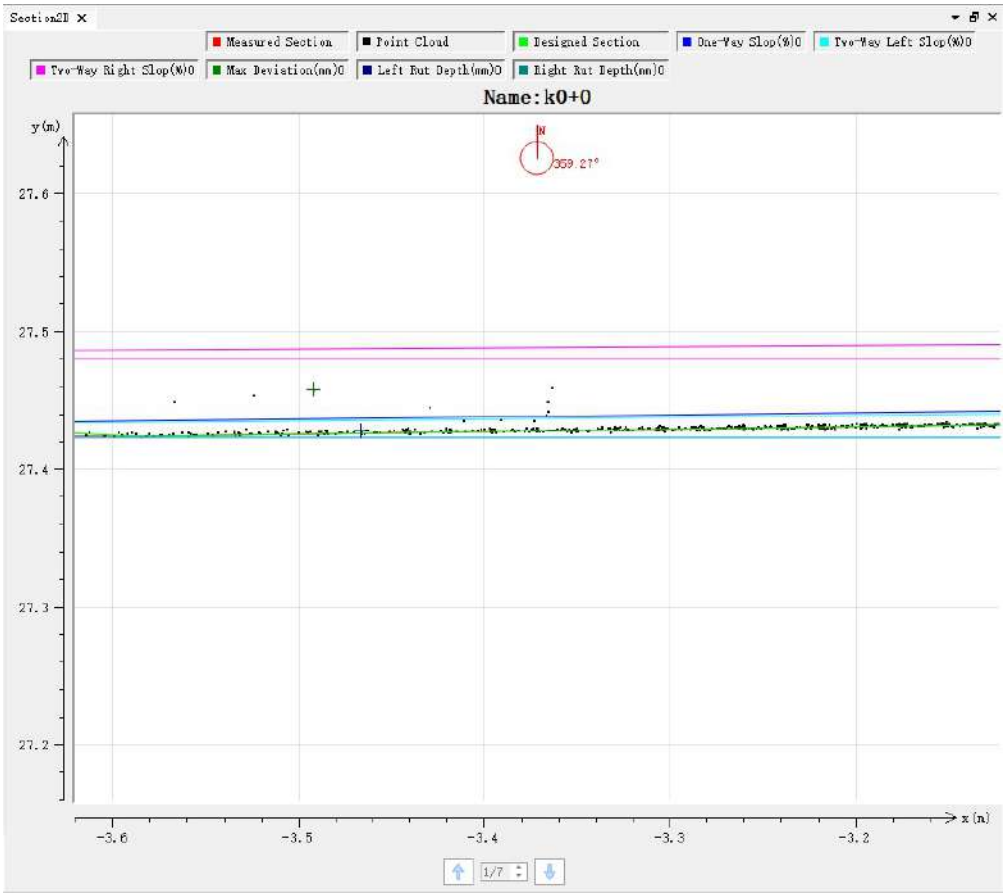
- **Cross-Section Parameters**:

> Can be displayed on the canvas. Clicking on the respective buttons controls the visibility of the result parameters.

  - **One-Way Slope**: The ratio of the height difference between the leftmost and rightmost points of the current cross-section to the road width.
  - **Two-Way Slope**: Calculate one-way slopes for both the left and right sides, using the center point as the dividing point.
  - **Cross-Section Roughness**: The average distance from all points to the line connecting the leftmost and rightmost points, measured in millimeters (mm).
  - **Maximum Deviation**: The maximum distance from all points to the line connecting the leftmost and

rightmost points, measured in millimeters (mm).

- **Left and Right Rutting Depths**: Rutting depths measured using the 3-meter straightedge method on the left and right sides of the road, measured in millimeters (mm).
- **Road Rutting Depth Index**: RDI.



Section Measurement

Measure the length and area of the generated sections.

- **Distance Measurement**: Click on the "Distance Measurement" button to enable distance measurement. You can use the left mouse button to click on the canvas to select points for measurement. While selecting points, you can zoom in, zoom out, and pan the canvas. Double-click with the left mouse button to complete the measurement. Click the "Distance Measurement" button again to end the measurement.
- **Area Measurement**: Click on the "Area Measurement" button to enable area measurement. You can use the left mouse button to click on the canvas to draw a rectangle for measurement. Double-click with the left mouse button to complete the measurement. Click the "Area Measurement" button again to end the measurement.
- **View Parameter Nodes**: Click on the "View Parameter Nodes" button to enable the viewing of parameter nodes. Click on a parameter node on the canvas, and the 3D view will navigate to and highlight the node position, achieving synchronization between the parameter nodes and the 3D view.

Settings

Set the colors for various displayed information according to your preferences.

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

Road Surface

The Road Surface module supports road classification, manual classification, terrain creation, terrain display, and vehicle passability analysis based on terrain and trajectory.

It includes the following:

- Classification Tools
  - Classification by Attribute
  - Classification by Ground Points
  - Classification by Deep Learning
- Manual Classification
- Terrain Creation
- Display
- Vehicle Simulation

Terrain Creation

- Road Surface Key Points By Grid
- Create TIN

Road Surface Key Points By Grid

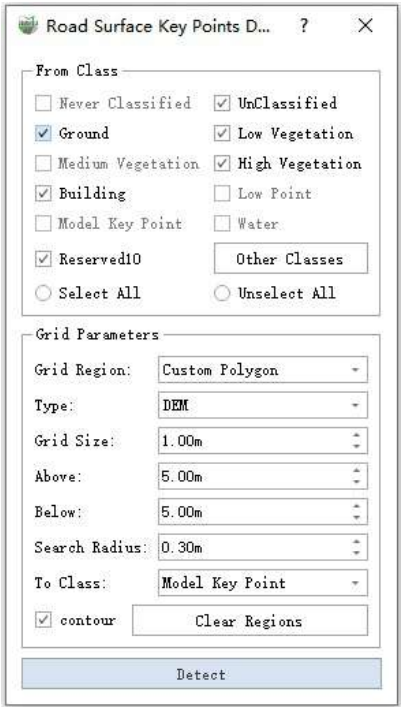
Function Description: Generate uniform grid key points for generating terrain based on the selected area or file.

> Note: It is strongly recommended to perform point cloud classification in advance to get the correct road surface TIN.

Steps

1.Click the **Road Surface Key Points By Grid** button.

2.The **Road Surface Key Points By Grid Extraction Dialog** pops up:

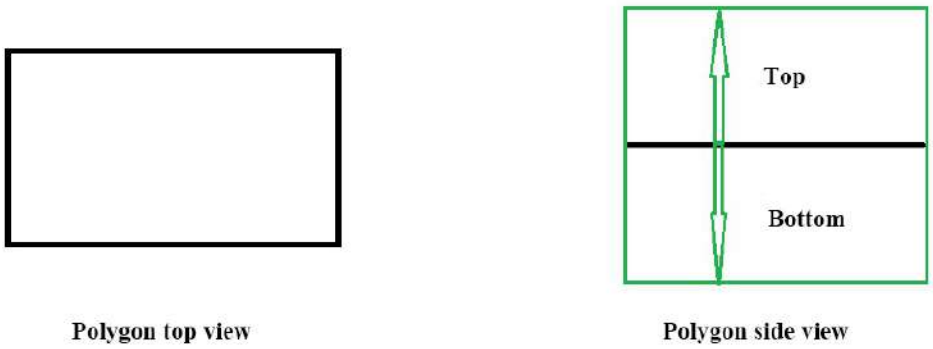


Road Surface Key Points Dialog

Parameter Explanation:

- **From Class**: Select the point cloud class involved in generating key points. To accurately express the road surface, it is recommended to perform point cloud classification in advance and only select the category where the road surface is located.
- **Grid Region**: Choose the range of the area where you need to generate key points.
  - **Custom Polygon**: Users can manually click on the point cloud according to their needs. Click the left mouse button to add a point, and double-clicking the left mouse button ends, thus forming a polygon.
  - **Existing Polygon**: Click the left mouse button to select an existing polygon as the grid area.
  - **All Files**: This option means to extract key points from all point clouds of the selected source category in the current project, without manually selecting/generating polygon areas.
- **Type**: Indicate the type of key points used to finally construct TIN. The default is DEM.
  - **DEM**: In this mode, the generated key points can be used to generate DEM (Digital Elevation Model).
  - **DSM**: In this mode, the generated key points can be used to generate DSM (Digital Surface Model).

- **Grid Size**: This function uses a uniform grid to generate key points, and this option represents the size of a single cell. The default is 1.0 meters.
- **Above, Below**: Used in conjunction with the grid area to capture the three-dimensional area above and below the grid area. See the illustration below for specifics.



Three-dimensional area illustration

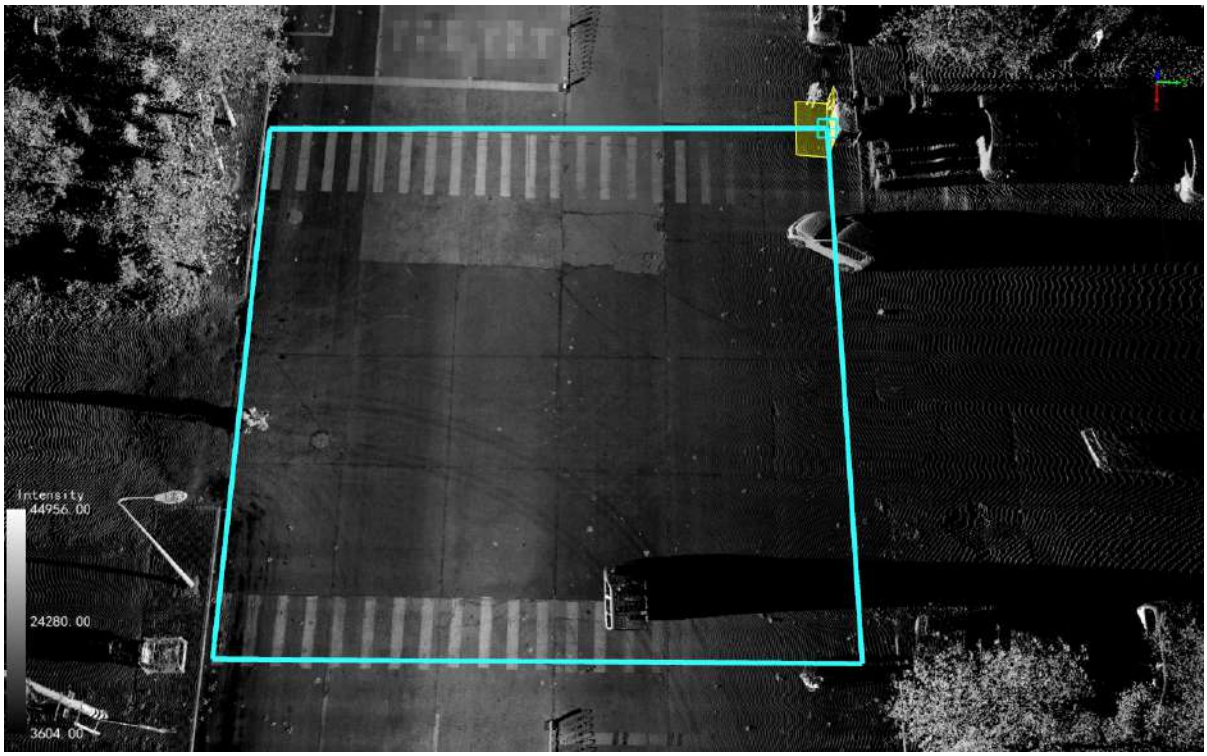
- **Search Radius**: Each grid key point needs to search around the point cloud when it is close to the ground. This parameter is used to determine the radius.
- **To Class**: The class in which the detected key points are to be saved.
- **Contour**: If checked, key points are also generated under the polygon contour line. If unchecked, key points only exist at uniform grid points.
- **Clear Regions**: Clear the custom polygon area.
- **Detect**: Start the detection function.

3.Grid Area Definition

- In the custom polygon mode, the left mouse button is clicked in sequence to select the road surface point cloud, and the mouse is double-clicked to end the polygon production.

> Click the left mouse button to add a polygon point; right-click the mouse to go back to one point; double-click the left mouse button to end the drawing.

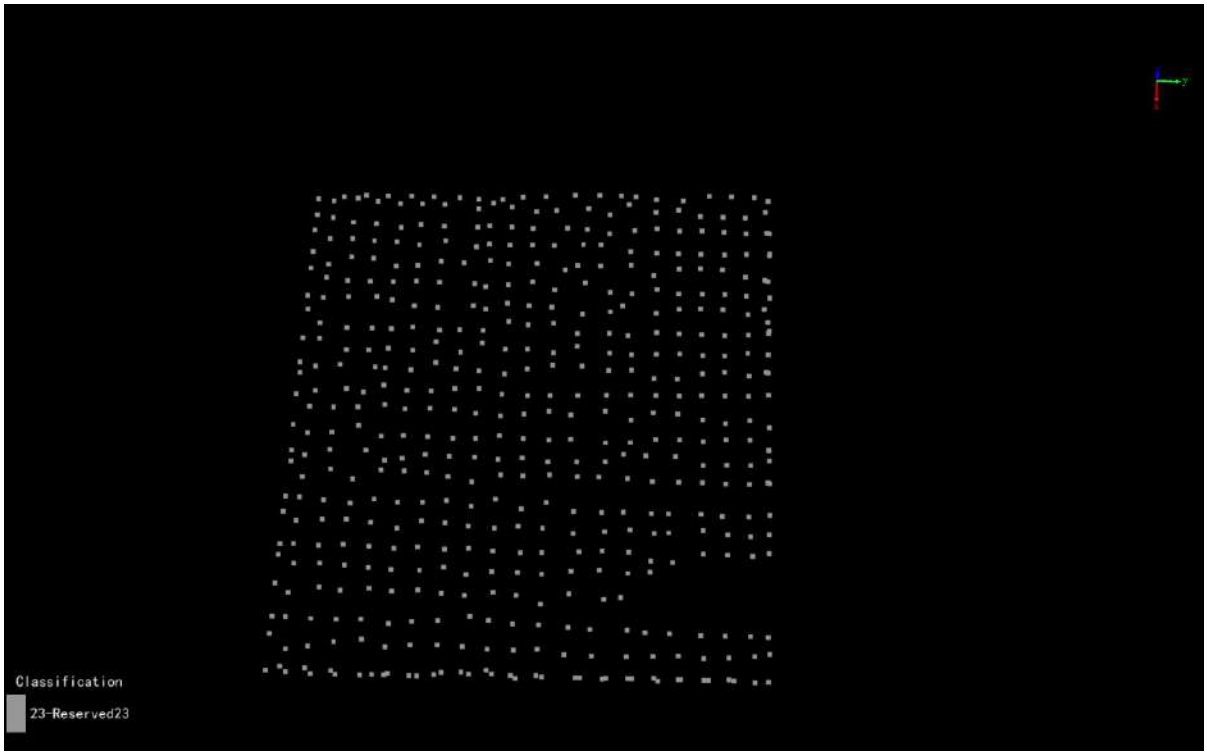
- In the existing polygon mode, click the left mouse button to select an existing polygon. Multiple selections can be made.
- In the all-files mode, there is no need to define the grid area.



Grid Area

4.Click the **Detect** button to open the probe and wait for the probe to finish.

5.Result check: Render the point cloud according to the class information and check whether the key point results are reasonable or have any errors. For any errors, you can use the sorting tool by attributes in the sorting tool (which can perform class reclassification and key point clearing operation), as well as use manual sorting to correct the key points.



Results Show

Create TIN

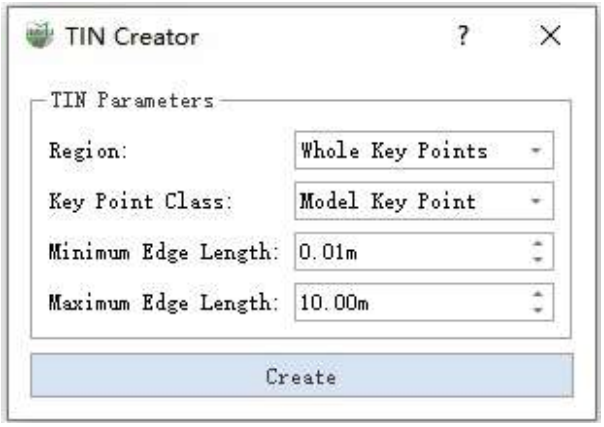
Function Description: Using existing road surface key points to construct TIN (Triangulated Irregular Network).

> Note: To obtain the correct road surface TIN, please ensure the correctness and completeness of the key points.

Steps

1.Click the **Create TIN** button.

2.The **Create TIN Dialog** pops up:



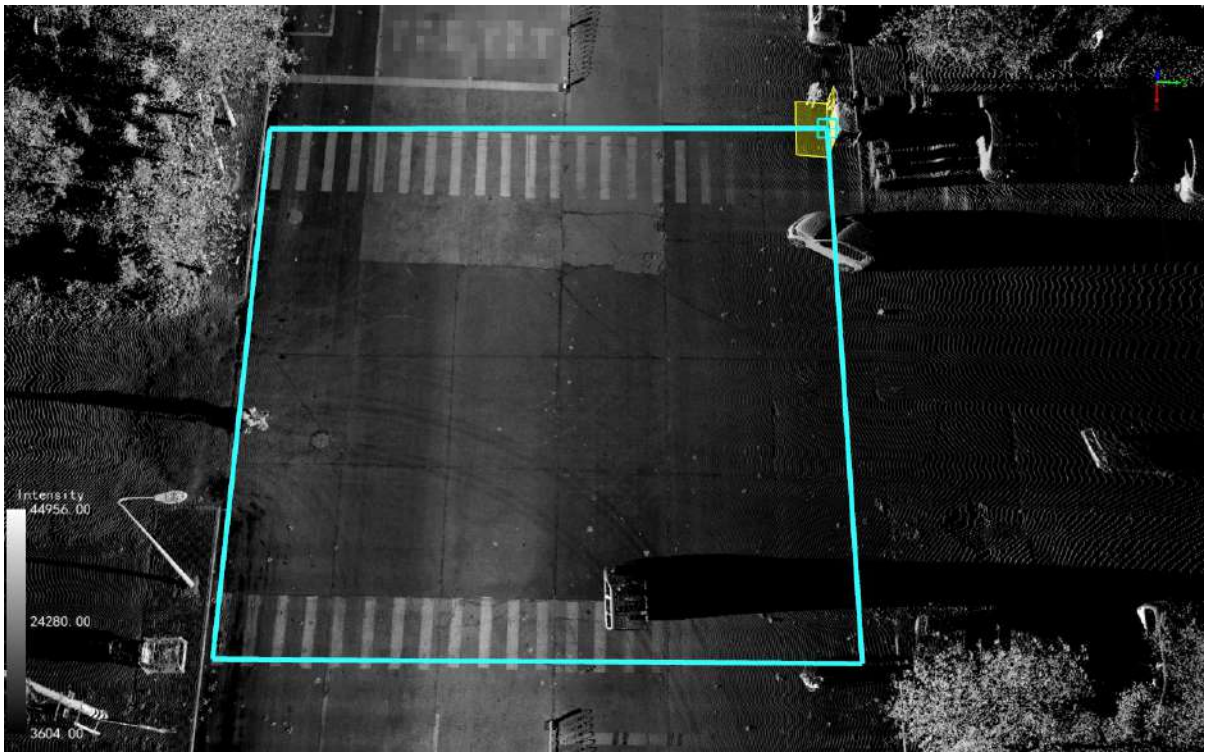
TIN Construction Dialog

Parameter Explanation:

- **Region**: Select the range of the area where you need to construct TIN.
  - **Whole Key Points**: All the points in the selected key point class to participate in the construction of TIN. No manual participation is required.
  - **Polygon Region**: Define a polygon area, the key points located in the area participate in the construction of TIN.
- **Key Point Class**: Select the key point class to participate in the construction of TIN.
- **Minimum Edge Length**: If there is an edge in the TIN mesh triangle that is shorter than the minimum edge length, the triangle will be deleted. Combined with the maximum edge length parameter, it ensures that the generated TIN does not contain triangles that are too short or too long.
- **Maximum Edge Length**: If there is an edge in the TIN mesh triangle that is longer than the maximum edge length, the triangle will be deleted. Combined with the minimum edge length parameter, it ensures that the generated TIN does not contain triangles that are too short or too long.
- **Create**: Generate TIN according to parameters.

3.Grid Area Definition

- **All Key Points Mode**: No manual participation is required.
- **Polygon Area Mode**: Click the left mouse button to add a polygon point; right-click the mouse to go back to one point; double-click the left mouse button to end the drawing.



Grid Area

4.Click the **Create** button to start to build, and wait for the detection to end.

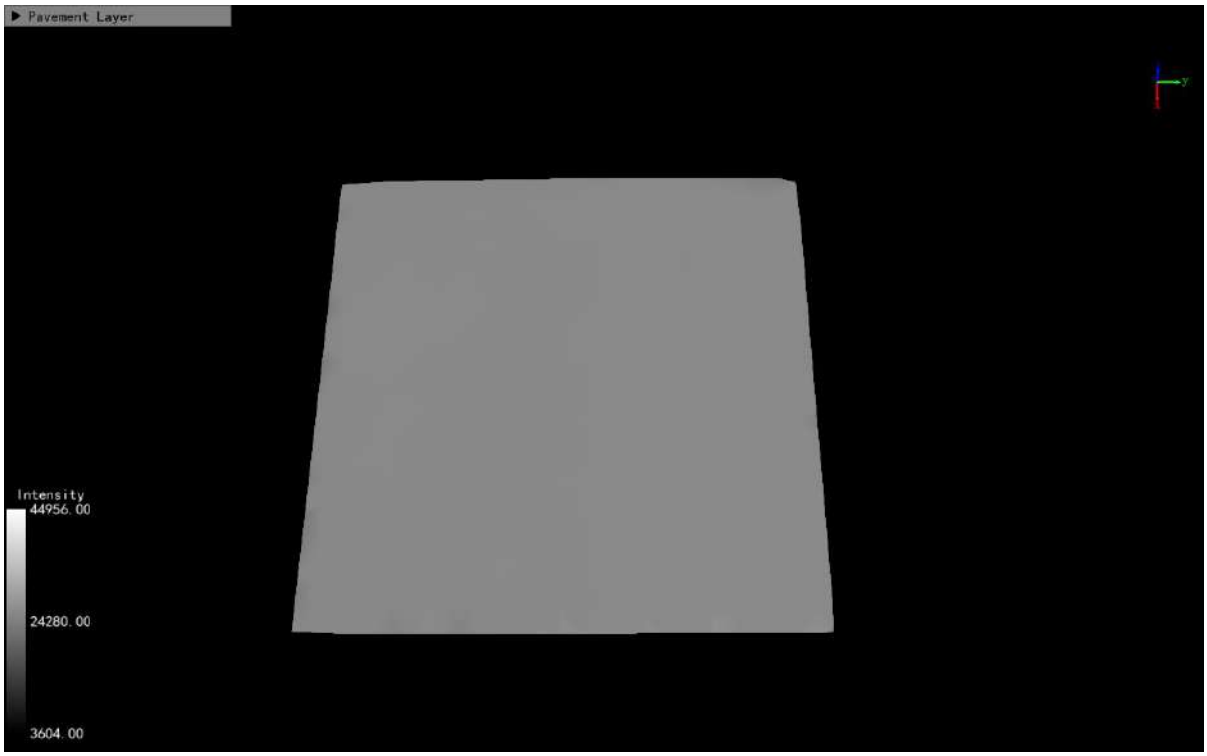
5.Result display, TIN can be viewed in different modes through Display.

Display

Display by Solid Model

Function Description: Display the TIN as a solid model based on an opaque surface.

Click on the **Display by Solid Model** button. Displayed by solid model:

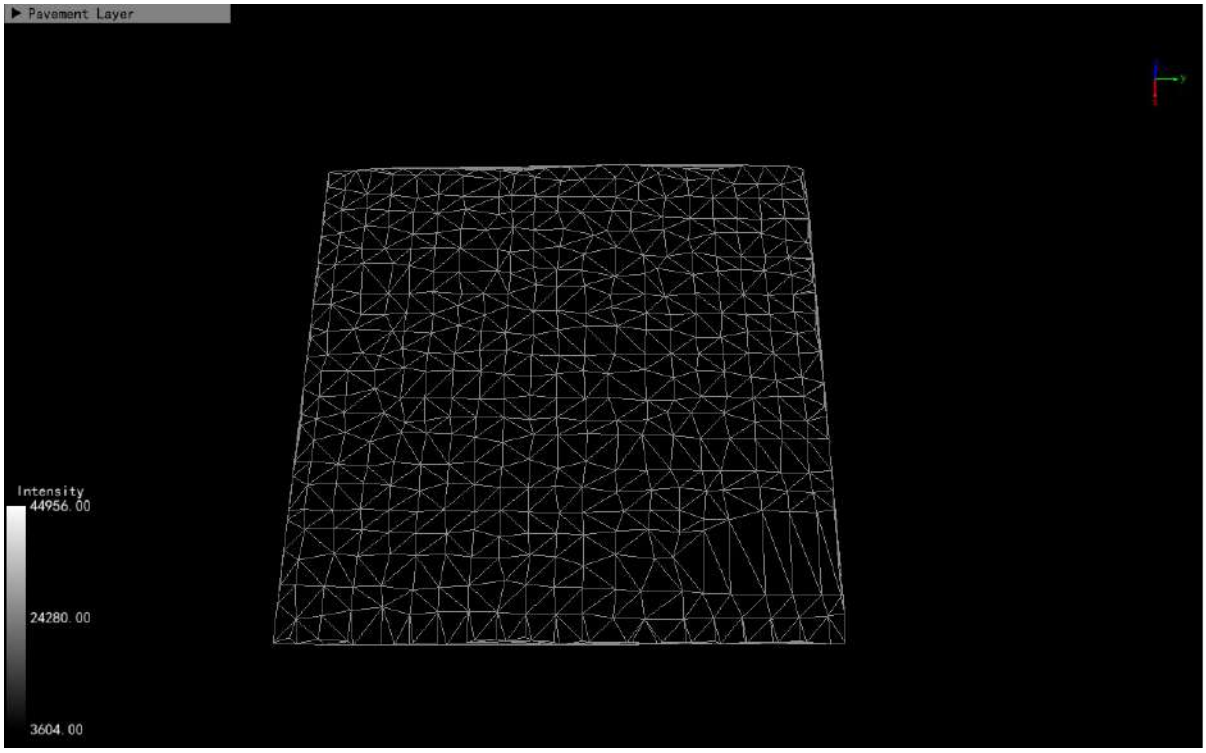


Display by Solid Model

Display by Wireframe

Function Description: Display the TIN as a frame by showing the mesh edges based on triangle vectors.

Click on the **Display by Wireframe** button. Results are displayed by Wireframe:

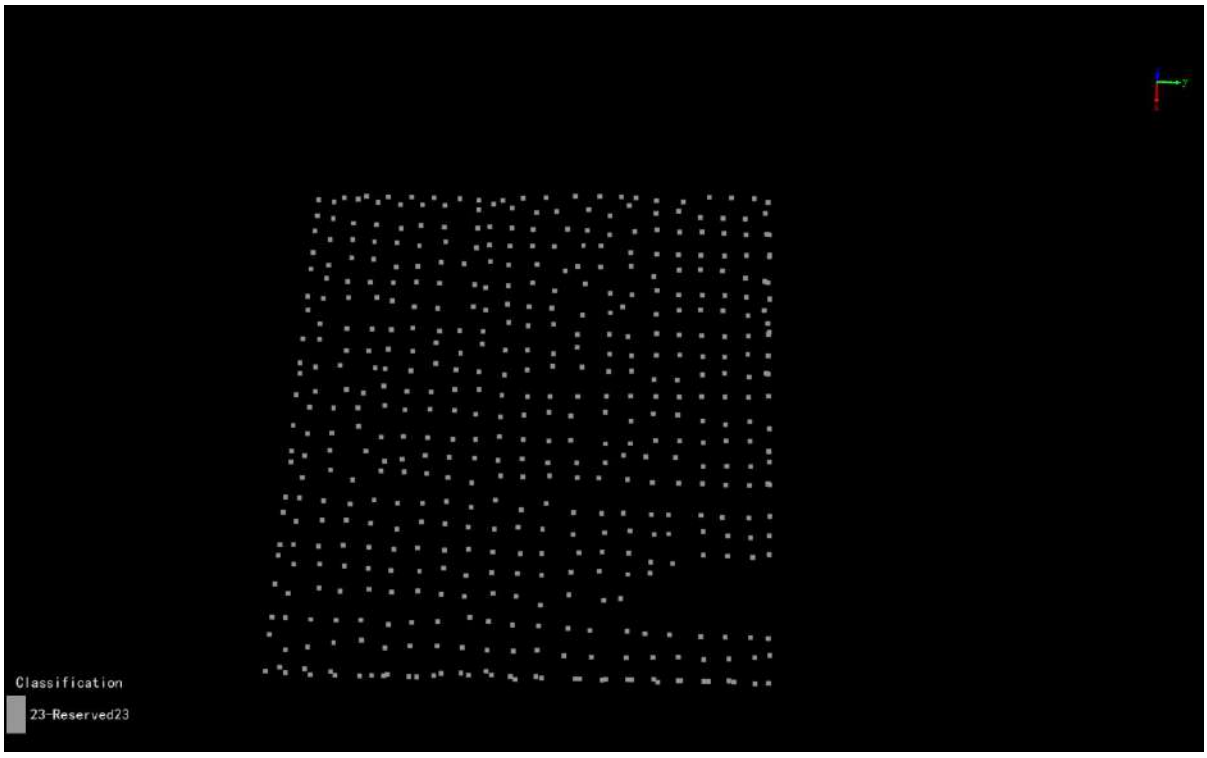


Display by Wireframe

Display by Vertices

Function Description: Display the TIN by showing the keypoints.

Click on the **Display by Vertices** button. Results are displayed by Vertices:



Display by Vertices

Clear All TIN

Function Description: Clear all the TIN in the current project.

Click on the **Display by Solid Model** button. That is, you can clear all TIN.

> Please note that this operation cannot be undone, so use it with caution.

Vehicle Passability Analysis

- Vehicle Passability Analysis
- Vehicle Designer

Vehicle Passability Analysis

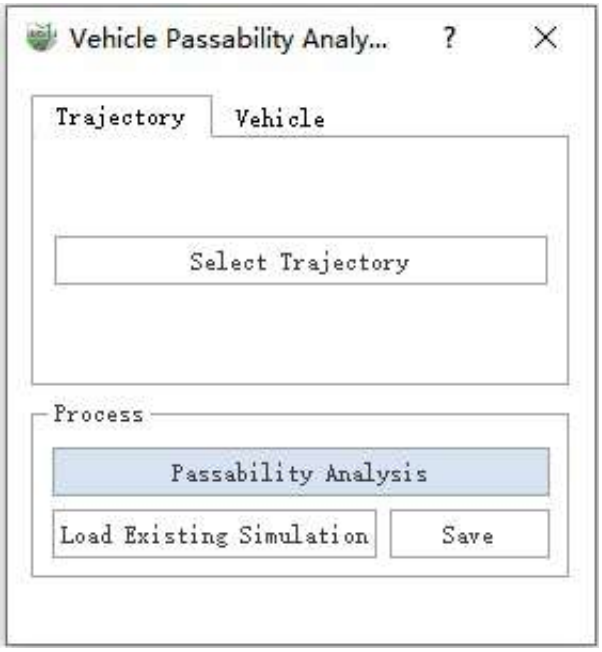
Function Description: Based on the road surface's TIN and trajectory data, combined with the vehicle's size information, simulate the vehicle's driving state and perform collision detection.

> Please note that it is necessary to construct the road TIN and input vehicle size information in advance.

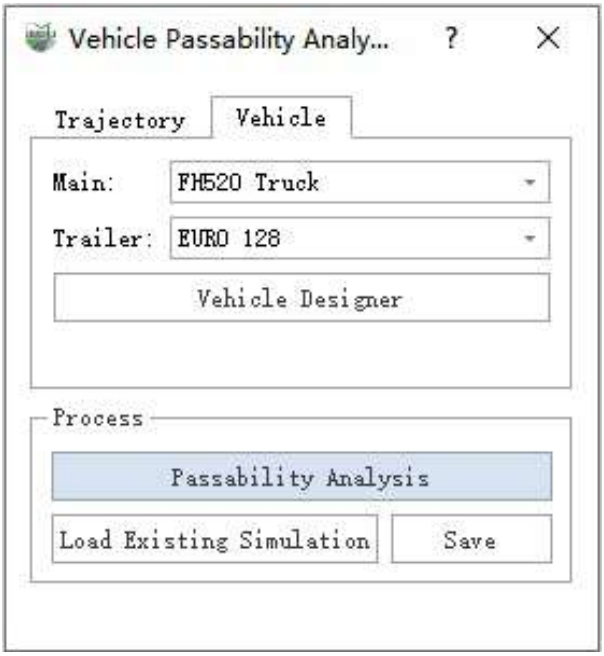
Steps

1.Click on the **Vehicle Passability Analysis** button in the road surface panel.

2.The **Vehicle Passability Analysis Dialog** pops up:



Vehicle Passability Analysis Dialog - Track Selection

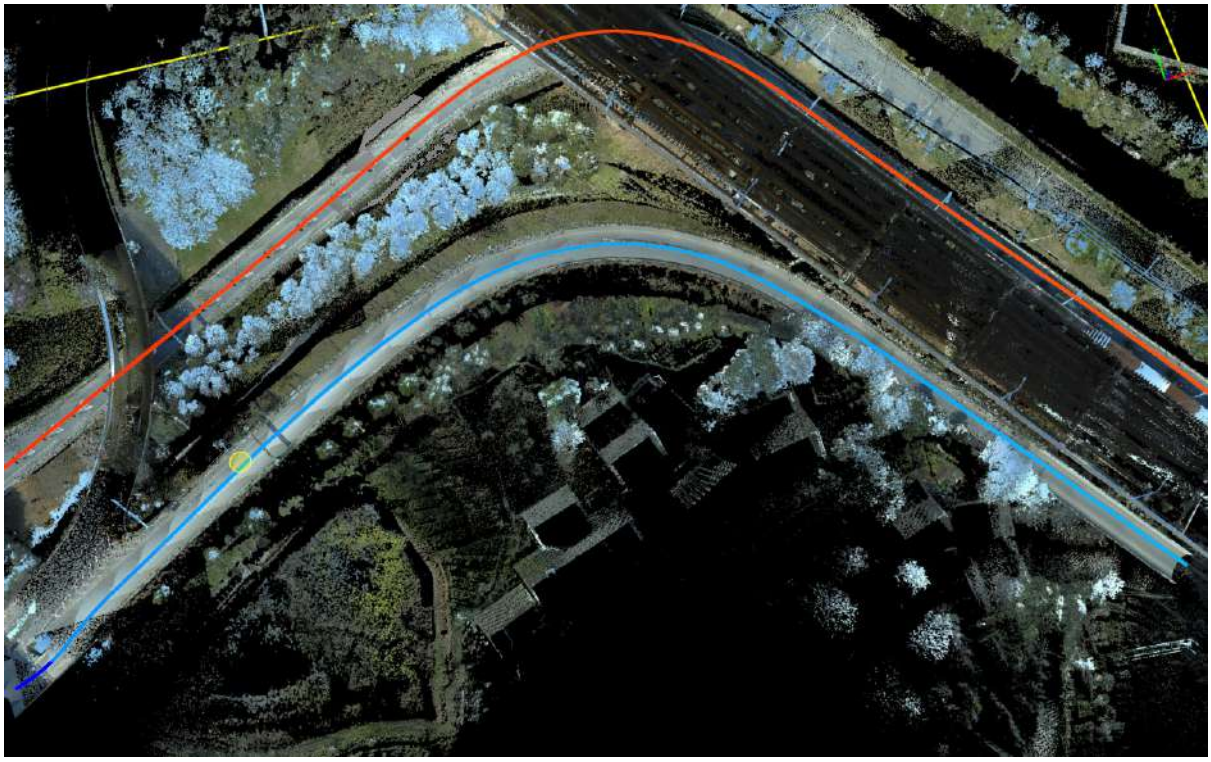


Vehicle Passability Analysis Dialog - Vehicle Size Selection

Parameter Description:

- **Select Trajectory**: Select a trajectory to simulate the vehicle's state within its range.

> Note: To ensure the accuracy of the simulation, please select a continuous trajectory without U-turns when choosing the trajectory.



Trajectory Selection - Selecting the Start of the Trajectory Segment

- **Vehicle**: Select or design the vehicle model needed for simulation.
  - **Main**: The head of the tractor or other integrated vehicle body.
  - **Trailer**: The carriage of the tractor.

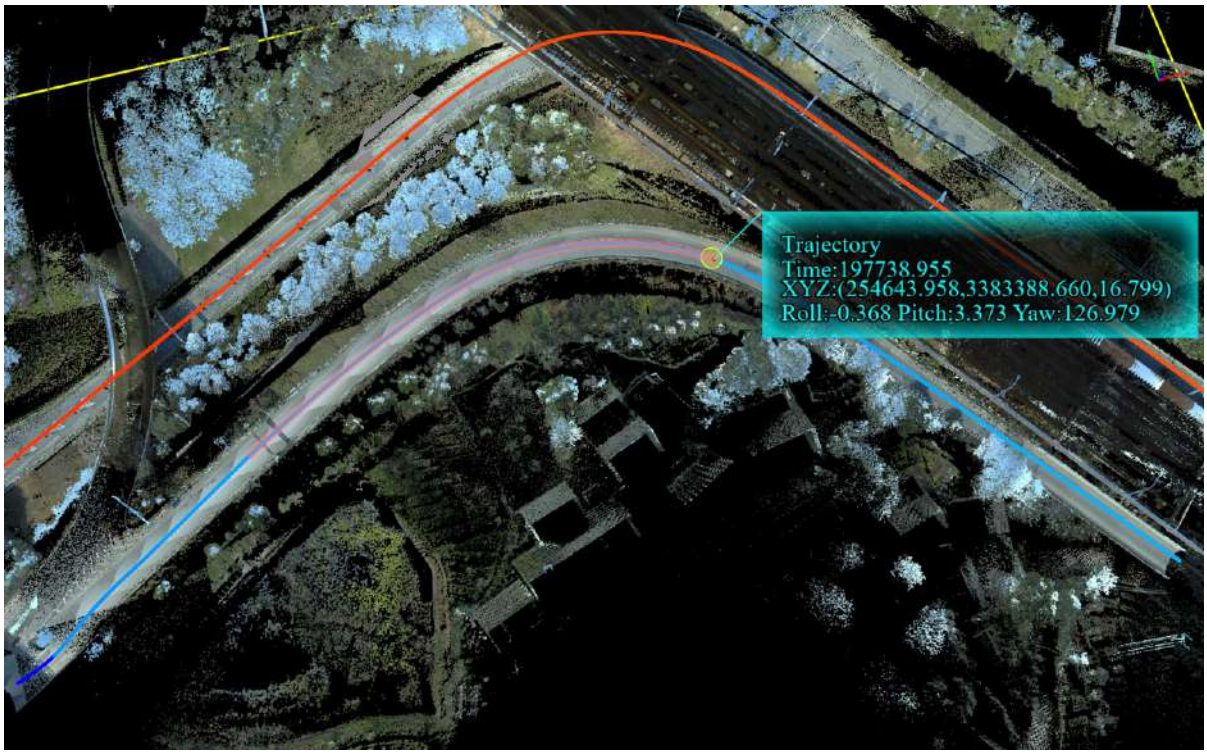
  > For passenger vehicle-style integrated vehicles, the carriage should be set to empty.
- **Vehicle Designer**: Design custom vehicles based on actual parameters, for details, please see Vehicle Designer.
- **Passability Analysis**: Perform analysis based on set parameters.
- **Load Existing Simulation, Save**: The path simulation of the vehicle in a single analysis can be stored as an independent simulation file. This file can be reloaded for vehicle simulation.

3.Trajectory Selection

Left-click to select the starting point of the trajectory segment:

Trajectory Selection

Left-click to select the endpoint of the trajectory segment:

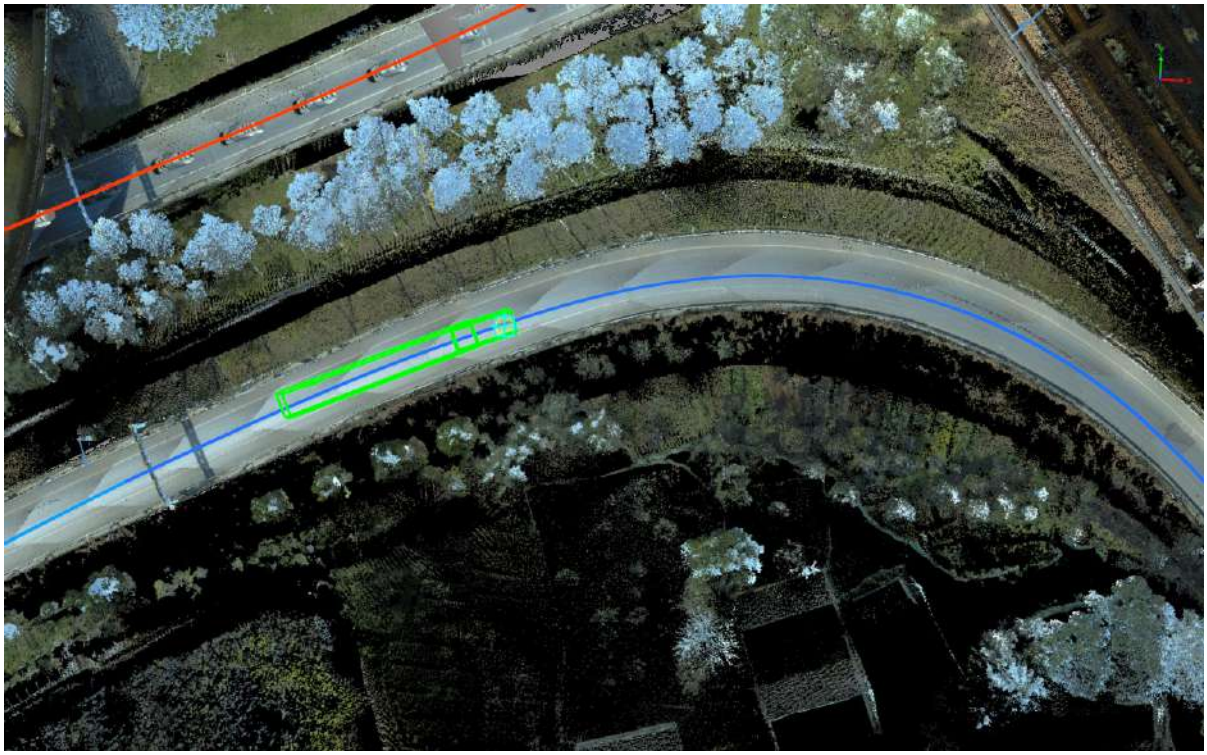


Trajectory Selection

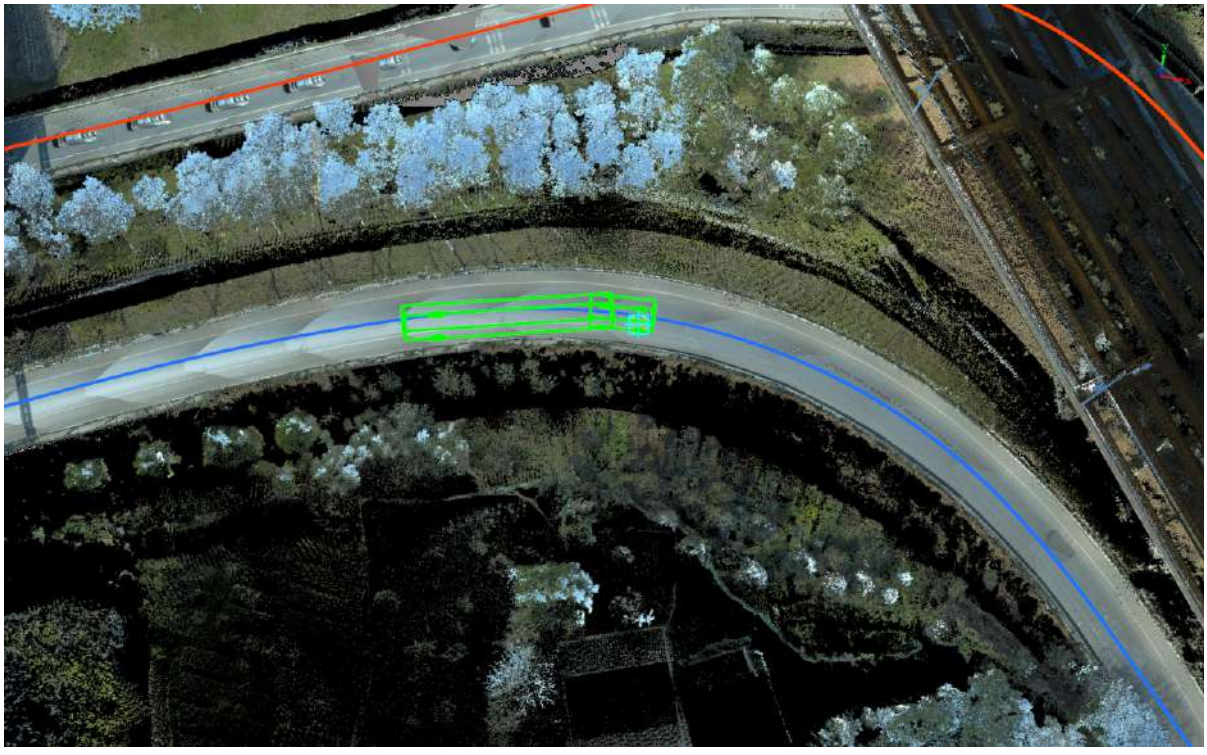
4.Select the vehicle model.

5.Automatic analysis.

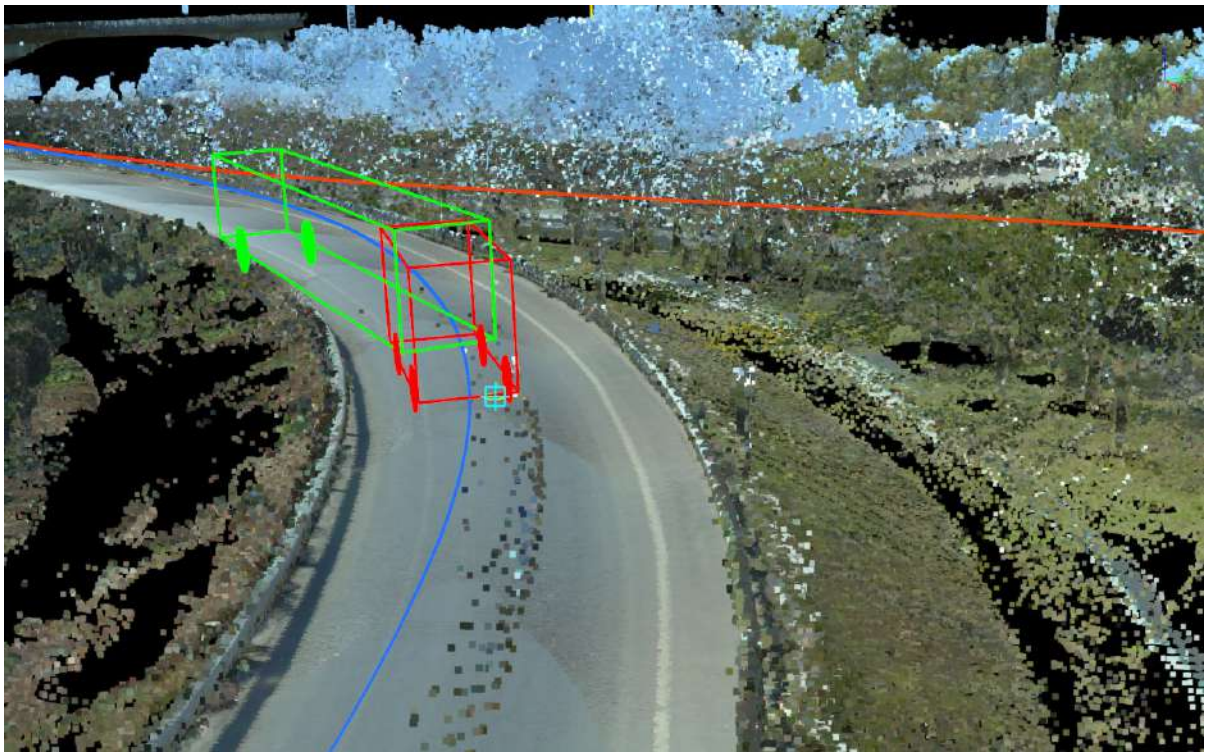
6.Result Viewing: Drag the mouse along the selected trajectory segment to see the wireframe model of the vehicle and the vehicle state at the nearest trajectory point currently pointed to by the mouse. At the same time, if the vehicle collides with the point cloud, the wireframe model of the vehicle will be marked in red, and the collision points will be saved in the Addition Attribute Cycle.



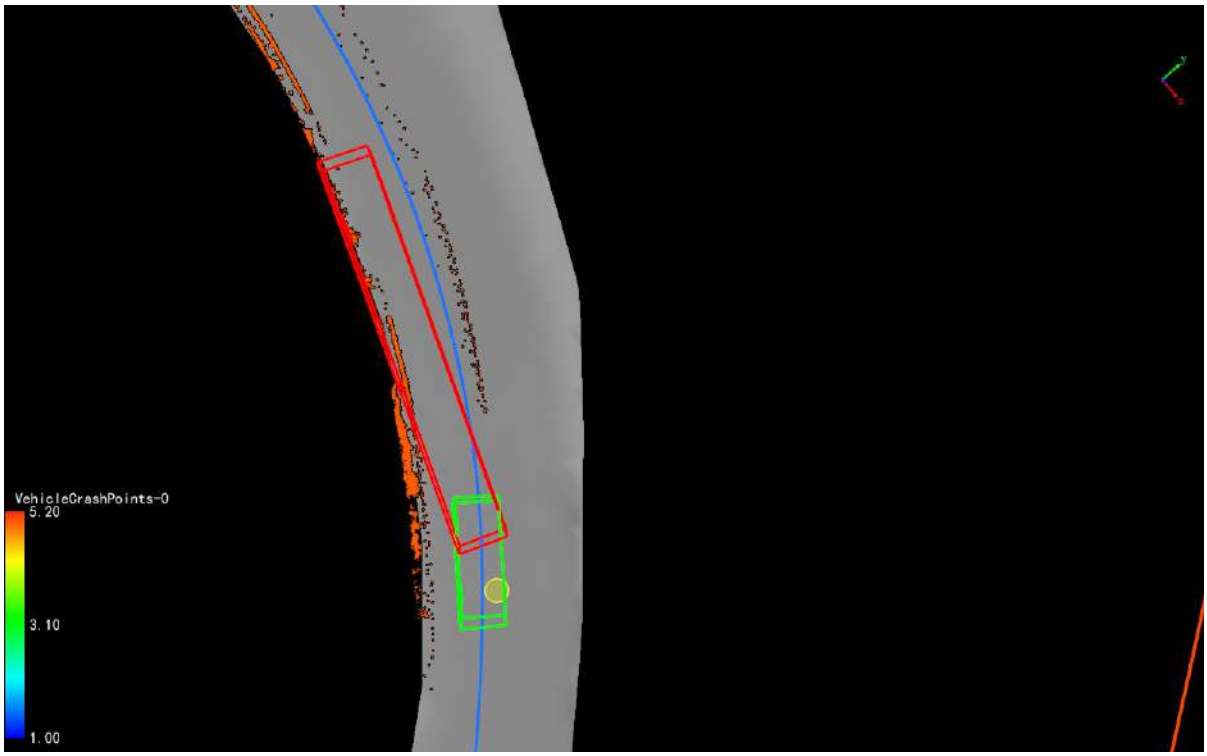
Vehicle Posture at Non-Collision Position 1



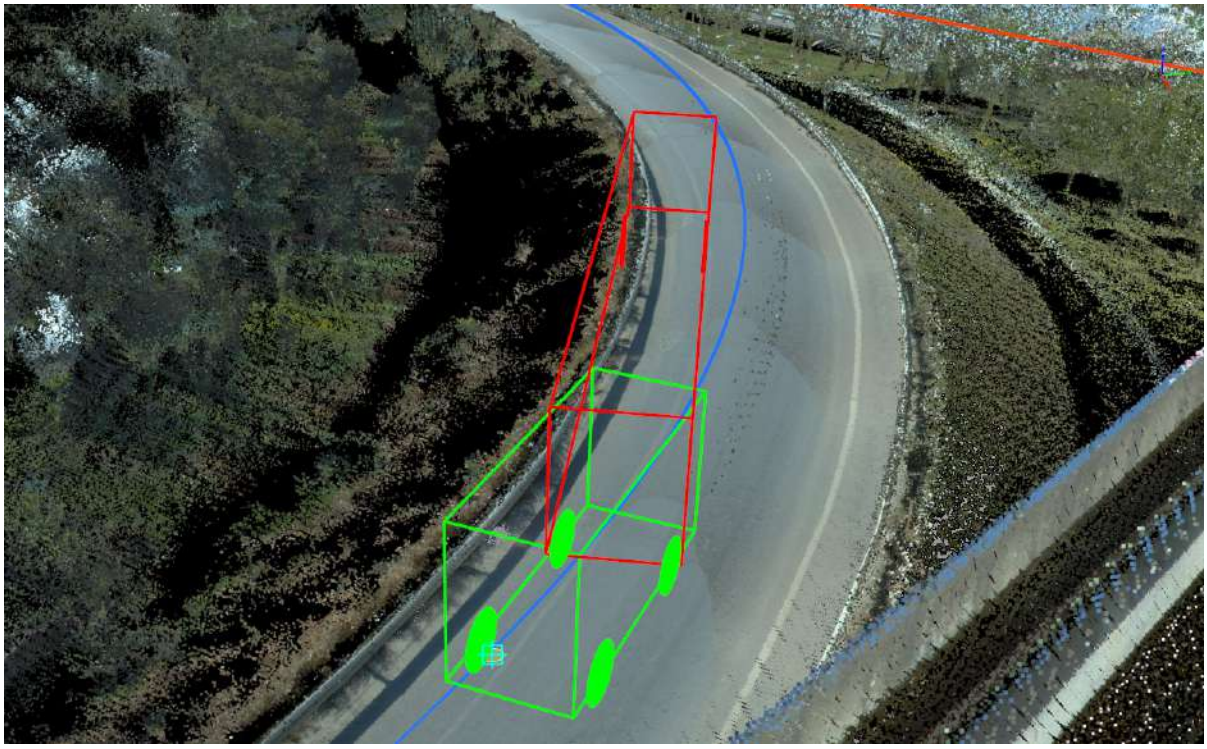
Vehicle Posture at Non-Collision Position 2



Collision Position



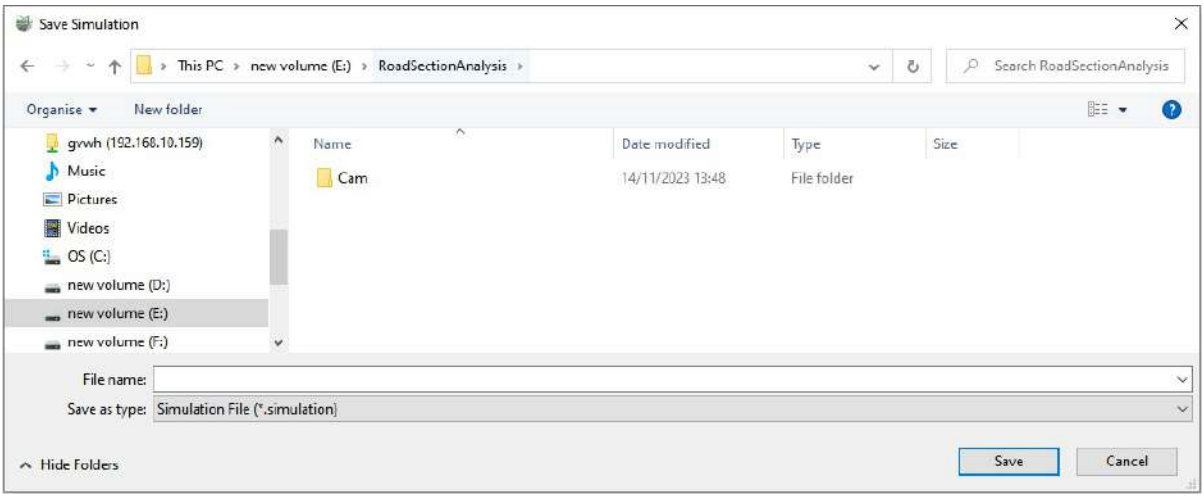
Problem Point Cloud and Vehicle Posture at Collision Position



Problem Point Cloud and Vehicle Posture at Collision Position

7.Save Simulation Result: Click the save button to record the vehicle simulation process of this analysis as an independent file (.simulation).

> Note: The simulation result only records the vehicle posture, not the collision point cloud.



Problem Point Cloud and Vehicle Posture at Collision Position

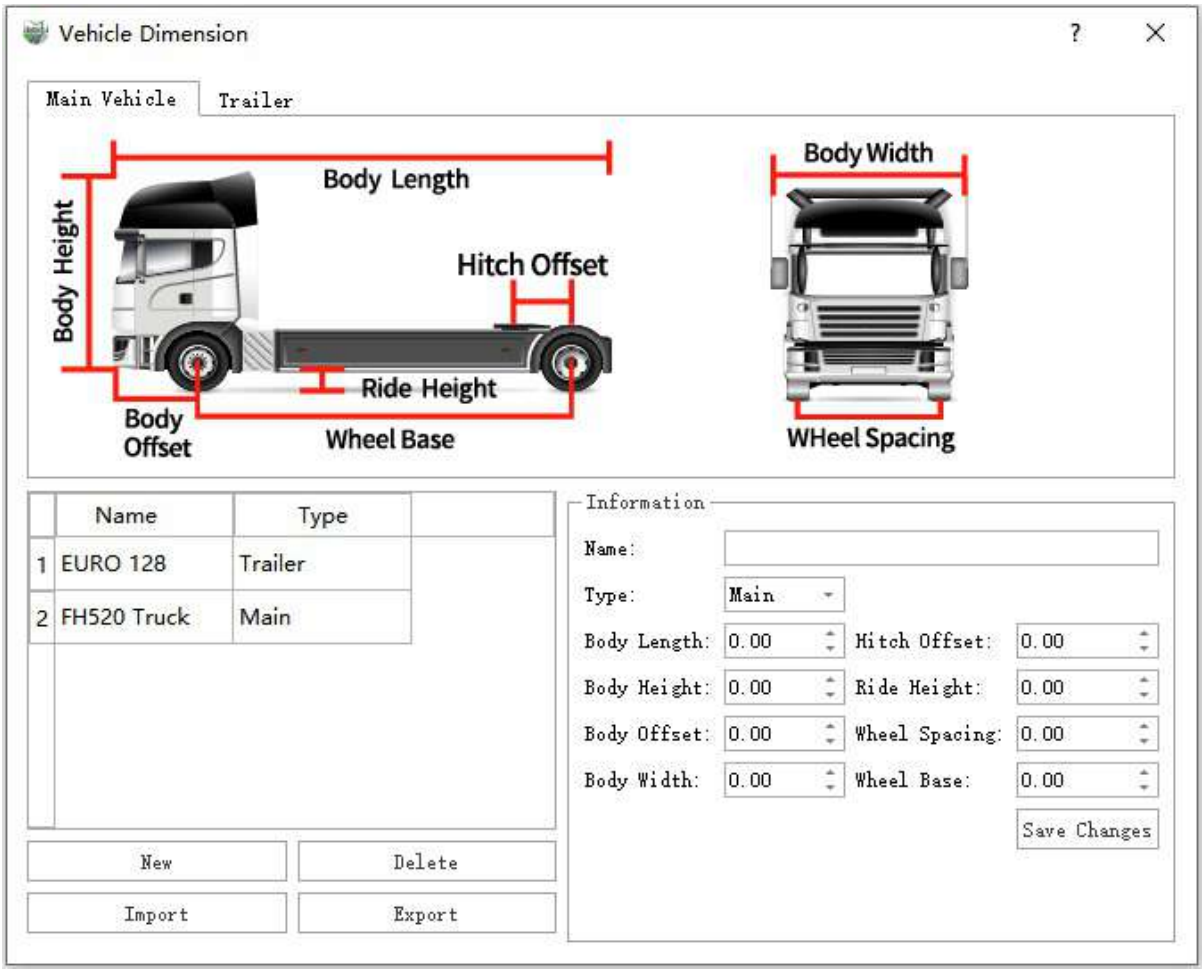
8.Load existing simulation files.

> Note: The simulation result only records the vehicle posture, not the collision point cloud.

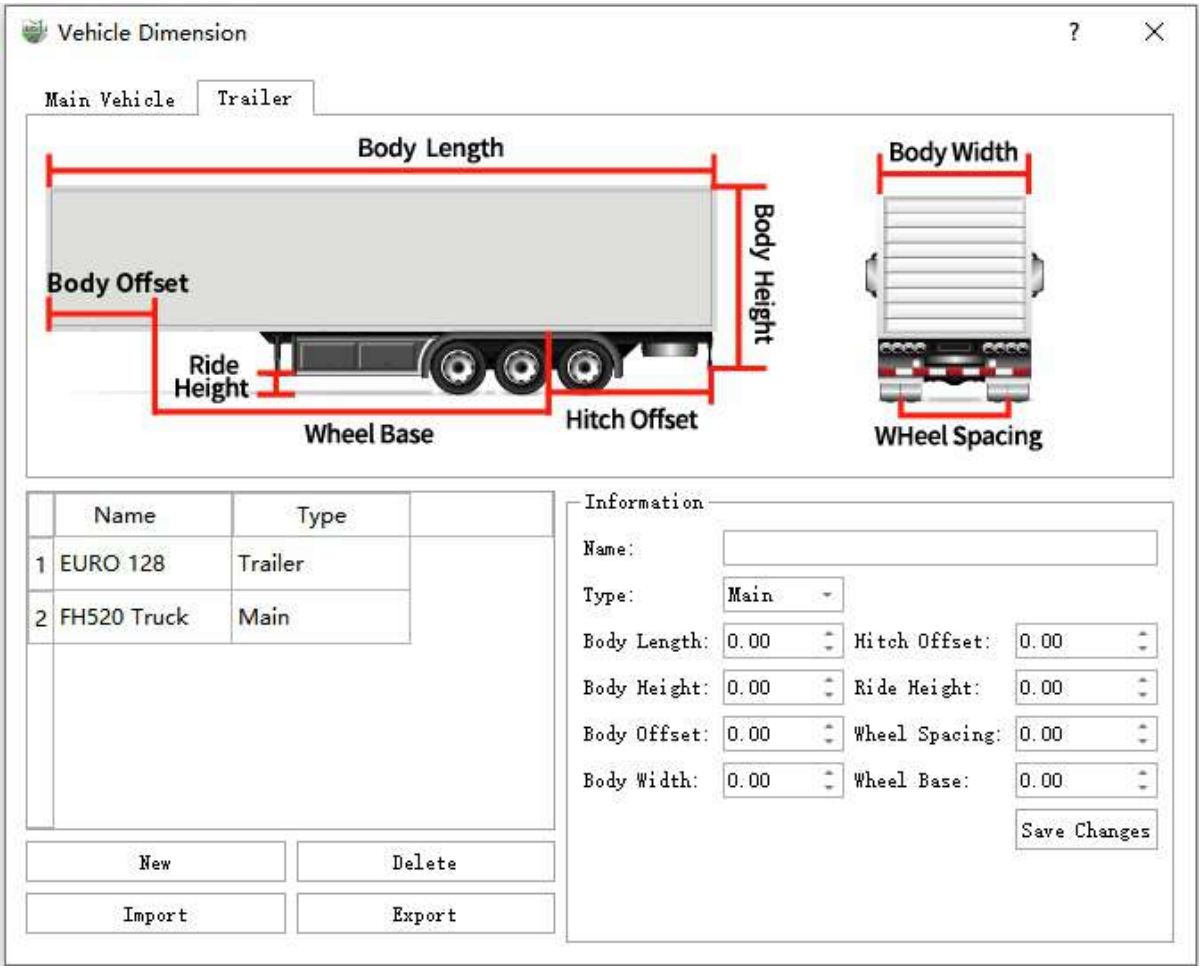
Vehicle Designer

Function Description: Design vehicles according to actual size parameters.
This function includes one built-in vehicle head and one vehicle carriage parameter file.

Dialog Parameters



Vehicle Head Parameter Dialog



Vehicle Carriage Parameter Dialog

- **Name**: Vehicle name
- **Type**: There are two options, vehicle head, and vehicle carriage.
  - **Main**: Refers to the whole of a one-piece vehicle or the head of a tractor.
  - **Trailer**: The carriage of the tractor.

> For passenger vehicles, SUVs, and one-piece trucks, the carriage is empty.

- **Body Length**: The length of the vehicle.
- **Body Height**: The height of the vehicle.
- **Body Width**: The width of the vehicle.
- **Body Offset**: The horizontal distance from the vehicle's front to the center of the front wheel.
- **Hitch Offset**: The distance from the center of the towing device to the center of the rear wheel. A positive value means that the center of the towing device is behind the center of the rear wheel, and a negative value means that the center of the towing device is in front of the center of the rear wheel.

> For vehicles without a towing device, this parameter is 0.

- **Ride Height**: Distance between the ground and the lowest point of the rigid object on the bottom of the vehicle
- **Wheel Spacing**: The distance measured on the support surface between the symmetric planes of the left

and right wheels on the same axle, i.e., the width of the left and right tires.

- **Wheel Base**: The distance from the center of the front axle to the center of the rear axle.

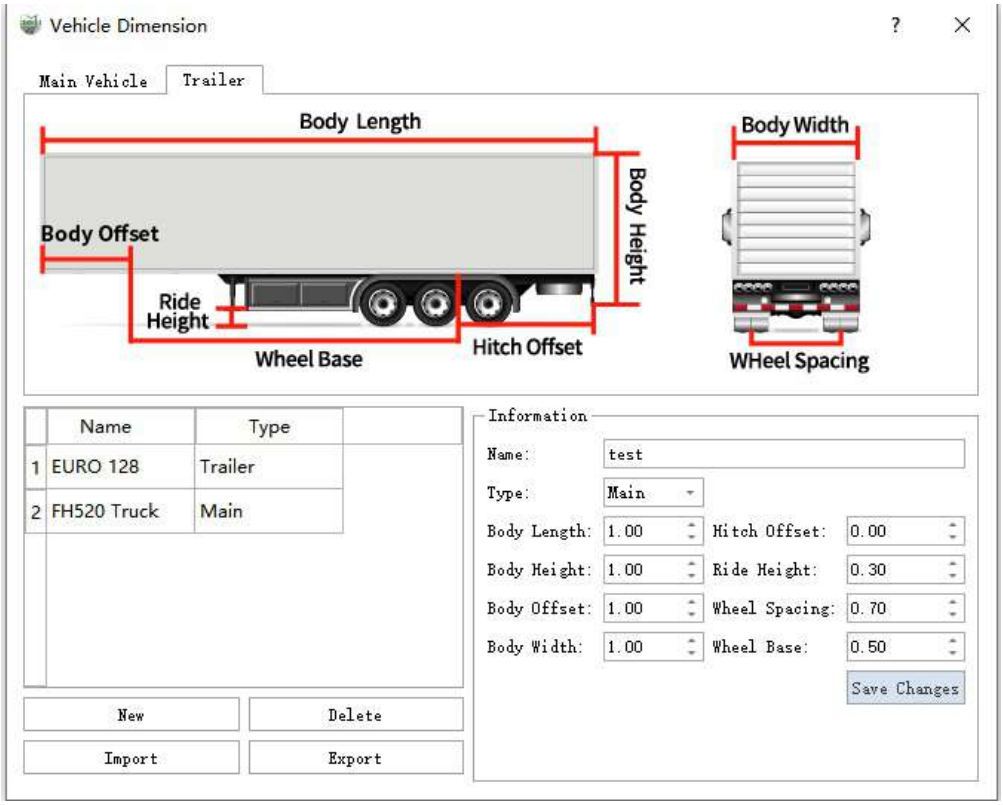
Operations

1.View Existing Files

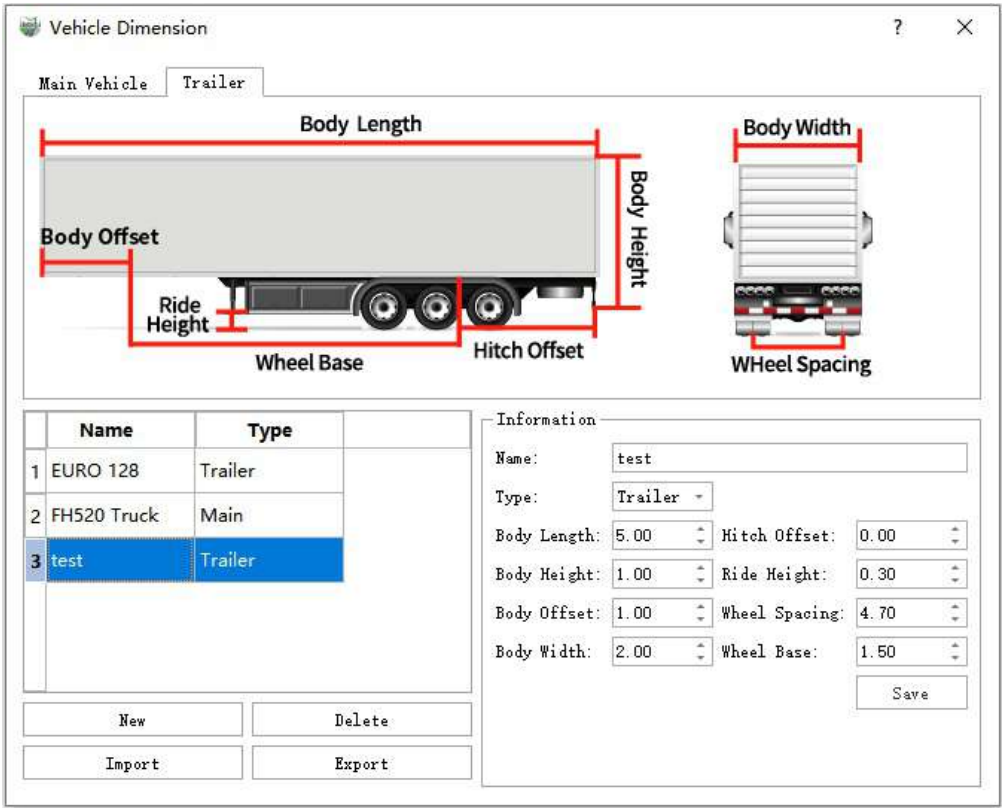
Select the name list with the left mouse button, and the right information bar will display size data in real time.

2.New Size File Creation

Click the **New** button, fill in the values in the **Information** module according to the actual size data, and click save to complete the creation. The vehicle list refreshes the newly created vehicle in real time.



New Creation



Creation Completion

3.**Size File Deletion**

Select a vehicle file from the vehicle size list with the left mouse button, and click the **Delete** button to delete the size file.

> Built-in files cannot be deleted.

4.**Size File Modification**

Select a vehicle file from the vehicle size list with the left mouse button, and directly modify the values in the right **Information** bar. After the modification is completed, click save the modification to complete the size file modification.

> Built-in files cannot be modified.

5.**Size File Export**

Export the selected vehicle from the vehicle size list as a file (.xml file).

6.**Size File Import**

Import the vehicle size file (.xml file) from external.

Urban Forestry

Urban Forestry supports classification, manual classification, individual tree measurement, tree editing, and ecological landscape analysis.

The main features include:

- Classification Tools
  - Classification by Attribute
  - Classification by Deep Learning
- Manual Classification
- Normalization By Ground
- Seed Editor
- Calculator
- Tree Editor
- DeNormalization
- Ecological Landscape Analysis

Seed Point Editor

- Seed Point Conversion
- Update
- Editor

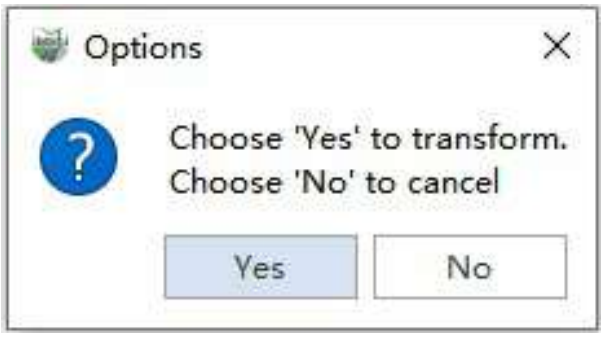
Seed Point Conversion

Function Description: Convert general vector points in the 'Tree' layer into individual tree points.

> This function is used to convert externally imported or manually drawn general vector points into individual tree points, allowing for individual tree segmentation and parameter measurement.

Steps

1.Click the **Seed Point Conversion** button. A dialog box will appear asking if you want to proceed with the conversion. Select **Yes** to complete the conversion of all general vector points in the *Tree* layer.

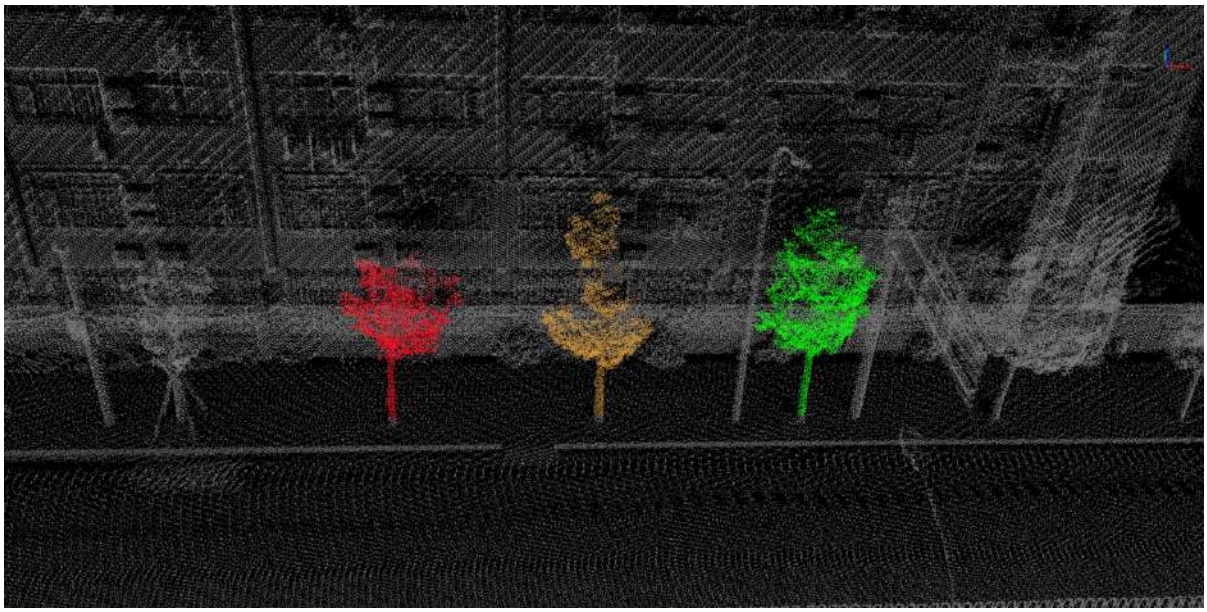


Updated seed points

Functional Description：Support for updating new seed points after deleting existing seed points in the presence of a single entity.

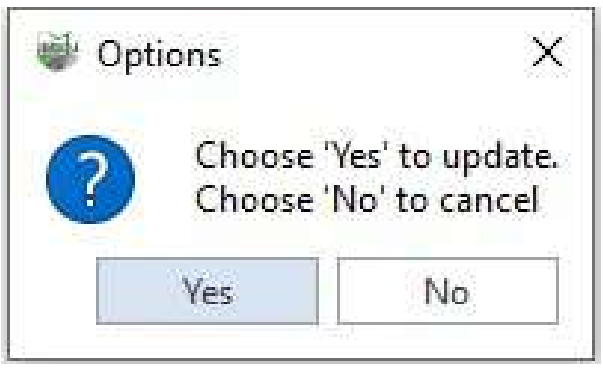
Steps

1.The previous point cloud data has already been segmented, it just lacks seed points.

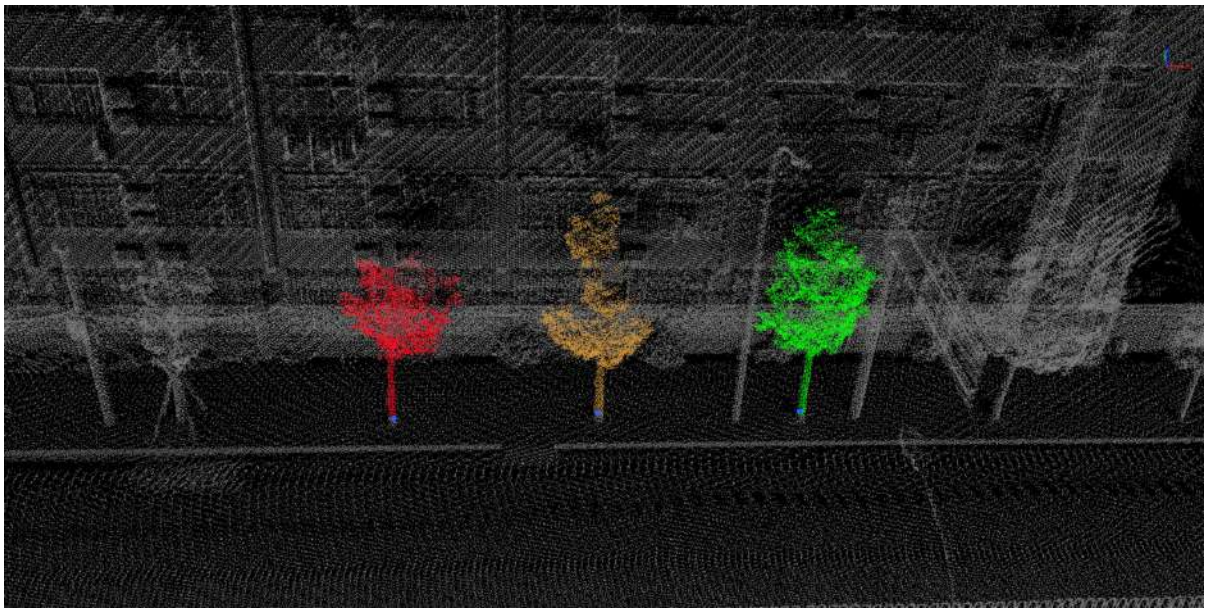


Before the update

2.Click on the **update** button if there is a singleton



3.Based on the available single entity, here are the updated seed points and results:



After the update

Seeds

Function ：This function supports operations such as adding, deleting, modifying, batch fitting of seed points, and updating seed points, among other actions.

Steps

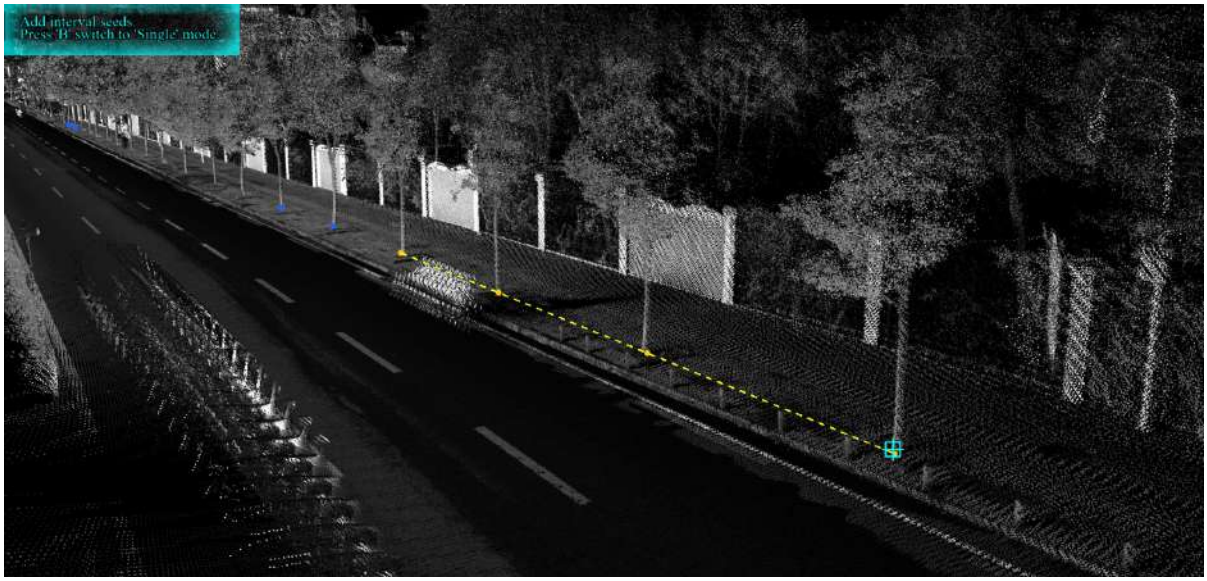
1.First, click the **Seed Point** button.

2.In the 3D window, you can see options for adding seed points , batch extracting seed points, global extract seed points , moving seed points , polygonal selection of seed points , seed point drape to ground , updating seed points , settings , and closing .

Furthermore, by default, the dialog boxes for section settings and georeferencing settings will pop up. It is the default behavior that when adding seed points, drape to ground is performed, especially when the point cloud has already undergone ground point classification. This way, when performing subsequent seed point fitting, the DBH (Diameter at Breast Height) positions will be accurate. Therefore, it is recommended to check the drape to ground option.

（1）The addition of add seed points supports two modes: single-point mode and equidistant mode, which can be switched using the 'B' (or 'b') key. By default, it's in single-point mode, where you can directly add seed points for the current tree at suitable locations on the trunk.

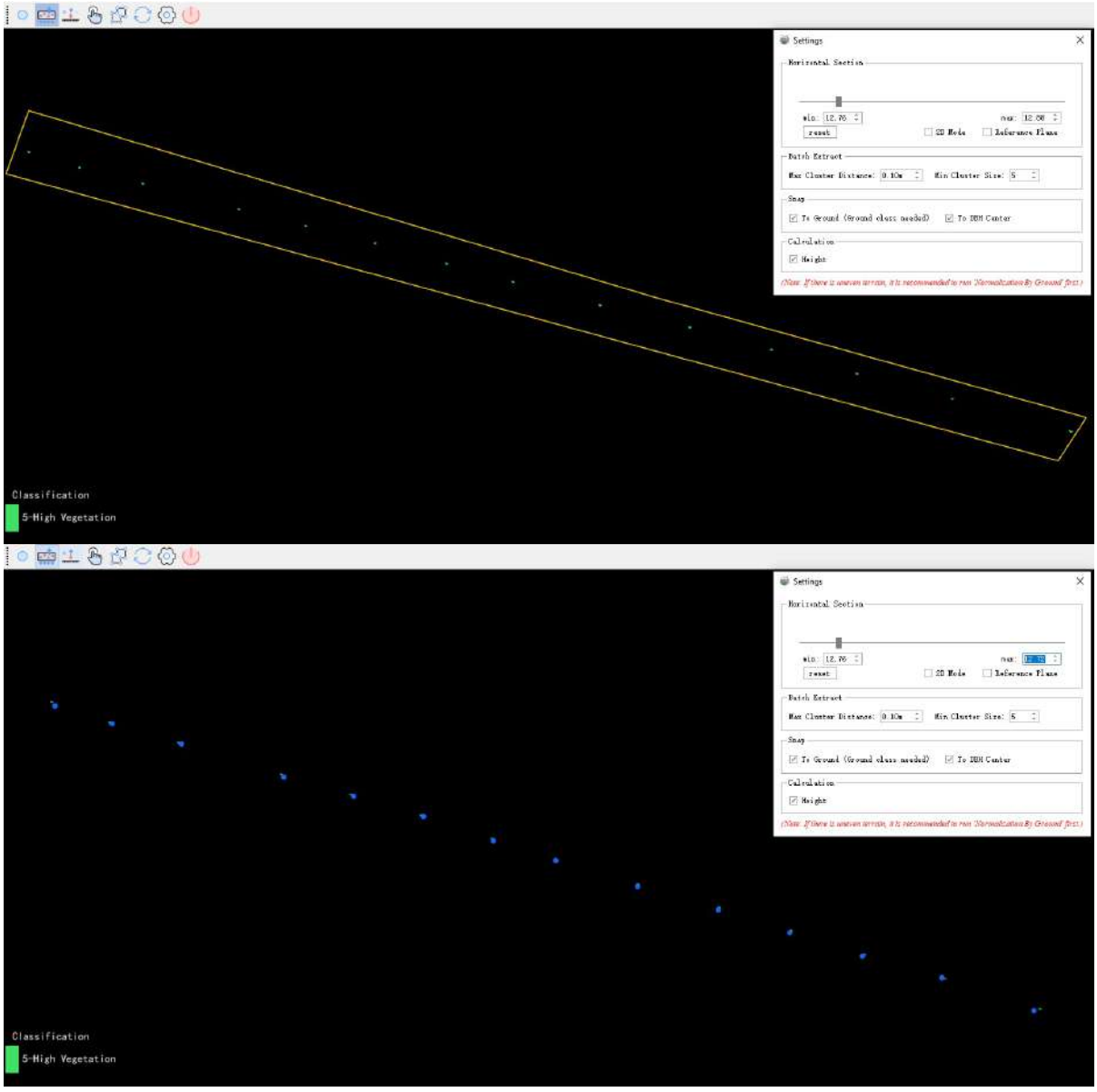
To switch to equidistant mode, press the 'B' key. In this mode, you'll start by clicking an initial seed point on the trunk of the first tree, which serves as the starting point. As you move the mouse, you'll see a dashed line indicating the distance between two trees. Then, select a seed point on the trunk of the second tree, defining the desired spacing. While moving the mouse, you can see in real-time how well the template matches the tree trunk. When you believe the fit is satisfactory, double-click the mouse to complete the batch addition of seed points.



Add seed points

(2)The batch seed point extraction function is more suitable for normalized data and can be used in conjunction with the horizontal cross-section tool. First, adjust the point cloud slicing to a suitable range, for example, setting it between 1.2 to 1.5 meters. Then, select the area to be extracted by drawing a box around it and double-click to confirm the selection. The software will automatically start the batch seed point extraction process.

> - Please note: If the point cloud data has been classified, it is recommended to switch to displaying by category and only show the category containing the trees when performing batch extraction. This allows you to filter out categories that do not need extraction and provides a clear view of the data.



Batch seed point

(3)global extract seed points , in this mode, based on the specified cross-section point cloud data, extraction is done automatically without the need for manual selection.

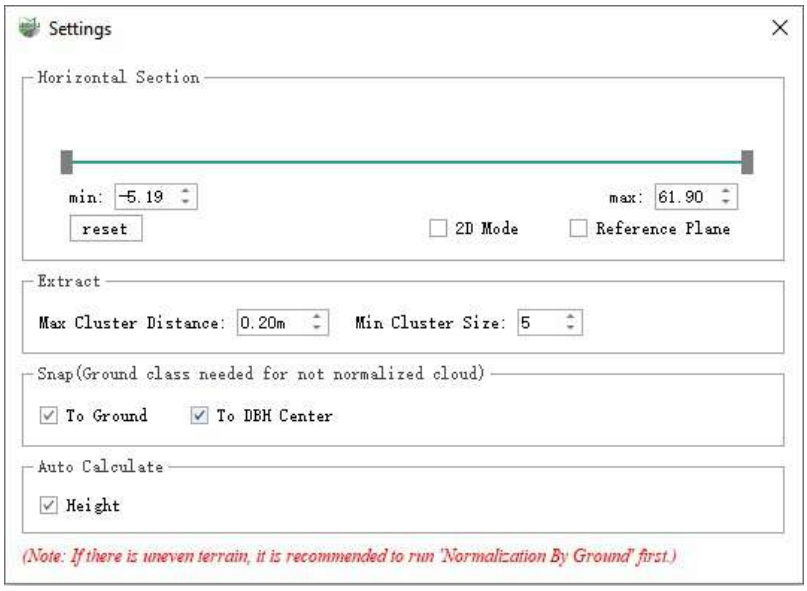
(4) Move Seed Point Function : Click on a seed point that needs modification, and then you can directly move the selected seed point's position with the mouse. You can also use 'Delete' to remove the selected seed point.

(5) Polygonal Seed Point Selection : Click on seed points that you want to select, and then use the mouse to create a selection area (box) around the desired seed points. Hold down the Ctrl key to add more seed points to the selection. The selected seed points will be displayed with their IDs in the selection dialog and will also be highlighted in the attribute table. You can directly use 'Delete' to remove the selected seed points.

(6) Select seed points for drape to ground : if the data is unclassified, ground points have not been separated, or ground points are not in category 2, the initially added seed points will not undergo ground fitting. After changing the point cloud categories, you can use the ground fitting function to select seed points that have not been fitted to the ground and perform ground fitting.

(7) The update seed points function is designed for scenarios where seed points are represented as vectors within the 'tree' vector layer. If you have cleared all the seed points from the directory tree layer, but your point cloud data has already undergone segmentation without seed points, you can use the update button. After clearing the seed points, you can choose whether to perform an update. Since the seed points have been cleared, the update will recreate the corresponding seed points for each individual.

(8) Setting Function , you to set the minimum and maximum values of the horizontal cross-section using dual sliders, or you can set custom minimum and maximum values. The 3D window will display the results in real-time, and you can reset the current settings by clicking the reset button. Checking the '2D mode' box allows you to switch between 2D and 3D modes in the 3D window, and checking the 'reference plane' box enables or disables the reference plane for the current 3D window, as shown in the diagram.



Setting

(8) Close the seed point functionality.

> Note: When adding and moving seed points, it is advisable to use the 'Hide Unselected Categories' and 'Quick Horizontal Section' features in combination for easier operation.

Calculator

Calculate single tree Diameter at Breast Height (DBH), height, and azimuth information.

The main components include:

- DBH And Height Calculation
- Azimuth

DBH and Height Calculation

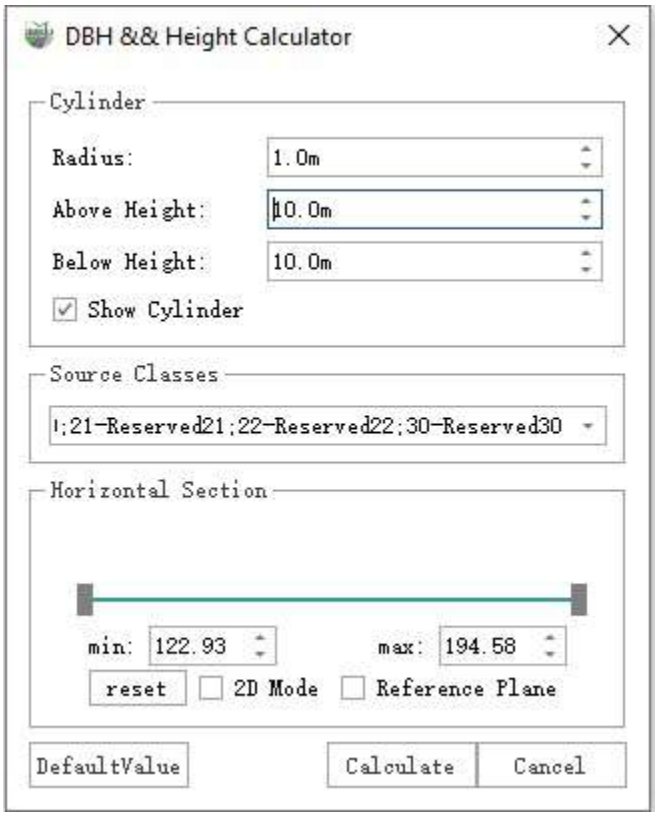
Feature Description: Semi-automatic measurement function for extracting individual tree DBH (Diameter at Breast Height) and tree height.

Function Description

This function will automatically generate individual vector points at the seed point and measure the DBH and tree height parameters within the cylindrical range.

Steps

1.Firstly, click the **Calculation** button, **DBH and Height Calculation** popup box pops up:

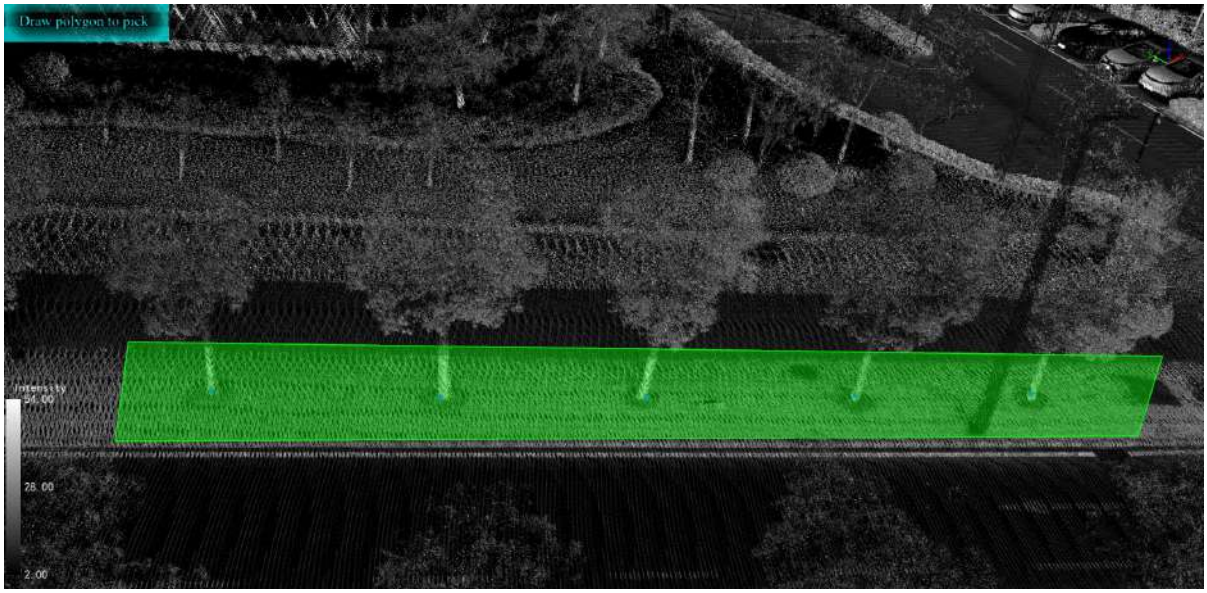


Parameter Description

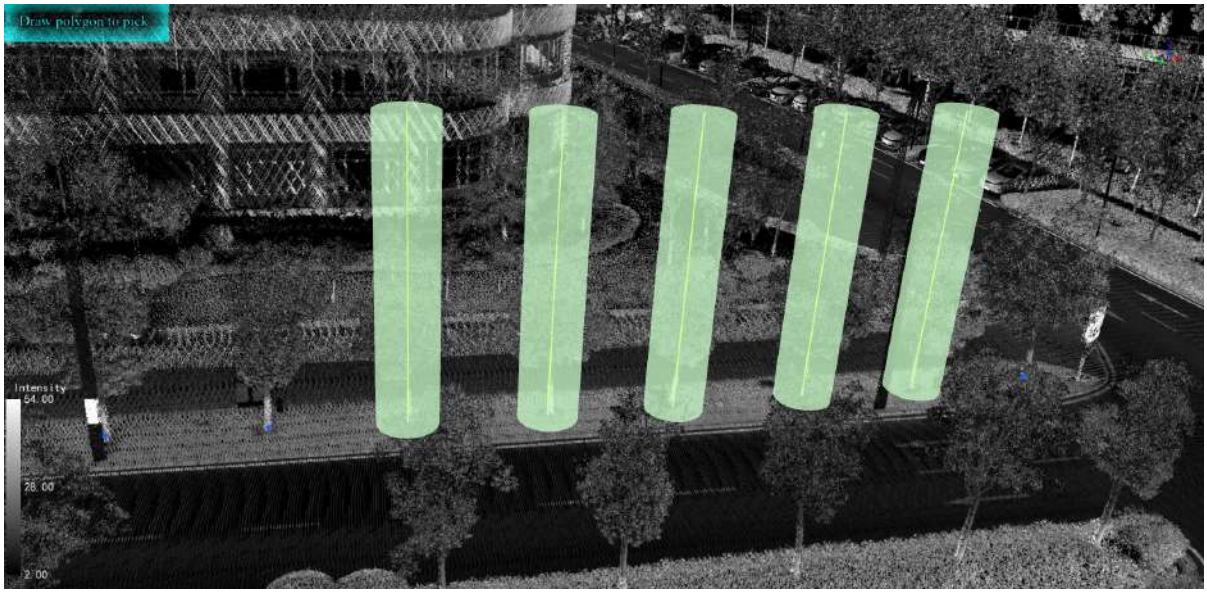
- **Cylinder Radius**: The radius of the cylinder with the seed point as the center.
- **Cylinder Upper Height**: The upward height based on the seed point.
- **Cylinder Lower Height**: The downward height based on the seed point.
- **Display Cylinder**：Whether to display in the form of cylinders according to the set parameters.
- **Source class**: Selection of road lines.
- **Horizontal Section Tool**：When the tree density is so high that the tree trunks cannot be clearly seen, use the slider to cut the horizontal section of the point cloud, making it easier to select points.

2.Adjust the cylinder parameters and single tree point cloud category settings in the DBH and height calculation pop-up dialog box.

3.Select the seed points to be calculated based on the requirements.

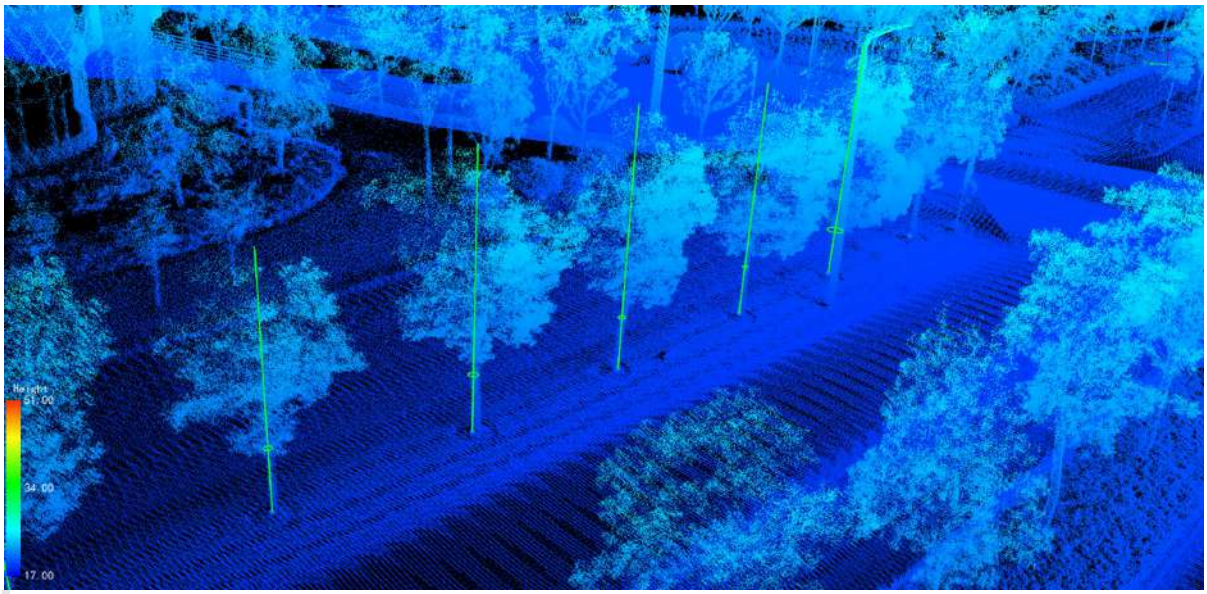


Boxed Seed Points



Selected Seed Points

4.Then click "Calculate" to complete the measurement of the selected seed points. Click the "Display" function in **Single Tree Editing** to view the results after calculation, as shown in the image.



Note: You can also select the seed points to be calculated in the attribute table of the tree layer.

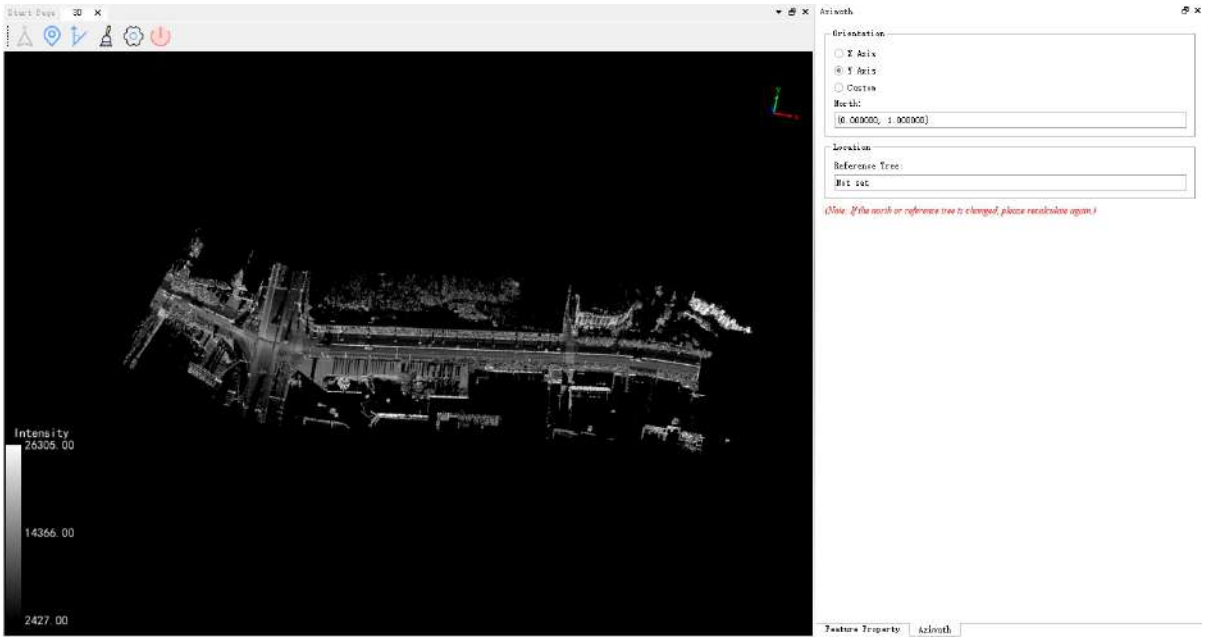
Azimuth

Function Description：Define a reference tree and north direction globally, and calculate the azimuth and horizontal distance for all trees.

> This function allows specifying a particular tree as a reference point, designating a direction as the north, constructing a polar coordinate system, and then calculating the azimuth angle and horizontal distance.

Steps

1.Click on **Azimuth** button, pop-up azimuth toolbar, and settings window:

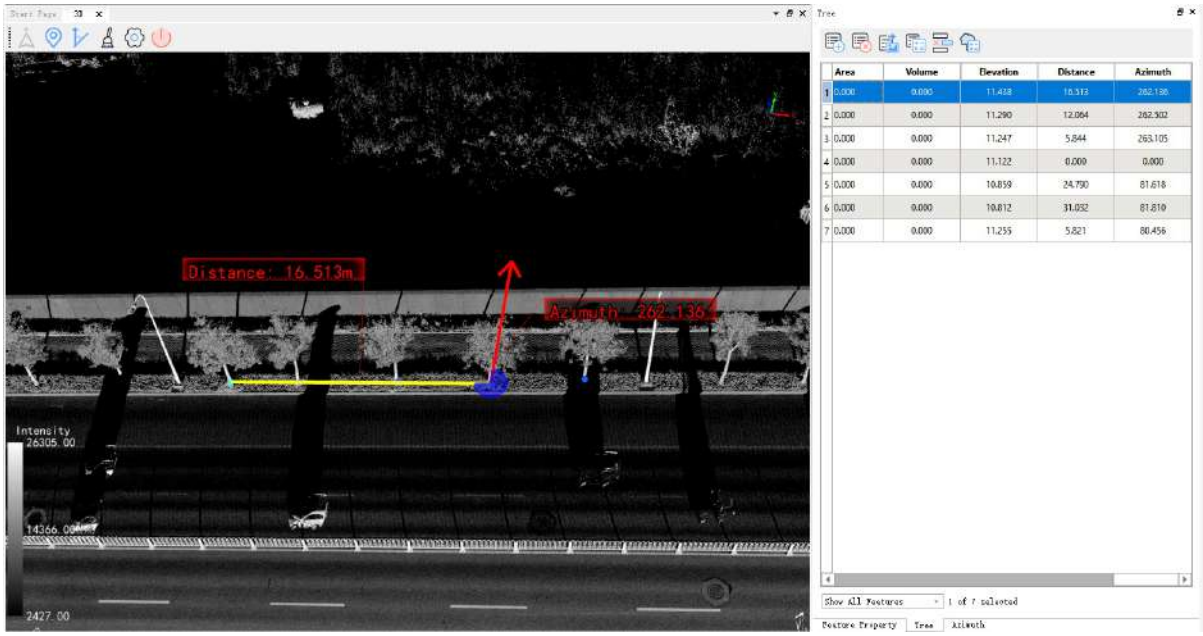


Function description

- **Set north**：Specify the north direction, providing X, Y, and custom modes. In the custom mode, determine the north direction by drawing two points.
- **Set reference tree**：Select a single tree seed point as the reference tree.
- **Calculate**：After completing the orientation and reference tree settings, you can calculate the azimuth angle and horizontal distance for all single tree seed points and fill them in the respective fields.
- **Clear**：Used to reset the north direction and reference tree, simultaneously clearing the azimuth angle and horizontal distance values for seed points.
- **Exit**：Exit function

2.First, click on the "Set Reference Point" button. Then, in the azimuth dialog, choose the direction as needed. Next, select a seed point as a reference. In the azimuth dialog, locate the parameter tree's ID. Finally, click on "Calculate."

3.After the calculation is complete, select the tree layer in the directory, right-click to open the dropdown menu, then click on "Attribute Table." In the table, select a seed point. You can find the azimuth and angle from the reference point to the selected seed point, and the result values will be stored in the corresponding fields.


Distance: 16.513m
Azimuth: 262.136
Intensity
26305.00
14366.00
2427.00
Tree
Area
Volume
Elevation
Distance
Azimuth
1 0.000 0.000 11.448 16.513 262.136
2 0.000 0.000 11.290 12.064 262.302
3 0.000 0.000 11.247 5.844 263.105
4 0.000 0.000 11.122 0.000 0.000
5 0.000 0.000 10.859 24.790 81.618
6 0.000 0.000 10.812 31.032 81.810
7 0.000 0.000 11.255 5.821 80.456
Show All Features
1 of 7 selected
Feature Property
Tree
Azimuth

Tree Editor

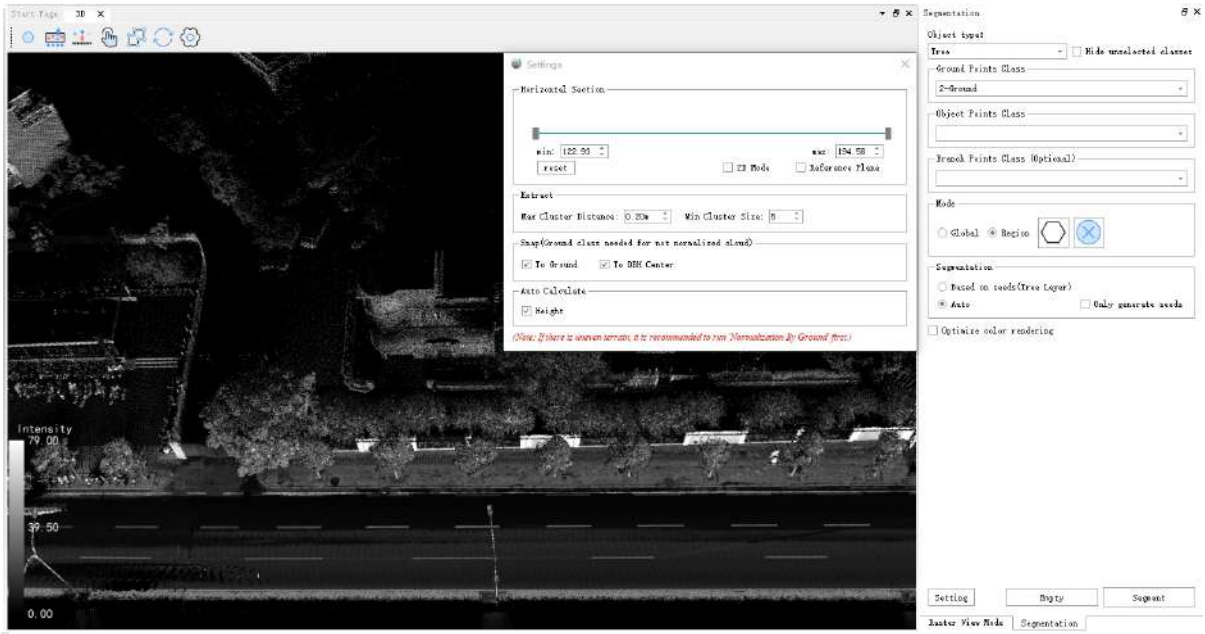
- Segmentation
- Editor
- Measurement
- Export Report
- Other Tools
- Settings
- Display

Segmentation

Function Description: Support the segmentation of individual trees, which will be introduced in detail below.

Steps

1.Click the **Segmentation** button and the following dialog box will pop up:



Note：

1.Segmentation function's seed point editing reference Editor.

2.If the data has been normalized, be sure to check the "Data Normalized" option before segmentation.

Parameter descriptions

- **Object type**: Tree.
- **Hide unselected olasses**: After checking, the points of unselected classes will be hidden, which can be used in conjunction with any rendering method to improve the convenience of seed point operations and point cloud observation.
- **Gromd Points Class**: The class where ground points are located (select the class where the ground points are located to assist segmentation and improve algorithm robustness).
- **Object Points Class**: The class where object points are located (for example, when segmenting trees, select the class where tree points are located).
- **Optimize color rendering**：Selecting the category where the trunk is located (for example, when segmenting the trunk) improves segmentation and position point calculation accuracy.
- **Mode**: It is divided into global and local modes. The global mode is to segment all point clouds under the project, and the local mode is to segment the point clouds within the selected area.
- **Seeds**: Including seed point addition, movement, generation, and growth. Tip: When adding and moving, you can use "**Hide Unselected Categories**" and "**Quick Horizontal Section**" together for easier operation.

- **Setting**: Initial trunk height (only effective for pole segmentation) is set to the minimum trunk height in the area to be segmented. Minimum clustering point number: the minimum number of points in a single cluster. Maximum clustering distance: the maximum distance between clusters.
- **Segment**: For the 'tree' segmentation process, it is divided into two steps: seed point generation and growth from the seed point, which is controlled by checking. For example, if only "**Generate Seed Points**" is checked, the result will only be seed points at the end of the segmentation. At this time, you can manually add seed points or modify the position of seed points, which is suitable for scenarios where users want to edit seed points a second time. Another example, after the seed points are determined, only "**Grow**" is checked and the final segmentation result is generated after the segmentation is over, ensuring the correctness of the number of trees.
- **Optimize color rendering**：Optimize the rendering of adjacent individual trees to ensure that the colors of neighboring trees are distinct.

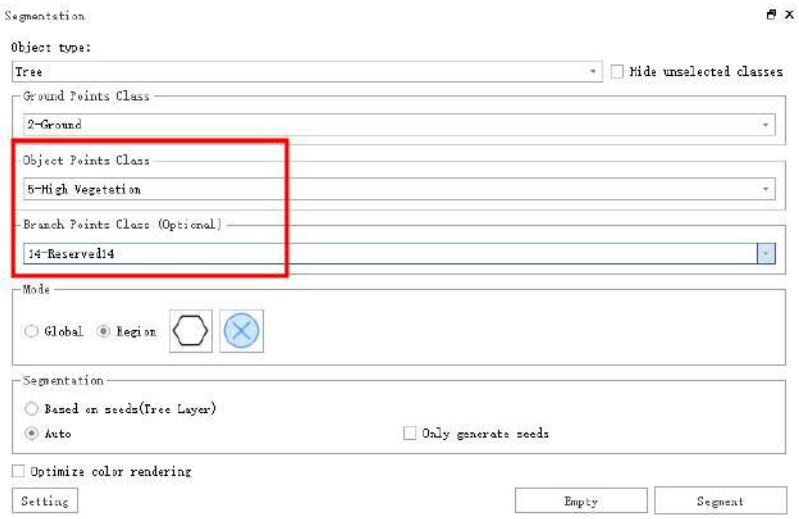
2.Click the Settings button to bring up the pop-up box.

Parameter descriptions

- **Initial Trunk Height**: the height of the center of the point cloud slice used to cluster the generated seed points.
- **Initial Trunk Thickness**: the thickness of the point cloud slice used for clustering to generate seed points.
- **Smallest Tree Height**: used to filter the smaller trees in the segmentation result, the trees lower than the set value will not be segmented.
- **Height above ground point (optional)**: when this parameter is set, the segmentation result will not start growing from the ground, but from the position that is away from the set value of the ground.
- **Maximum Cluster Radius**: is the maximum spacing of the clusters, points larger than this spacing will not be clustered together. If the trees are denser, you can turn this value down.
- **Minimum Cluster Points**: can act as a filter, if the point cloud within a cluster is smaller than the set value, it will not be considered as a cluster and will not be segmented.

3.After the settings are complete, click **Segment**. The segmentation result is shown in the figure:

4.If you need to combine trunk segmentation with individual tree segmentation, select the category where the crown is located for the individual point category and the category where the trunk is located for the branch point category.



5.If you want to clear individual trees, select the ones you want to remove, then click **Clear**, and the cleared result will be as shown in the figure.

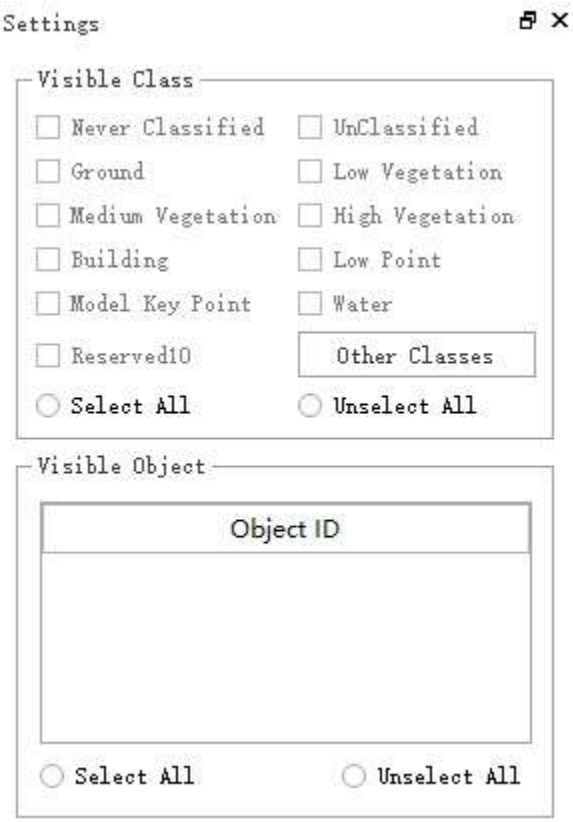
Editor

Function Description: Support manual editing of individual trees, including creation, deletion, and setting of individual tree points. Detailed instructions are provided below.

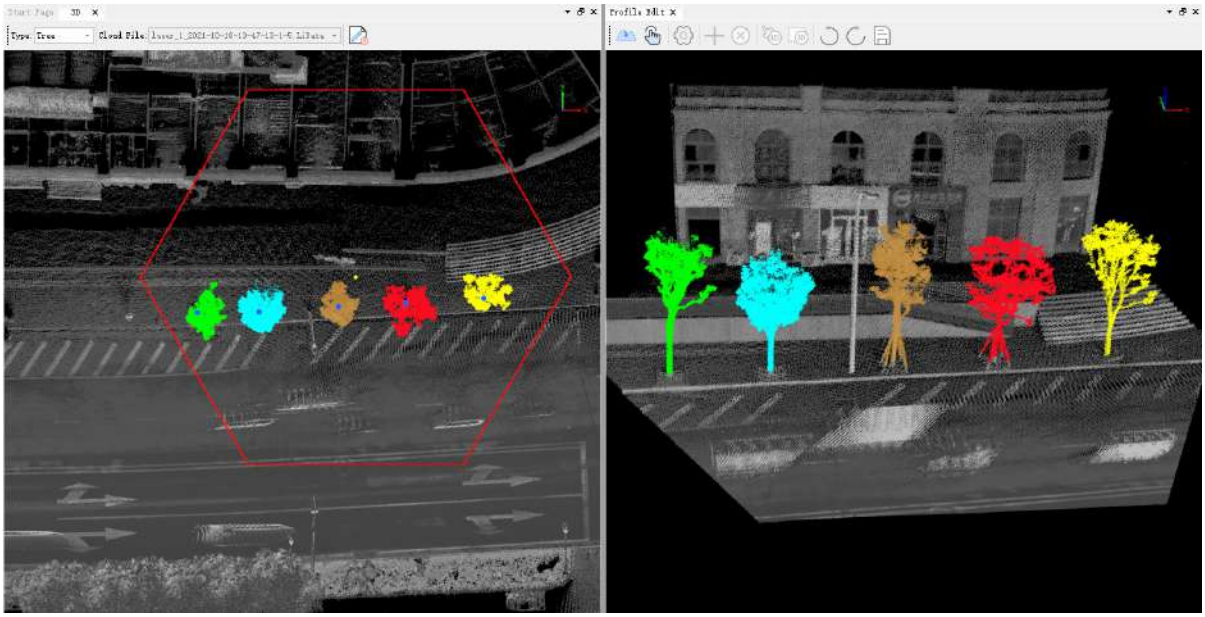
Steps

1.Click the **Editor** button and select the category and point cloud file to be edited. And then click the **Start Edit** button. The point cloud in the 3D window will then be displayed according to the selected individual tree type ID.

2.After starting the edit, an individual tree editing window will automatically pop up. In this window, you can add, delete, or modify individual trees.

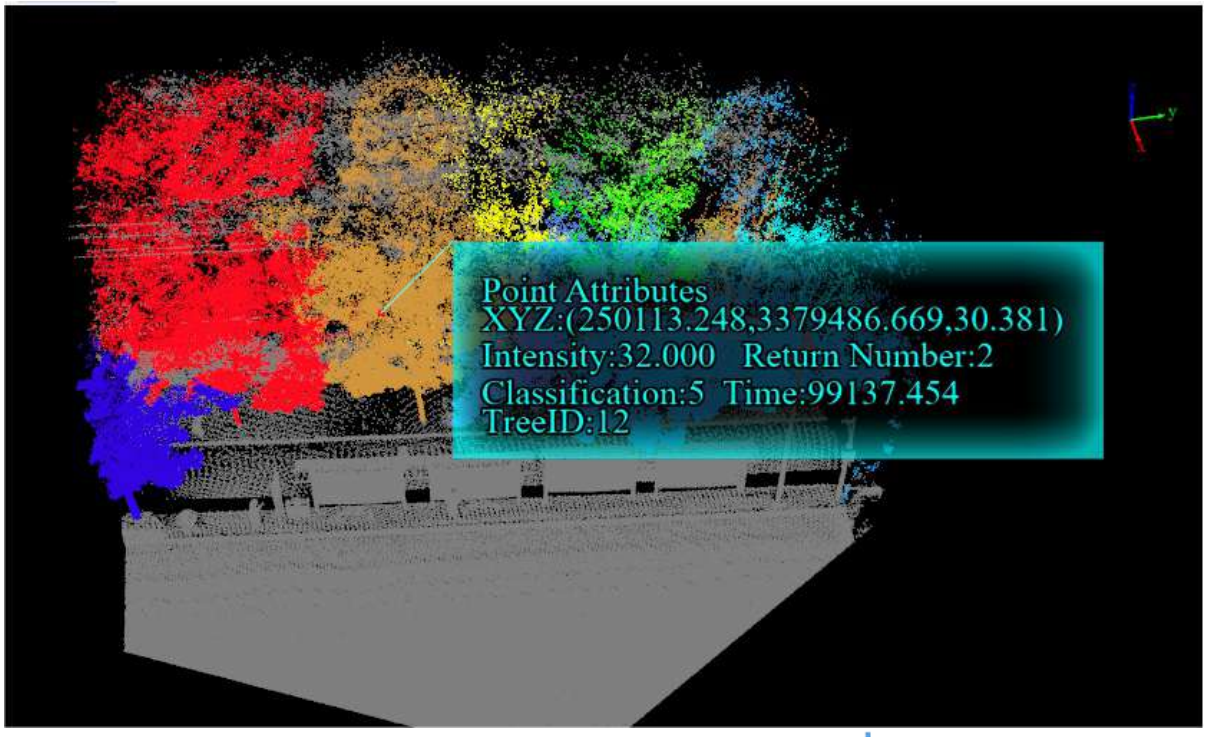


3.In the 3D window, click the left mouse button to draw a hexagonal area to be edited. The points in this area will be displayed in the individual tree editing window.Click the **Load Edit Region** button in the individual tree editing window to load the point cloud of the area to be edited into memory for editing. If it's not loaded, editing is not supported.

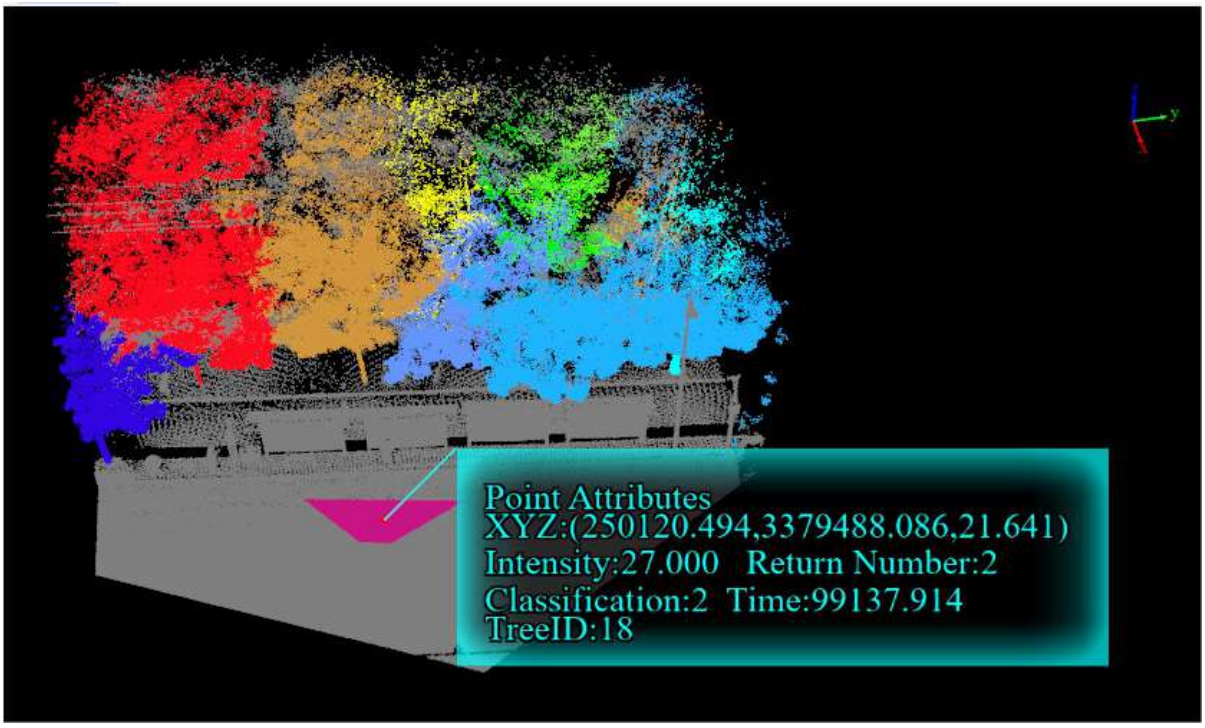


4.If the selection frame was not positioned correctly and missed part of the tree, you can either redraw it or use the "Move" function in the 3D window to adjust the frame's position.

5.Click the **pick Object** button in the individual tree editing window to view the attribute information of the selected point, including three-dimensional coordinates, GPS time, intensity, RGB, object ID, etc.



6.**Create Object**: In the individual tree editing window, click the **Create Object** button and select the point cloud of the individual tree by frame to create the individual tree. At this time, the object ID of the points within the selected area will be set to a new object ID, and a new object record will be automatically added to the attribute table. If you want to finely edit the point cloud of an individual tree, you can use the "**Set ID by Point Selection**" and "**Set ID by Frame Selection**" operations.



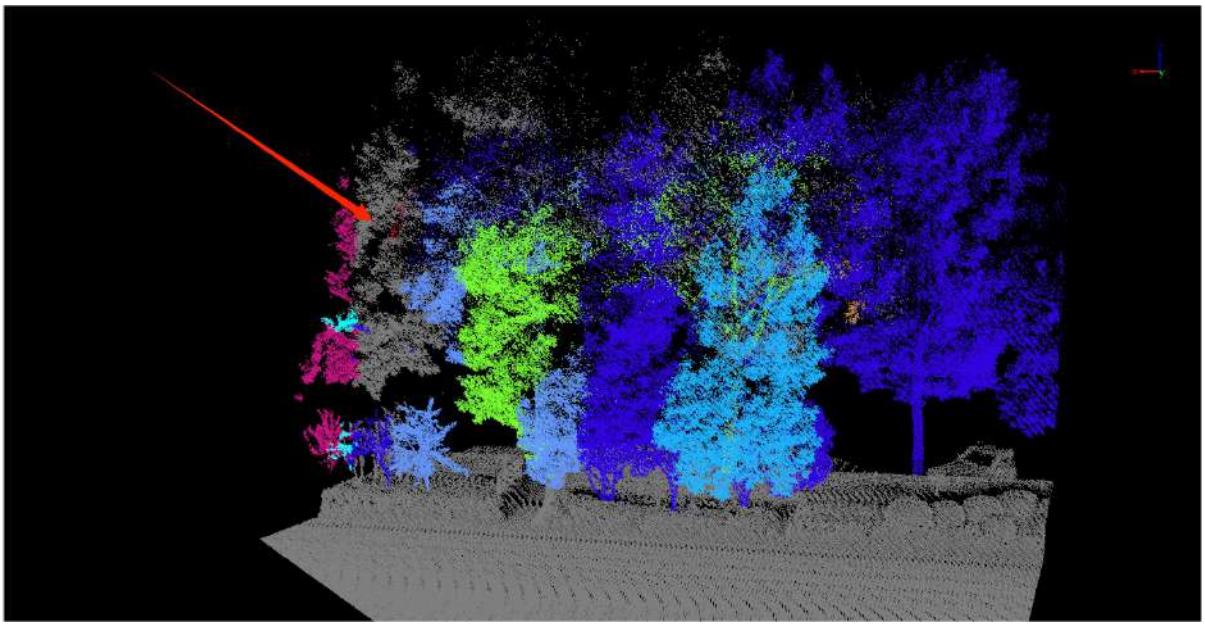
New Individual Tree

7.**Settings**: In the individual tree editing window, click the **Settings** button. In the settings window, you can select to display only the points that need to be edited from two dimensions: category and individual tree ID, which can reduce the interference of background point clouds on the editing process and enhance the convenience of operation.

8.**Zoom**：In the single tree editing window, if, during the editing process, the point cloud is moved out of the current window display range and cannot be zoomed back, you can use the zoom button to scale the selected area of the point cloud back into the current window view.

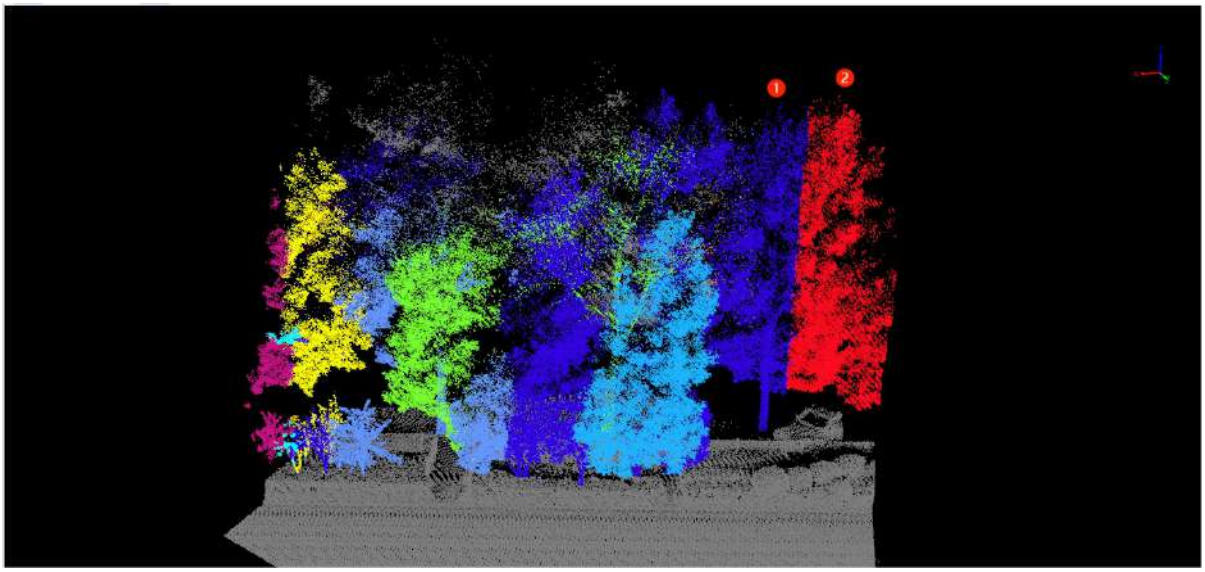
9.**Hide Single Tree**：In the single tree editing window, you can use the "Hide Single Tree" function to selectively hide trees that do not need editing. The hidden trees are linked to the visible trees in the settings window. To cancel the hiding, you can recheck the corresponding checkbox.

10.**Delete Object**: In the individual tree editing window, click the **Delete Object** button to delete an individual tree by point selection. The object ID of the points of this tree in the edit area will be reset to zero, and the object record in the attribute table will be deleted.

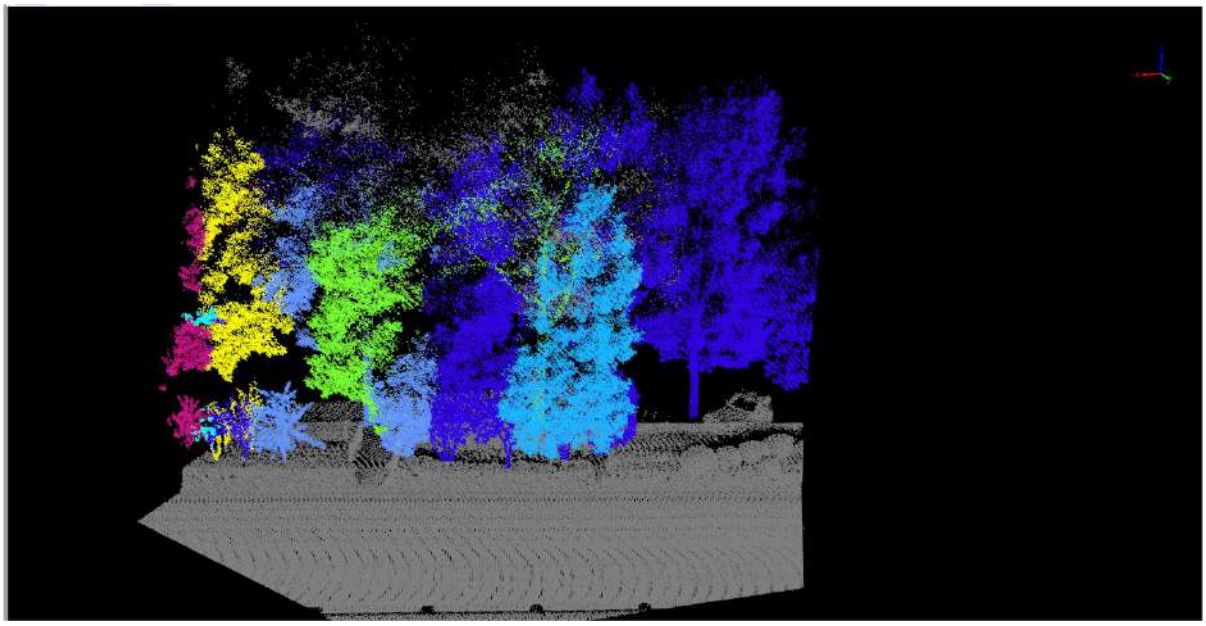


Delete Individual Tree

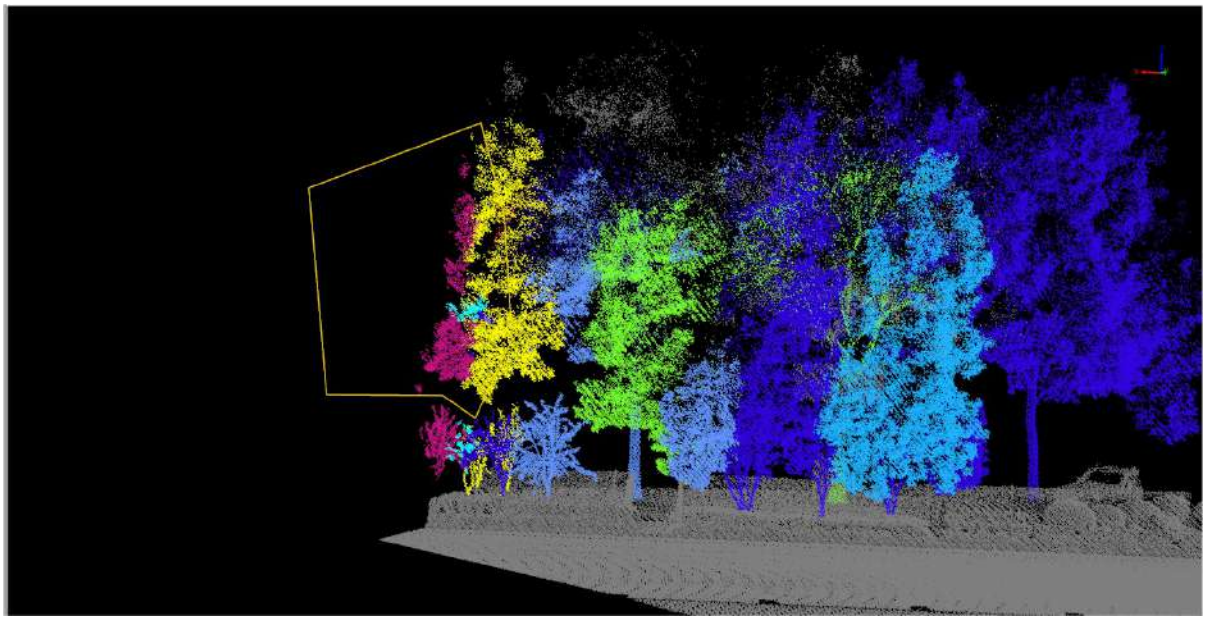
11.**Set ID By Pick**: In the individual tree editing window, click the **Set ID By Pick** button, and then click to select two individual tree points in turn. The object ID of the first clicked tree will be set to the object ID of the second clicked tree. This operation can be used to merge two individual tree point clouds in the edit area or delete a point cloud of an individual tree.



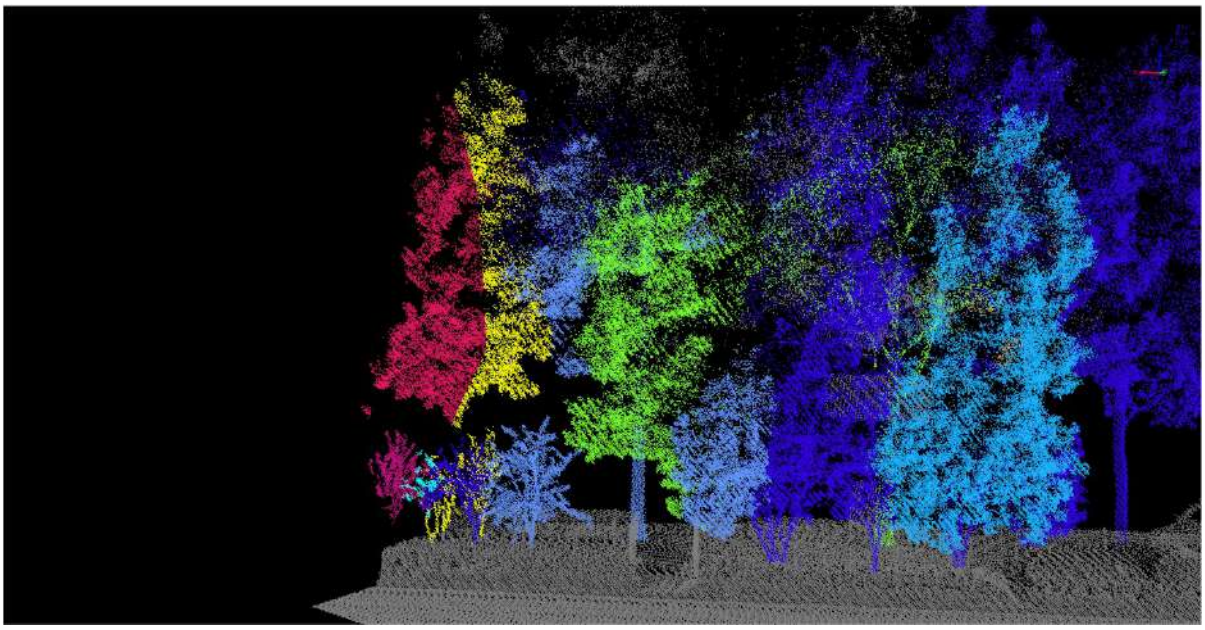
Before Set ID by Point Selection


After Set ID by Point Selection

12.**Set ID By Polygon**: In the individual tree editing window, click the **Set ID By Polygon** button, firstly, frame-select a part of the point, and then click an individual tree (or non-tree point). The object ID of the points in the frame will be set to the object ID of the clicked tree. This operation can be used for fine-editing individual tree points and deletion of framed individual tree points.



Before Set ID by Frame Selection



After Set ID by Frame Selection

13.**Undo**: In the individual tree editing window, click the **Undo** button to revert to the previous step. Support the "Ctrl+Z" shortcut key.

14.**Redo**: In the individual tree editing window, click the **Redo** button to advance to the next step. Support the "Ctrl+Y" shortcut key.

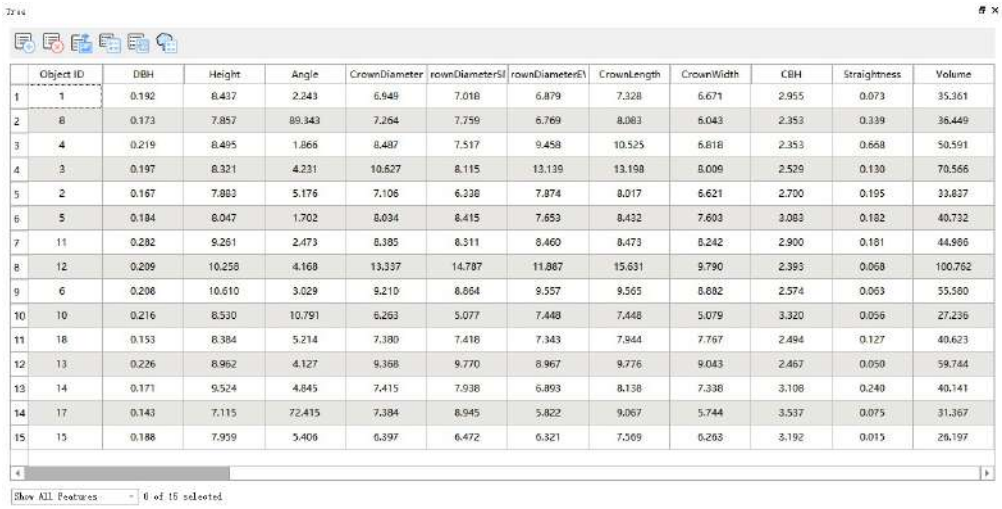
15.**Save Edit**: In the individual tree editing window, click the **Save Edit** button to save the edit to a file. The 3D view on the left will refresh the result. Support the "Ctrl+Shift+S" shortcut key.

Measurement

Function Description: Support the measurement of parameters of 'trees', including DBH, height, inclination angle, crown diameter, lower branch height, straightness and etc.

Steps

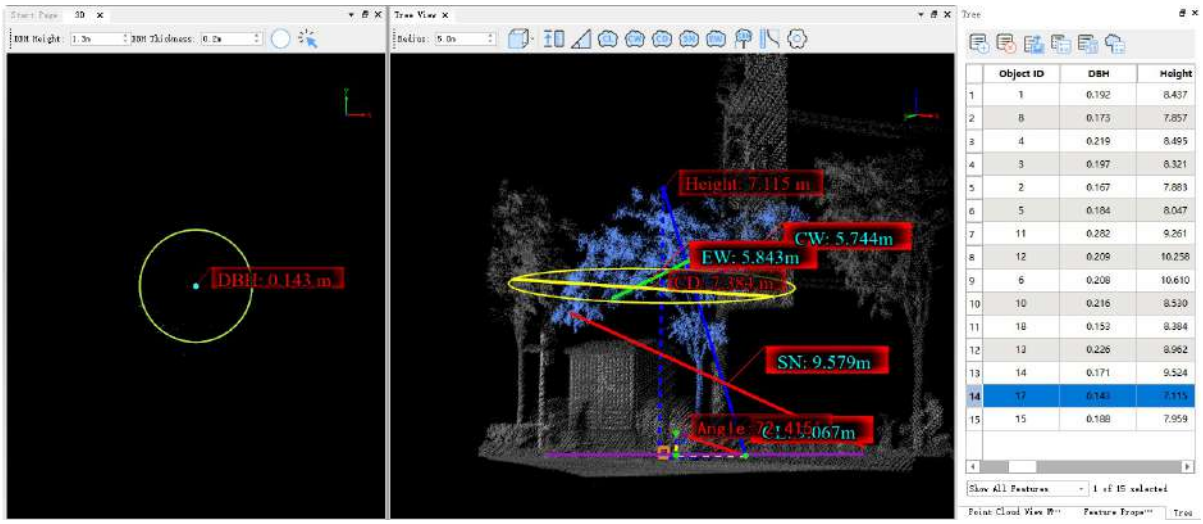
1.Select one or more rows in the attribute table (hold ctrl for multiple selection). Click on the **Calculate** button and then the calculation of the parameters is started. The results of the parameters are displayed in real time in the table as shown below:



	Object ID	DBH	Height	Angle	CrownDiameter	rownDiameterSl	rownDiameterEl	CrownLength	CrownWidth	CBH	Straightness	Volume
1	1	0.192	8.437	2.243	6.949	7.018	6.879	7.328	6.671	2.955	0.073	35.361
2	8	0.173	7.857	89.343	7.264	7.759	6.769	8.083	6.043	2.353	0.339	36.449
3	4	0.219	8.495	1.866	8.487	7.517	9.458	10.525	6.818	2.353	0.668	50.591
4	3	0.197	8.321	4.231	10.627	8.115	13.139	13.198	8.009	2.529	0.130	70.566
5	2	0.167	7.883	5.176	7.106	6.338	7.874	8.017	6.621	2.700	0.195	33.837
6	5	0.184	8.047	1.702	8.034	8.415	7.653	8.432	7.603	3.083	0.182	40.732
7	11	0.282	9.261	2.473	8.385	8.311	8.460	8.473	8.242	2.900	0.181	44.986
8	12	0.209	10.258	4.168	13.337	14.787	11.887	15.631	9.790	2.393	0.068	100.762
9	6	0.208	10.610	3.029	9.210	8.864	9.557	9.565	8.882	2.574	0.063	55.580
10	10	0.216	8.530	10.791	6.263	5.077	7.448	7.448	5.079	3.320	0.056	27.236
11	18	0.153	8.384	5.214	7.380	7.418	7.343	7.944	7.767	2.494	0.127	40.623
12	13	0.226	8.962	4.127	9.368	9.770	8.967	9.776	9.043	2.467	0.050	59.744
13	14	0.171	9.524	4.845	7.415	7.938	6.893	8.138	7.338	3.108	0.240	40.141
14	17	0.143	7.115	72.415	7.384	8.945	5.822	9.067	5.744	3.537	0.075	31.367
15	15	0.188	7.959	5.406	6.397	6.472	6.321	7.569	6.263	3.192	0.015	26.197

> Note: Clicking on the white box in the upper left corner is a full selection of the single ID.

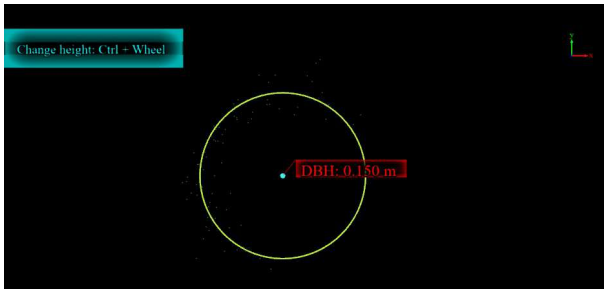
2.After the parameters are calculated, click the **Measurement** button. Open the property sheet of 'Tree' layer, click a record, two windows will jump to the actual position of the vector object, the left 3Dview view will show DBH parameters, the right Tree View window will show height, crown width, angle, crown length, crown width 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 the diameter at breast, which are 1.3m and 0.2m by default. Displayed as shown:



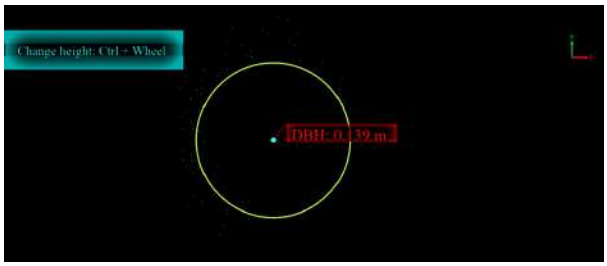
> Note: The tree segmentation and editing operation must be done first.

3.**DBH Measurement**:The DBH is refitted in two ways:

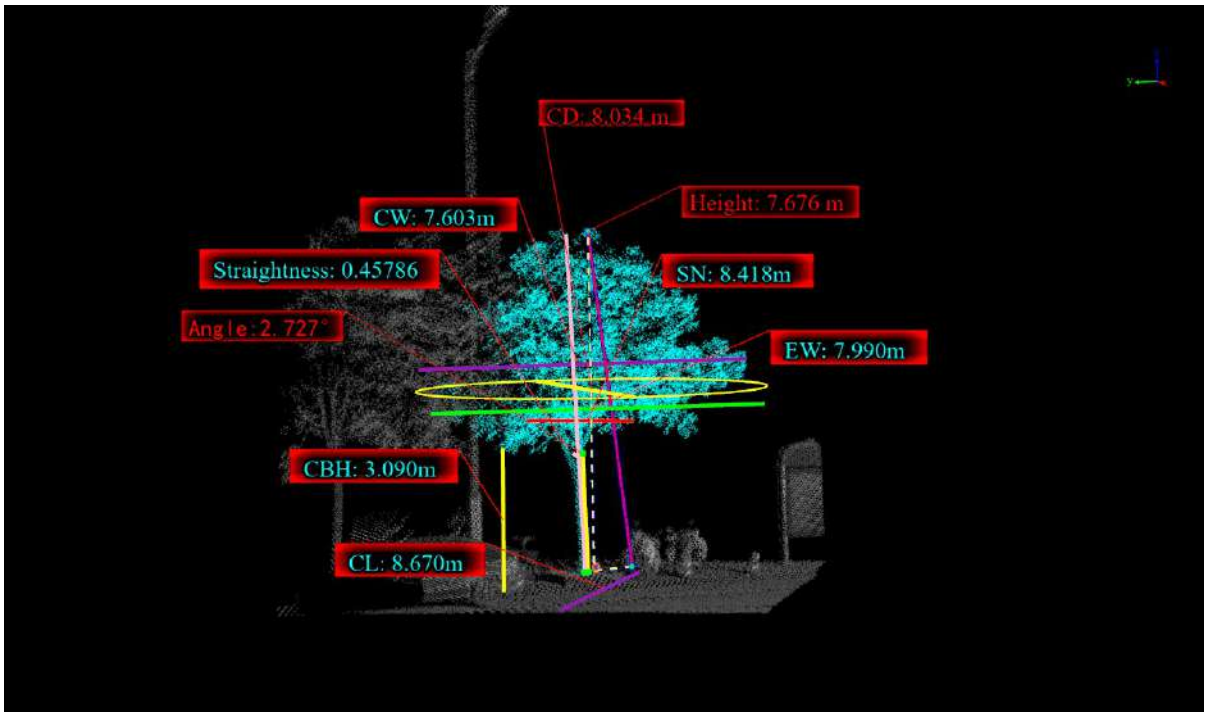
- **Window Introduction**：The left window is the DBH measurement window, displaying sliced point clouds.
- **DBH Height**：It is used to control the elevation of the sliced point cloud, allowing users to choose the point cloud at their desired height position for DBH measurement. It's important to note that this height value is the relative elevation based on the vector points, not the absolute elevation value, and it defaults to 1.3m.
- **DBH Thickness**：It is used to control the thickness of the sliced point cloud. When the point cloud is sparse, increasing the thickness can enhance the point cloud density.
- **DBH Three-point Fitting**:First click on the single ID you want to view, and then click on **DBH Three-point Fitting** button in the 3Dview view window. Left mouse click on the three points to determine the refitted circle, as shown in the figure.



- **DBH Rectangle Fitting**：Also after switching the tree, click on the **DBH Rectangle Fitting** button in the 3D window. Box the point cloud that needs to be refitted with DBH, as shown in the figure:



4.In the Object View window, after switching the point cloud to the appropriate angle via the switch button (top view, front view, rear view, left view, right view) of the view, perform the measurement of the parameter to be modified and update the result of the measurement in the attribute table.



5.**Height Measurement**: Click on the **Height Measurement** button, using a two-point approach, to re-measure the height of the monolith on the point cloud, and the new measurements are updated in real time in the corresponding monolith table list.

6.**Angle Measurement**: Click on the **Angle Measurement** button. Re-measure the angle and update the result of the angle in the parameter table.

7.**Crown Length Measurement**: Click on the **Crown Length Measurement** button. Re-measure the crown length and update the crown length result in the parameter table.

8.**Crown Width Measurement**: Click on the **Crown Width Measurement** button. Re-measure the crown width and update the crown width result in the parameter table.

9.**Crown Diameter Measurement**: Click on the **Crown Diameter Measurement** button. Re-measure the crown diameter and update the crown width result in the parameter table.

10.**North-South Crown Diameter Measurement**: Click on the **North-South Crown Diameter Measurement** button. Re-measure the North-South crown diameter and update the North-South crown width result in the parameter table.

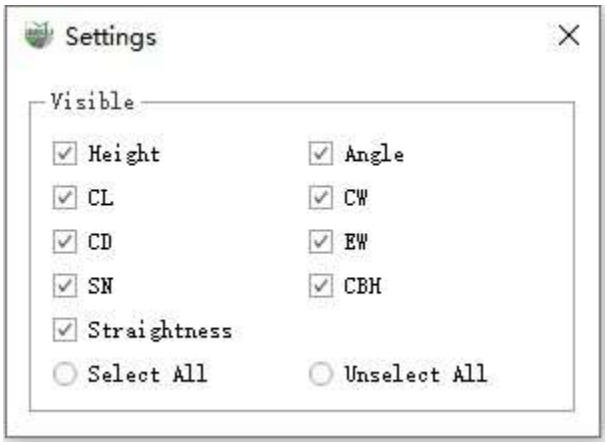
11.**East-West Crown Diameter Measurement**: Click on the **East-West Crown Diameter Measurement** button. Re-measure the East-West crown diameter and update the East-West crown width result in the parameter table.

12.**Lower Branch Height Measurement**: Click on the **Lower Branch Height Measurement** button. Re-measure the lower branch height and update the lower branch height result in the parameter table.

13.**Straightness Measurement**: Click on the **Straightness Measurement** button. Re-measure the straightness and update the straightness result in the parameter table.

14.**Area Measurement**：Click on the **Area Measurement** button，In the top-view perspective, use the multi-point mode to measure the area, and update the area result in the parameter table.

15.Click on the **Setting** button，A pop-up dialog appears, allowing you to check the parameter values you want to display. It provides the option to show or hide parameters within the Tree View window.



Export Report

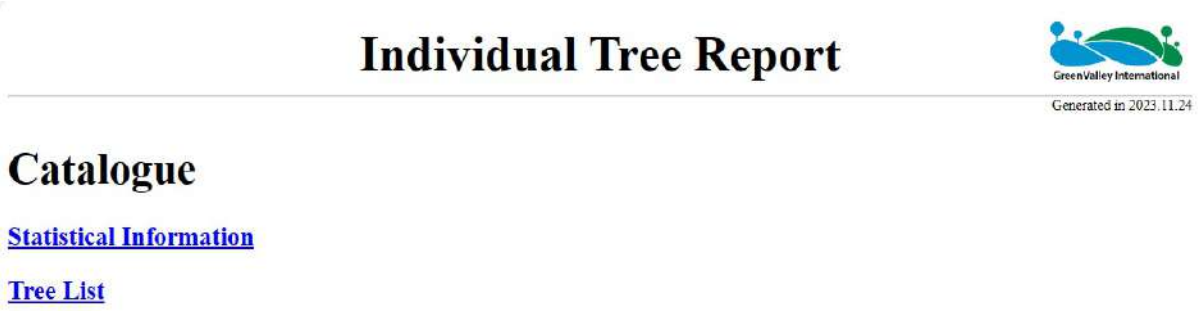
Function Description：Export all individual tree information in the scene, including fixed fields such as tree height, crown diameter, area, azimuth, and all custom fields.

Steps

1.Click the **Export Report** button to open the save file dialog. Choose the report file name and the location where you want to save it.

2.Report Content: header, Divided into two main parts - Statistical Information and Individual Tree List.

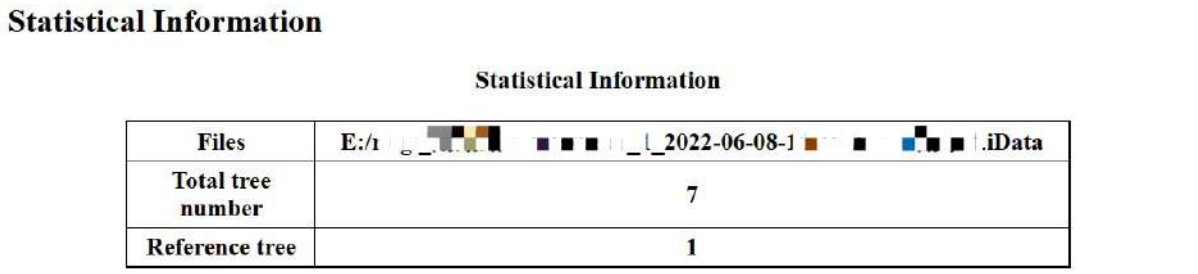
（1）Header and Table of Contents



Header and Table of Contents

Click on **Statistical Information** or **Tree List** under the directory to jump to the corresponding location.

（2）Statistical Information: Includes statistical details about the project point cloud file, the number of individual trees, and information related to the reference tree.

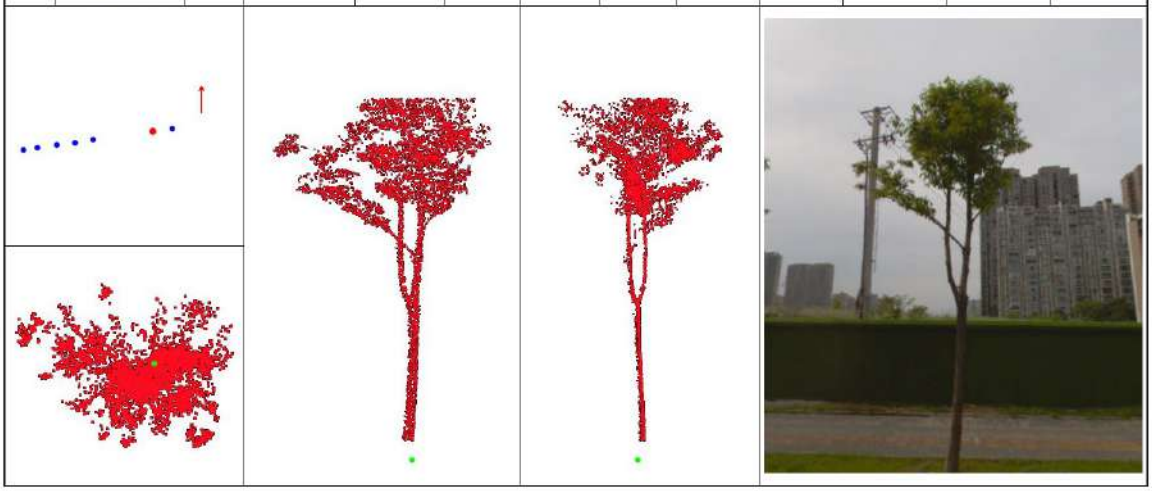


Statistical Information

（3）Individual Tree List: Includes fixed fields such as tree height, crown diameter, area, azimuth, and all custom fields for each individual tree.

Tree List

Tree ID	Location(m)	Z(m)	Elevation(m)	Height(m)	DBH(m)	CD(m)	SN(m)	EW(m)	Area(m^2)	Volume(m^3)	Distance(m)	Azimuth(°)
1	532224.769, 3373647.190	10.859	10.859	7.074	0.158	3.177	3.737	2.812	7.929	15.766	0.000	0.000



Individual Tree List

Other Tools

Function Description：Other tools include:Update Seed Points、Clear Tree ID and Extract by Tree ID

Clear Tree ID

Function Description： This tool is used to clear the Tree ID values in the point cloud data, but it will not clear any related vector points.

Steps

1.Click on the **Other Tools** button and select **Clear Tree ID** . This will open the following dialog box:

Clear Tree ID

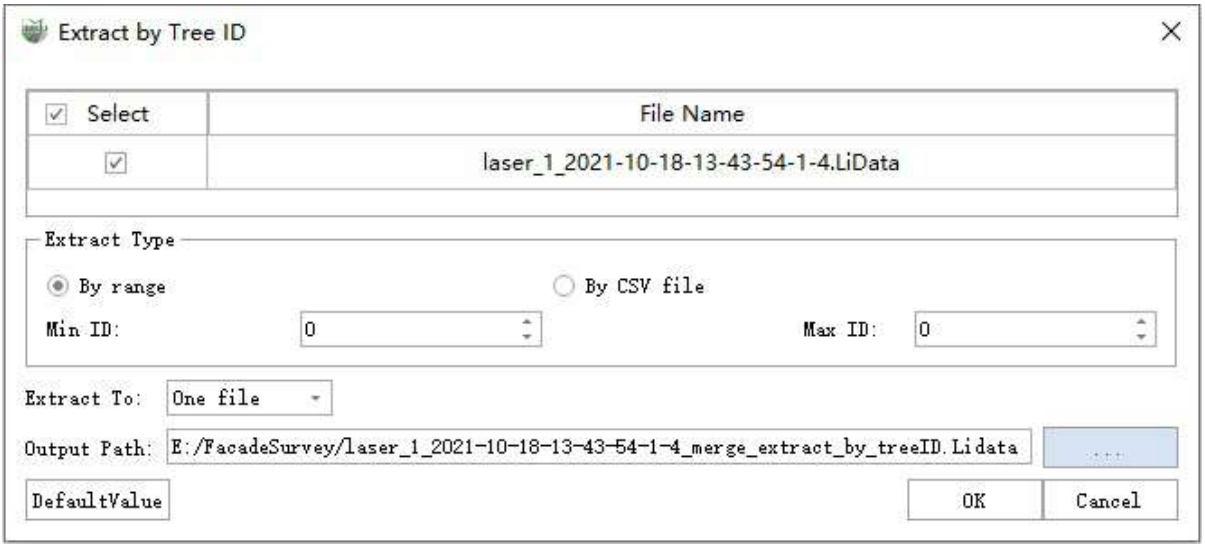
2.Set the point cloud file that you want to clear the Tree ID for, and click **OK**. Wait for the clearing process to complete.

Extract by Tree ID

Function Description: According to the set target tree ID, the corresponding points are extracted from the point cloud file and generated into a new file.

Steps

1.Click on the **Other Tools** button, and then click on **Extract by Tree ID** button to open the dialog box as shown below:

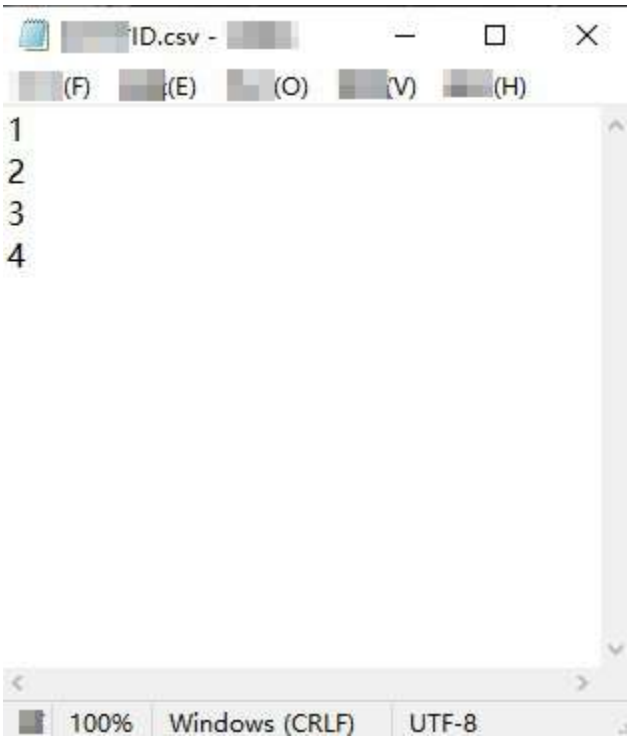


Empty Tree ID

2.Set the point cloud file that needs to be extracted;

3.Set the extraction type. There are two ways:

- **By range**: Set the minimum and maximum target tree ID.
- **By CSV File**: Edit all target tree IDs in the CSV file and import it into the function. The specific file format is as follows:

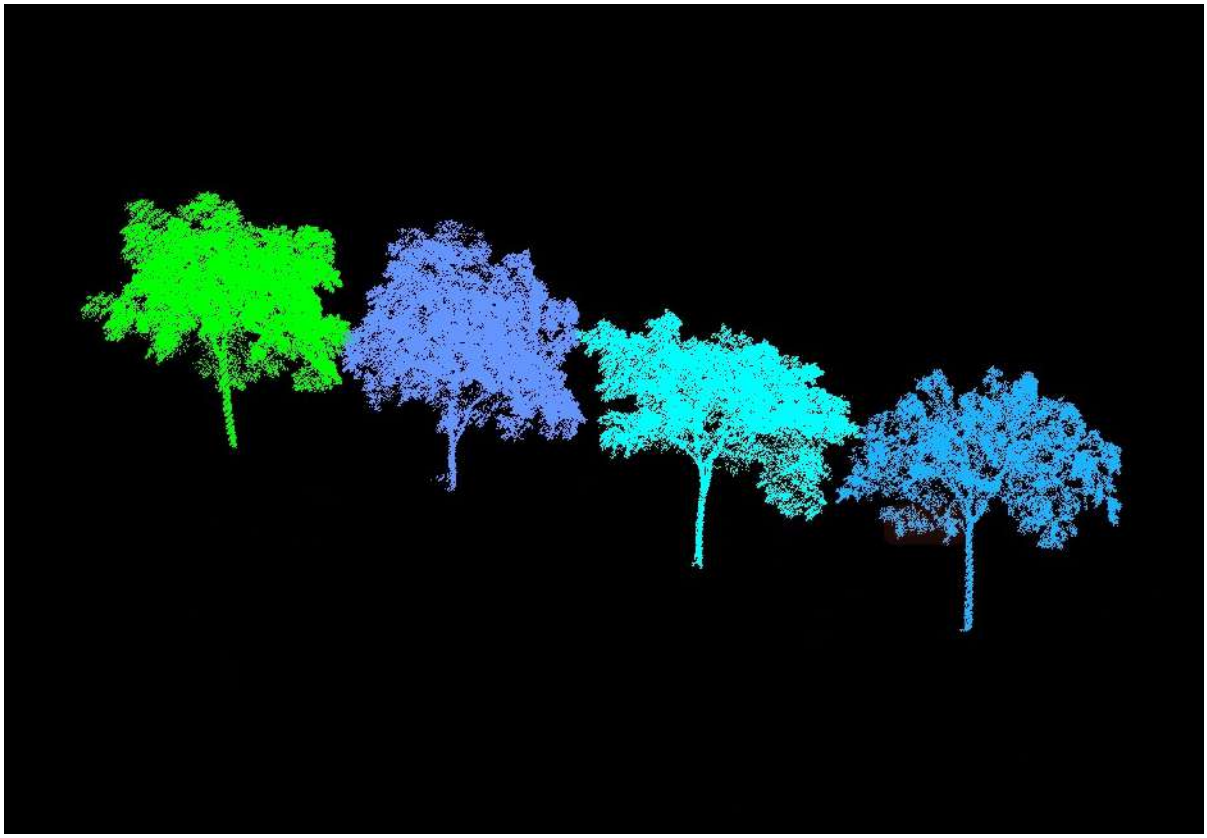


Tree ID CSV File Format

4.Set the output type. There are two ways:

- **One file**: All points are exported to one point cloud file.
- **Multi files**: The target tree ID points of each file are exported to their respective point cloud result files.

5.Set the output file or folder, and click **OK**. The export result is as follows.



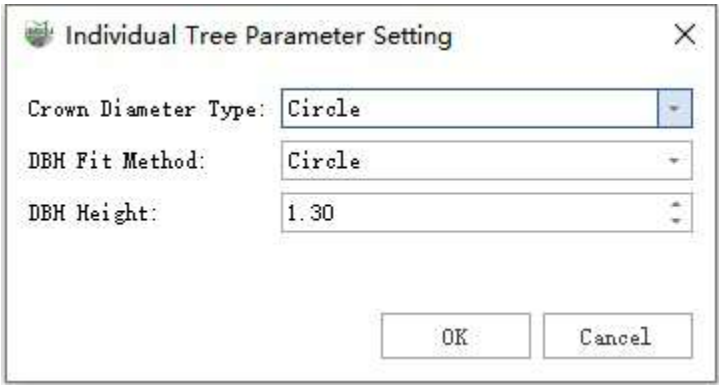
Extraction Result

Settings

Function Description: Allow you to set the individual tree position, crown radius calculation type, and DBH fitting method. These settings apply to individual tree measurement, editing, parameter calculation, and parameter measurement.

Steps

1.Click the **Settings** button in the **Urban Forestry** page under the **Tree Editor** group. A dialog box will appear as shown below:



Parameter Explanation

- **Crown Diameter Type**: Provide three options for crown radius calculation: Average Crown Radius, Circle Fitting and Ellipse Fitting.
  - **Average**: (North-South Crown Radius + East-West Crown Radius) / 2.
  - **Circle**: 2 * sqrt(Crown Projection Area / PI).
  - **Ellipse**: sqrt(North-South Crown Radius * East-West Crown Radius).
- **DBH Fit Method**: Provide two options for DBH fitting: Circle Fitting and Ellipse Fitting.
  - **Circle**: Fit a circle to the point cloud at a height of 1.3 meters on the tree trunk.
  - **Ellipse**: Fit an ellipse to the point cloud at a height of 1.3 meters on the tree trunk.
  - **Cylinder**: Perform cylinder fitting on the sliced point cloud at the set DBH (Diameter at Breast Height) tree trunk height.
- **DBH Height**: Customize the DBH (Diameter at Breast Height) height according to your specific requirements.

Display

Function Description: Display tree height, DBH (diameter at breast height) and crown radius at the location of individual trees.

Steps

1.Check the **Display** button in the **Tree Editor** group of the **Urban Forestry** page. The display will show tree height, DBH, and crown radius as shown below:

Ecological Landscape Analysis

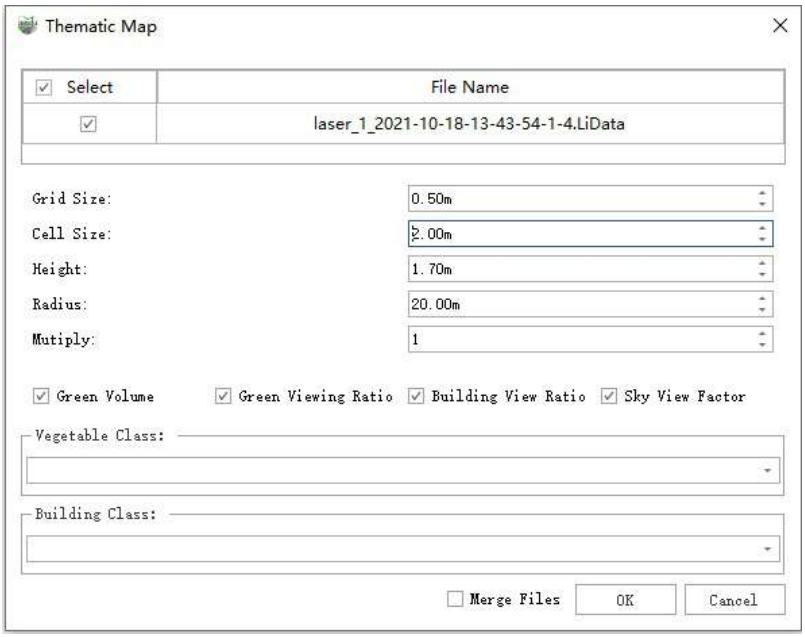
- Thematic Map
- Export Report
- Visual Analysis

Thematic Map

Function Description: Calculate and generate thematic maps for evaluating the three-dimensional visual quality of streets. The available indicators are: green volume, green view ratio, building view ratio and sky view factor.

Steps

1.After launching the software, click on the **Thematic Map** button to open the parameter settings window. Select the desired indicators by checking the corresponding checkboxes:

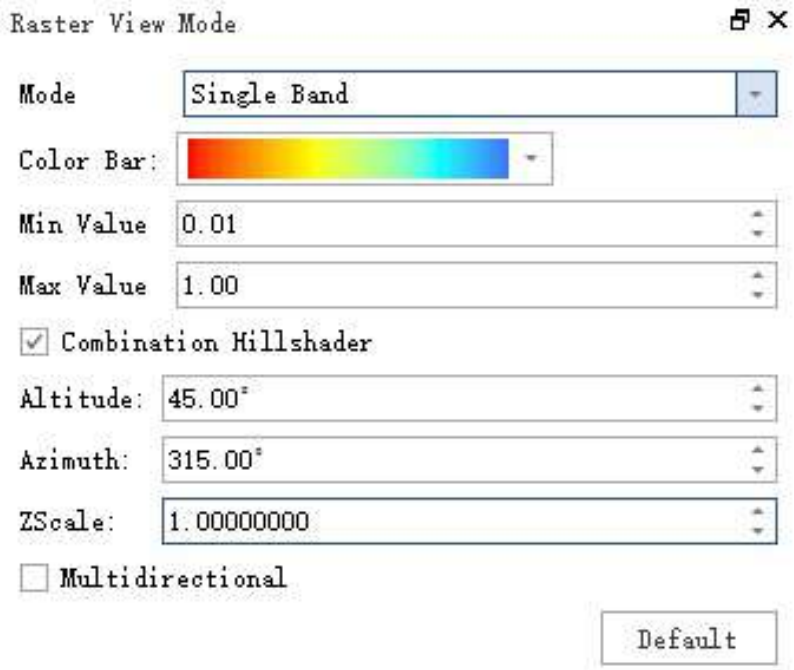


Parameter Description

- **Grid Size**: Voxel size for point cloud voxelization.
- **Cell Size**: The size of each pixel in the generated thematic map.
- **Height**: The height of the viewpoint above the ground.
- **Radius**: The visual distance used for calculating the three-dimensional visual quality indicators.
- **Mutiply**: The subsampling factor for visual sampling during the calculation of three-dimensional visual quality indicators. A higher factor results in faster computation but lower accuracy.
- **Green Volume**: Calculate the volume of green vegetation for each ground grid. Check this checkbox to calculate this indicator.
- **Green View Ratio**: Calculate the ratio of visible green vegetation to total vegetation. Check this checkbox to calculate this indicator.
- **Building View Ratio**: Calculate the ratio of visible buildings to total buildings. Check this checkbox to calculate this indicator.
- **Sky View Factor**: Calculate the ratio of unobstructed sky view to total view. Check this checkbox to calculate this indicator.
- **Vegetation Class**: Specify the vegetation class for calculating the three-dimensional visual quality indicators. Low vegetation points, medium vegetation points and high vegetation points can all be classified as vegetation.
- **Building Class**: Specify the building class for calculating the three-dimensional visual quality indicators.

- **Merge Files**：Unchecked: Generate thematic maps separately for multiple files.Checked: After checking, support generating thematic maps separately for multiple files.

2.After clicking **OK**, the corresponding images will be generated and saved inside the raster according to the checked indicators: green volume, green view ratio, building view ratio and sky view factor. The **Ortho** window allows you to see how the images are displayed. Right-click **Raster View Mode** to bring up a pop-up box that allows you to view the generated images in different modes of display.



Raster View Mode Picture

Parameter Description

- **View**：Select to set the raster color.
- **Mode**：Display by Single Band stretch or Hill Shade effect(Only single-band raster is supported, multi-band is displayed in regular rgb).
- **Color Bar**：Select color sampling for single-band stretching.
- **Max Value**：Maximum value of single-band stretching.
- **Min Value**：Minimum value of single-band stretching.
- **Combination Hillshader**：Single-band superimposed hill shade effect.
- **Altitude**： The altitude angle is the angle of the sun's elevation above the horizon, which ranges from 0 to 90 degrees. A value of 0 degree indicates that the sun is on the horizon, i.e., in the same horizontal plane as the reference frame. The 90 degree value indicates that the sun is directly above the head. The default value is 45 degrees above the horizon.
- **Azimuth**：Azimuth is the relative position (in degrees) of the sun along the horizon. This position is indicated by the sun angle measured clockwise from due north. 0 degrees azimuth indicates due north, and 90 degrees due east, as well as 180 degrees due south and 270 degrees due west. The default azimuth is 315 degrees northwest.
- **Z Scale**：The ZScale is the scale factor used to convert the elevation values. Its purpose is to convert height units (e.g. meters or feet) to horizontal coordinate units (possibly feet, meters or degrees) of the dataset, adding vertical exaggeration for achieving visual effects.
- **Multidirectional**：Multi-directional hill shading improves terrain visualization. This type blends light from six

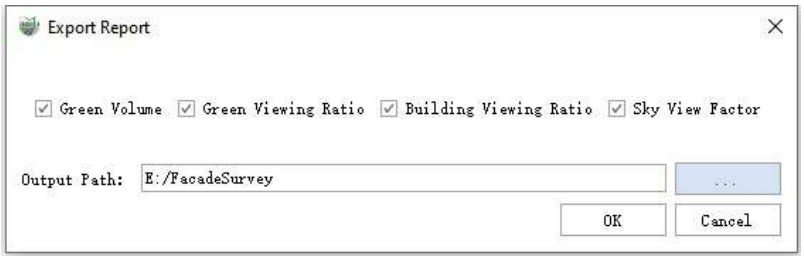
different directions to enhance the visualization of the terrain and improve the appearance of lower areas of the terrain. What's more, it results in the output of topographic backgrounds suitable for use as topography, soils, hydrology, land cover, or other thematic maps enhanced with data from topography.

Export Report

Function Description：After calculating the three-dimensional visual space quality evaluation indicators for streets, it can generate result files. The Export Report button can organize these result files into an HTML report.

Steps

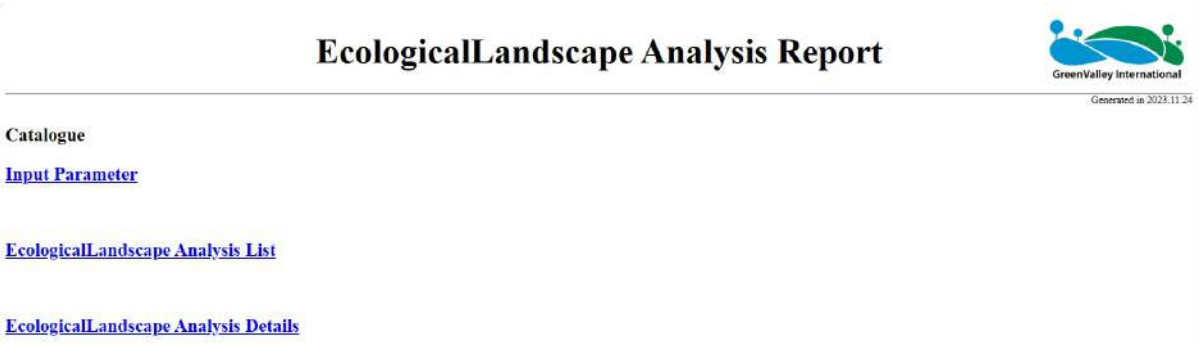
1.Launch the software and click on the **Export Report** button. This will bring up the parameter settings interface for exporting the report.



- **Green Volume**: Calculate the volume of green vegetation for each ground grid. Check this checkbox to calculate this indicator.
- **Green Viewing Ratio**: Calculate the ratio of visible green vegetation to total vegetation. Check this checkbox to calculate this indicator.
- **Building Viewing Ratio**: Calculate the ratio of visible buildings to total buildings. Check this checkbox to calculate this indicator.
- **Sky View Factor**: Calculate the ratio of unobstructed sky view to total view. Check this checkbox to calculate this indicator.
- **Output Path**: Set the path for storing the report.

2.The report content is divided into three main parts: the header, the table of contents, and the specific content.

（1）Header and Table of Contents.



（2）Clicking on "Pavement" in the table of contents will navigate you to the corresponding section. The specific content includes:

Input Parameter

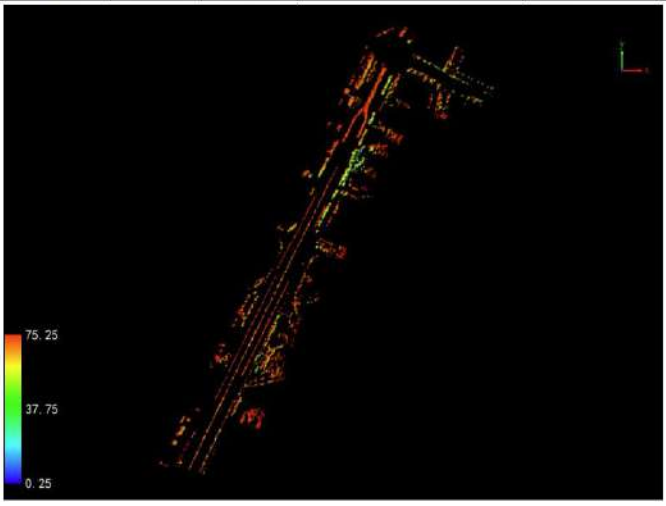
Laser File	E:/laser_1_2021-10-18-13-43-54-1-4.LiData
Radius(m)	20
Height(m)	1.7
Grid Size(m)	0.5
Cell Size(m)	2
Green Class	Low Vegetation、High Vegetation
Building Class	Building

EcologicalLandscape Analysis List

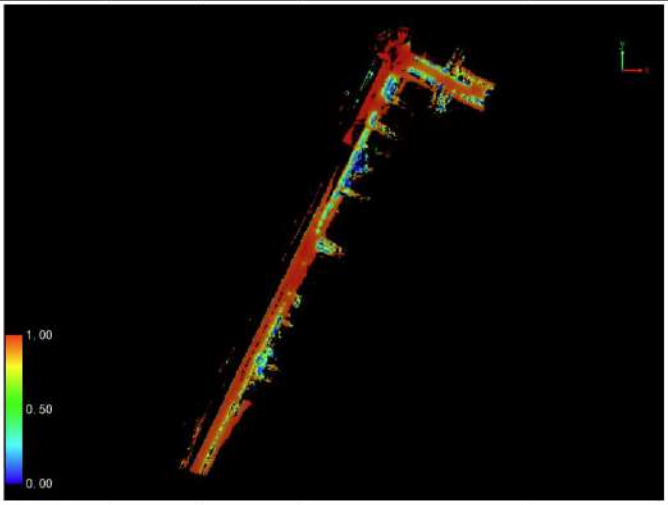
Laser File	E:/laser_1_2021-10-18-13-43-54-1-4.LiData			
Type	Min Value	Max Value	Min Coordinate	Max Coordinate
Green Volume	0.000	75.250	(185528.297,3381983.751)	(185880.297,3381423.751)
Green Viewing Ratio	0.000	1.000	(185528.297,3381983.751)	(185708.297,3381709.751)
Building Viewing Ratio	0.000	1.000	(185528.297,3381983.751)	(185868.297,3381469.751)
Sky View Factor	0.000	1.000	(185528.297,3381983.751)	(185868.297,3381353.751)

EcologicalLandscape Analysis Details

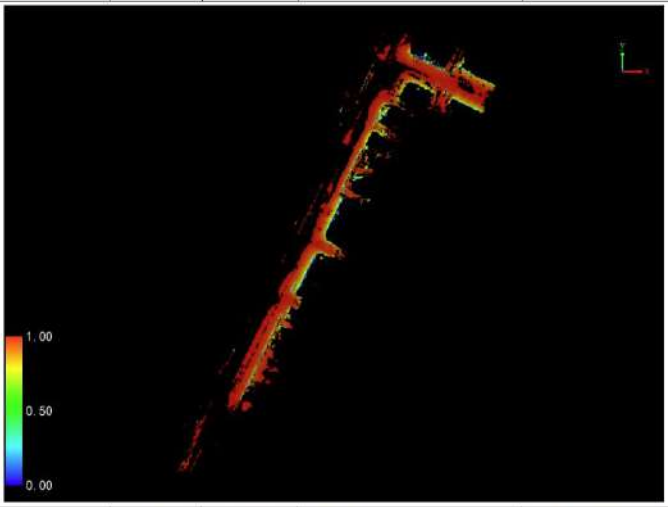
Laser File	E:/laser_1_2021-10-18-13-43-54-1-4.LiData			
Type	Min Value	Max Value	Min Coordinate	Max Coordinate
Green Volume	0.000	75.250	(185528.297,3381983.751)	(185880.297,3381423.751)



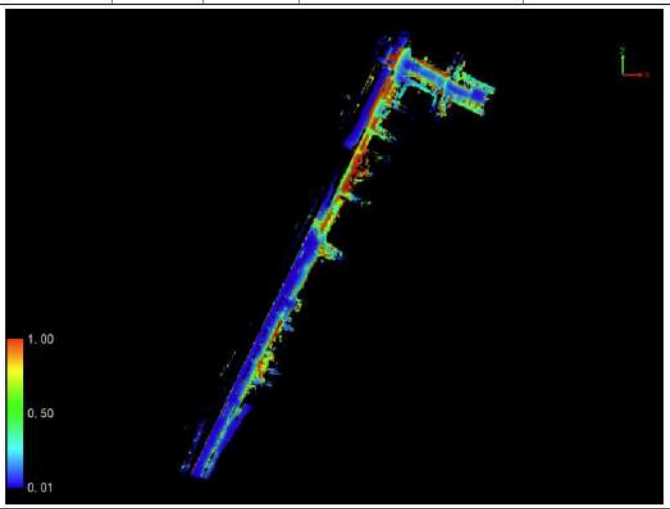
Green Viewing Ratio	0.000	1.000	(185528.297,3381983.751)	(185708.297,3381709.751)



Building Viewing Ratio	0.000	1.000	(185528.297,3381983.751)	(185868.297,3381469.751)



Sky View Factor	0.000	1.000	(185528.297,3381983.751)	(185868.297,3381353.751)



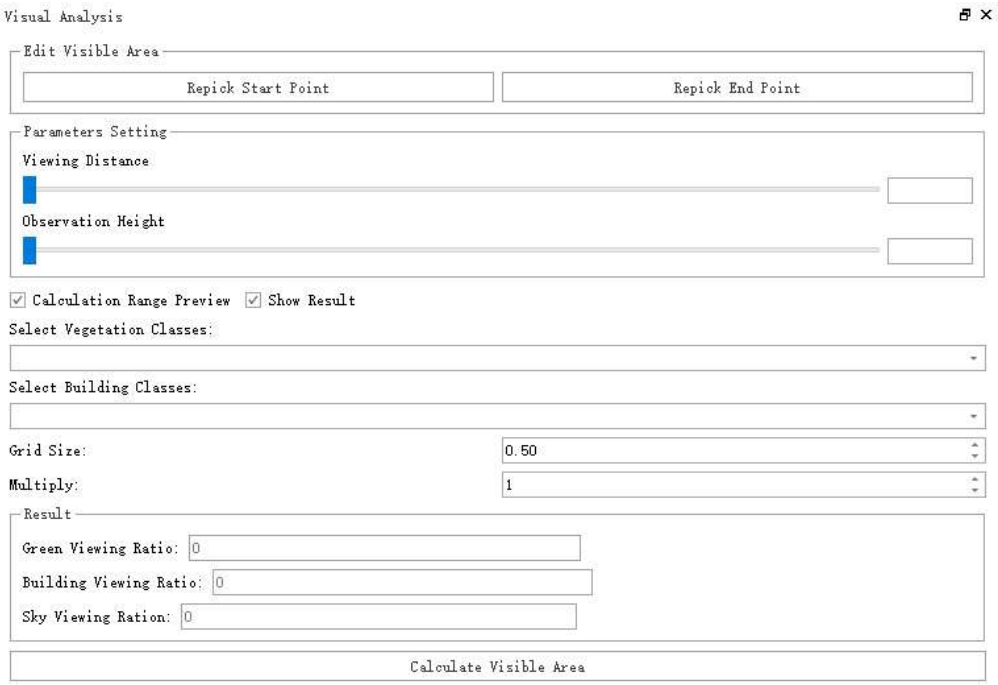
3.Multiple point clouds are supported, allowing the merging of multiple thematic map reports. It also supports multiple point clouds without merging, generating a separate result report for each individual point cloud.

Visual Analysis

Function Description: This tool allows you to analyze the visible areas of selected point cloud categories within a specified observation range from an observation point.

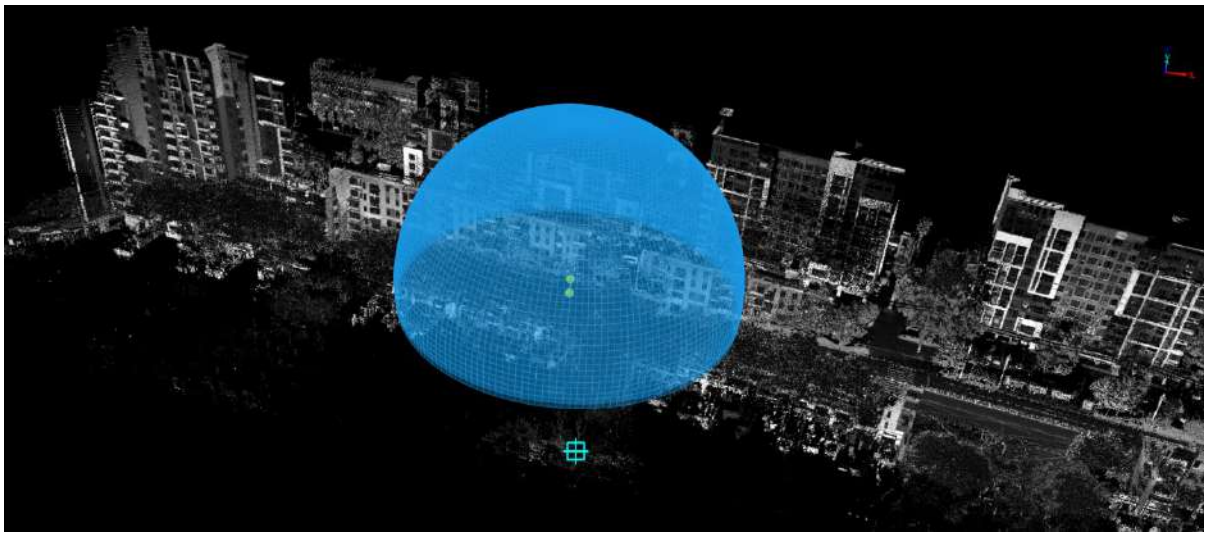
Steps

1.Click on the **Visual Analysis** button. The visibility analysis settings window will appear on the right side of the software, as shown below:



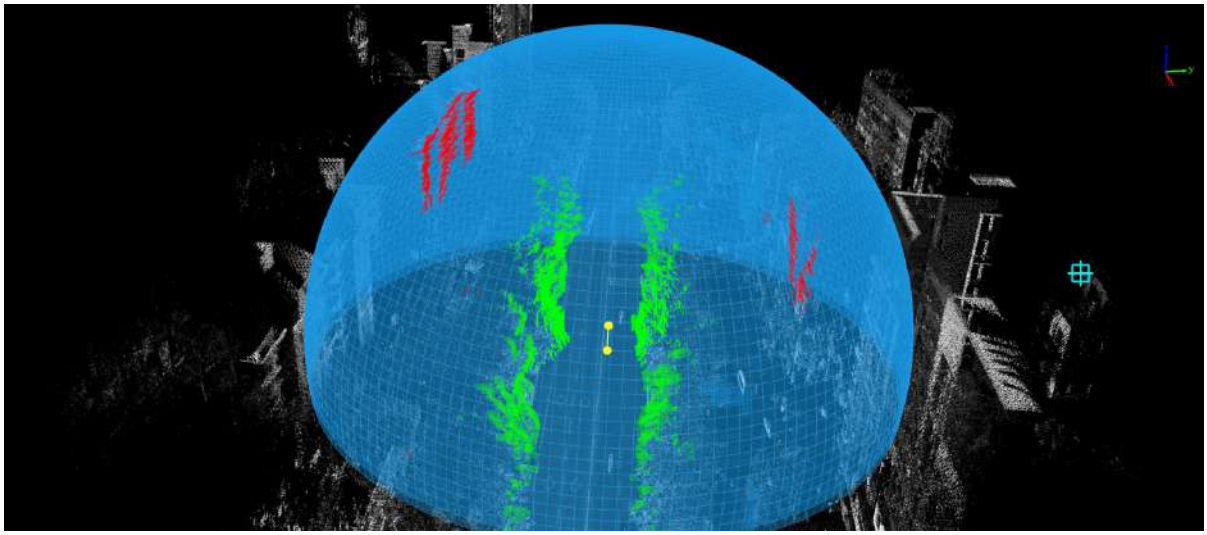
Visibility Analysis Settings Window

2.You can select the observation point by left-clicking in the 3D view. After selecting the observation point, you can preview the observation range by moving the mouse. Once the range is set, double-click the left mouse button to confirm the observation range. In addition, the view distance and the height of the observation point can be reselected according to the requirements. See the example below:



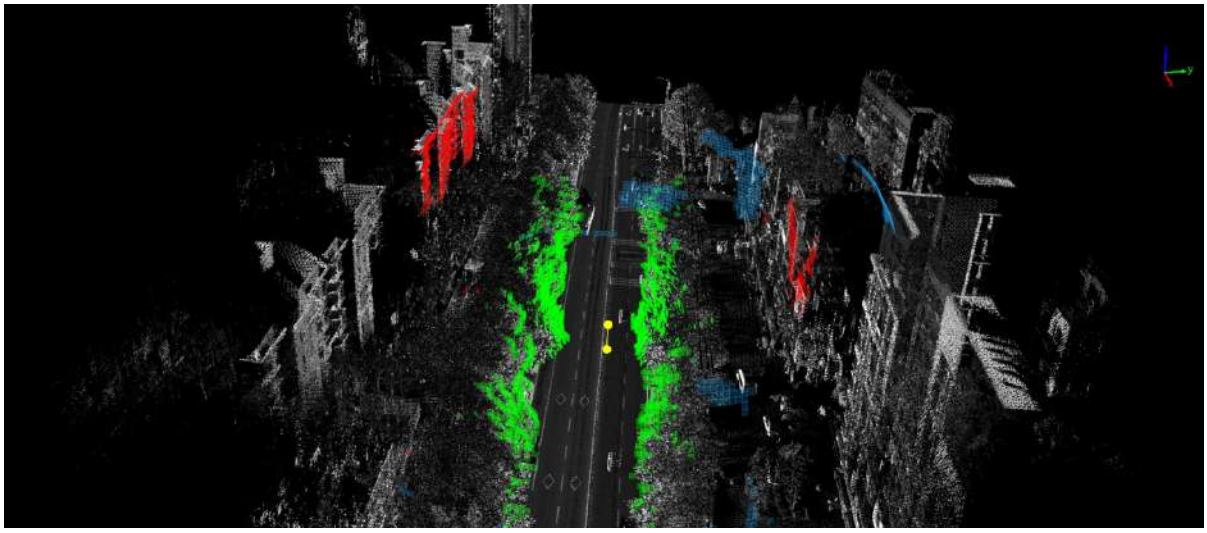
Selecting the observation point and observation range

3.If the observation point is selected incorrectly, you can click the "**Repick Start Point**" button in the visibility analysis settings window to choose a new observation point or click the "**Repick End Point**" button to choose a new endpoint for the observation range. Once the observation point and observation range are finalized, the next step is to select the point cloud vegetation type and point cloud building type in the settings window. You can also set the grid size and point cloud subsampling factor for the calculation. After setting all the parameters, click on the "**Compute Visibility Area**" button. Once the progress reaches 100%, the 3D view will display the computed visibility results. See the example below:



Calculation range and results

4.You can hide the calculation range by unchecking the **Preview Calculation Range** option in the settings window to only display the computed results. You can also toggle the visibility of the results by checking or unchecking the "Show Results" option. See the example below:



Computed results

View tool

Controls the display of the currently active view.

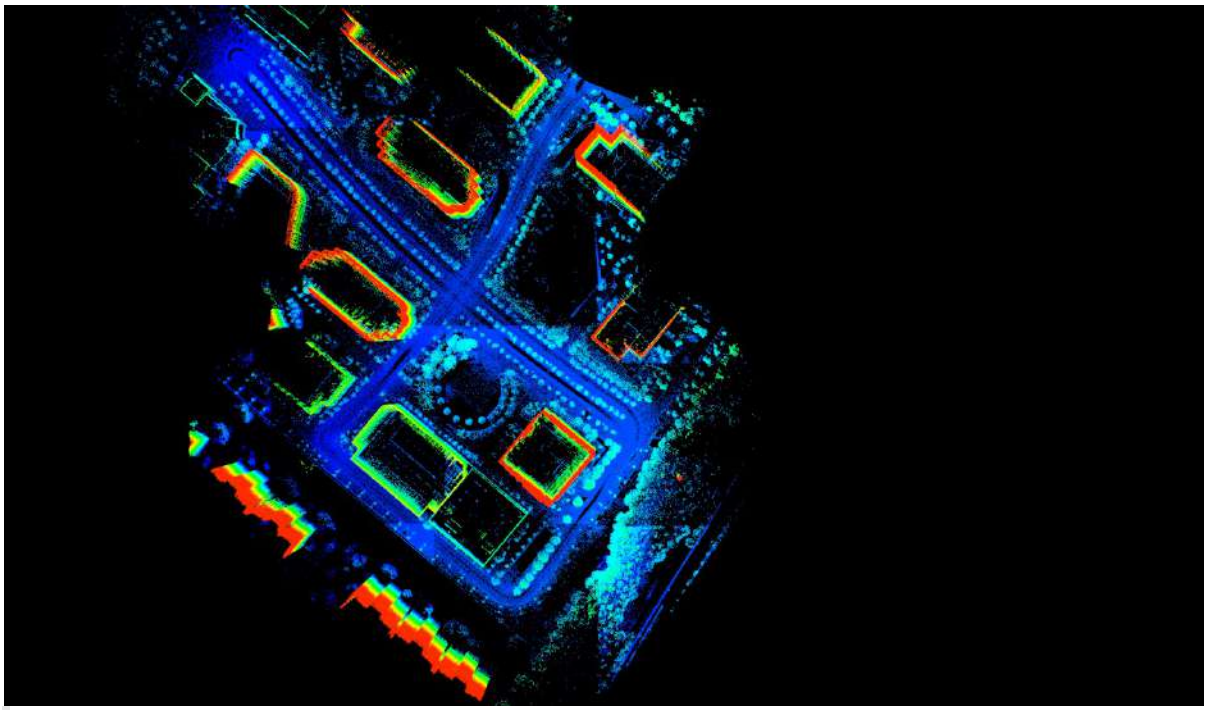
- Top View
- Bottom View
- Left View
- Right View
- Front View
- Back View
- Front Isometric View
- Back Isometric View
- Full Extent
- Set Projection Mode
- Configure Point Size
- Capture Image

Top View

Function Description: This tool allows the user to set the camera position to view from the top, that is, from +z to-z to view the three- dimensional data, the plane is the x-y plane.

Steps

1.Click **Top** button, the current activation form is displayed as shown in the figure：



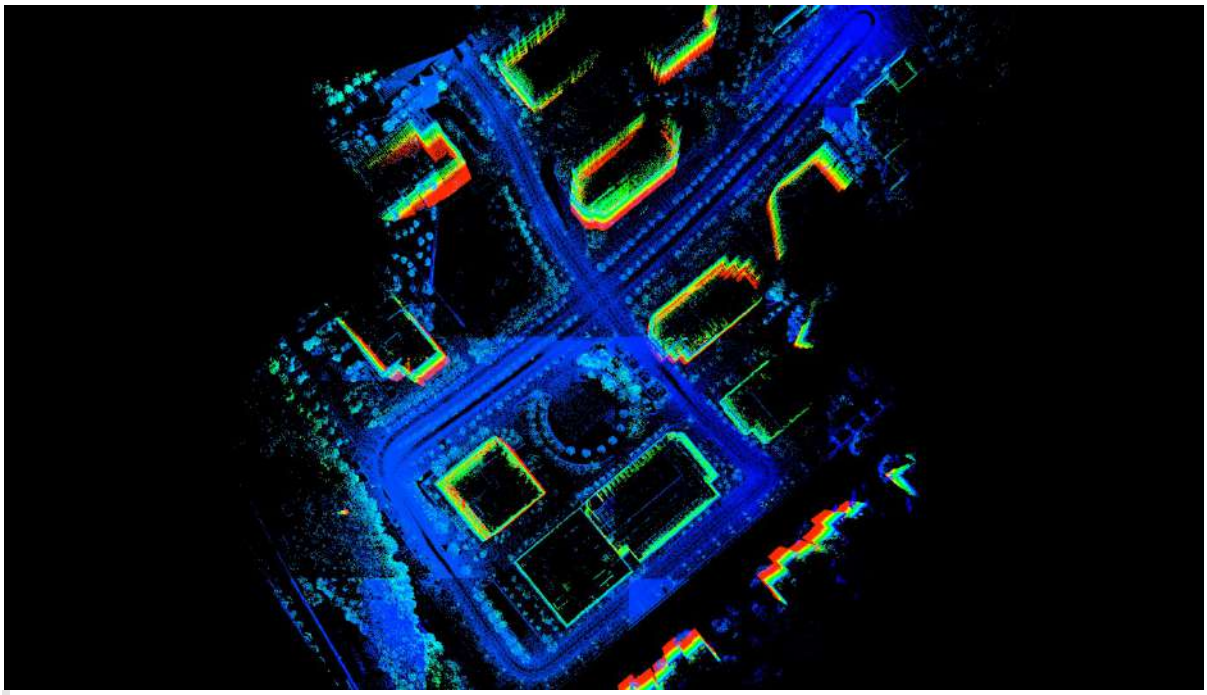
> Note: This function is only for views displayed in 3D; this function does not reset the center position of the viewpoint. If you need to reset to the default viewpoint, please click Global Display.

Bottom View

Function Description: This tool allows the user to set the camera position to view from the bottom, that is, from-z to +z to view the three- dimensional data, the plane is the x-y plane.

Steps

1.Click **Bottom** button, the current activation form is displayed as shown in the figure：



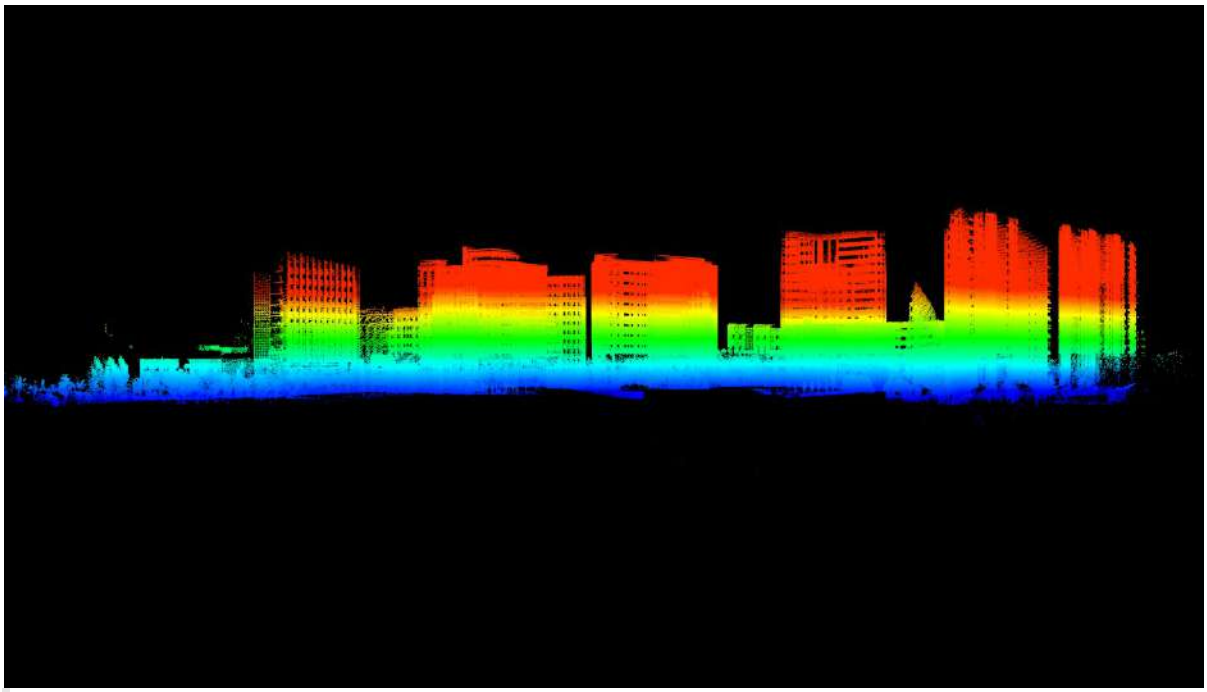
> Note: This function is only for views displayed in 3D; this function does not reset the center position of the viewpoint. If you need to reset to the default viewpoint, please click Global Display.

Left View

Function Description: This tool allows the user to set the camera position to view from the left, that is, from -x to +x to view the three- dimensional data, the plane is the y-z plane.

Steps

1.Click **Left** button, the current activation form is displayed as shown in the figure:



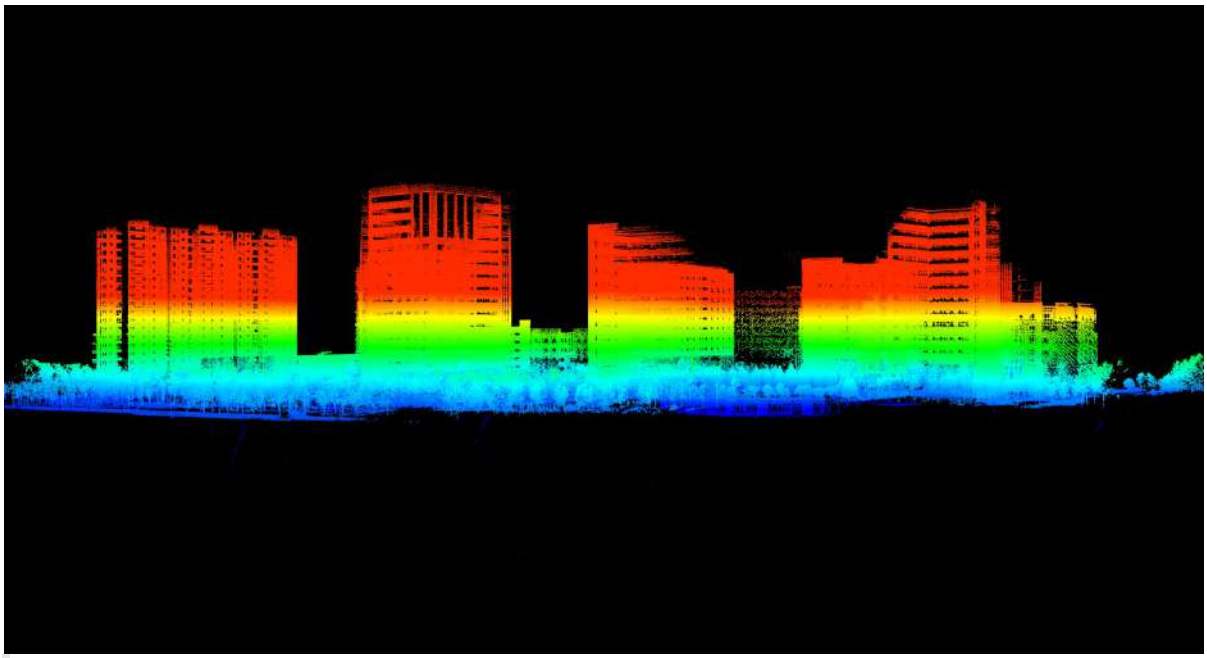
> Note: This function is only for views displayed in 3D; this function does not reset the center position of the viewpoint. If you need to reset to the default viewpoint, please click Global Display.

Right View

Function Description: This tool allows the user to set the camera position to view from the right, that is, view the three-dimensional data from +x to-x direction, and the plane is the y-z plane.

Steps

1.Click **Right** button, the current activation form is displayed as shown in the figure：



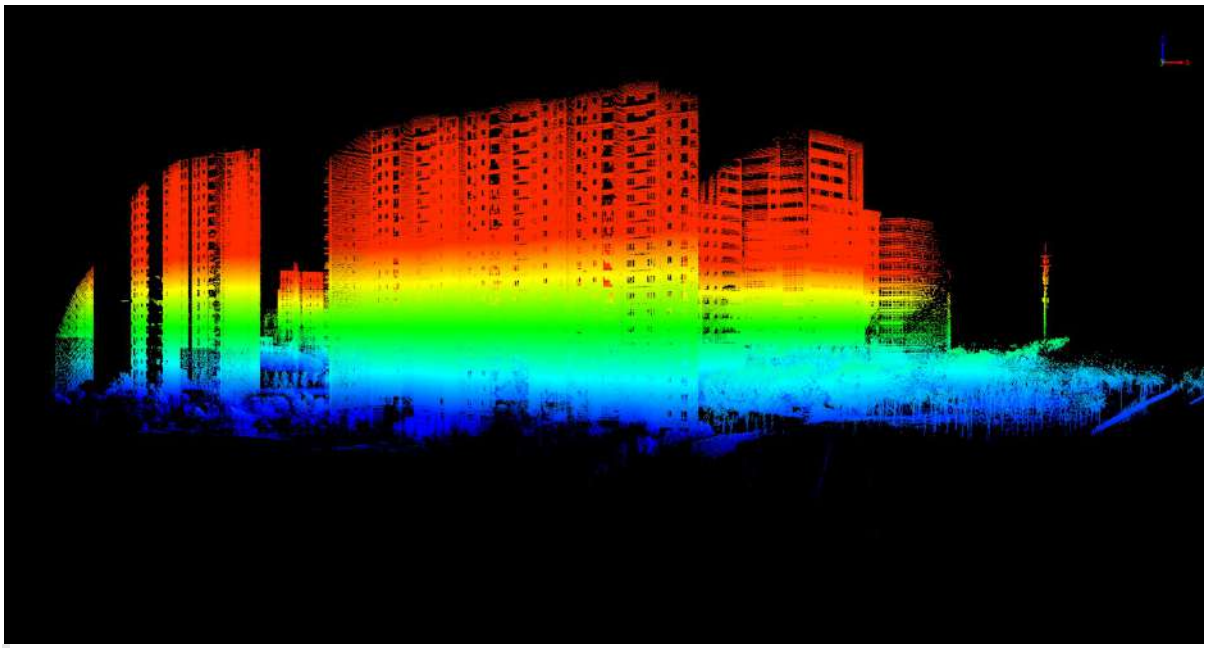
> Note: This function is only for views displayed in 3D; this function does not reset the center position of the viewpoint. If you need to reset to the default viewpoint, please click Global Display.

Front View

Function Description: This tool allows the user to set the camera position to view from the front, that is, view the three-dimensional data from the-y to +y direction, and the plane is the x-z plane.

Steps

1.Click **Front** button, the current activation form is displayed as shown in the figure：



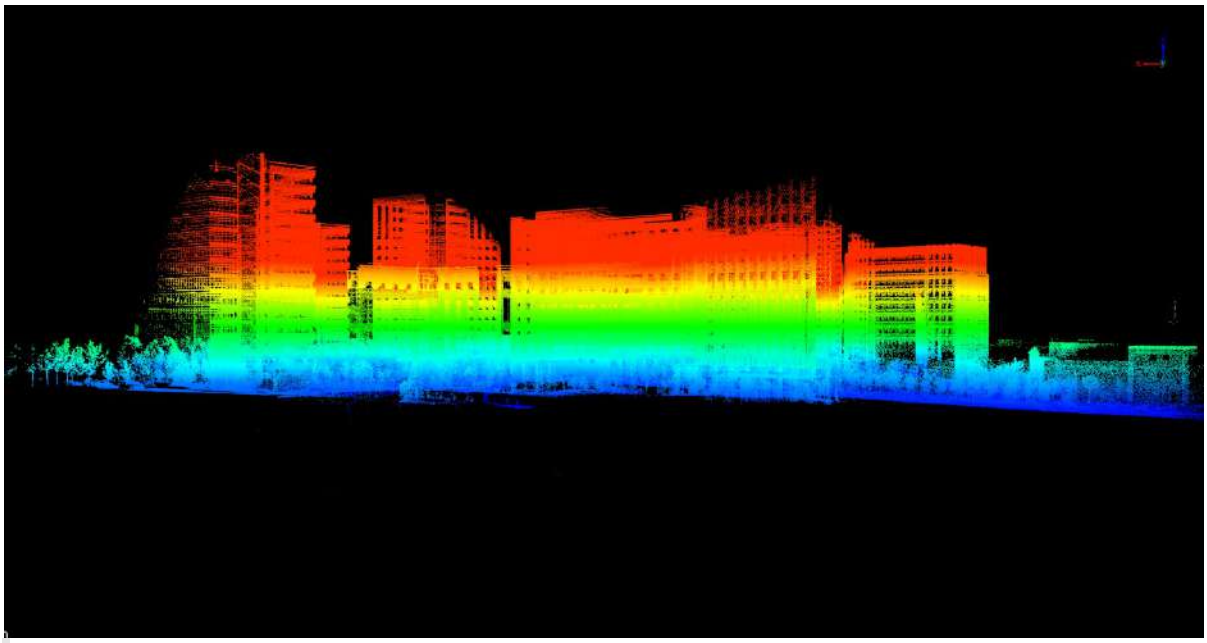
> Note: This function is only for views displayed in 3D; this function does not reset the center position of the viewpoint. If you need to reset to the default viewpoint, please click Global Display.

Back View

Function Description: This tool allows the user to set the camera position to view from the back, that is, view the three-dimensional data from +y to-y direction, and the plane is the x-z plane.

Steps

1.Click **Back** button, the current activation form is displayed as shown in the figure：



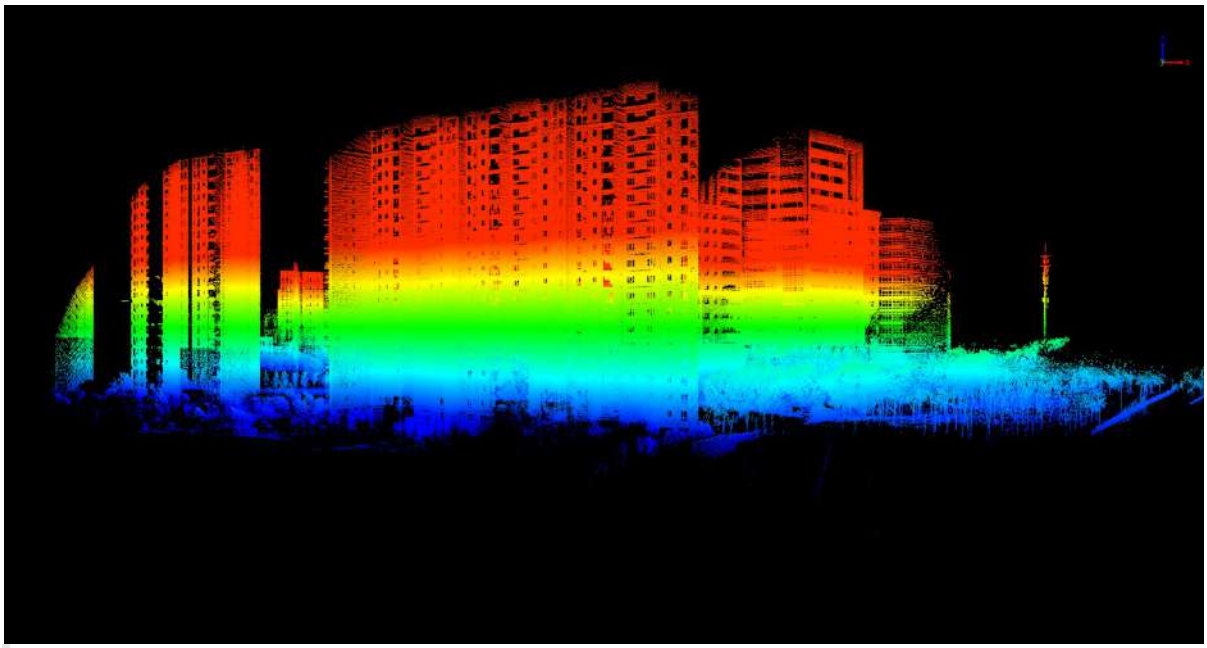
> Note: This function is only for views displayed in 3D; this function does not reset the center position of the viewpoint. If you need to reset to the default viewpoint, please click Global Display.

Isometric Front View

Function Description: This tool allows the user to set the camera position to tilt 45° in front of x-y.

Steps

1.Click **Front Isometric** button, the current activation form is displayed as shown in the figure:



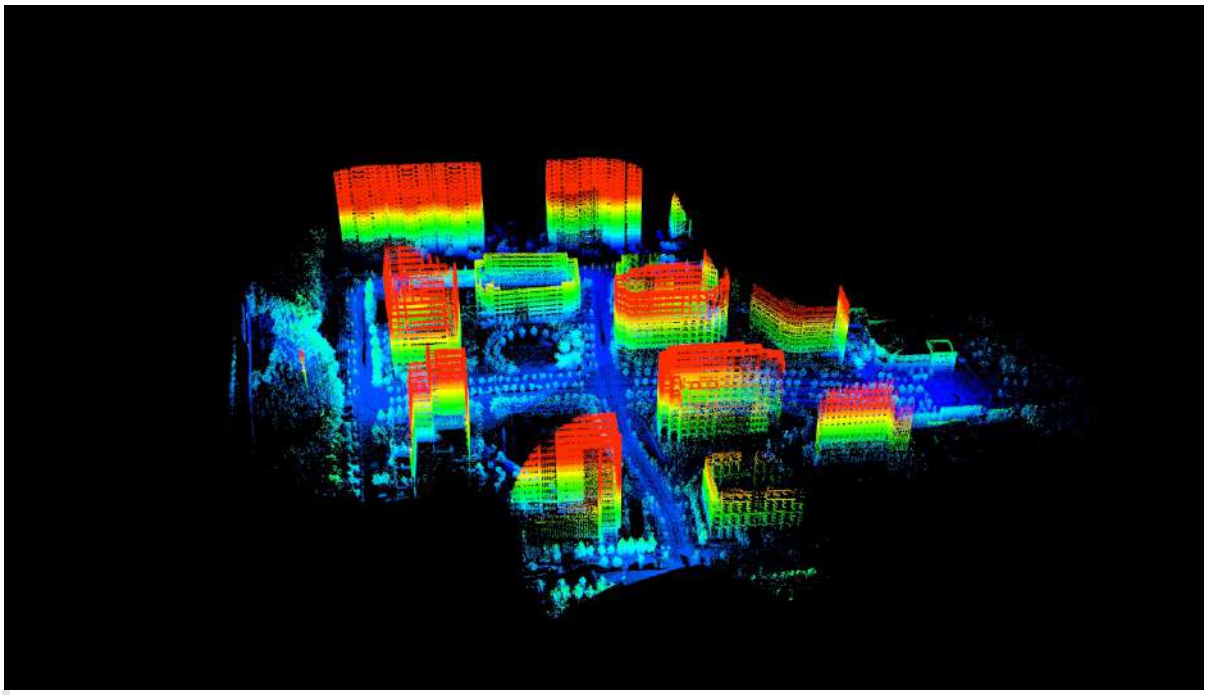
> Note: This function is only for views displayed in 3D; this function does not reset the center position of the viewpoint. If you need to reset to the default viewpoint, please click Global Display.

Isometric Back View

Function Description: This tool allows the user to set the camera position to tilt 45° backward on the x-y surface.

Steps

1.Click **Back Isometric** button, the current activation form is displayed as shown in the figure：



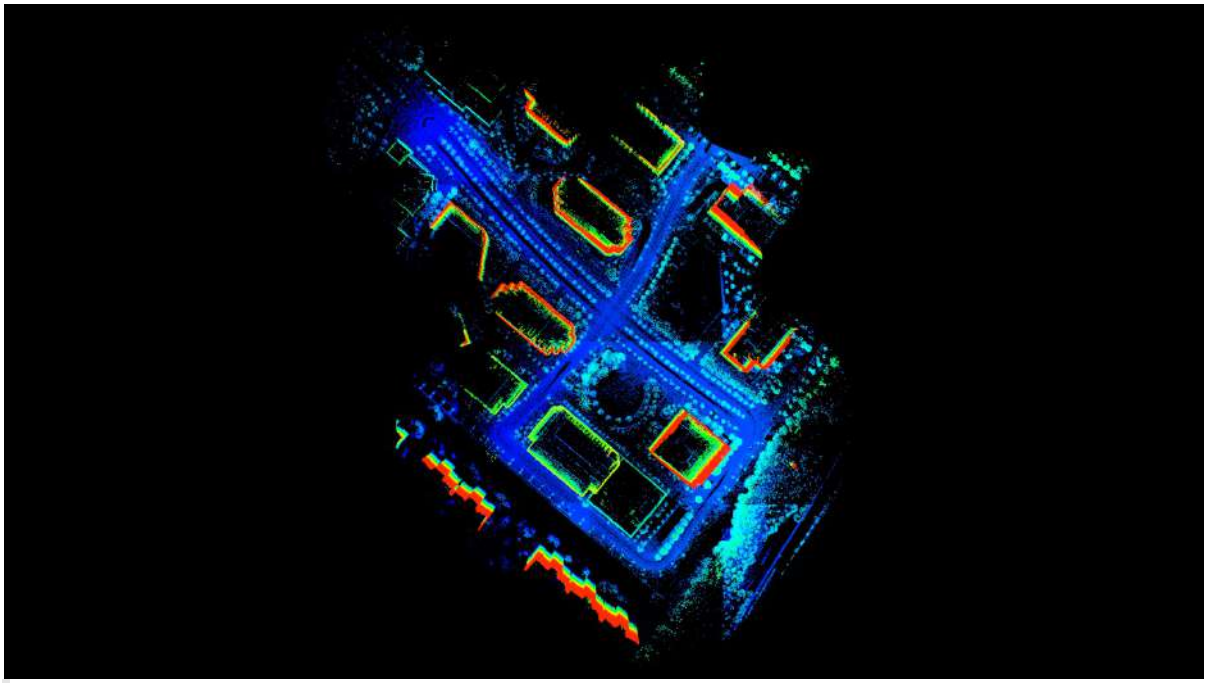
> Note: This function is only for views displayed in 3D; this function does not reset the center position of the viewpoint. If you need to reset to the default viewpoint, please click Global Display.

Full Extent

Function Description: The global display function is suitable for the 3D window of the LiDAR360 MLS software, which is used to make all the data in the 3D window cover the entire window in the form of a top view, in order to achieve the purpose of global browsing of the data.

Steps

1.Click on the toolbar **Full Extent** button, the data in the 3D window will be automatically scaled to cover the entire window, as shown in the figure：



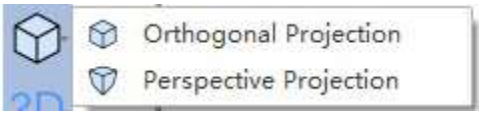
> Note: This function is only for views displayed in 3D; this function does not reset the center position of the viewpoint. If you need to reset to the default viewpoint, please click Global Display.

Set Projection Mode

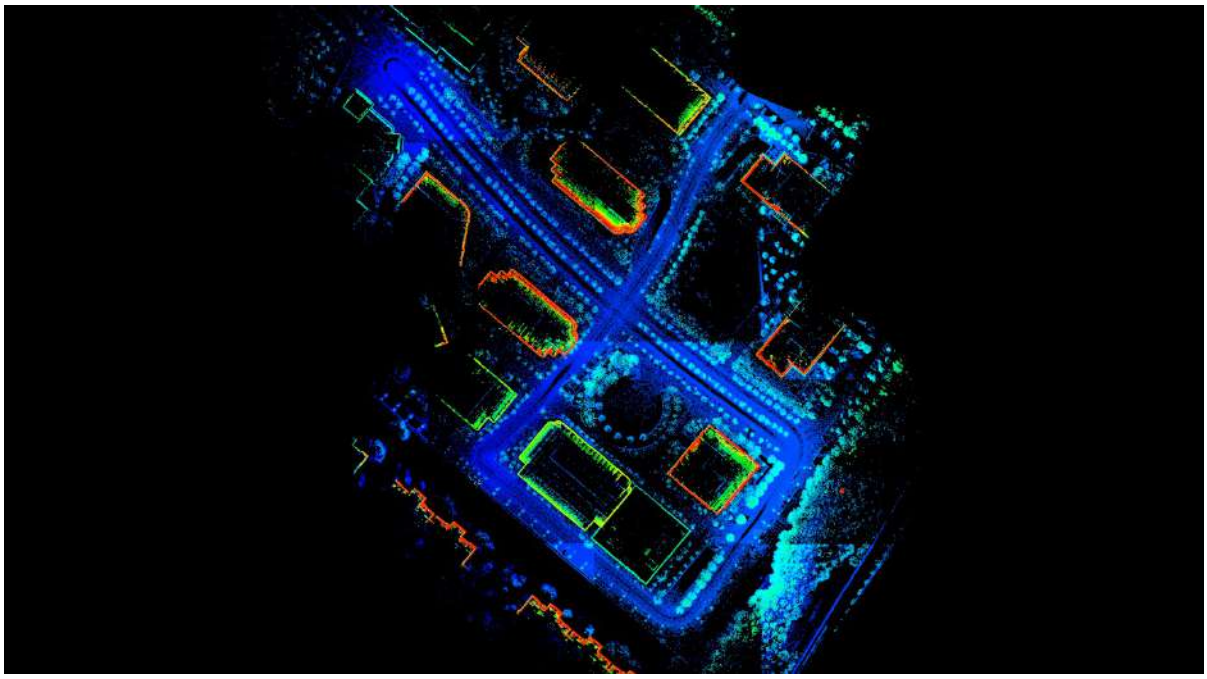
Function Description: This tool allows the user to change the projection method of the view and supports two projection methods, orthogonal and perspective.

Steps

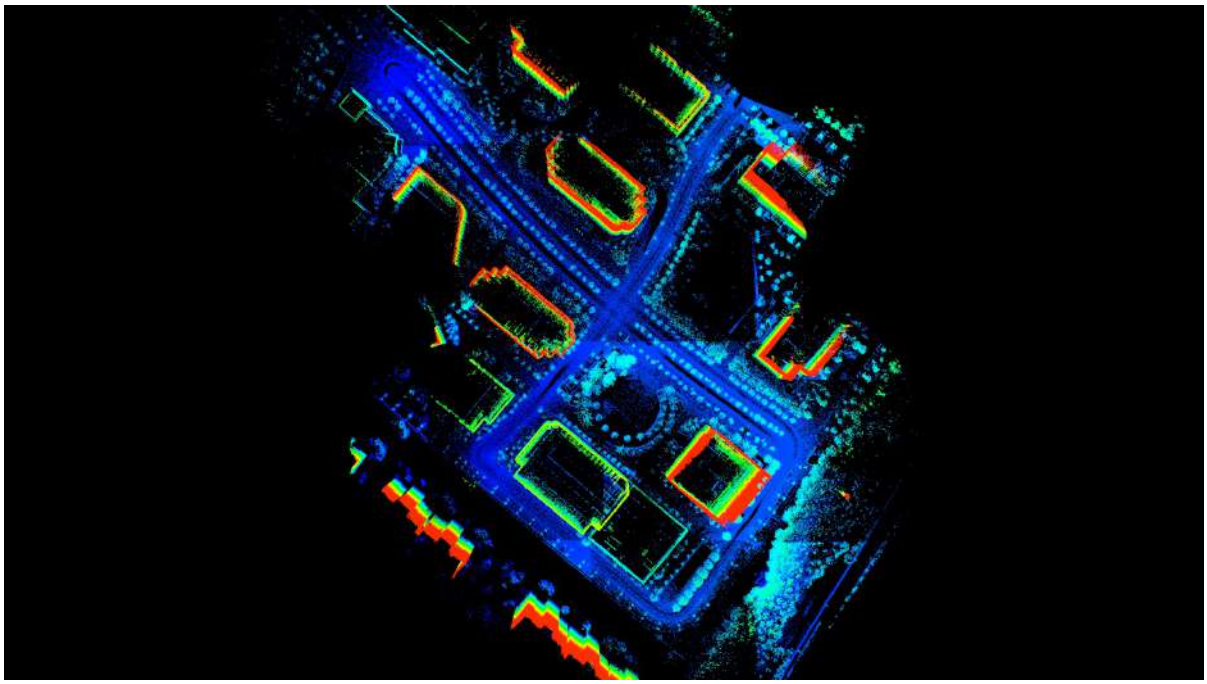
1.Click **Set Projection Mode** button, pop up the option to select the projection method.



2.If you select orthogonal projection, the point cloud window is orthogonal projection.



3.If perspective projection is selected, the point cloud window is perspective projection.

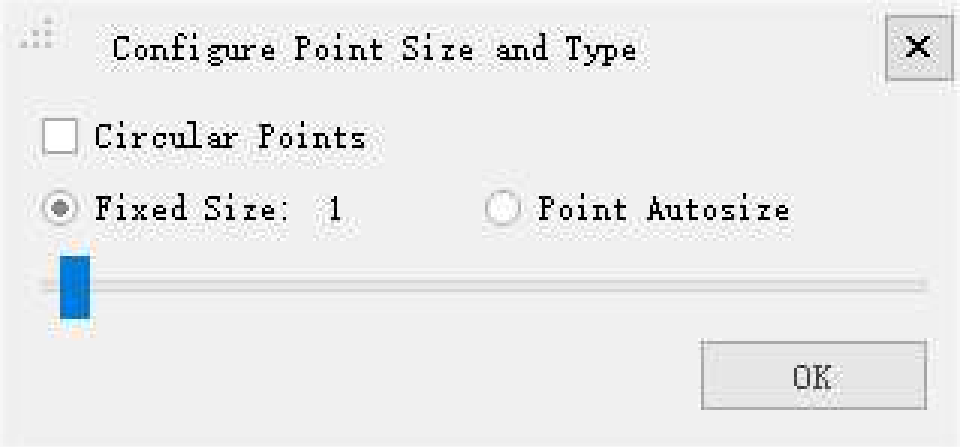


Configure Point Size

Function Description: Set the size and type of points in the three-dimensional point cloud in the entire software system.

Steps

1.Click **Configure Point Size** button, the interface shown in the figure below pops up:



Parameter setting

- **Circular Points (Circle)**: This parameter defines the display type of point cloud. If checked, click to display according to the circle, if not checked, click to display according to the rectangle.
- **Fixed Size (Fixed Size)**: This parameter defines the point of the software system to be displayed in a fixed size. You can slide the slider below to set the point to be displayed between 0-50 pixels.
- **Point Autosize (Auto Size)**: This parameter defines the depth display of the visual vertebral body in the midpoint adaptive form of the software system.

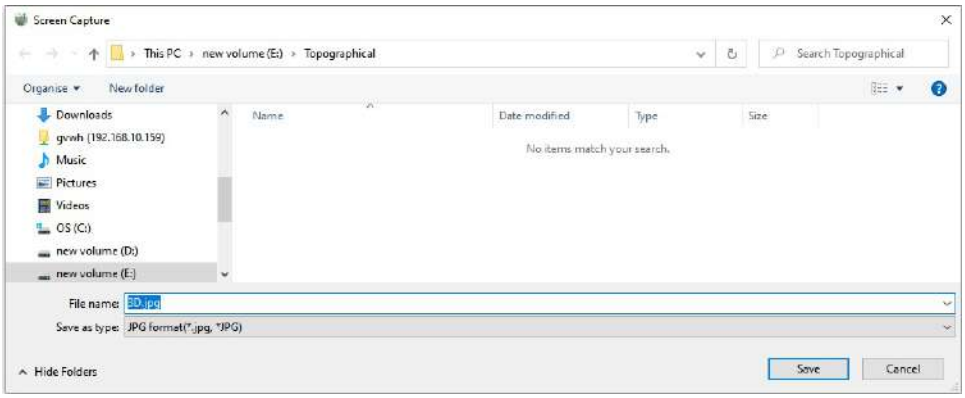
2.Set the size and display type of the point, click OK.

Capture Image

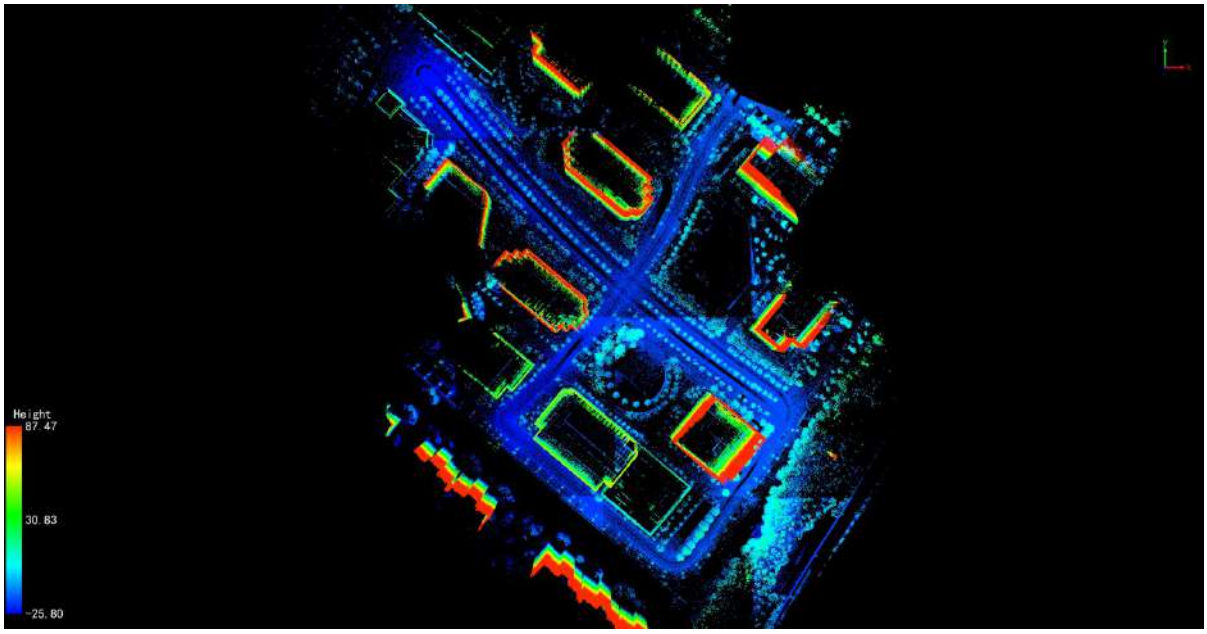
Function Description: This tool allows the user to directly take a screenshot of the point cloud that the user has customized or set a certain viewing angle in the 3D window.

Steps

1.Click **Capture Image** button, pop up the save path：



2.Click save, you can see the scene of the 3D window screenshot in the corresponding folder.



Color bar tool

The color toolbar provides several color display modes for the visualization for large point cloud datasets, and the most suitable display method can be selected for different analysis functions (such as display by intensity, display by GPS time, display by echo frequency, etc.). In addition, LiDAR360 MLS provides EDL tools to enhance the display effect, more intuitively reflect the characteristics of the data, and also help to check the quality of the data.

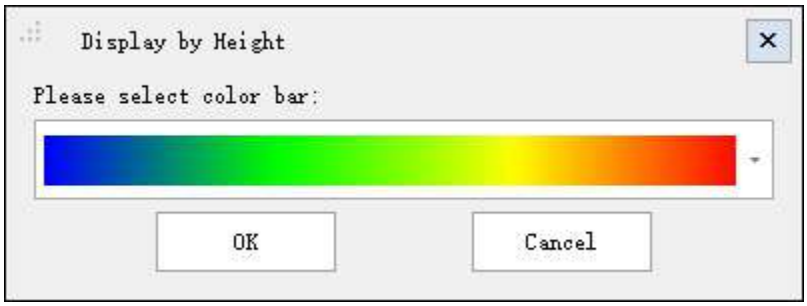
- Display by Height
- Display by Intensity
- Display by Classification
- Display by RGB
- Display by Return
- Display by Time
- Display by EDL
- Display by Xray

Display by Height

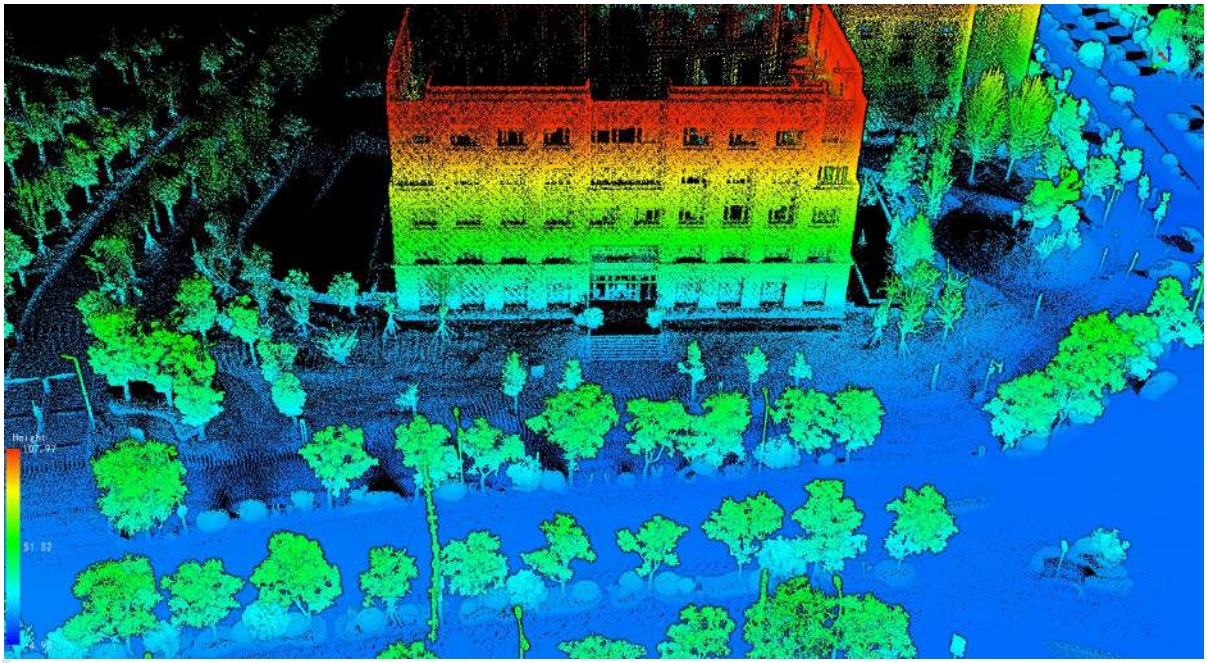
Function Description: It can be used for the display of point cloud data, mapping the elevation attributes of point cloud data to several evenly varying color intervals, and more intuitively showing the changes in the elevation value of point cloud data.

Steps

1.Click button and the Display by Height dialog box should pop up, as shown in the figure.



2.Select the appropriate color bar in the drop-down box and click the OK button. The color indicator in the lower left corner of the window automatically maps the elevation change range of the point cloud data to the selected color bar. At the same time, the point cloud data in the scene is displayed by elevation, which is enhanced with EDL display.



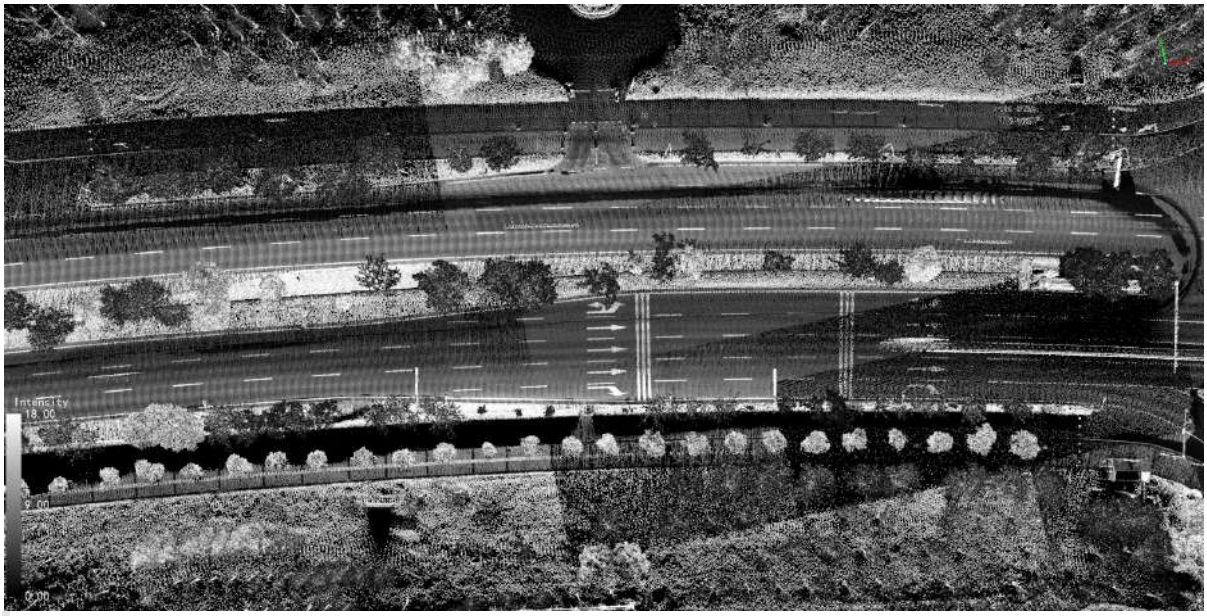
> Note: This function only works on point cloud data.

Display by Intensity

Function Description: This tool can be used for the display of point cloud data, mapping the intensity attributes of point cloud data to evenly varying color intervals, and more intuitively showing the changes in the intensity value of point cloud data.

Step

1.Click Button, the color indicator in the lower left corner of the window automatically maps the range of changes in the intensity of the point cloud data to the color bar, and the user can choose the color of the color bar. The point cloud data in the scene will be displayed by intensity according to the corresponding color bar.



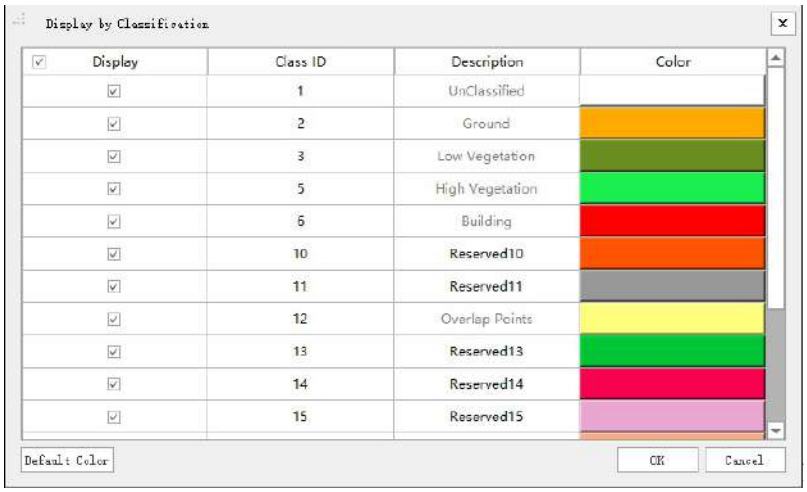
> Note: This function only works on point cloud data.

Display by Classification

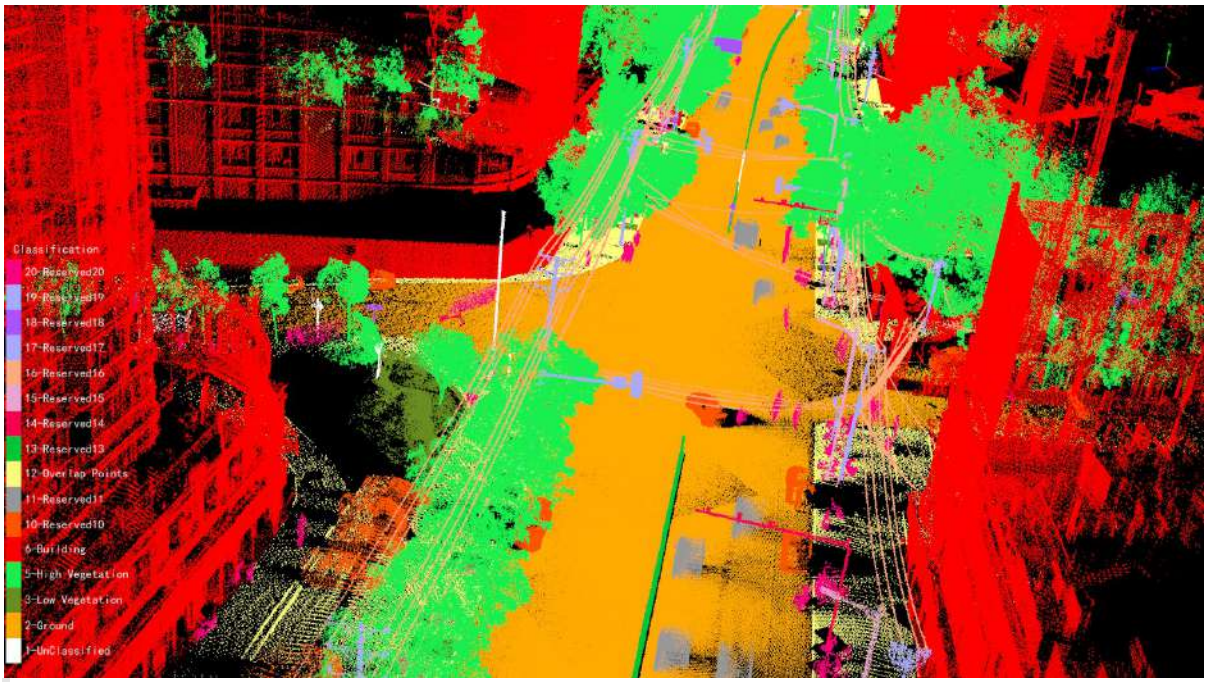
Function Description: This tool can be used for the display of point cloud data, mapping the category attributes of point cloud data to different color values, and more intuitively distinguishing various categories of point cloud data.

Steps

1.Click button and the Display by Classification dialog box should pop up, as shown in the figure.



2.Select different colors for different categories, click the OK button, and the color indicator in the lower left corner of the window automatically maps different categories of point cloud data to the corresponding colors. At the same time, the point cloud data in the scene is displayed by category, which is enhanced with EDL display.



> Note: This function only works on point cloud data.

Display by RGB

Function Description: This tool can be used for the display of point cloud data as well as the point cloud data drawn with the RGB color attributes of existing point cloud data.

Steps

1.Click button, the point cloud data in the scene is displayed according to its own RGB value, the effect is shown in the figure.

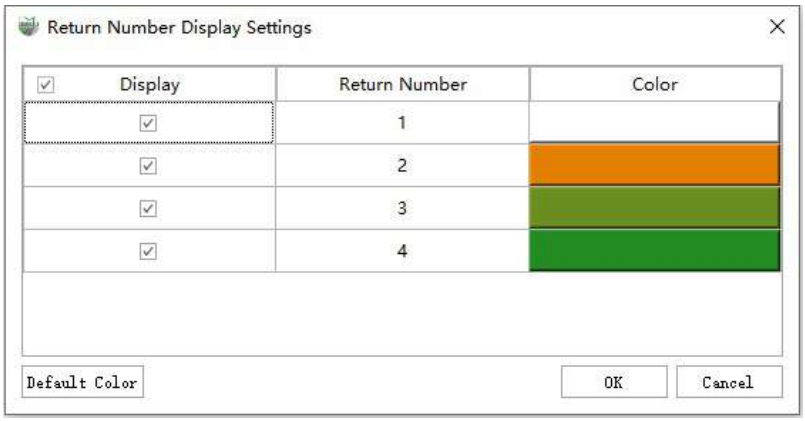
> Note: This function only works on point cloud data that contains RGB attributes.

Display by Return

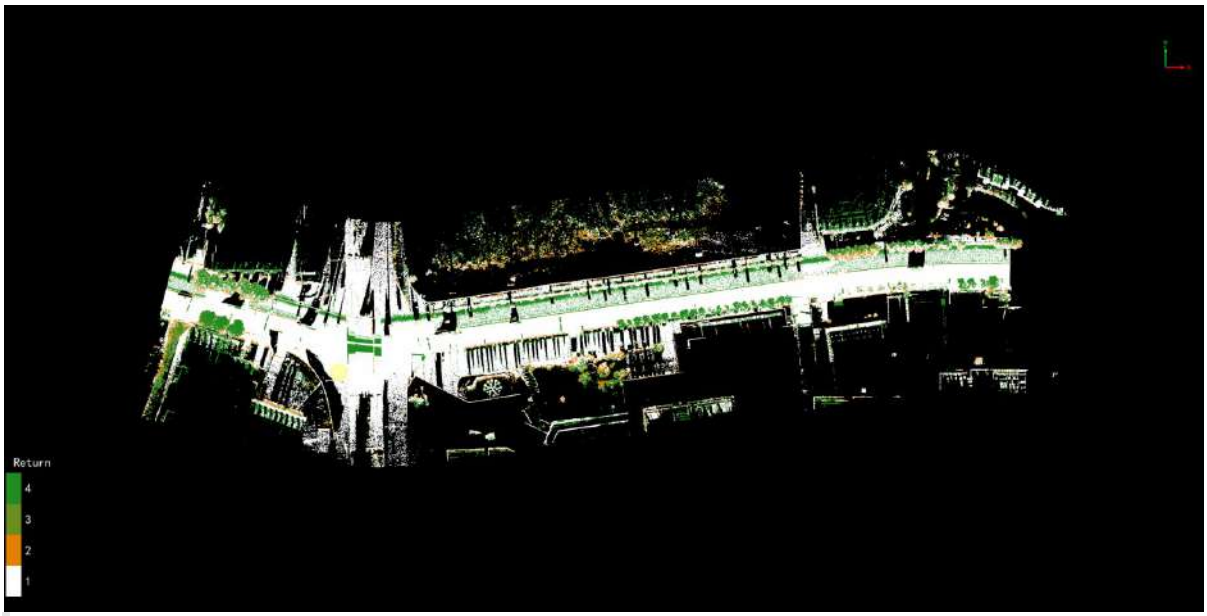
Function Description: This tool can be used for the display of point cloud data, mapping the return number attributes of point cloud data to different color values, and more intuitively distinguishing the echo number of point cloud data.

Steps

1.Click button and the Display by Return Number dialog box should pop up as shown in the figure.



2.Select different colors for different return number, click the OK button, and the color indicator in the lower left corner of the window automatically maps the different return number of the point cloud data to the corresponding color. At the same time, the point cloud data in the scene is displayed according to the return number, which is enhanced with EDL display, the effect is as shown in the figure.



> Note: This function only works on point cloud data.

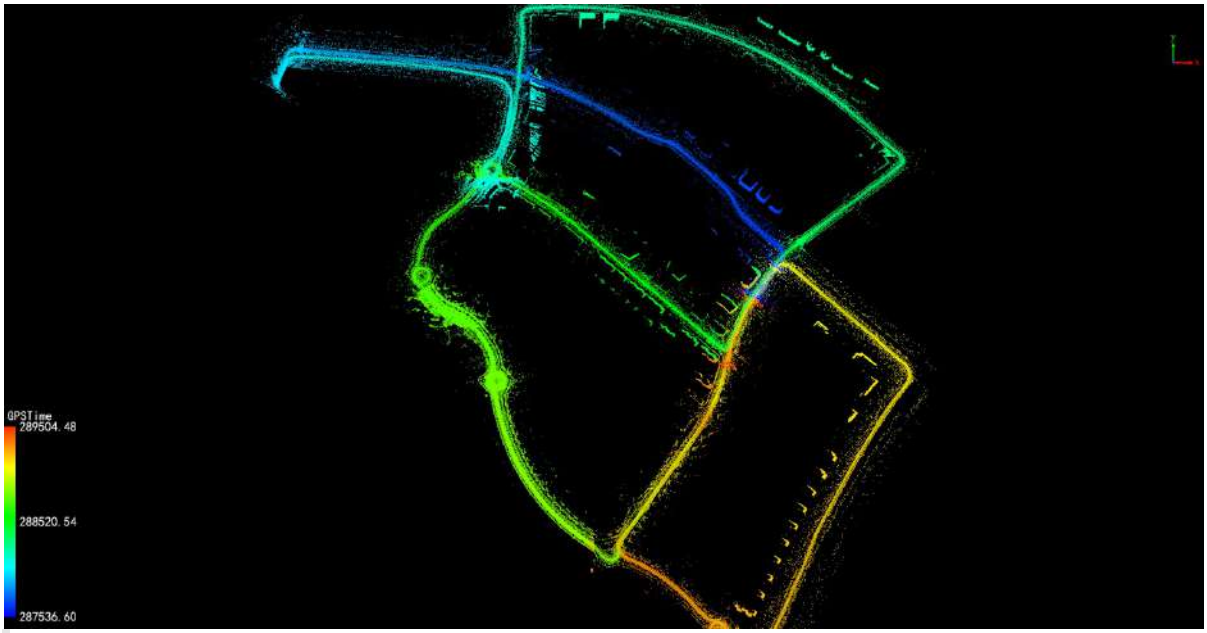
Display by Time

Function Description: This tool can be used for the display of point cloud data, mapping the GPS time attributes of point cloud data to evenly varying color values, and more intuitively showing the changes in the GPS time attributes of point cloud data.

Steps

1.Click button and the Display by GPS Time should pop up.

2.Select the appropriate color bar in the drop-down box, click the OK button, the color indicator in the lower left corner of the window automatically maps the GPS time change range of point cloud data to the selected color bar, and the scene point cloud data is displayed according to GPS time.



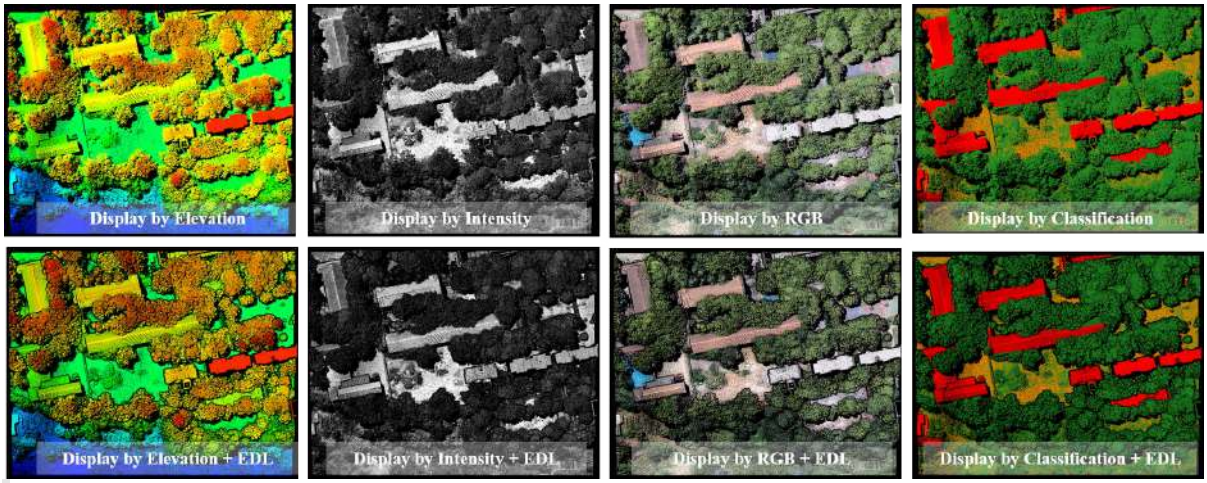
> Note: This function only works on point cloud data.

Display by EDL

Function Description: This tool can be used for the display of point cloud data and used in conjunction with other display methods to enhance the display of the contour characteristic of point cloud objects.

Steps

1.Click button, the point cloud data in the scene will be displayed in the EDL manner to enhance the display effect. The comparison of the point cloud display effect before and after using EDL in different display methods is shown in the figure.



> Note: This function only works on point cloud data.

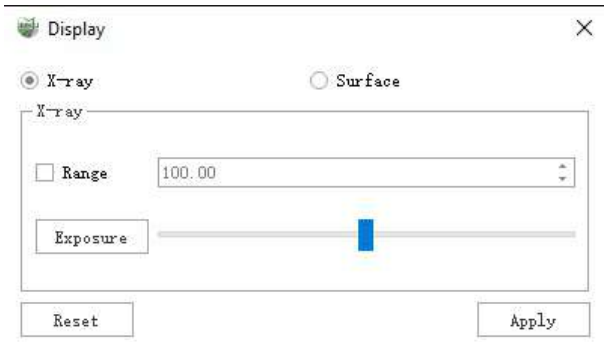
Display By Xray

Function Description：It can be used for point cloud data display, and can be used in conjunction with other display methods to render data with a transparent effect.

Steps

1.Click the button

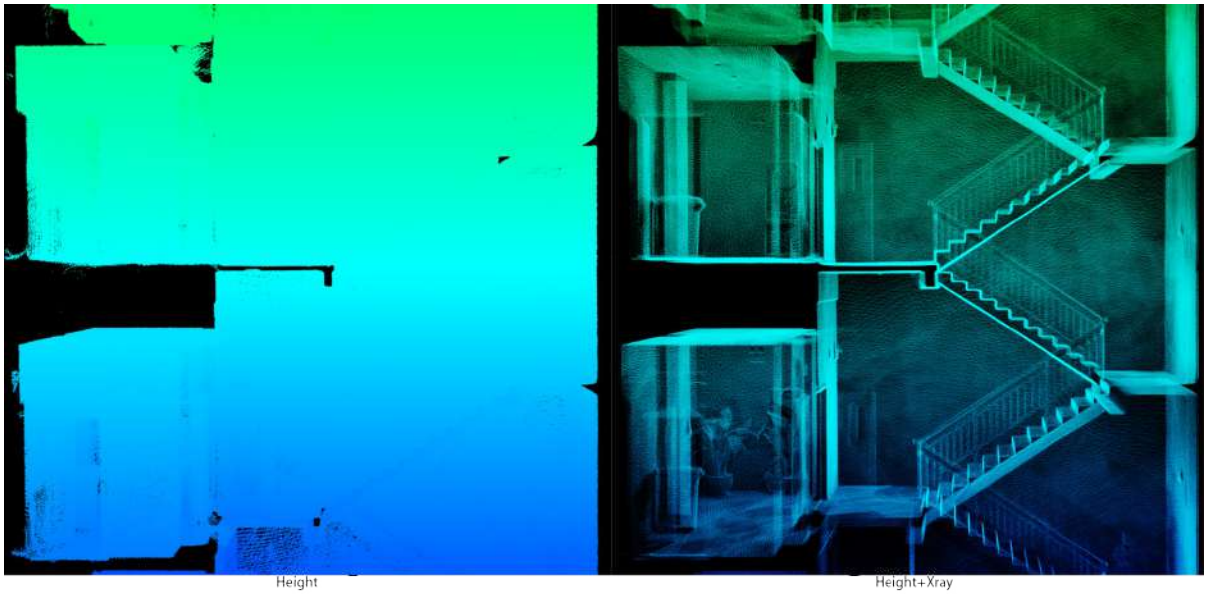
on the toolbar to bring up the parameter setting interface:



Parameter description：

- Render Mode：The software provides two types of enhanced rendering in Xray mode, where Xray can make the point cloud appear transparent, and Surface mode can be used with RGB for a cleaner color display
- Xray-Range：The range does not need to be adjusted by default, but when the data is in bands, the point cloud rendering variable near the display viewpoint can be made smaller by adjusting the range.
- Xray-Exposure：Adjust the brightness of the data rendering in Xray mode.
- Surface-Point Size：When smoothing the rendering, the point size is adjusted to achieve the best possible rendering.
- Surface-Smooth：Check the box to enable smooth rendering.
- Apply: After clicking Apply, the point cloud rendering will be superimposed according to the set effect, if you want to cancel the superimposed effect, you can choose to directly close the parameter window or click Apply and then click the button again.

Staircase scene by height and height overlay Xray effect comparison:



Comparison of the smoothing effect of color assignment data by RGB and RGB overlay:

> Note: This function only works on point cloud data.

Project Management

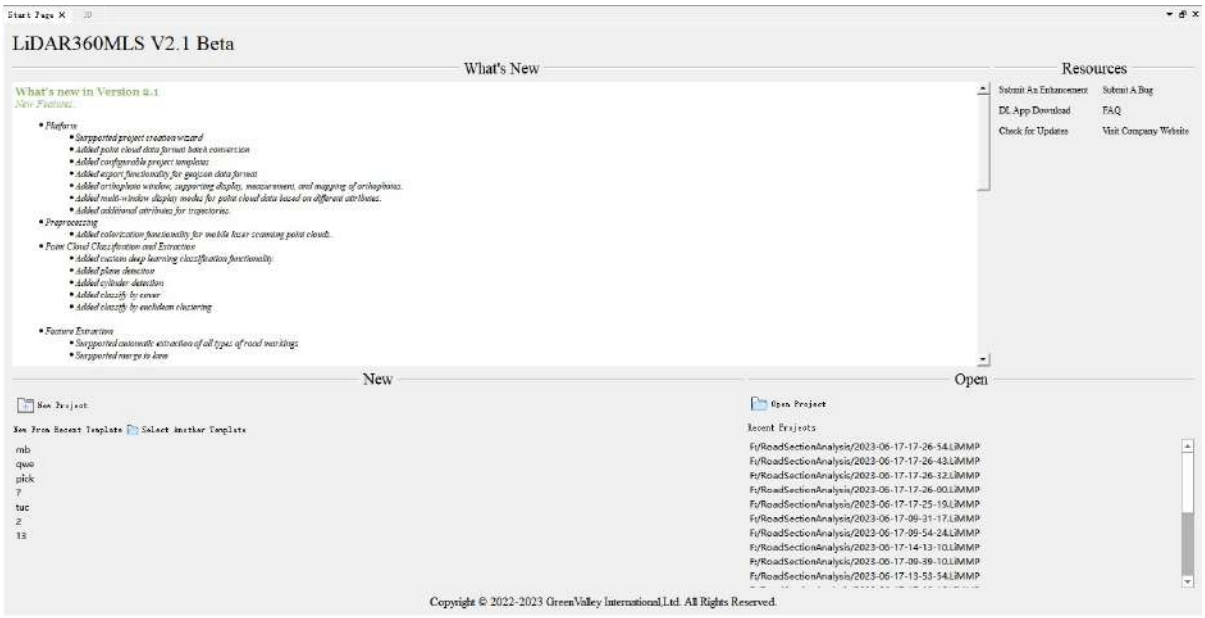
Project management consists of five components: Start Page, Layer Management, Layer Settings, Window Management, Display Mode, and Feature Property.

- Start Page
- Layer Management
- Layer Settings
- Window Management
- Display Mode
- Feature Attribute
- Attribute Table
    - Status Bar
    - Field Calculator
    - Export Images

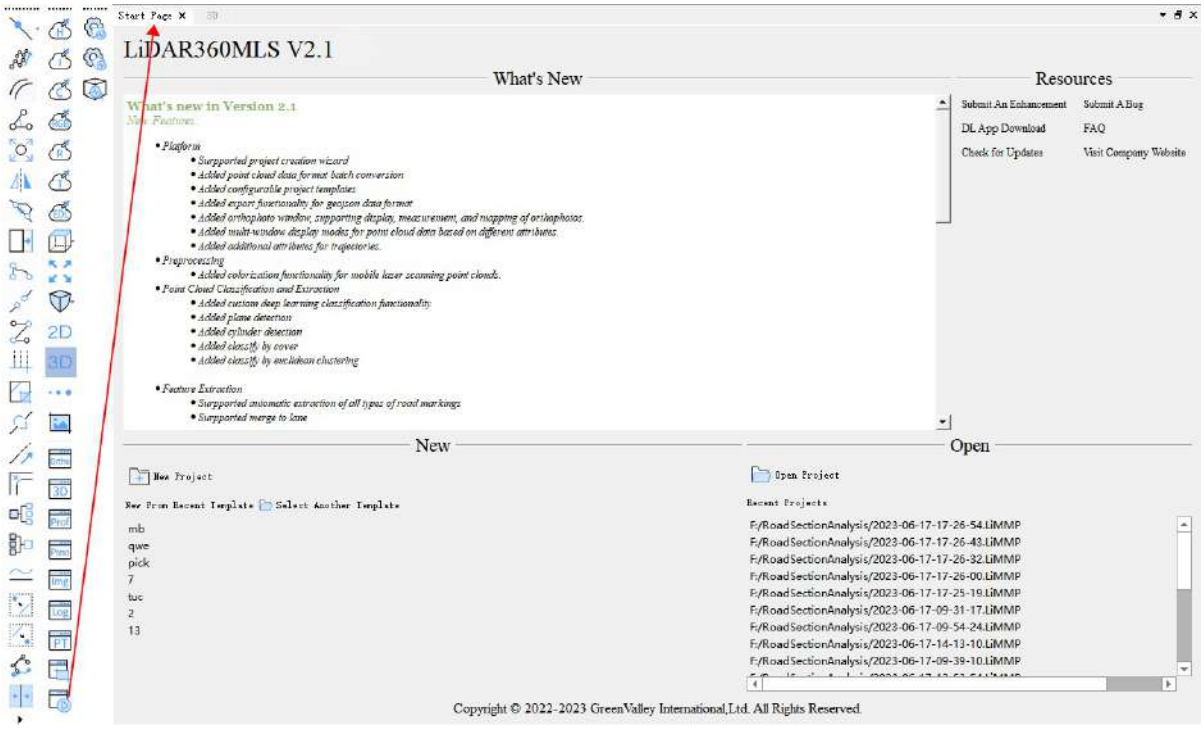
Start Page

Function Description: As the default home page after the software starts, it provides functions such as accessing recent projects, creating new projects, accessing help and learning resources.

1.The entire interface is shown in the following image:



2.The start page can be closed by clicking the close button next to the title. If you need to open it again, you can click the **Show Start Page** button on the toolbar.



3.Recent News: Display software update information, including new modules and features, as well as improvements of existing features.

4.Resources: Provide functions such as submitting requirements, reporting bugs, downloading deep learning applications, frequently asked questions, checking for updates, and accessing the company's website.

5.New: Provide three ways to create a new project:

（1）Click the **New Project** button on the left side to open the dialog for creating a new project, similar to the File-New Project function.

（2）Click on one of the recent templates below **New from Template** to create a project based on the selected template, similar to the File-New from Template function.

（3）Click the **Open Project** button on the right side to open the browse dialog. Select a template file and create a project based on the selected template, similar to the File-Open Project function.

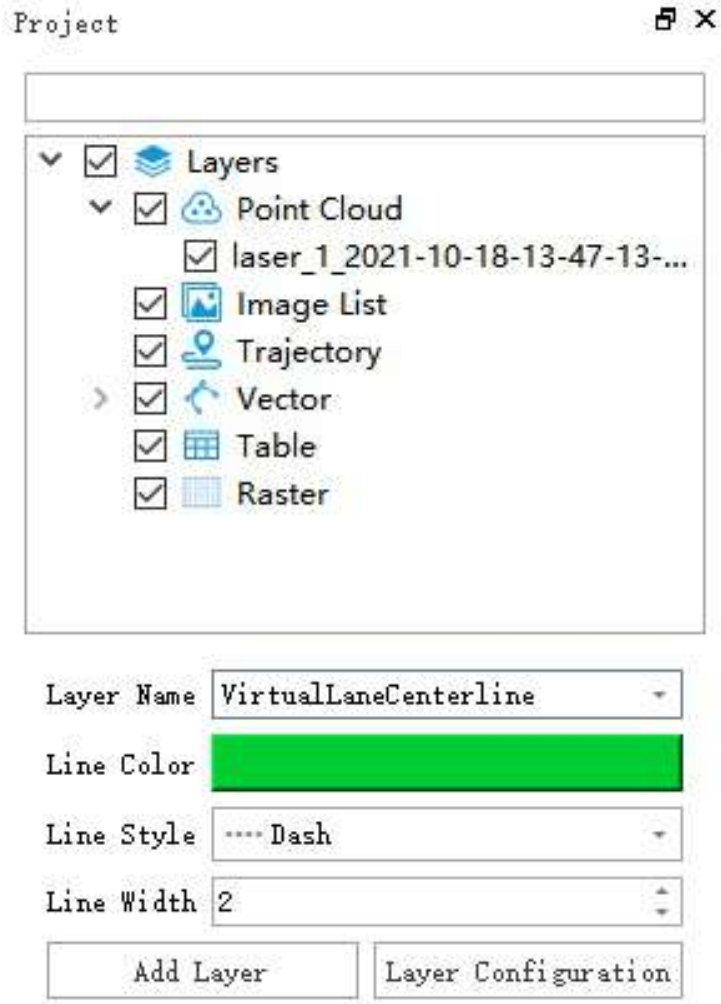
Layer Management

Function Description: Allow for grouping and management of data included in the project.

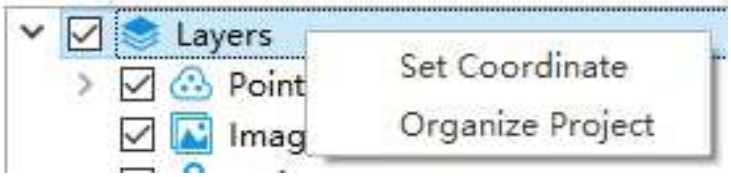
Steps

1.By checking or unchecking the checkboxes of tree nodes, you can control the display or hiding of the current data and all child nodes' data (if any). The right-click menu of a data node provides options for querying, displaying, analyzing, exporting, removing, and etc. Different types of data (such as point clouds, trajectories, vectors, etc.) have different right-click menus.

2.The Layer Management window is initially open. If it is closed, you can click the button in the toolbar to display the layer list again, as shown below:



3.Layer right mouse button:



layer

Function Description:

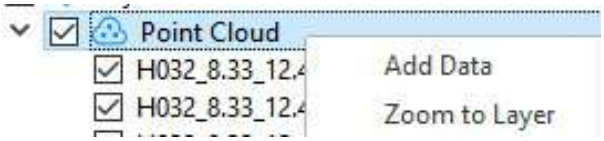
- **Set Coordinate**: Set the coordinate system according to your requirements.
- **Organize Project**: Organize the point cloud, trajectory, image, project, GPKG, and other components included in the current project into one folder.

> Note: Define Projectionin the Reference Point Cloud Tool Module.

4.The layers can be categorized into six parts based on their data sources:

- Point Cloud
- Image List
- Trajectory
- Vector
- Table
- Raster

Point Cloud Layer

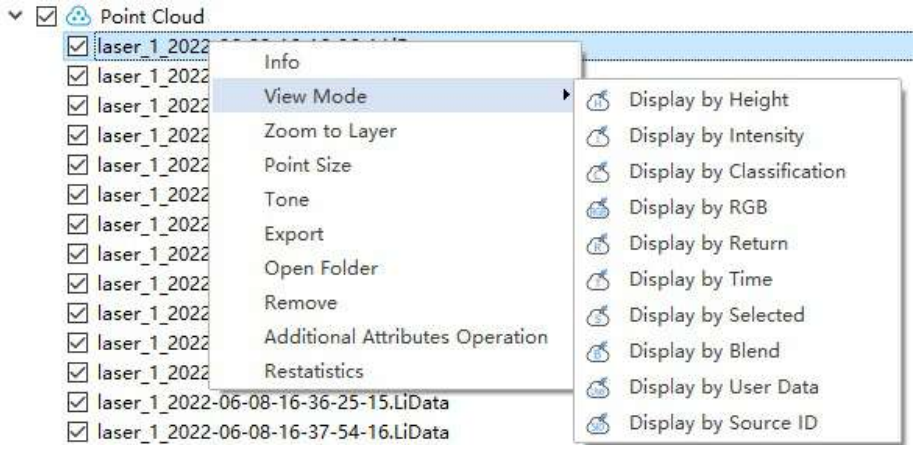


Parameter Settings:

- **Add Data**: Add point cloud data files, multiple selections are supported.
- **Zoom to Layer**: Calculate the bounding box of all current point cloud data and display globally within this bounding box range in all windows that open this data.

Right-Click Menu

Point Cloud Right-click menu includes the following options:

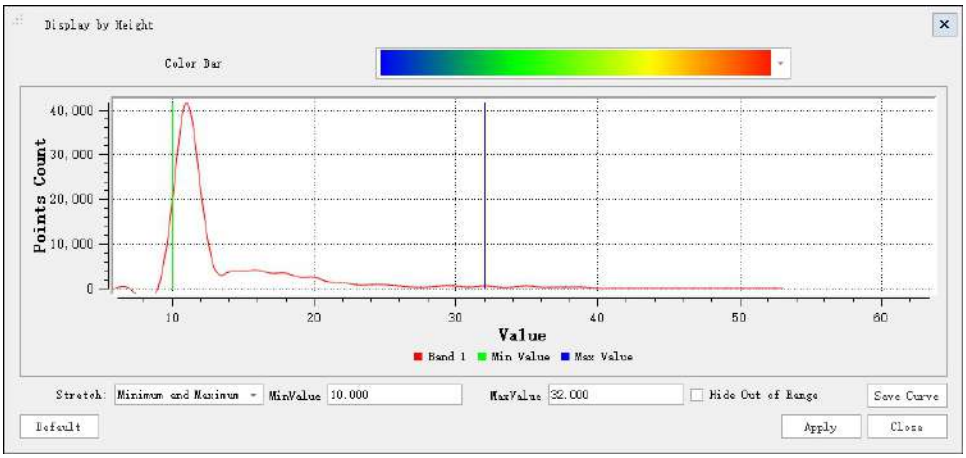


> Note: A is a counterclockwise rotation point cloud, G is a clockwise rotation point cloud.

1.**Info**: Display basic information about the point cloud, including the data path, Coordinate, minimum and maximum values of X, Y, and Z, the mean and std deviation of Z, minimum and maximum GPS time, minimum and maximum values, the mean and std deviation of intensity, Box Dimensions, Total Points Count, Classification Statistics and Return Number Statistics. Clicking the "Export" button allows you to export the basic information of the point cloud as a txt file.

2.**View Mode**：Set the display mode for a single point cloud file, including the following types:

（1）**Display by Height**：Link to the point cloud color mode. Display by Height can be used for displaying point cloud data, which will map the Height of point cloud data to several uniformly changing color ranges and provide a more intuitive representation of the changes in height values.



Parameter Settings:

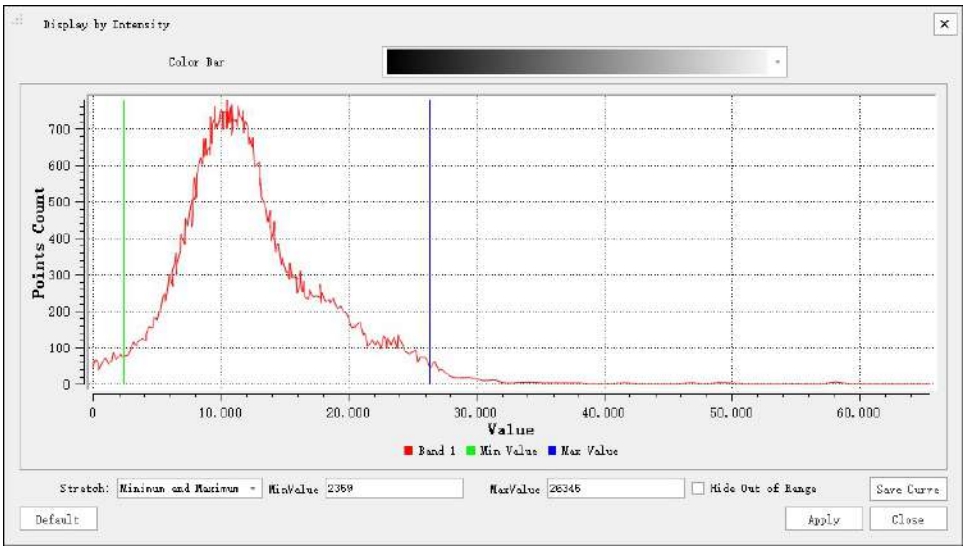
- **Color Bar**：A Color Bar used to map point cloud the attribute of height.
- **Display Histogram**：Display the histogram as needed.
- **Stretch**：Set the stretch mode of the histogram: Min, Max and Std deviation.
  - **Minimum and Maximum Values (Default)**：Linear stretch based on minimum and maximum pixel values, with minimum and maximum pixel values as endpoints of the histogram. For example, if the minimum and maximum values of the image are 2488 and 2656 respectively, linear stretch distributes pixel values between 0-255, increasing image brightness and contrast by distributing pixel values across the entire histogram range, making elements in the image easy to distinguish.
  - **Standard Deviation**：The Std deviation stretch type applies for linear extension between values defined by standard deviation n. For example, if the minimum and maximum values of the image are 2488 and

2656 respectively and n is 2, values exceeding ±2 times the standard deviation will become 0 or 255, and other values are stretched between 0-255.

- **Hide out-of-range data**: Point cloud data with corresponding values less than the minimum value and greater than the maximum value is not displayed.

For the specific display effect, refer to Display by Height.

(2) **Display by Intensity**: Linked with point cloud color mode function. Display by Intensity can be used for the display of point cloud data, mapping the Intensity of point cloud data to evenly changing color intervals, more intuitively showing the change of point cloud data intensity values.



Parameter Settings:

- **Color Bar**: A Color Bar used to map point cloud the attribute of height.
- **Display Histogram**: Display the histogram as needed.
- **Stretch**: Set the stretch mode of the histogram: Min, Max and Std deviation.
  - **Minimum and Maximum Values (Default)**: Linear stretch based on minimum and maximum pixel values, with minimum and maximum pixel values as endpoints of the histogram. For example, if the minimum and maximum values of the image are 2488 and 2656 respectively, linear stretch distributes pixel values between 0-255, increasing image brightness and contrast by distributing pixel values across the entire histogram range, making elements in the image easy to distinguish.
  - **Standard Deviation**: The Std deviation stretch type applies for linear extension between values defined by standard deviation n. For example, if the minimum and maximum values of the image are 2488 and 2656 respectively and n is 2, values exceeding ±2 times the standard deviation will become 0 or 255, and other values are stretched between 0-255.
- **Hide out-of-range data**: Point cloud data with corresponding values less than the minimum value and greater than the maximum value is not displayed.

For the specific display effect, refer toDisplay by Intensity.

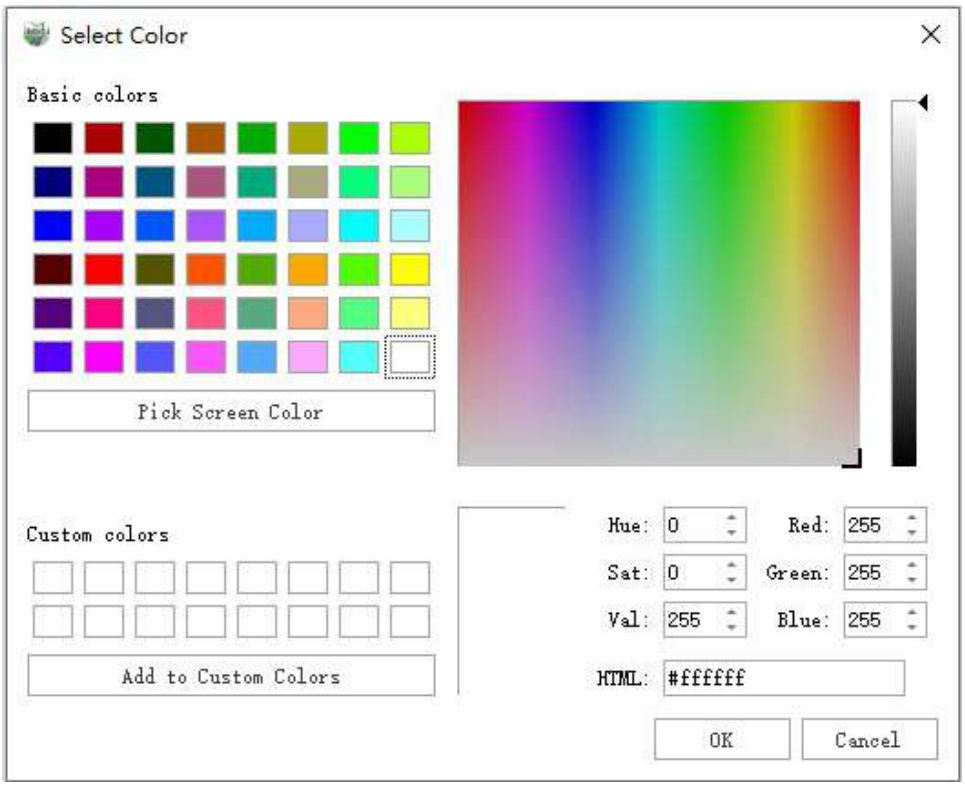
(3) **Display by Classification**: Please refer to Display by Classification for specific display effects.

(4) **Display by RGB**: Please refer to Display by RGB for specific display effects.

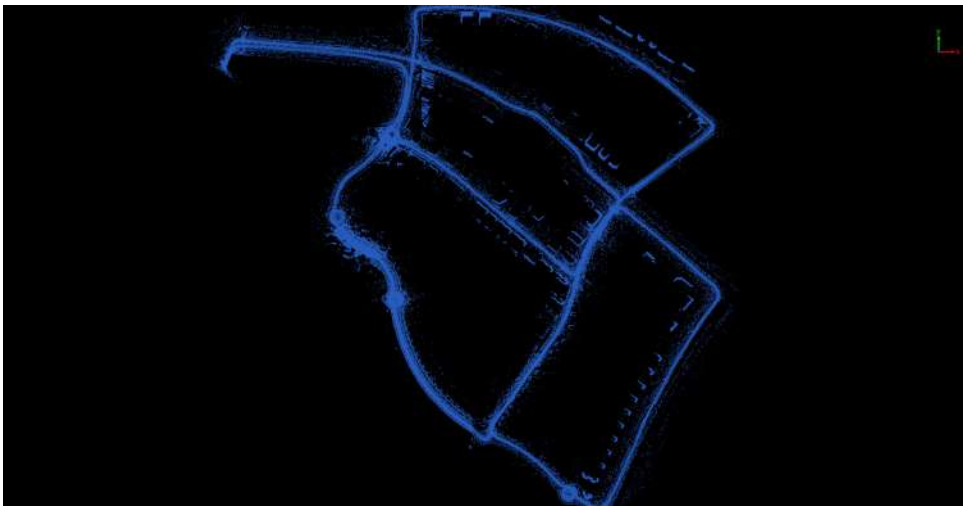
(5) **Display by Return**: Please refer to Display by Return Number for specific display effects.

(6) **Display by GPS Time**: Please refer to Display by GPS Time for specific display effects.

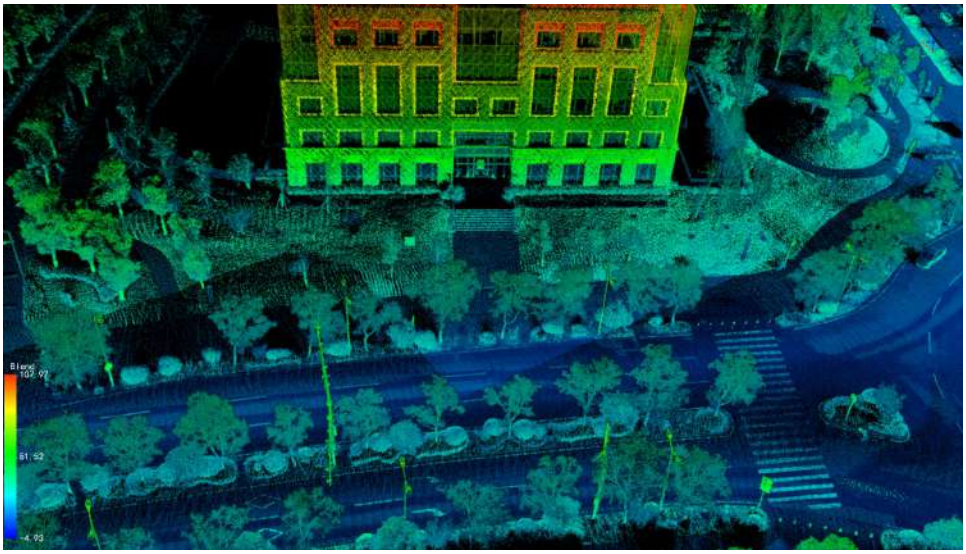
(7)**Display by Selected Color**: It allows you to choose a color in the color dialog and the selected point cloud will be displayed in a unified color. The color dialog is shown below:



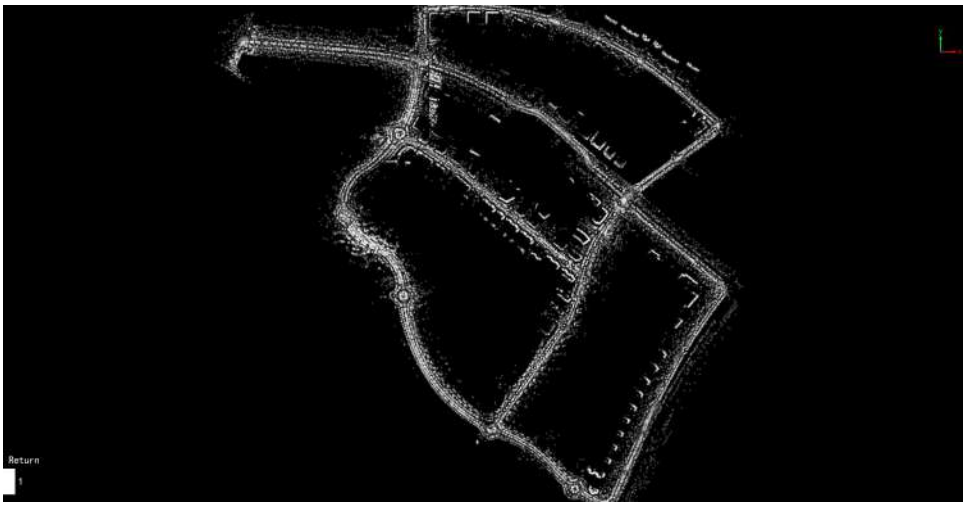
The specific display effect is shown below:



(8)**Display by Blend**: Used for displaying point cloud data by blending the elevation attribute and intensity attribute of the point cloud. The attributes are mapped to a uniformly changing color range, providing a more intuitive representation of the combined changes in elevation and intensity of the point cloud, as well as a clearer display of object classes and boundaries. The specific display effect is shown below:



（9）**Display by User Data**：Used for displaying point cloud data by mapping the user data attribute of the point cloud to different color values, providing a more intuitive differentiation of point cloud data with different user data. The specific display effect is shown below:



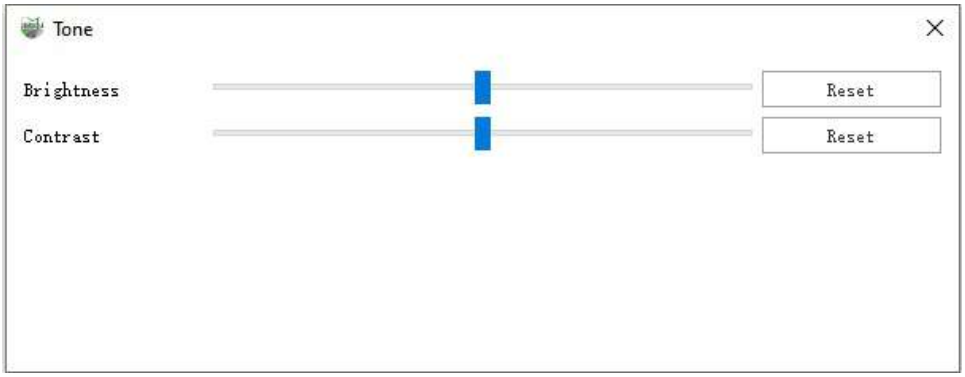
（10）**Display by Source ID**：If the point cloud data contains information about the source of the data, it can be displayed and rendered according to this attribute.

3.**Zoom to Layer**：Calculate the bounding box of the current point cloud data and display all open windows with this bounding box.

> The root node of the point cloud also includes the "Zoom to Layer" menu, which has similar functionalities and is not described here.

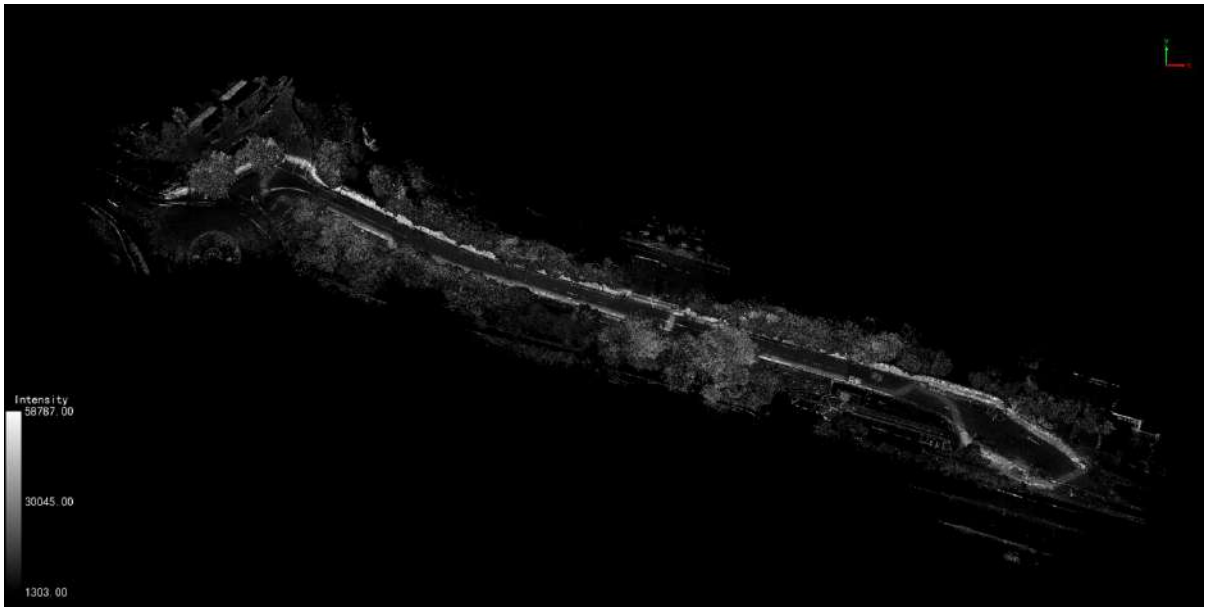
4.**Point Size**：Set the size of the point symbol for point cloud display.

5.**Tone**:Adjusting Point Cloud Display with Brightness and Contrast.:

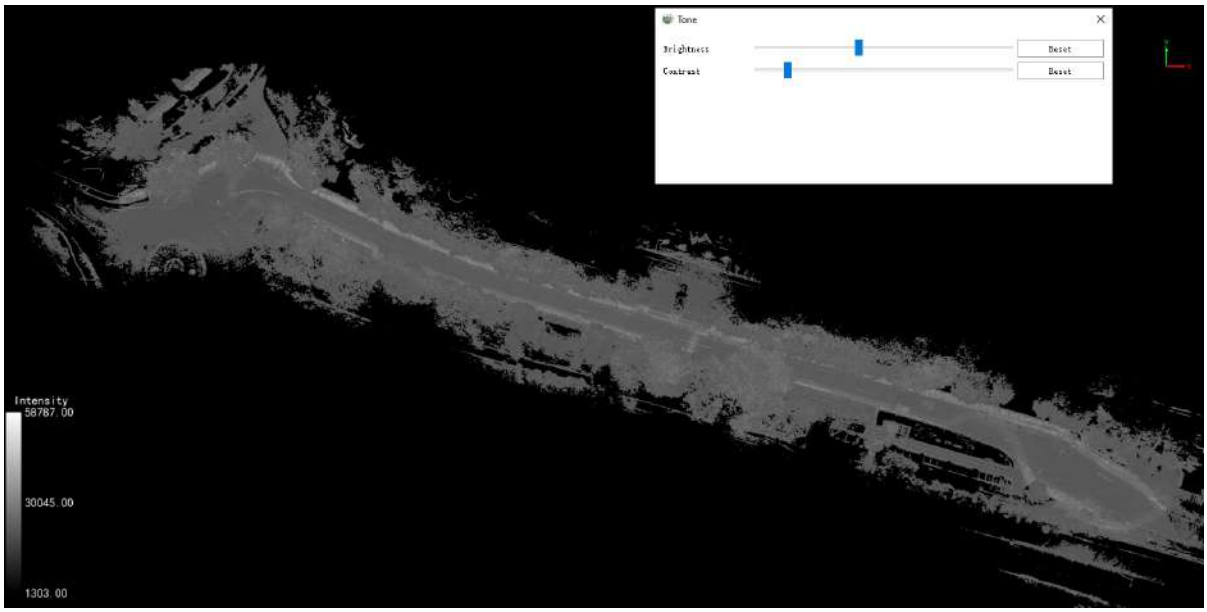


Parameter Description:

- **Brightness**:The brightness represents the intensity of color, with higher brightness corresponding to brighter colors. As the values of the RGB components decrease, the brightness decreases, and as the values increase, the brightness increases. Therefore, to control brightness, it is only necessary to adjust the sizes of the RGB values.
- **Contrast**:Contrast measures the different brightness levels between the brightest white and the darkest black in the image. A higher contrast results in greater color differences, while lower contrast leads to a more subdued and grayish appearance in the image.
- **Reset**:After clicking, return to the default value, which is the middle position.



Before adjustment



After adjustment

6.**Export**：Export the individual point cloud data as LAS (*.las,* .laz) format.

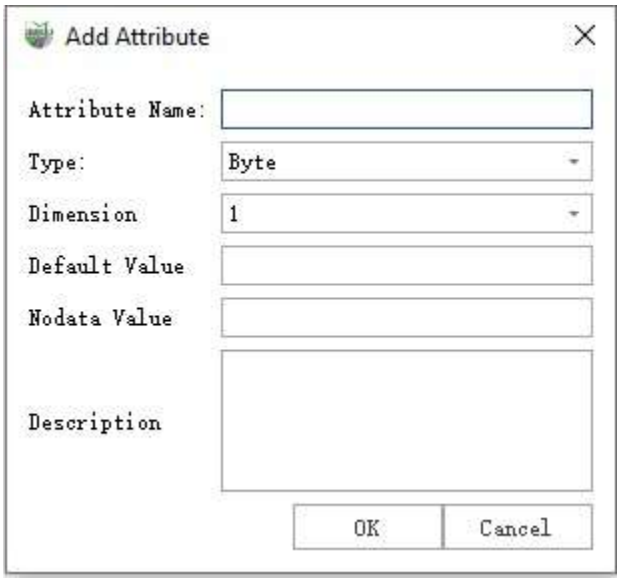
7.**Open Folder**：Open the folder where the current point cloud data is located.

8.**Remove**: Remove the current point cloud data.

9.**Additional Attributes Operation**：To operate on the additional attributes of a single point cloud data, you can clear a specific attribute data or delete a particular attribute, as shown in the figure.

(1)To add an additional attribute: Click on 'Add Additional Attribute,' and a dialog will pop up. Set the additional attribute's name, type, dimensions, default value, NoData value, and description. After clicking 'OK,' the additional attribute will be successfully added.

> The additional attribute names must be unique. If the newly added additional attribute already exists, you will be prompted that the additional attribute already exists. Please define another additional attribute.



(2)Select the additional attribute you want to operate on from the attribute list box.

(3)(Optional) Click on "Clear Data" to set the selected additional attribute component data to their corresponding invalid values. If the current point cloud color display mode is related to this additional attribute, switch the point cloud display to show by height.

(4)(Optional) Click on "Delete" to remove the selected additional attribute. If the current point cloud color display mode is related to this additional attribute, switch the point cloud display to show by height.

(5)Click the close button to exit the operation interface.

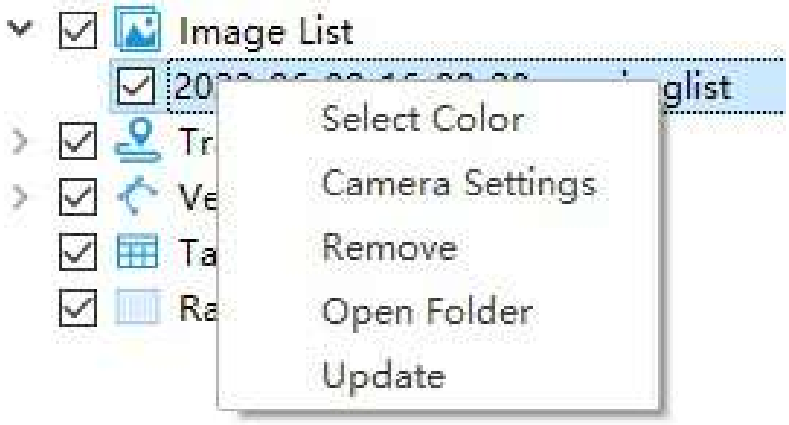
10.**Restatistics**：Recalculate the Mean Z, Std Z, Mean Intensity and Std Intensity of point cloud data. This function is used to repair older versions of LiData which may contain incomplete information.

Image List Layer

Function Description：ettings for Image Category Layers.

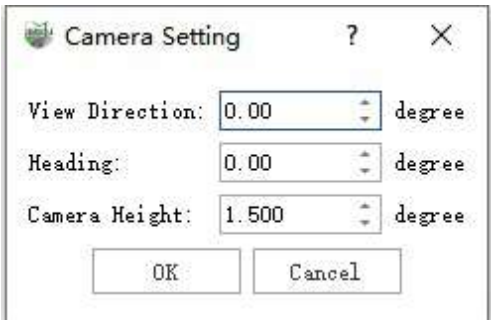
Steps

1.Right-click the mouse to open the menu：



Parameter Description：

- **Select Color**：Set the display color for the frame.
- **Select Color**：Set the camera parameters, including the view direction, heading angle, and camera height.

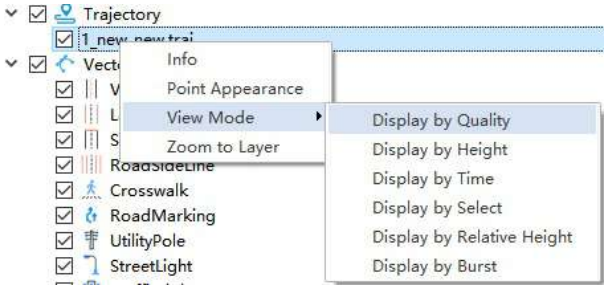


- **Remove**：Remove the current image data.
- **Open Folder**：Open the folder where the current image data is located.
- **Update**：Update ImageList based on the trajectory.

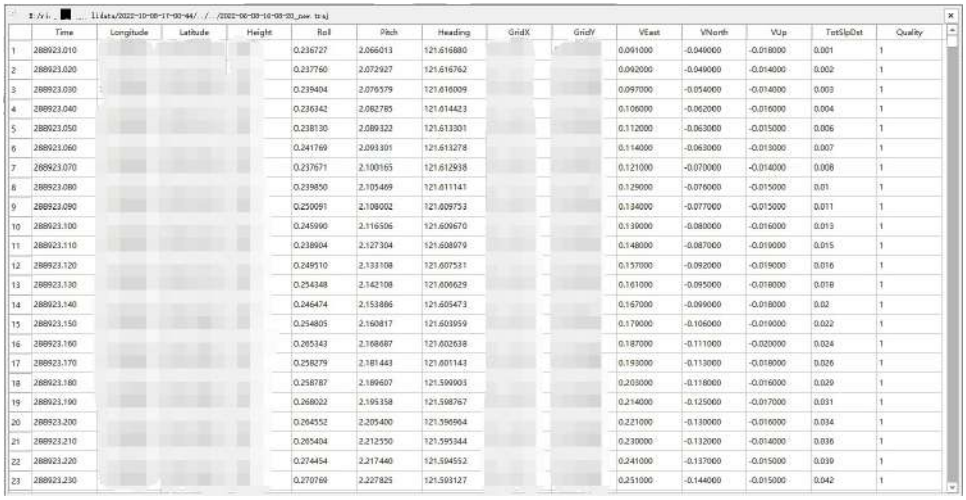
Trajectory Layer

Function Description: Manage the loaded trajectory data.

Trajectory Right-Click Menu



1.**Info**：View basic information about the point cloud, including the path where the data is located, coordinate information, minimum, maximum, average, and standard deviation values of X, Y, and Z, the minimum and maximum GPS times, the minimum, maximum, average, and standard deviation values of intensity, point cloud bounding box, total points, point cloud category statistics, and echo times statistics. Clicking the **Export** button can export the basic information of the point cloud as a txt file.



2.**Point Appearance**：Set the display mode size of a single trajectory:

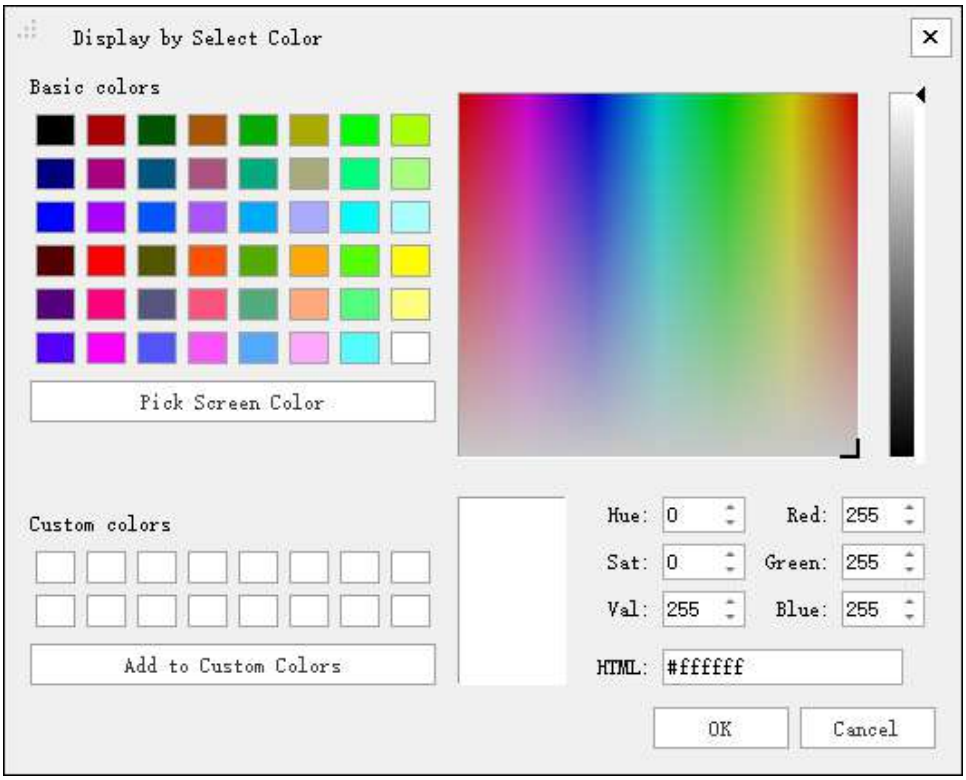
3.**View Mode**：Set the display mode of the trajectory, including the following types:

- **Display by Quality**：A dialog box pops up as shown in the figure. The trajectory can be displayed by randomly choosing colors or selecting a color bar.

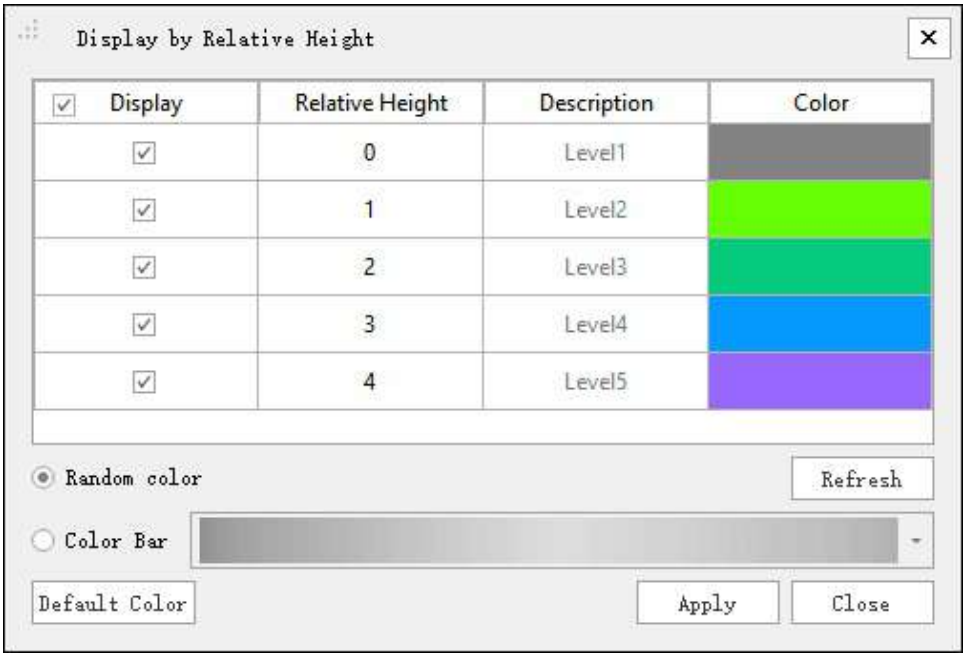
- **Display by Height**: A dialog box pops up as shown in the figure. The trajectory can be displayed by selecting different color bars.

- **Display by Time**: The user data attributes of the trajectory data are mapped to different color values, more intuitively distinguishing trajectory data of different user data.

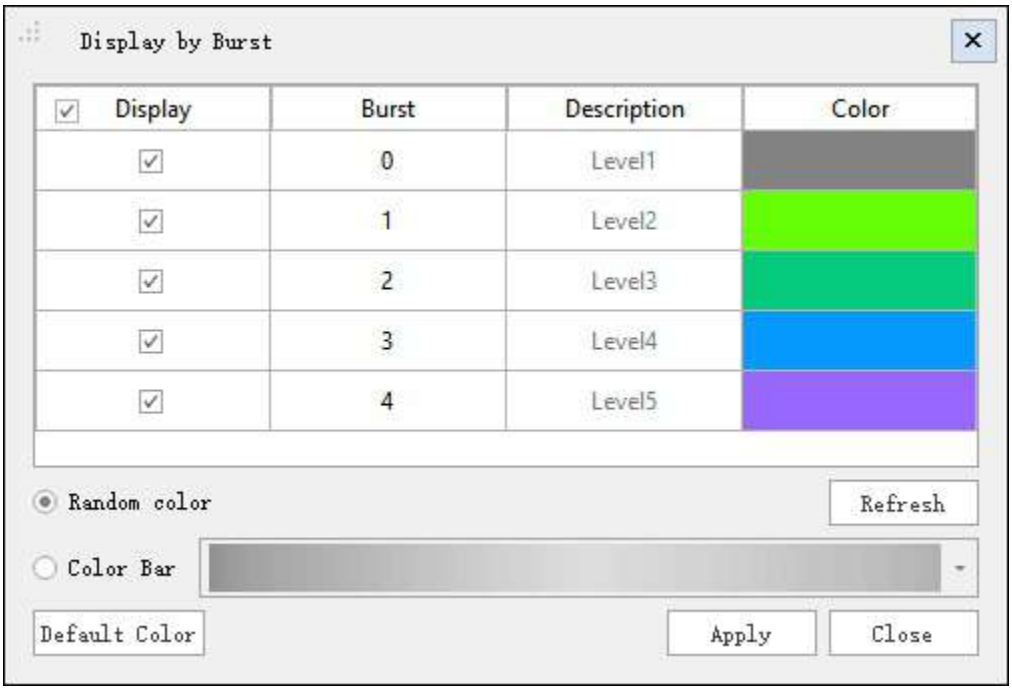
- **Display by Selected Color**: Each trajectory data is displayed in a specified color.



- **Display by Relative Height**: It can be used for the display of the trajectory, mapping the height difference of the trajectory at the same location on the road to the height difference level. Different height difference levels are shown in different colors.



- **Display by Burst**: It can be used for the display of the trajectory, displaying the values of the trajectory burst level field in different colors.



> For specific calculation of trajectory burst level, see Trajectory Quality Detection.

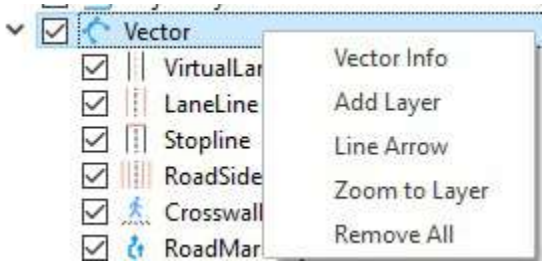
4.**Zoom to Layer**：Center the display based on the current trajectory range.

> The right-click menu of the root node also contains the "Zoom to Layer" menu, which functions similarly and will not be described again here.

Vector Layer

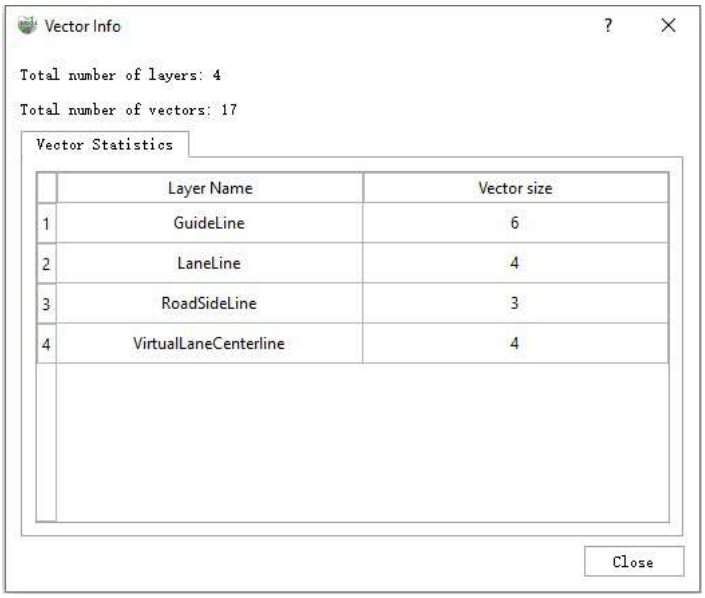
Function Description: Include four types: point, line, polygon and annotation. Right-clicking on the root node of the vector layer allows you to add layers, zoom to layers and remove all layers.

Root Node Right-Click Menu



Root node right-click menu for vector layer

- **Vector Info**:



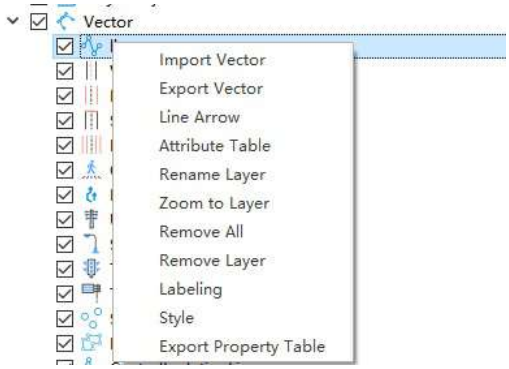
Vector Info

  Description:

  - **Total number of layers**: Count all the layers that contain vector objects.
  - **Total number of vectors**: Count the total number of vector objects.
  - **Yector Statistics**: Display the layers with vector objects that have been counted, along with the corresponding number of vector objects for each layer.
- **Add Layer**: Add a custom vector layer.
- **Line Arrow** : Display arrows on lines, with the direction consistent with the drawing order. **This feature is not available for non-line layers.**
- **Remove All**: Delete all vector data.
- **Zoom to Layer**: Zoom to the center of the window to display all vectors.

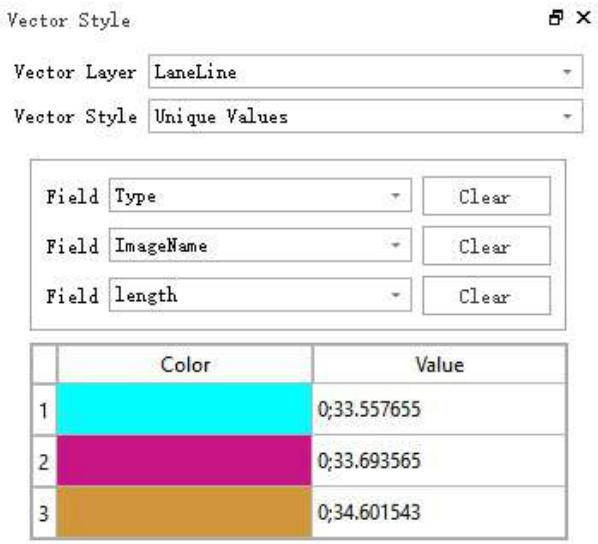
Layer Right-Click Single Selection Menu

The right-click menu for different types of vector layers is generally the same. However, the line layer has an additional **Line Arrow** menu, and the custom vector layer has an additional **Remove Layer** menu. Here is an example of a user-defined **Line Layer** right-clicks menu:

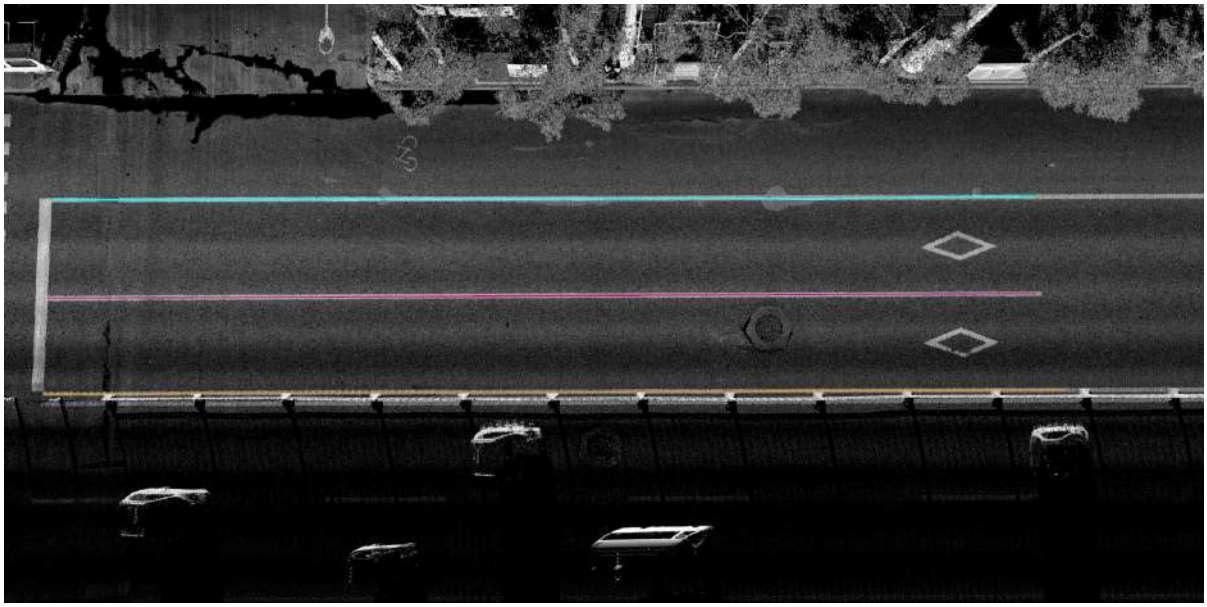


Right-click menu for line layer

- **Import Vector**: Import an external vector into the selected layer. If the geometry types do not match, a prompt will be given. In the dialog box that appears, if you check "Coordinate Transformation," a coordinate transformation will be performed.
- **Export Vector**: You can export the vector layer in a different format, supporting shp, GeoJson, KML and DXF. In the dialog box that appears, select the export format and path. If you need to perform a coordinate transformation, check **Coordinate Transformation** and complete the relevant parameters. The point cloud layer supports exporting in CSV format.
- **Line Arrow**: Display arrows on lines, with the direction consistent with the drawing order. **This feature is not available for non-line layers.**
- **Attribute Table**: Display the attribute table of the current layer. For details, refer to the operation instructions in the Attribute Table section.
- **Rename Layer**: Modify the name of the layer. Alternatively, you can press F11 to edit it directly when the layer is selected.
- **Zoom to Layer**: Zoom to the center of the window to display the vector data of the selected layer.
- **Remove All**: Delete all vector features in the selected layer.
- **Remove Layer**: Delete the selected layer.
- **Labeling**: Activate **Labeling** function to display different attribute values of the layer.
- **Style**: Support styling vector layers based on attribute information.



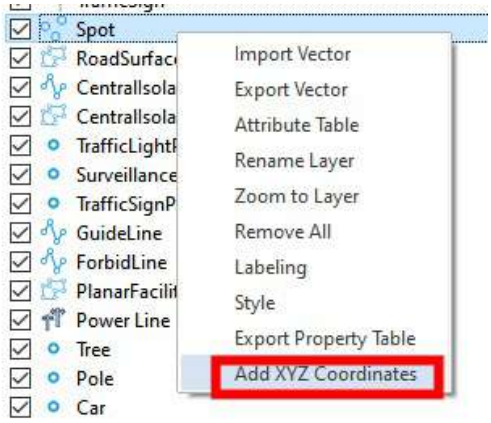
Style Settings



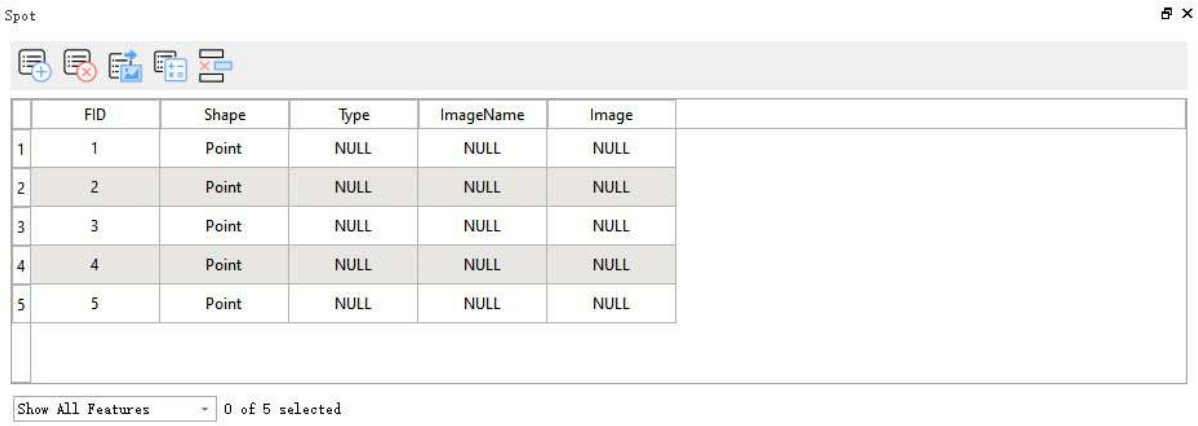
Visual Representation

- **Export Property Table**: Export attribute table information, supporting .DBF and .CSV formats.

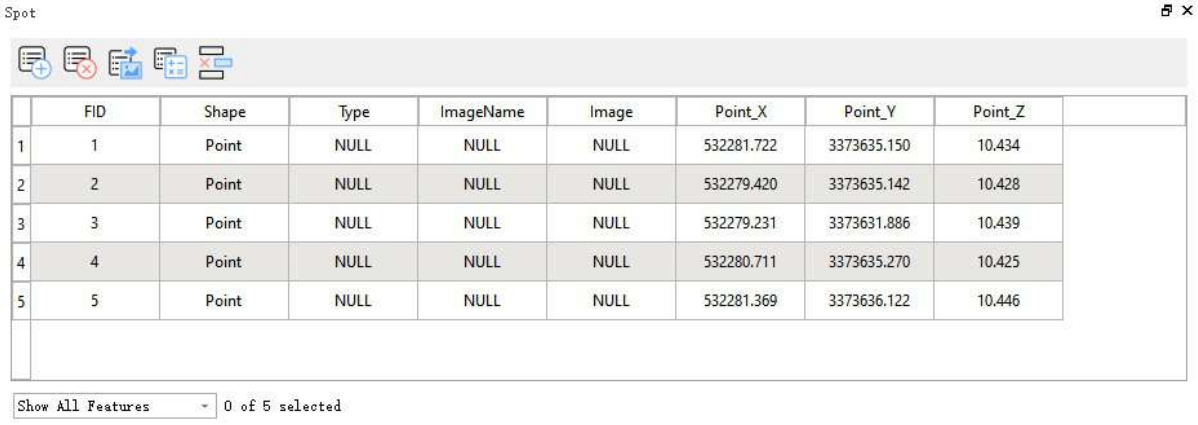
2.Point layer, can add XYZ information to the attribute table.



Right-Click Menu for Point Layer



Before Addition



After Adding

Layer Right-Click Multi-Selection Menu

The menu includes the following options:

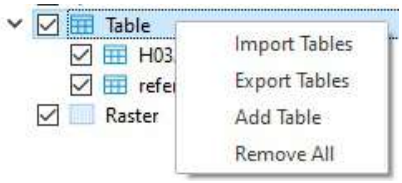
Right-click menu for multiple selections of vector layers

- **Remove All**: Delete all vector features in the selected layer.
- **Show Selected Object**: Show the selected layers. Clicking this option will check the checkboxes next to the layers.
- **Hide Selected Object**: Hide the selected layers. Clicking this option will uncheck the checkboxes next to the layers.

Table Layer

Description: Manage user-defined table layers (without geometry information).

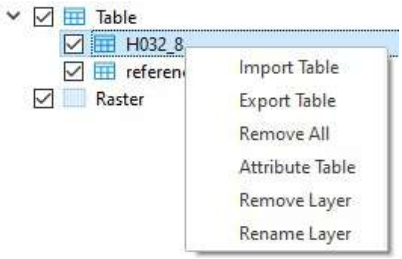
Root Node Right-Click Menu



Root node right-click menu for table layer

- **Import Tables**: Pop up a dialog to select the table file (*.dbf,* .csv) you want to import, and import it into the table.
- **Export Tables**: Pop up a dialog to select the table file you want to export, then choose the export format (*.dbf,* .csv), select the output path, and export the desired table file.
- **Add Table**: Add a table layer.
- **Remove All**: Delete all table layers.

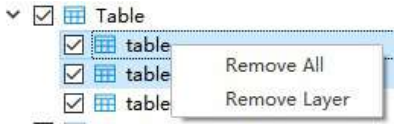
Layer Right-Click Single Selection Menu



Right-click menu for single selection of table layer

- **Import Tables**: Pop up a dialog to select the table file (*.dbf,* .csv) you want to import and import it into the table.
- **Export Tables**: Pop up a dialog to select the table file you want to export, then choose the export format (*.dbf,* .csv), select the output path, and export the desired table file.
- **Remove All**: Select "Remove All" and clear all contents in the table.
- **Attribute Table**: Select "Attribute Table" opens the attribute table interface, where users can add or delete fields, add or delete records, and edit attribute values.
- **Remove Layer**: Select "Remove Layer" to remove the current table from the directory tree.
- **Rename Layer**: Modify the name of the layer. Alternatively, you can press F11 to rename it.

Layer Right-Click Multi-Selection Menu



Right-click menu for multi-selection of table layer

- **Remove All**: Clear all records in the selected layers.
- **Remove Layer**: Remove the selected layers from the directory tree.

Raster Layer

Function Description: Manage user's raster data.

Root Node Right-Click Menu

Root node right-click menu for raster layer

- **Add Layer**: Add a custom vector layer.
- **Zoom to Layer**: Zoom to the center of the window to display all vectors.
- **Remove All**: Delete all vector data.

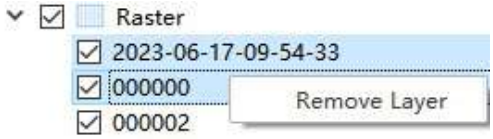
Layer Right-Click Single Selection Menu

Right-click menu for single selection of raster layer

- **Information**: Display basic information about the image, including width, height, number of bands, statistics, etc.
- **Open Folder**: Open the folder where the image is located.
- **Show Histogram**: Delete the histogram of the image.
- **Zoom to Layer**: Zoom to the extent of the current layer.
- **Zoom to Native Resolution**: Display the image at its original resolution 1:1.
- **Remove Layer**: Remove the layer.

Layer Right-Click Multi-Selection Menu

- **Remove Layer**: Delete the selected layers.



Right-click menu for multi-selection of raster layer

Layer Manager

Layer Explanation: Used to manage different types of data. Based on geometric shapes, it can be categorized into three main types: point layers, line layers and polygon layers. For example, VirtualLaneCenterline, LaneLines, and Stopline belong to line layers, while Trafficlight and SurveillanceCameras belong to point layers, and ForbidLine and Crosswalk belong to polygon layers.

> Note: Point vector features can only be saved in point layers, and line vector features and polygon vector features should be saved in line layers and polygon layers, respectively.

Description: The Layer Manager module has the following functions:

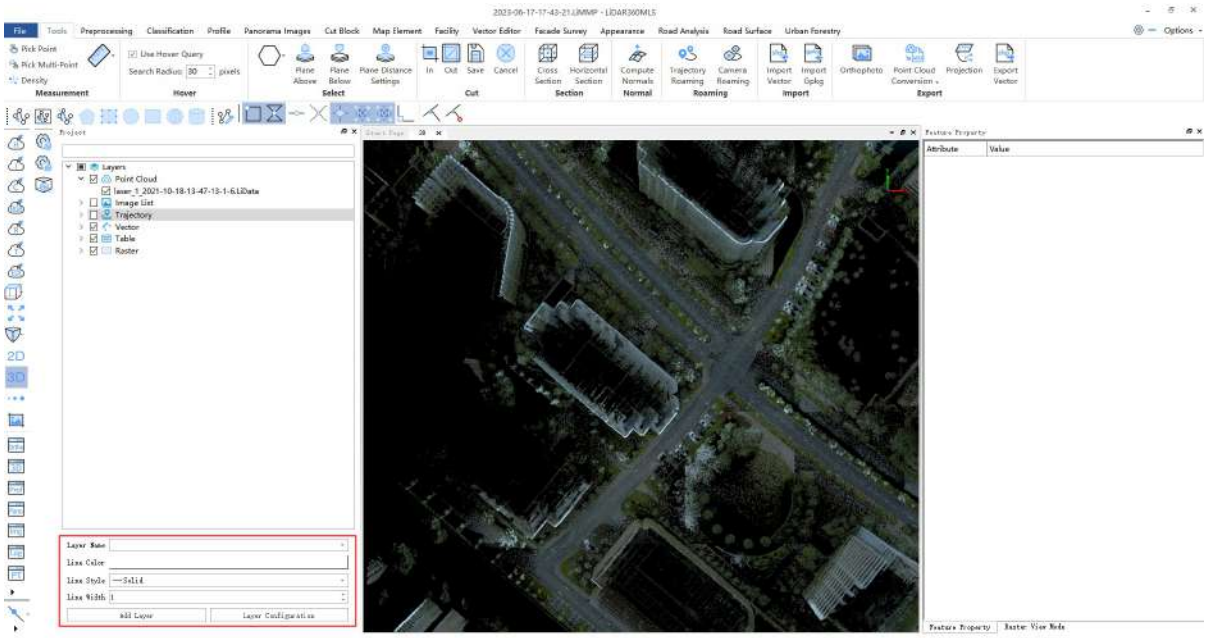
(1) Switching the currently active layer;

(2) Setting the line type, line width, and color of the layer;

(3) Adding custom layers;

(4) Configuring the attribute fields of the layer.

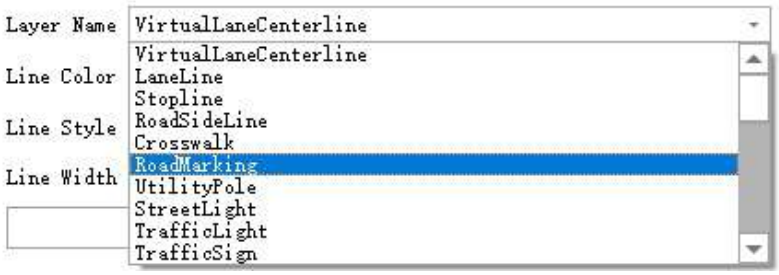
> You can also switch the current layer by left-clicking on the layer node in the directory tree.



Layer Manager Interface

Instructions

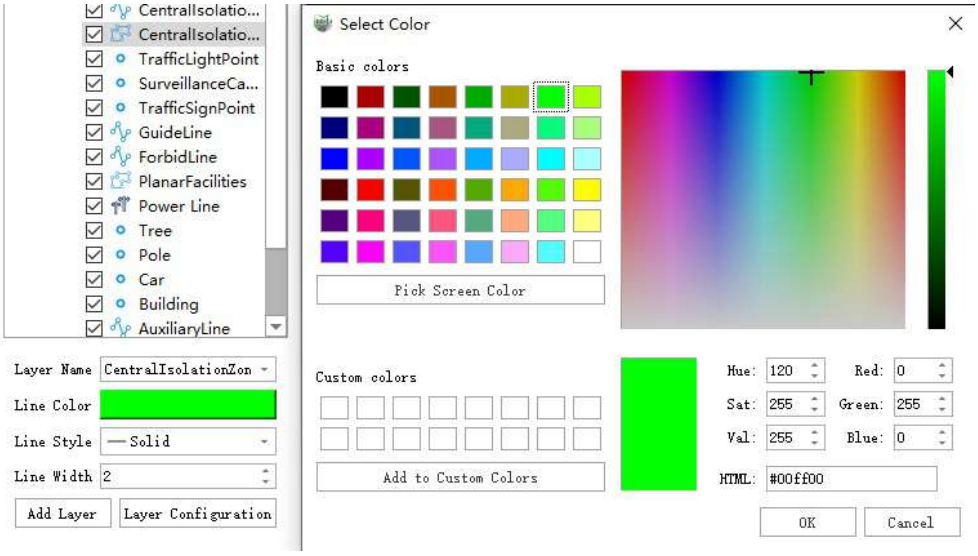
1.**Layer Selection**：Click the drop-down arrow next to the **Layer Name** in the **Layer Manager** to select the layer you want to switch to.



Layer Switching Interface

2.**Layer Attribute Settings**:

- **Vector Color Setting**: Click the **Line Color** in the Layer Manager to open the color palette. You can choose from the Basic colors or Custom colors.



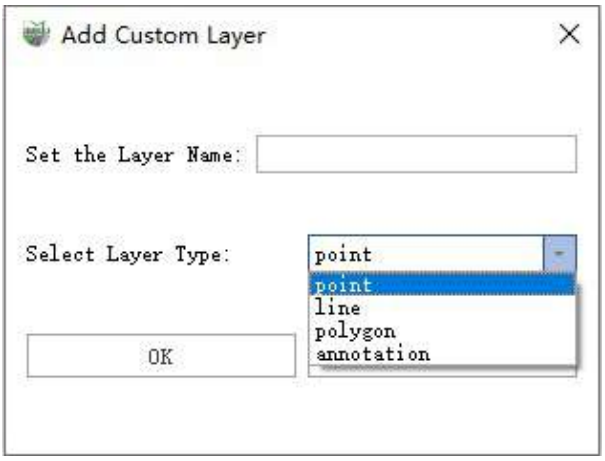
Line Color Setting Interface

- **Vector Line Type Setting**: Click the drop-down arrow next to **Line Style** to switch to the desired line type.

Layer Line Type Setting Interface

- **Vector Line Width Setting**: Click the adjustment button next to **Line Width** to set the line width.

3.**Add Layer**: Click the **Add Layer** button to open the dialog for adding a custom layer.



Add Layer Interface

Parameter Settings:

- **Set the Layer Name**: Enter the name of the layer. It is recommended to use a combination of characters and numbers and avoid special characters.
- **Select Layer Type**: Based on the geometric features of the vector features (point, line, polygon, annotation), select the geometry type for the custom layer.
- **OK**: After clicking, the corresponding layer will be created based on the settings, and the **Custom Layer Node Tree** will be updated in **real-time**.
- **Cancel**: Cancel creating a new layer.

Window Management

Function Overview

The Window Management Tool controls the display of project windows.

Details

Show Ortho Viewer: Used to display point clouds, vectors and raster images in 2D mode. It supports common operations such as drawing, selecting and modifying vectors through the Ortho Viewer. By default, vectors drawn through the Ortho Viewer have an height of 0.

Show 3D View.

Show Profile Viewer.

Show Panorama Viewer.

Show Image Viewer.

Show Log.

Show Project Tree.

Show Screenshot Viewer.

Show Start Page.

Display Mode

Function Description: The Display Mode provides single attribute display and multi-attribute combined display for point cloud data in different views. The best display method can be selected according to different analysis functions.

Single Attribute Display

1. Display by Height: For specific display effects, see Display by Height.

2. Display by Intensity: For specific display effects, see Display by Intensity.

3. Display by Blend: It can be used for point cloud data display. Integrating the height and intensity attributes of the point cloud data, map them to a uniformly changing color interval. It helps display the comprehensive change of point cloud data elevation and intensity more intuitively and display the categories and boundaries of ground objects more clearly.

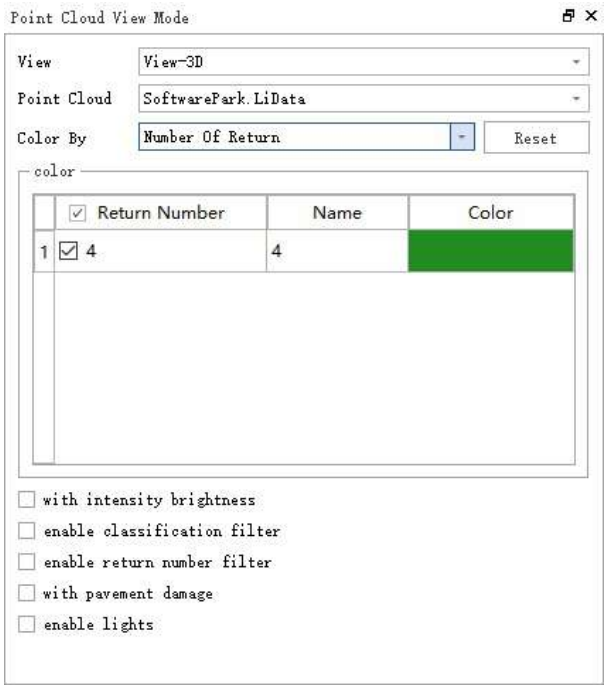
4. Display by Classification: For specific display effects, see Display by Classification.

5. Display by RGB: For specific display effects, see Display by RGB.

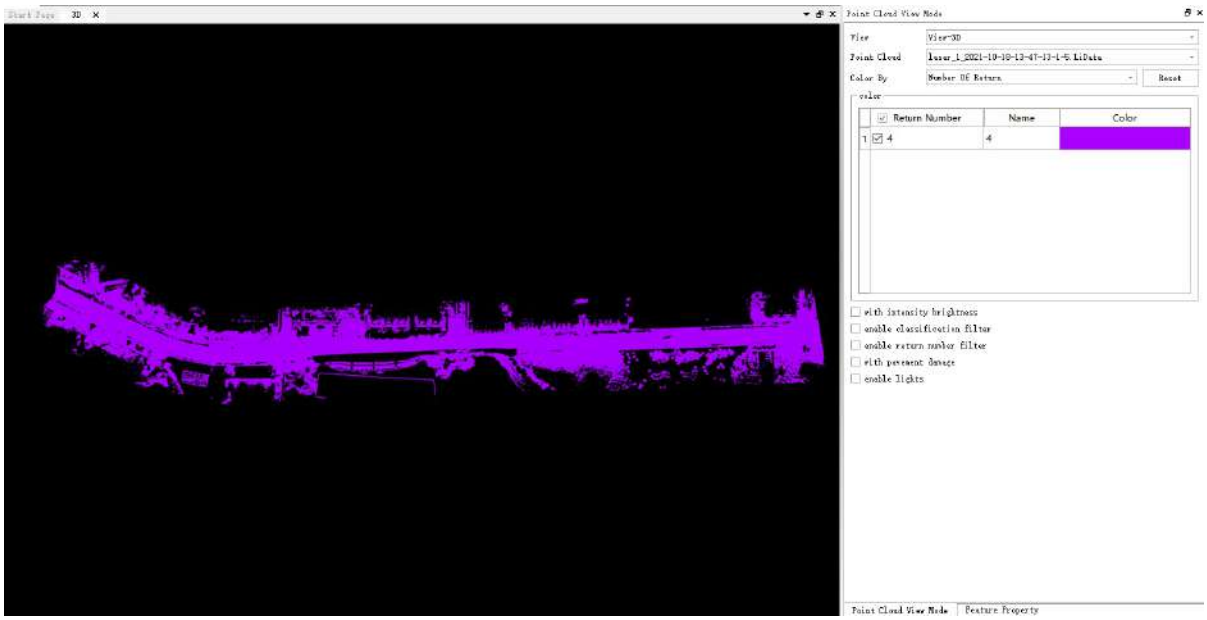
6. Display by Return: For specific display effects, see Display by Return Num.

7. Display by Number of Return: Can be used for point cloud data display, mapping the last return count attribute of the point cloud data to the point cloud data.

7.1 Click the dropdown box of the Return Number to pop up the display dialog box by number of return.



7.2 You can modify the display color and the effect which is better with EDL display, as shown in the figure.



8. Display by GPS Time：For specific display effects, see Display by Time.

9. Display by Tree ID：Can be used for the display of point cloud data, mapping the tree ID attribute of the point cloud data to different color values, more intuitively distinguishing point cloud data of different tree IDs.

> You can choose to display the minimum and maximum ID of the tree ID.

10. Display by Edge of Flight Line：If the point cloud data records Edge of Flight Line information, it can be displayed and rendered according to this attribute.

11. Display by Point Source ID：If the point cloud data records Point Source ID, it can be displayed and rendered according to this attribute.

12. Display by Scan Angle Rank：If the point cloud data records the Scan Angle attribute, it can be displayed and rendered according to this attribute.

13. Display by Scan Direction：If the point cloud data records Scan Direction information, it can be displayed and rendered according to this attribute.

14. Display by Scan Channel：If the point cloud data records Scan Channel information, it can be displayed and rendered according to this attribute.

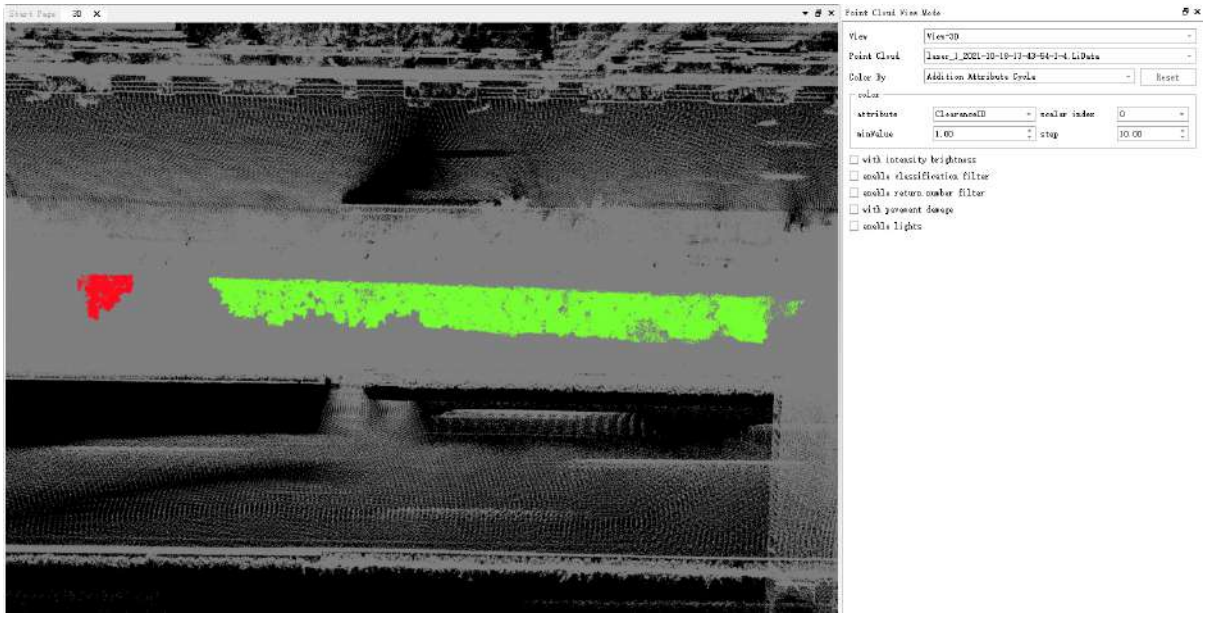
15. Display by Near Infrared：If the point cloud data records Near Infrared information, it can be displayed and rendered according to this attribute.

16. Display by User Data：Can be used for the display of point cloud data, mapping the user data attribute of the point cloud data to different color values, more intuitively distinguishing point cloud data of different user data.

17. Display by Selected：Select a color in the color dialog box, and then display the Selected point cloud data in a uniform color.

18. Display by File: Each point cloud data is colored by one color, showing the multi-point cloud connections.

19. Display by Additional Attribute Cycle：If there are additional attributes in the point cloud data, as shown below, there is a single feature ID (PoleID). You can display the color cyclically according to a certain additional attribute value. The display effect is similar to the TreeID display effect.

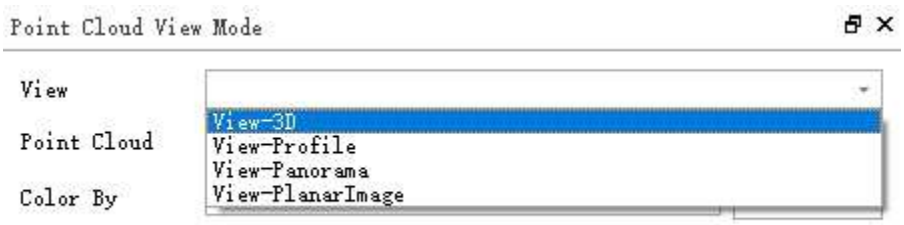


20. Display by Additional Attribute：If there are additional attributes in the point cloud data, you can set a color bar to stretch and render according to a certain additional attribute value.

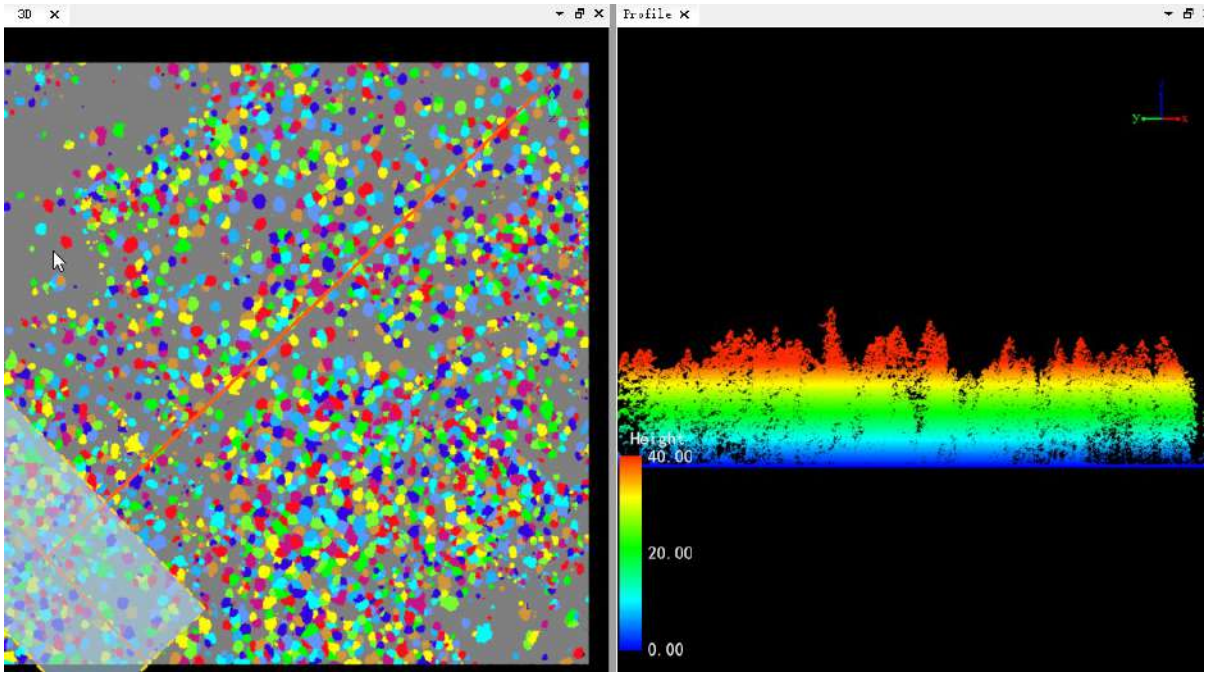
Multi-View Display

1.The same point cloud can be displayed in different windows with different colors.

2.Click **Point Cloud Color Mode---Viewport** to select the color display mode of different windows.



3.Display effect:



3D window is displayed by Tree ID, and the cross-section is displayed by height.

Saving and Resetting Display Parameters

After setting the color of the point cloud, the corresponding display parameters will be recorded. When switching back from other color modes, the previously set parameters will be used for displaying. If you want to restore to the default state, click the **Reset** button.

Multi-Attribute Combination Display

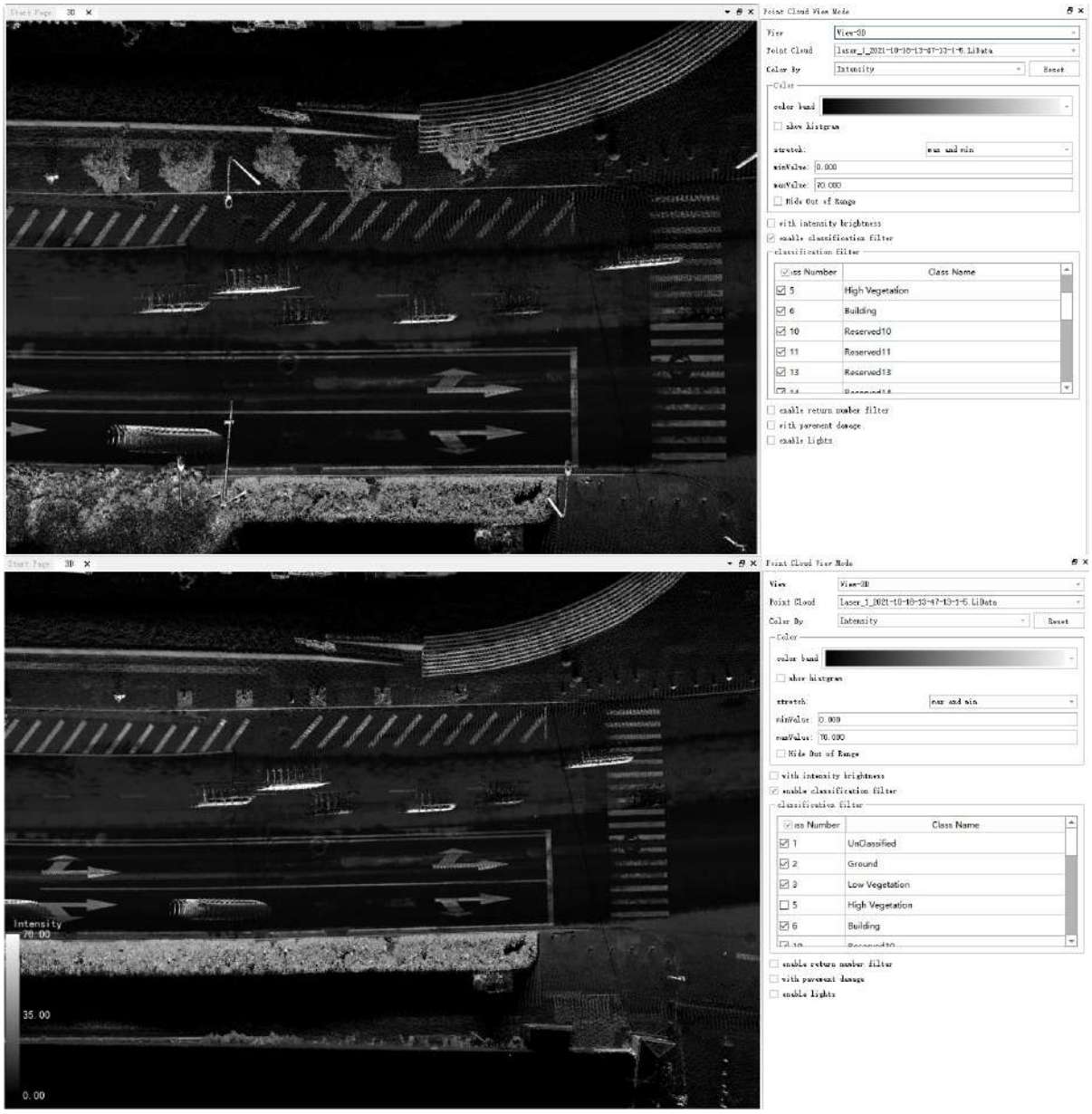
You can set the category to be displayed in the color display in the Display Mode window. In the checkbox at the bottom of the window, check the category to be needed.

Instructions

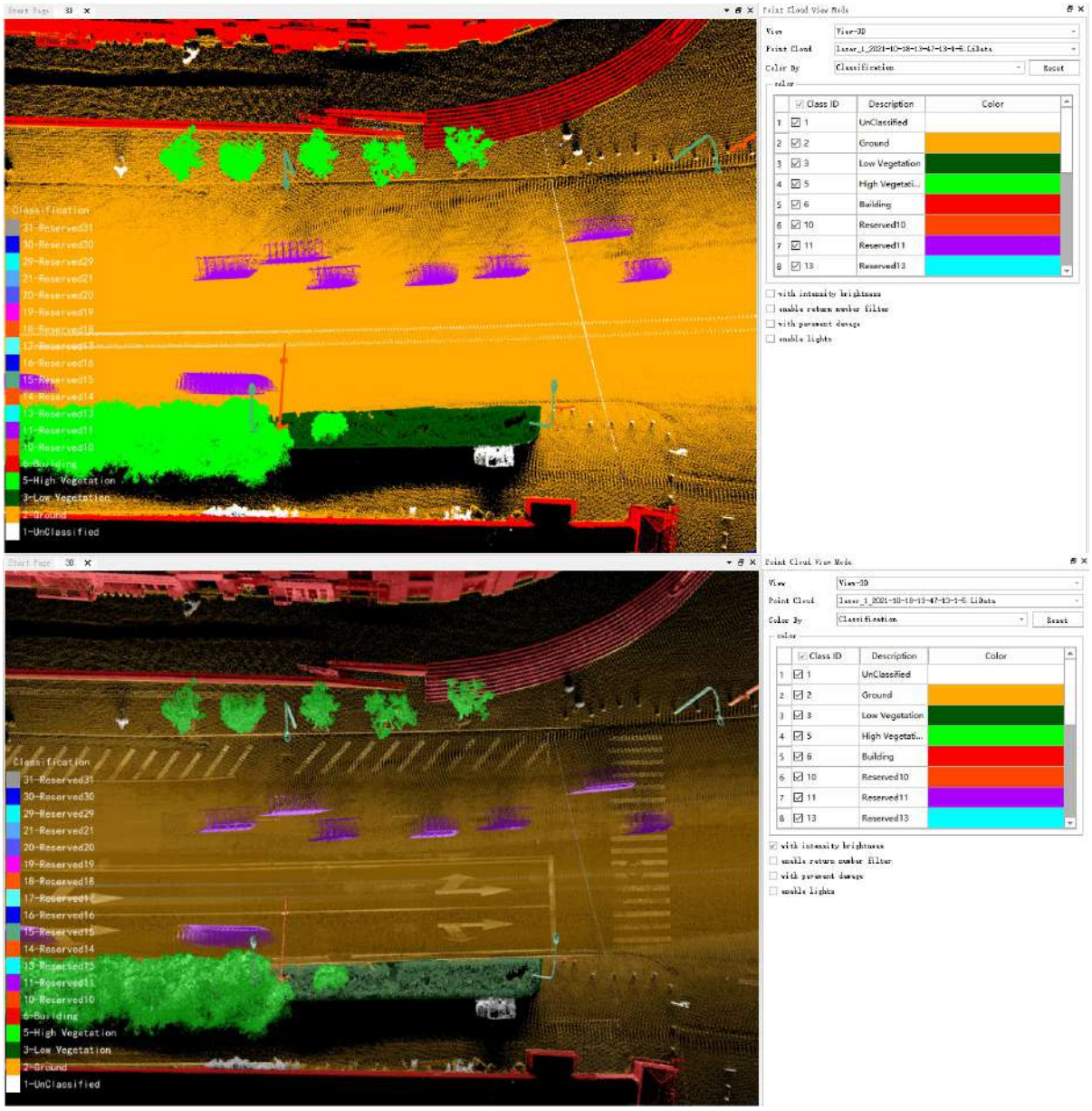
- **Blend Intensity**: If checked, the point cloud will blend the results of intensity stretching based on the main mode.
- **Enable Classification Filtering**: If checked, it filters the categories of point cloud data.
- **Enable Echo Filtering**: If checked, it can be used for the display of point cloud data, mapping the echo count of the point cloud data to different color values, more intuitively distinguishing point cloud data of different echo counts.
- **Enable Road Damage Filtering**: If checked, it can be used for the display of point cloud data, mapping the road damage attribute of the point cloud data to different color values, more intuitively distinguishing the situation of road damage.
- **Enable Lighting**: The data needs to have an additional attribute named Normal with 3 components. The color overlays the lighting effect of parallel light.

Common combination display modes include:

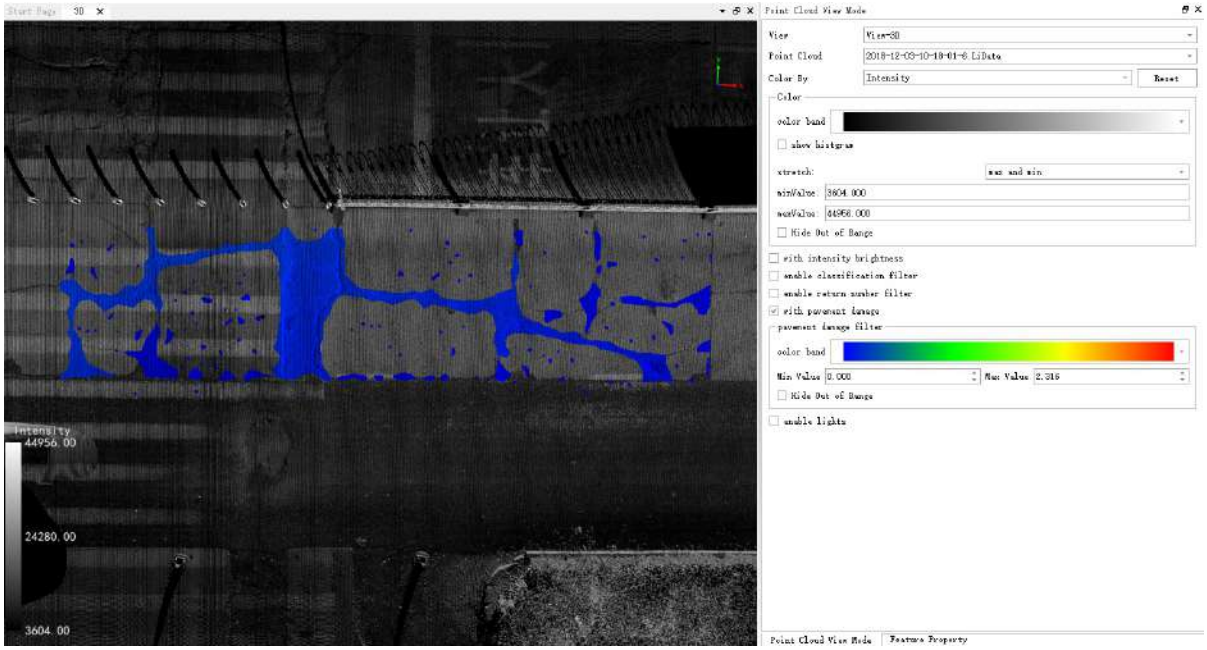
（1）Intensity Overlaid with Class Filtering:



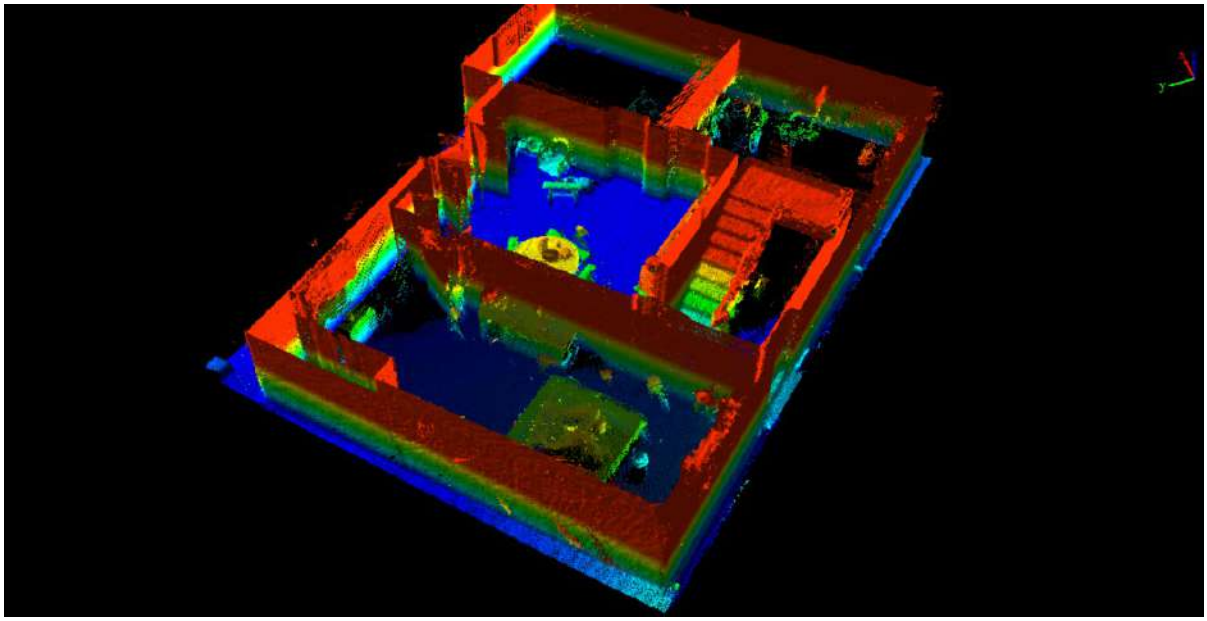
(2) Classification Overlaid with Intensity Filtering:



(3) Intensity Overlaid with Road Damage Filtering:



(4) Enable Lighting:



Feature Attributes

Function Description: It is used to display and edit the currently selected vector features. When the layer is a point layer, symbols can be set.

Description

1.**Feature Attributes**: Select a vector to display related vector information.

Feature Attributes

Basic Attributes

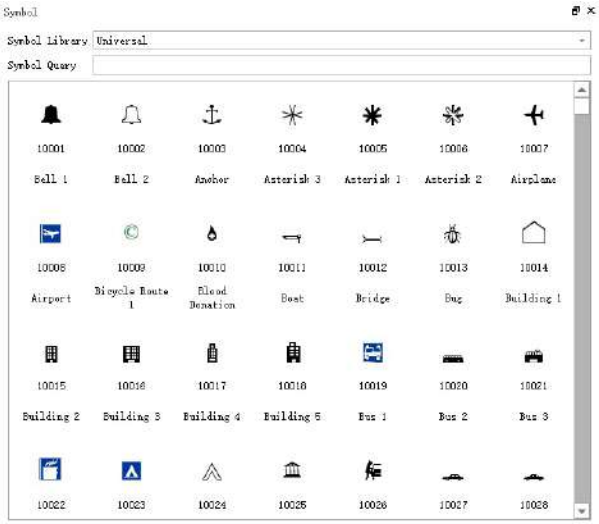
- **Symbol**: Displays the symbol code of the point feature. If it is not set, it will be displayed as blank.
- **Screenshot**: Click the button on the right, you can complete the screenshot based on the current screenshot window, and the picture is saved as a png format in the Image field of the current feature.

  > This function requires the panorama window and screenshot window to be open.

- **Layer ID**: The ID corresponding to the layer, read-only.
- **Layer Name**: The name of the layer it belongs to, read-only.
- **Extended Attributes**: All attribute fields contained in the feature.
- **fid**: The ID, which cannot be modified, generated by the database automatically.
- **Other attribute fields**

Setting Symbols

Click the blank button in the row where the symbol is located to pop up the symbol table:



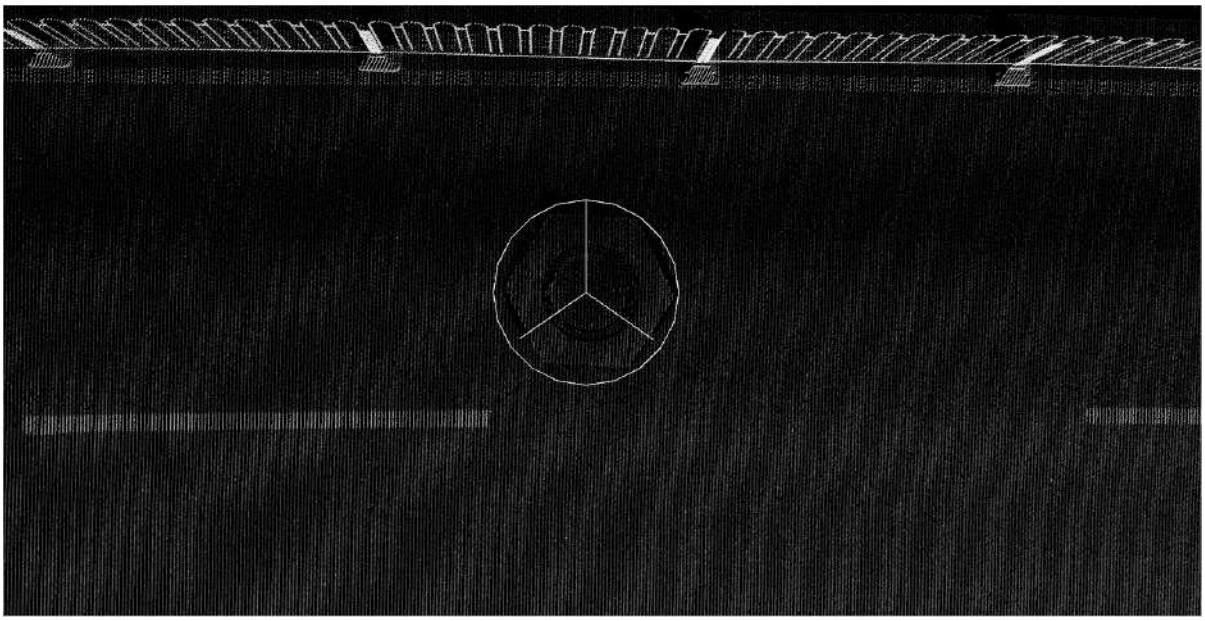
Symbol Table

> If the element is not a point type, the related setting button will be grayed out.

Parameter Setting

- **Symbol Library**: Divided into standard and common symbols. You can choose according to your needs.
- **Symbol Lookup**: You can precisely find symbols based on name and code.

After selecting the required symbol with the mouse, click "Apply". To view the symbol effect, click on Display Symbol in Appearance.



Symbol Effect

Modifying Attribute Values

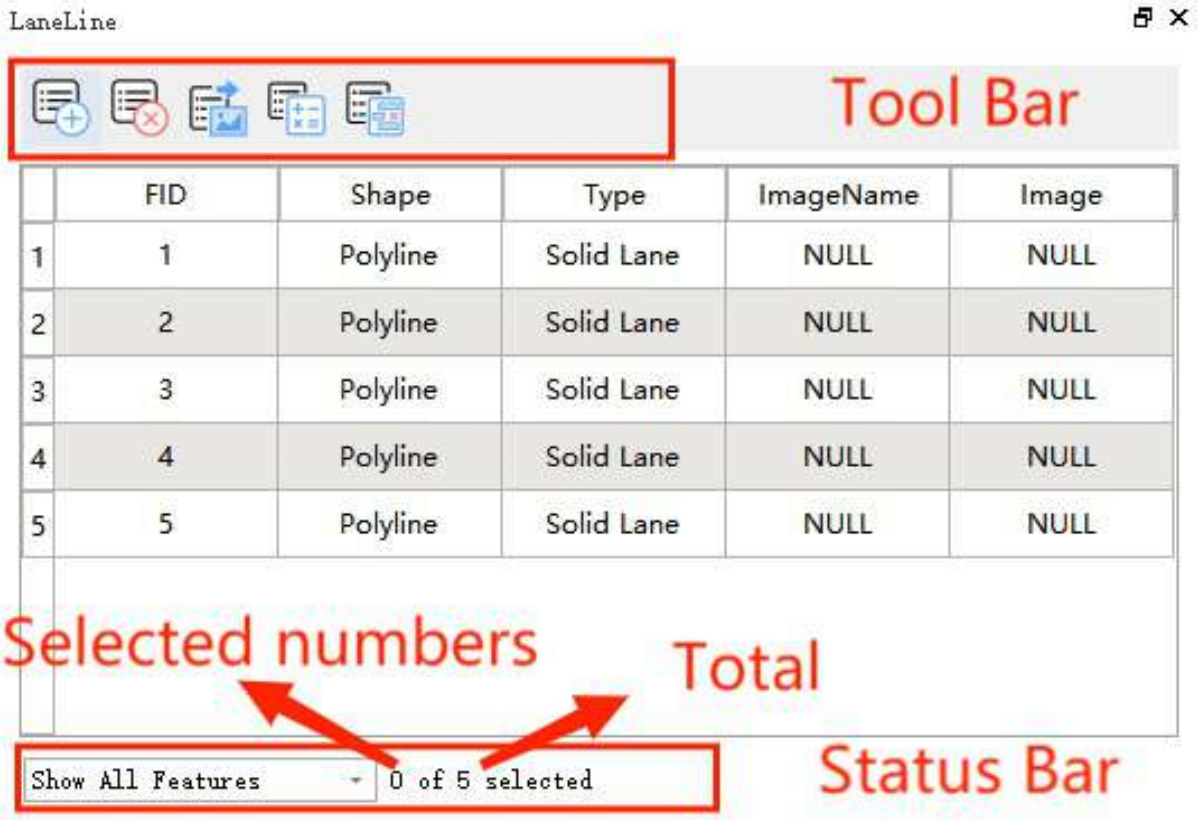
The mouse can directly click the editing control in the second column of the extended properties to modify it.

Attribute Table

The attribute table is an important component in the LiDAR360MLS software, allowing you to view and edit attribute information in vector datasets. The attribute table can display various types of data, including attributes of points, lines, surfaces, and other features.

Overall Interface

As shown in the figure below, the title in the top left corner displays the name of the layer to which it belongs. Below it, the toolbar arranges commonly used operation buttons, including the addition and deletion of fields, image export, and field calculator. The rectangular area under the toolbar is used to display attribute information, where the FID column displays the unique identification number of the feature in the database, usually an integer; the Shape column only exists in the vector layer, used to display the geometry type of the layer. The drop-down box on the left side of the status bar at the bottom is used to switch between various display modes—including displaying all features and only displaying selected features; the panel on the right is used to intuitively display the total number of features in the current layer and the number of currently selected features.



Attribute Table Description

1.Steps to open the attribute table

（1）Right-click on "Attribute Table" on any vector layer or table layer in the directory tree to open the attribute table.

（2）The attribute table will be displayed on the right side of the software in the form of a docked window.

2.Select a row: You can quickly select a row by clicking the number column on the left with the mouse. You can select multiple non-continuous rows through CTRL, and select multiple continuous rows through the SHIFT key.

> To select all rows, you can directly click on the blank area in the upper left corner of the attribute table.

3.Delete a row: After selecting a row, right-click and click "Delete selected row".

4.Edit attribute information: In the attribute table, you can edit the attribute information of features. To edit the attribute information of a feature, perform the following operations:

（1）Select a feature in the attribute table.

（2）Click any cell in the row where the feature is located to enter edit mode.

（3）Edit the attribute value of the feature.

（4）If you want to undo the modification, you can use the combination key of Ctrl+Z; if you want to redo the modification, use the combination key of Ctrl+Y.

5.Add a field: In the attribute table, you can add a new attribute field to the vector dataset by adding a field. You can specify the type and length for each field. Here are the steps to add a field:

（1）Click the button in the toolbar to pop up the Add Field dialog box.

（2） In the "Add Field" dialog box, enter the name, type, and length of the new field. You can choose the following different types of fields.

- **Short Integer**: Used to store short integer data.
- **Long Integer**: Used to store long integer data.
- **Float**: Used to store 32-bit floating-point data.
- **Double**: Used to store 64-bit floating-point data.
- **Text**: Used to store text data.
- **Date**: Used to store date data

（3）Click the **OK** button to add a new field.

> Support shortcut keys Ctrl+Z/Ctrl+Y to undo/redo.

6.Delete a field: In the attribute table, you can complete the deletion of attribute fields through the following steps:

（1）Click on the header where the field name to be deleted is located to select the entire field.

（2）Click on the button in the toolbar above.

> FID and Shape are auxiliary information columns and are not allowed to be deleted; attribute fields required by built-in layers (such as Type, Image, ImageName, etc.) are also not allowed to be deleted.
> Support shortcut keys Ctrl+Z/Ctrl+Y to undo/redo.

7.Export pictures: In the attribute table, you can export the image information stored in the Image field as a png format. The saved png is named after the value in the ImageName field by default. Here are the specific operation steps:

（1）Click the button in the toolbar to pop up the dialog box for selecting the export folder path.

（2）After selecting the folder to be saved, click **Choose** Folder.

> Currently, exporting pictures only supports point layers, and other types of layers are not supported. In addition, if the ImageName and Image fields are empty, it will prompt that the export failed.

8.Field calculator: Click the button in the toolbar to pop up the field calculator dialog box. For specific field calculator operation instructions, refer to Field Calculator.

9.Close the attribute table: To close the attribute table, please perform the following operations:

（1）Click the "X" button in the upper right corner of the window.

（2）Right-click the tab label at the bottom of the attribute table, supporting the following closure methods:

- **Close the current attribute table**: Only close the currently selected attribute table
- **Close non-current attribute tables**: Close all non-current attribute tables
- **Close all attribute tables on the left**: Close all attribute tables arranged on the left of the current attribute table
- **Close all attribute tables on the right**: Close all attribute tables arranged on the right of the current attribute table

10.Right-click function

（1）Select a column and support the following right-click menu functions:

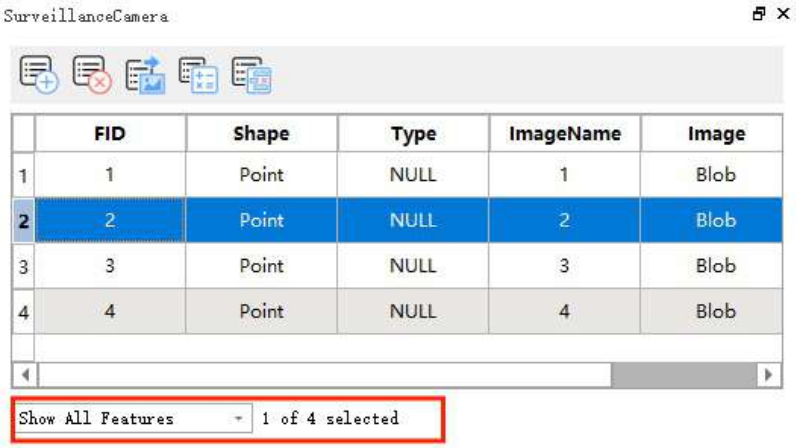
- **Ascending order**: Sort the attribute table in ascending order of the selected field.
- **Descending order**: Sort the attribute table in descending order of the selected field.
- **Organize column**: Control the visibility of the header.
- **Hide column**: Hide the currently selected column.
- **Detect duplicate values**: Filter out all rows with the same attribute value in the current column, and keep them highlighted.
- **Detect illegal values**: Filter out all rows containing illegal characters (mainly for text fields) in the current column attribute value, and keep them highlighted.
- **Properties**: Display basic property information of the current column, including field type, length, etc.
- **Copy**: Copy the content of the current column to the clipboard
- **Paste**: Paste the content from the clipboard into the current column
- **Generate NanoID**：Generate a unique identifier.

（2）Double-click a cell and support the following right-click menu functions:

- **Undo**: Go back to the previous step.
- **Redo**: Cancel.
- **Cut**: Put the selected information into the clipboard.
- **Value**: Set the value according to the demand.
- **Paste**: Paste the content from the clipboard to the current.
- **Delete**: Delete the data.
- **Select All**: Select all data.
- **Increase**: Increase a value.
- **Decrease**: Decrease a value.

Status Bar

Function Description: The Status Bar is primarily used to display the current status information of the attribute table of the layer. It includes the number of currently selected records and the total number of records. It also supports toggling between displaying all records and displaying only selected records.



Steps

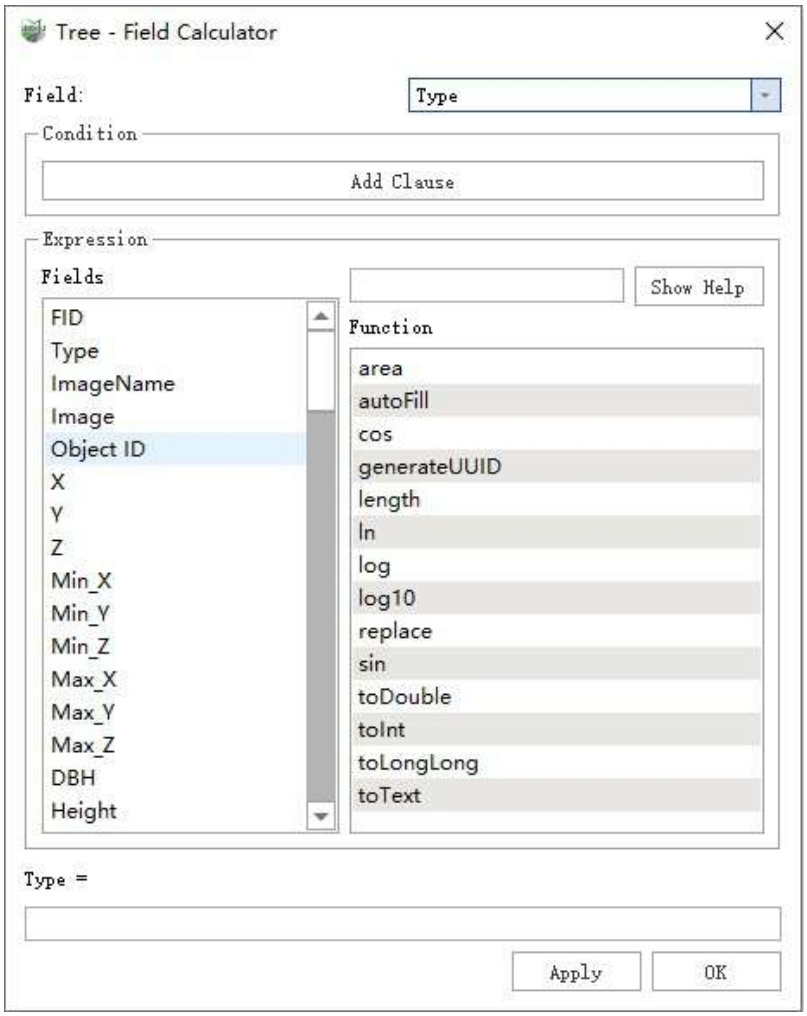
Click on the left dropdown list. If you select "Show All Features," it will display the attribute information of all features. If you select "Show Selected Features," it will only display the attribute information of the selected features.

Field Calculator

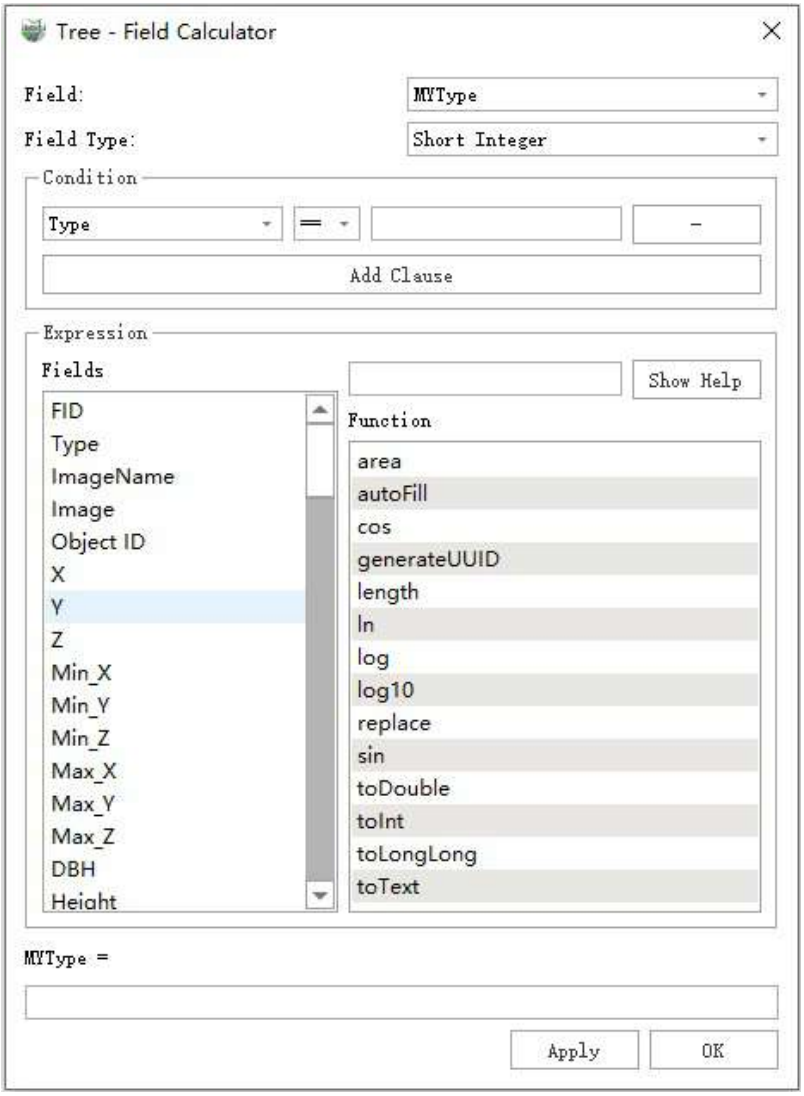
Description: Used for calculating and editing fields in a feature class or table. It allows users to write simple code to perform various calculations and operations, including calculating field values, updating field values, and calculating geometry properties.

Steps

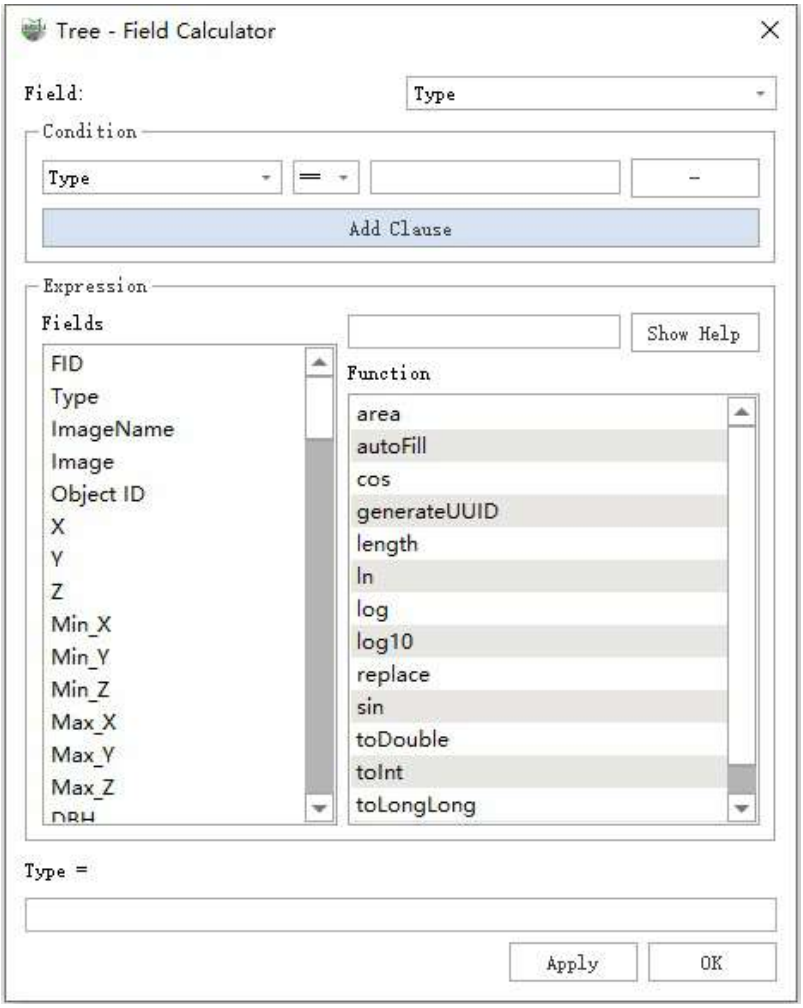
1.Open the attribute table of the feature class or table. In the top toolbar, click on **Field Calculator** to open the Field Calculator. Select the field you want to calculate, as shown in the following image:



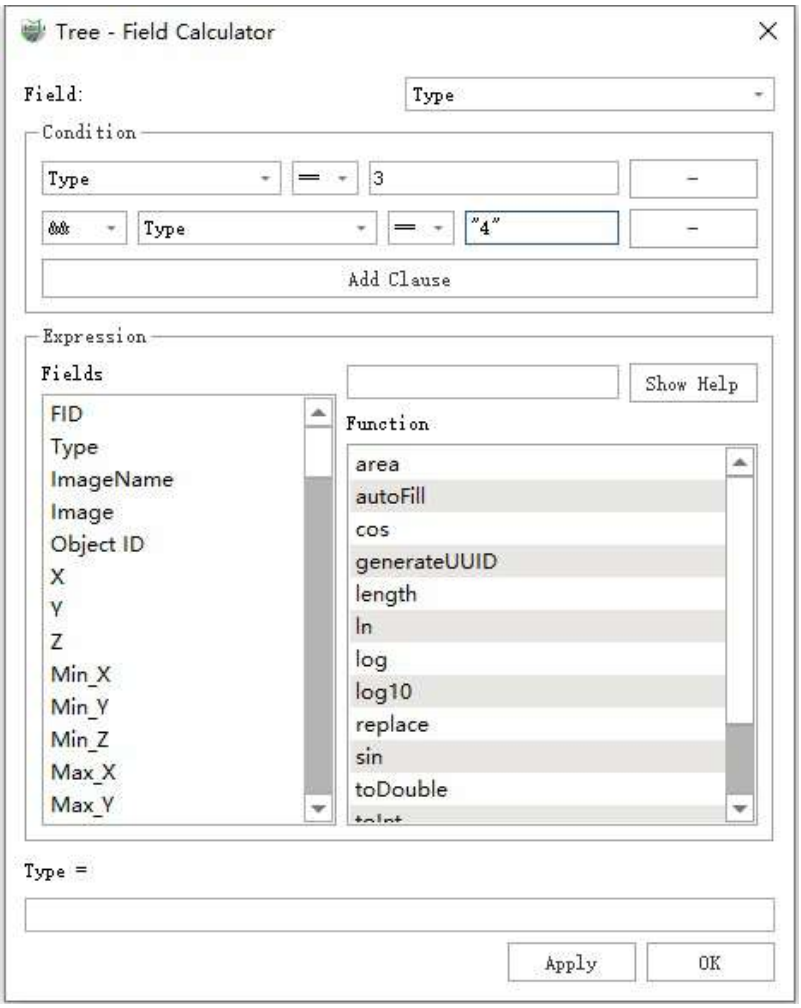
> If you need to apply the expression to a newly added field, you can directly enter the name of the target field in the field dropdown box. After editing, press Enter and click elsewhere with the left mouse button. A control will appear to set the type of the new field, as shown in the image below:



2.Add conditions: Click the "Add Condition" button to open the condition expression control. You can set the filtering conditions as needed. The software supports combination conditions. The entire interface is shown in the following image:



The first column is used to select the field name to which the condition applies, the second column selects the comparison operator (greater than, less than, equal to, etc.), and the third column enters the corresponding value. For example, the condition expression "Type=3" means that only records where Type equals 3 will have the subsequent calculation performed, and the ones that do not satisfy the condition will be ignored. The fourth column is the delete button, which deletes the current expression when clicked. Clicking the "Add Expression" button at the bottom allows you to add new expressions, as shown in the image below:



3.Write the expression: The left list in the expression section displays the field names that can be used as input variables. The input box on the top right can be used to search for matching functions based on the entered function name. Clicking on "Show Help" will display the usage of the currently selected function, including a description of its functionality, input parameters, etc. The function list below displays the built-in functions of the software:

- **area**- Calculate the geometry area.
- **autoFill**- Automatically fill the target field based on the start value and step size.
- **generateUUID**- Generate a UUID.
- **length**—— Calculate the length.
- **replace**—— Replace characters.
- **sin**—— Calculate the sine value.
- **cos**—— Calculate the cosine value.
- **toDouble**—— Convert to floating-point.
- **toInt**—— Convert to integer.
- **toLongLong**——Convert to long integer.
- **toText**—— Convert to text.

Double-clicking on a field name in the left list will automatically fill the field in the expression editor below. Double-clicking on a function in the function list on the right will automatically fill the function in the editor below. For example, if you want to use autoFill on the "type" field with a start value of 0 and a step size of 1, the expression in the text box would be: autoFill(0,1)

4.View the results: Click "Apply" to execute the calculation. If the input expression is invalid, relevant error messages will be displayed in the log window. After the calculation is complete, the dialog box will not close. If you click "OK," the current dialog box will automatically close after the calculation is finished.

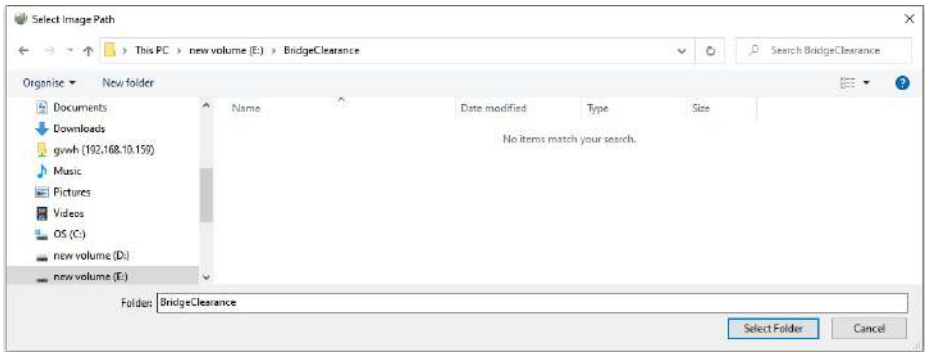
> For string constants, they need to be enclosed in double quotation marks. If there is a selection set on the layer, the calculation will only be performed on the records in the selection set.

Export Image

Function Description: Export all the images stored in the Image field to a specified directory, using the ImageName field value as the filename with the ".png" extension.

Steps

1.Open the attribute table of the feature class or table. In the top toolbar, click on **Export Image** to open the output folder selection dialog. Choose the destination folder where you want to save the images and click the "Select Folder" button, as shown in the following image:



2.If there are Image fields or empty ImageName fields in the current attribute table, a dialog box will appear, indicating that there are empty fields and the export will fail.

Appendix

As shown in the list below, the file page contains the following functions:

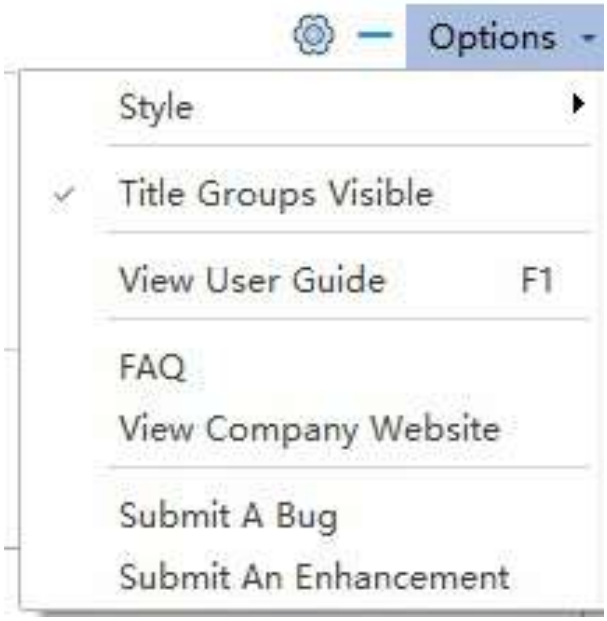
- Options
- Display Options
- Adjusting to High-Performance Display Mode
- FAQ
- Version Update Subscription

Options

Feature Description: The options function in the main interface of the software provides auxiliary operations such as style setting, group name display/hide, and feedback for bugs and feature requests.

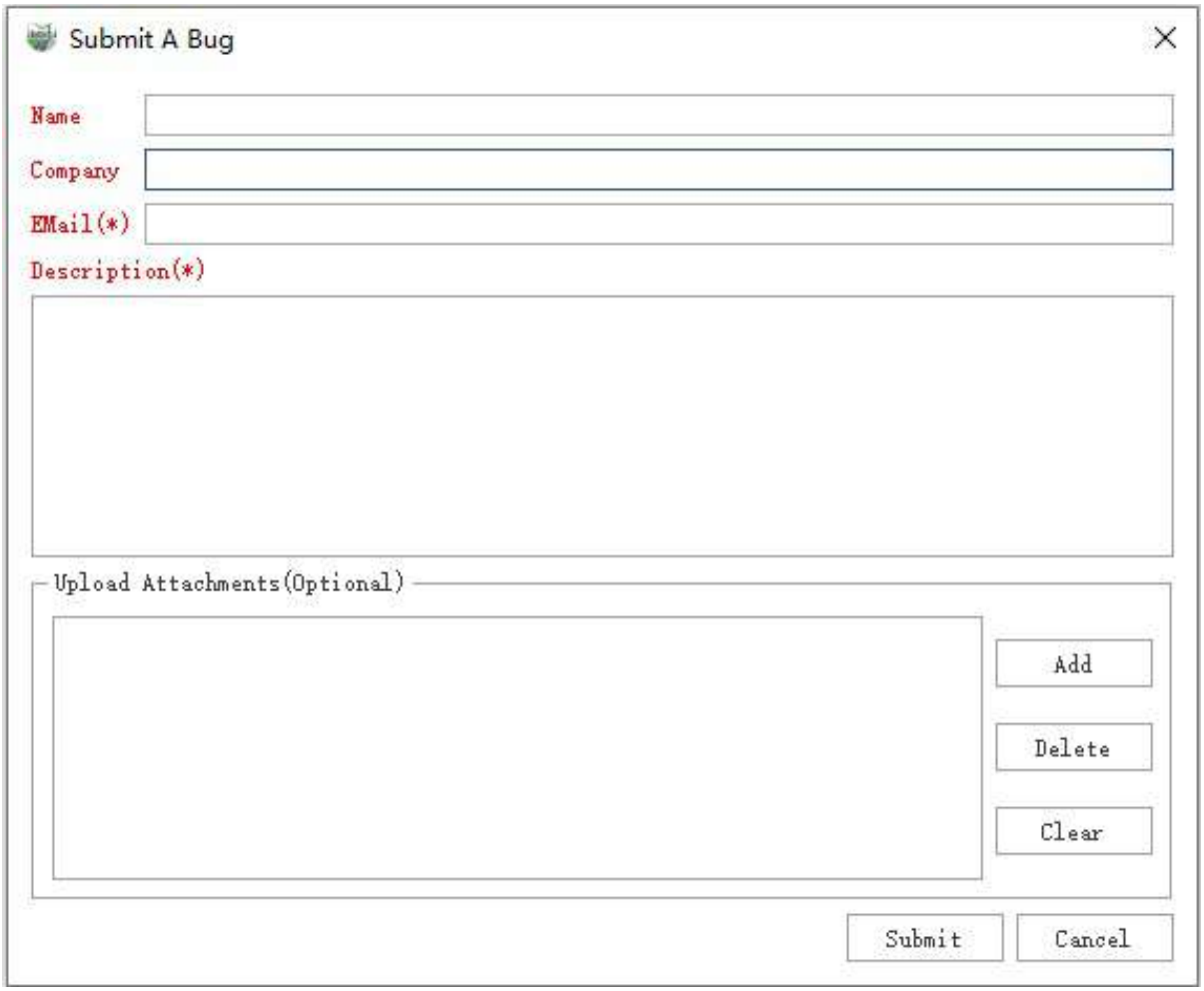
Steps

1.In the top right corner of the main interface, click "Options". Here you can modify the interface style and control the visibility of group names in the menu bar.



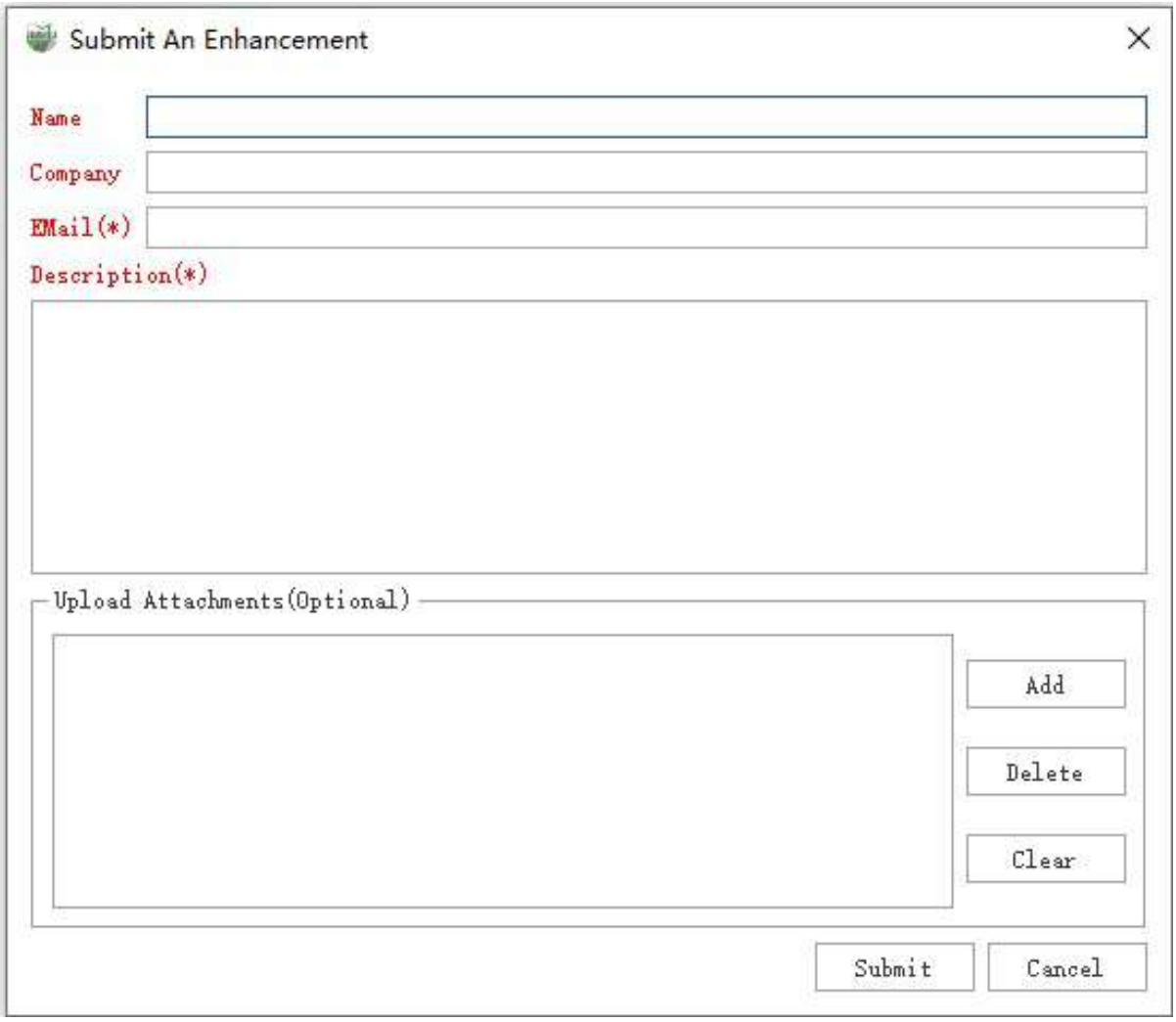
2.Submit BUG: Click "Submit Bug". In the pop-up window, please describe your basic information and the steps that led to the bug in detail. This will help us reproduce the problem and contact you.

> Attachment upload and other related content are supported.



3.Submit Request: Click "Submit Request". In the pop-up window, please describe your basic information and software requirements related to your business needs in detail. This will help us contact you.

> Attachment upload and other related content are supported

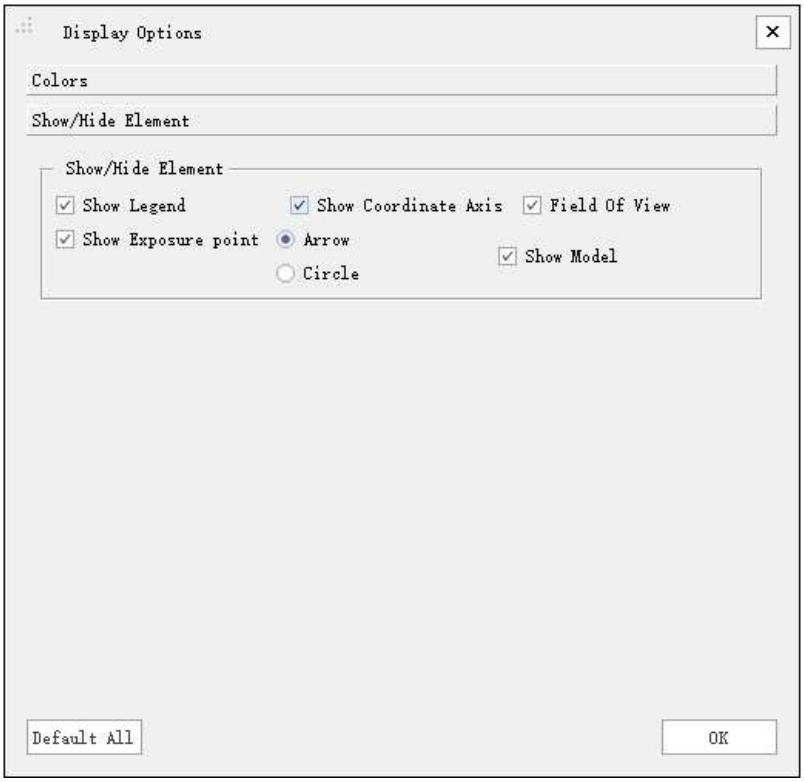

Submit An Enhancement
Name
Company
EMail(*)
Description(*)
Upload Attachments(Optional)
Add
Delete
Clear
Submit
Cancel

Display Options

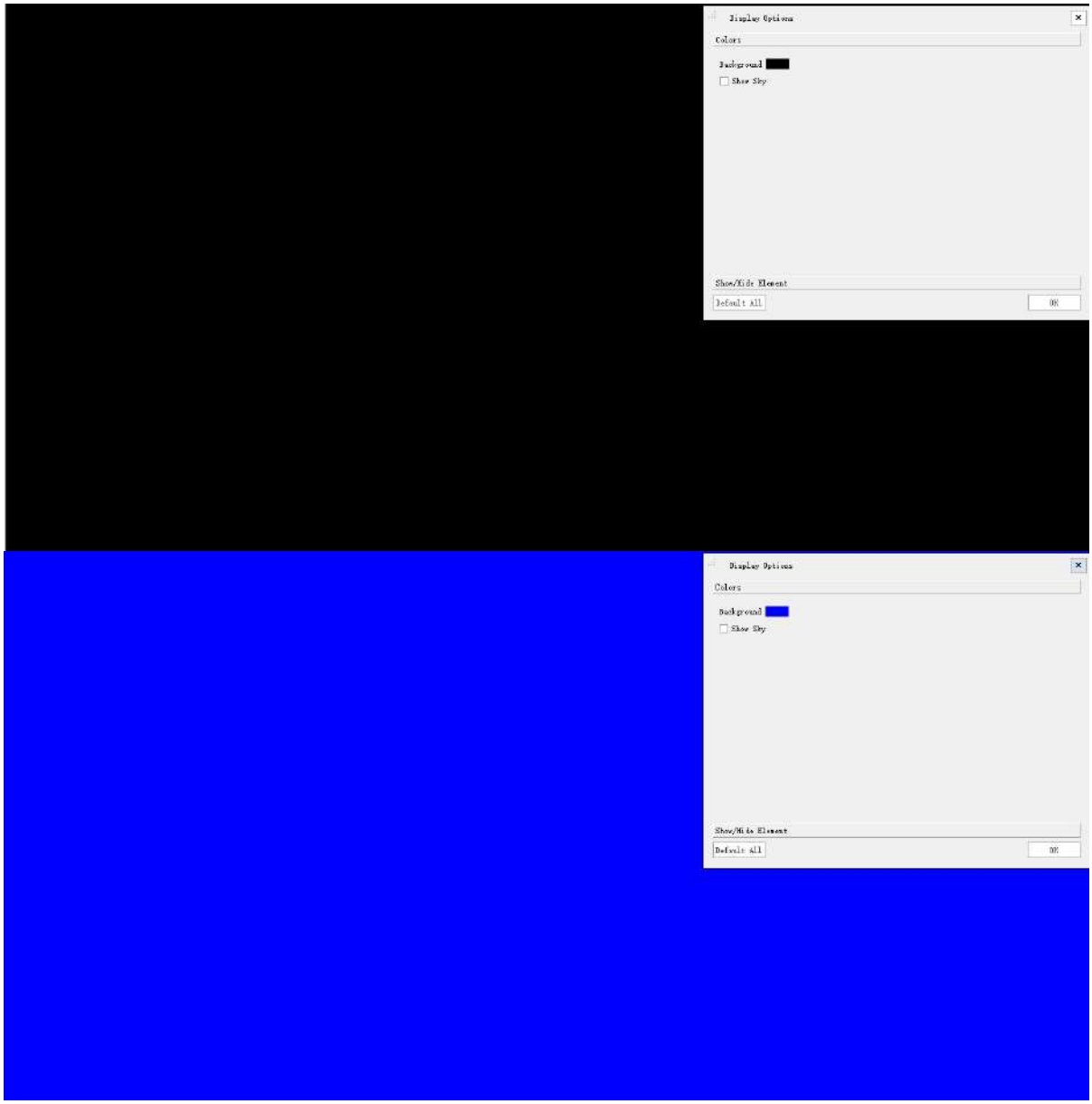
Function Description: The display options feature in the software's main interface provides settings for background color, element visibility, camera height, and orientation.

Steps

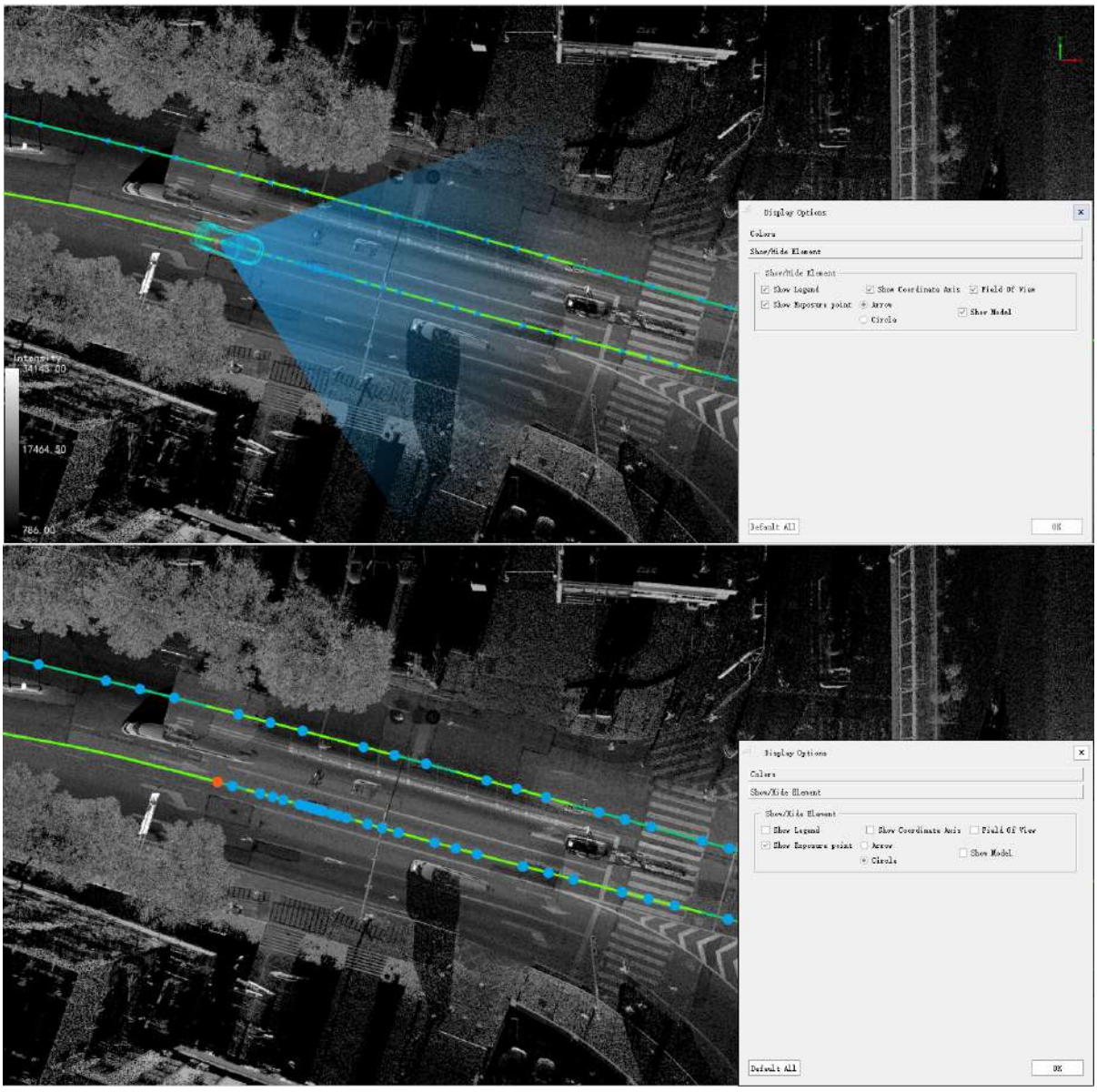
1.Click on the display option settings button located at the top right corner of the software, a display option settings dialog box will pop up.



2.Color: Allow you to set the background color of the 3D window.



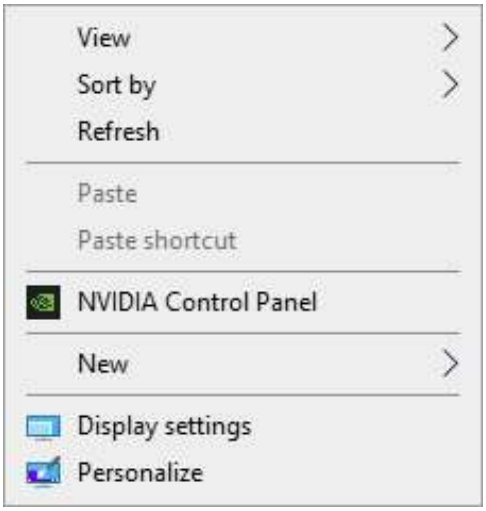
3.Show/Hide Elements: Used for showing/hiding the legend, coordinate axis, field of view, arrows, and whether the model and panorama view is displayed in the 3D window.


Intensity
34143.00
17464.50
786.00
Display Options
Colors
Show/Hide Element
Show/Hide Element
Show Legend
Show Coordinate Axis
Field Of View
Show Exposure point
Arrow
Circle
Show Model
Default All
OK

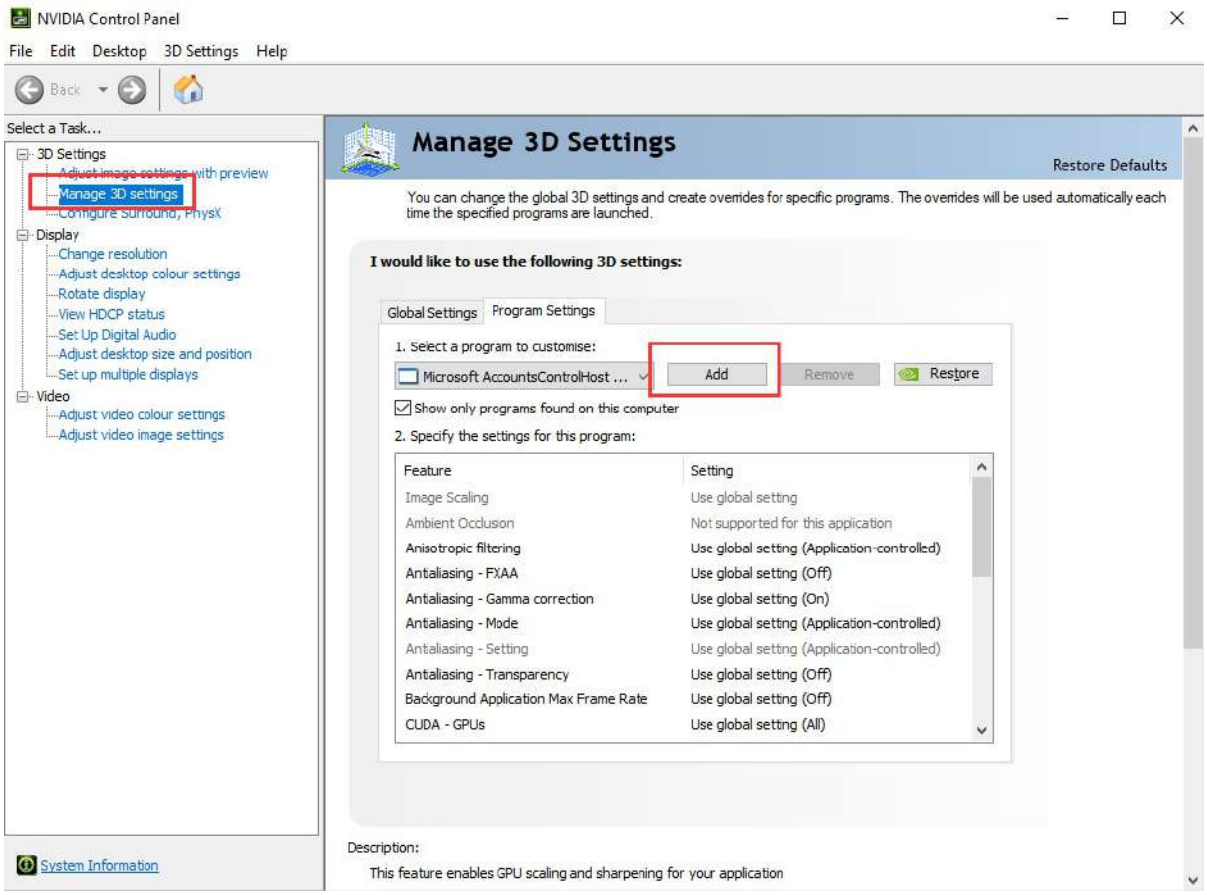
Adjusting to High-Performance Display Mode

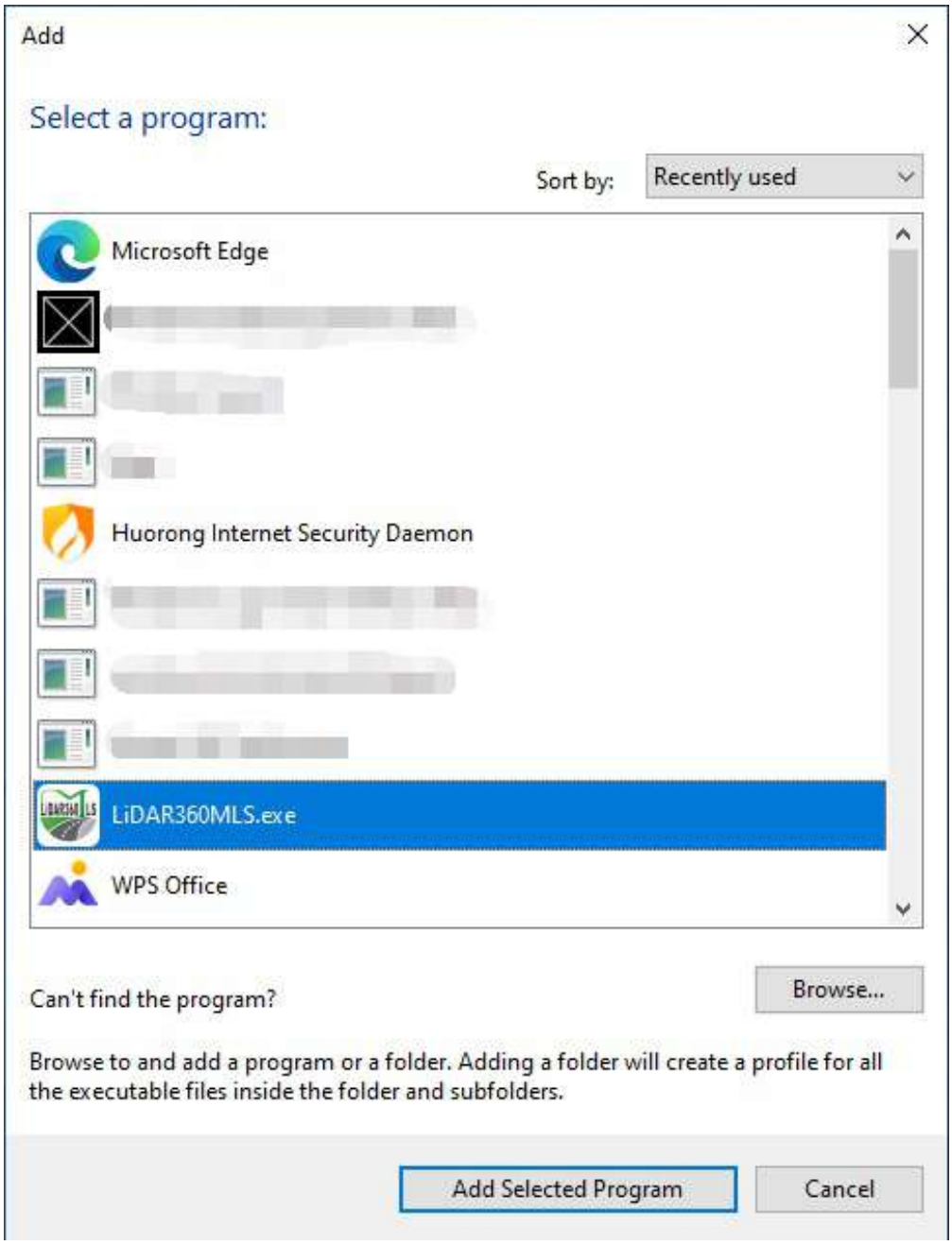
Follow the steps below to optimize the graphics mode of LiDAR360MLS.exe (for NVIDIA graphics cards).

1.Right click on the desktop and select NVIDIA Control Panel.



2.Select Manage 3D settings -> Program Settings -> Add **LiDAR360MLS.exe** to the list of high-performance graphics modes, click **Apply**.




Add
Select a program:
Sort by:
Recently used
Microsoft Edge
Huorong Internet Security Daemon
LiDAR360MLS.exe
WPS Office
Can't find the program?
Browse...
Browse to and add a program or a folder. Adding a folder will create a profile for all the executable files inside the folder and subfolders.
Add Selected Program
Cancel

FAQ

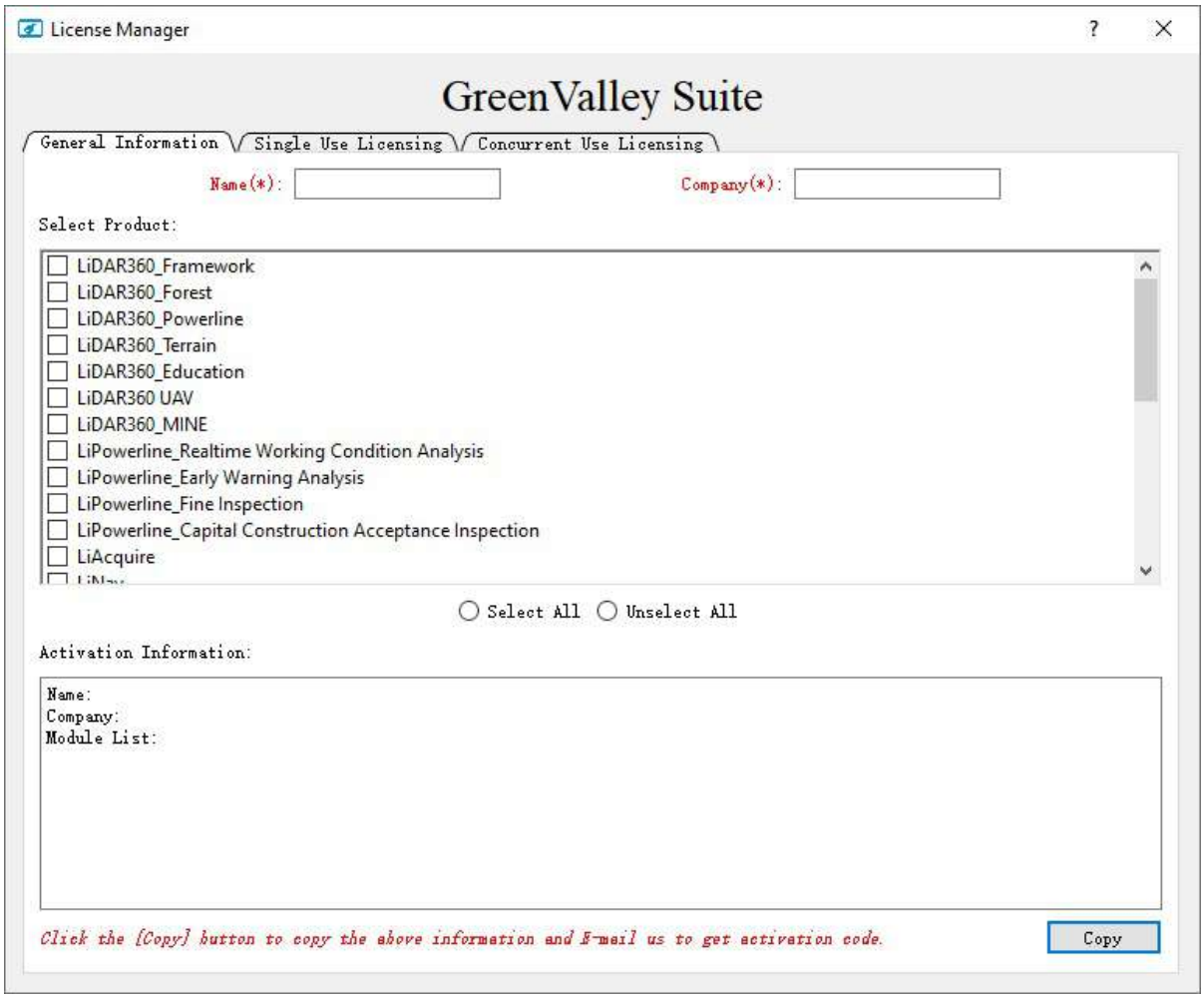
Function Description: This section introduces common problems and their respective solutions during the use of LiDAR360 MLS software.

Why does the software prompt that the trial license has expired after installing LiDAR360 MLS?

- LiDAR360 MLS cannot be tried out for the following reasons:
    - Change of system time: During the trial of LiDAR360 MLS, there may be some reasons that may cause the system time to change, causing the trial to expire.
    - Trial expiration: The trial period of each version of LiDAR360 MLS is 15 days. If the same version is installed for more than 15 days, the trial will expire.
    - Other reasons: Please send an email to info@greenvalleyintl.com to apply for an extension of the trial license.

How does LiDAR360 MLS send activation information? How to activate after receiving the license file?

- Send activation information: Click File > License or double-click License Manager.exe in the installation directory.



- enter name;
- enter the company;
- select the activation module;
- Click Copy;
- Ctrl+V paste the content into an email and send it to info@greenvalleyintl.com.
  - License activation:
- Copy the authorization code in the email to the local, and the activation method can refer to the license manager.

What features can I still use after the LiDAR360 MLS trial expires?

- After the trial period ends, the LiDAR360 MLS software cannot be used normally, and the function needs to be activated before it can be used.

How to view the help manual?

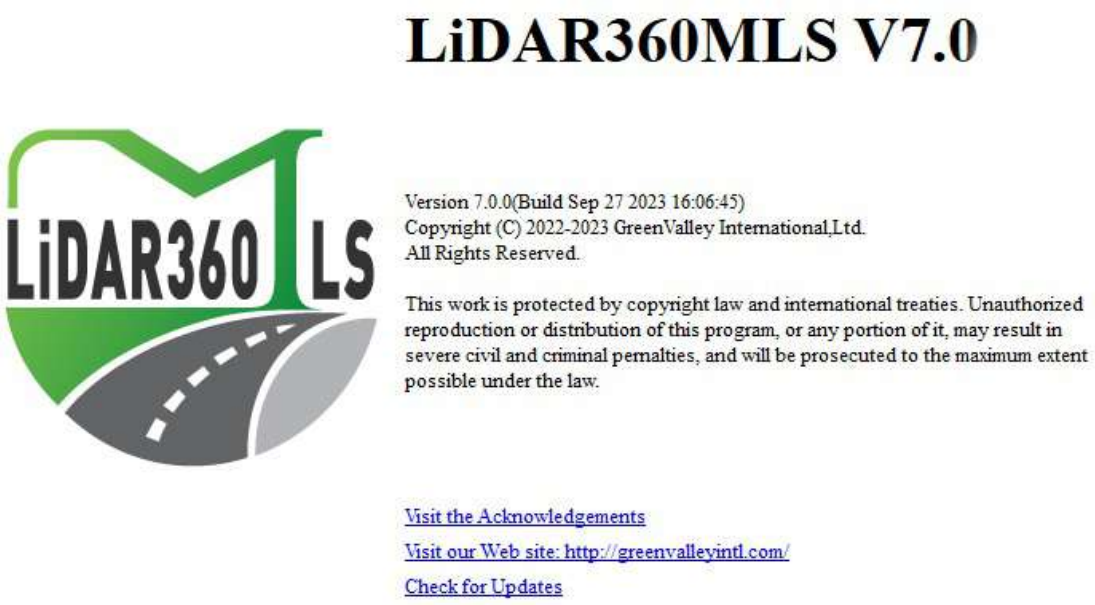
- Click File > Help

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, 2.1.0 is the software version number, and Sep 27 2023 16:06:45 is the compilation date.



How to change the language settings?

- Click File > Options > Language to switch between English and Chinese.

Why does the software exit abnormally?

- When the software exits abnormally, please check: (1) Whether the on-screen word-taking software (such as Youdao Dictionary) is opened; (2) Whether there is enough hard disk space or memory space.

What data formats does LiDAR360 MLS support?

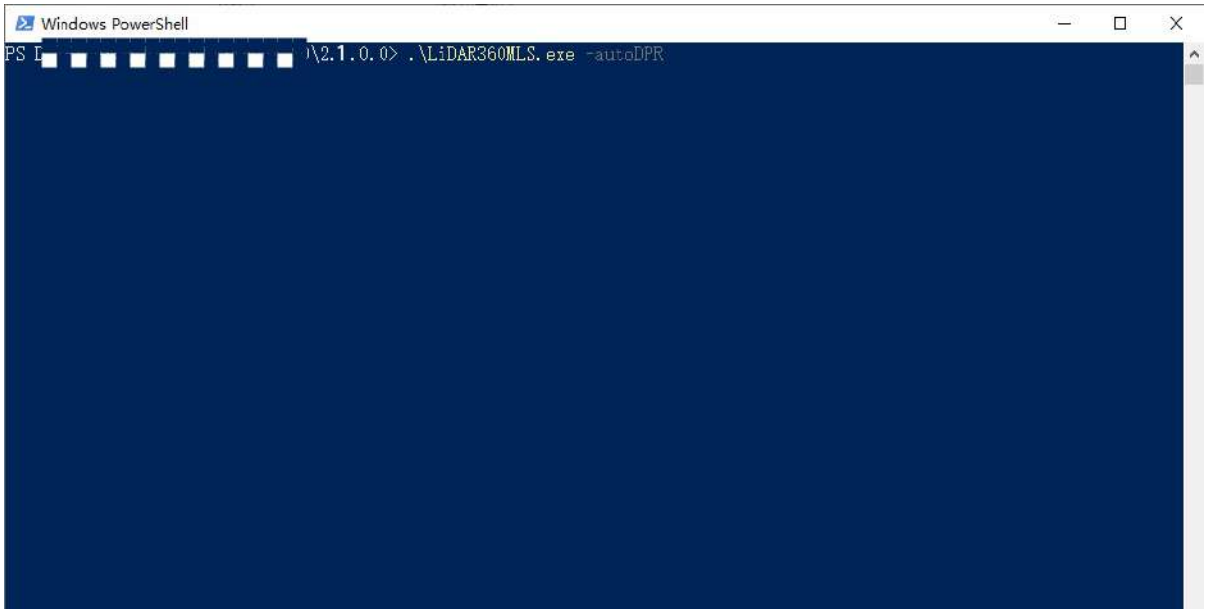
- The data types that can be imported into LiDAR360 MLS are divided into four categories: point cloud, trajectory, image and vector. The specific formats are as follows:
  - **Point cloud**: LiData file (.LiData custom point cloud format), LAS file (.las, .laz), PLY file (.ply), E57 file (*.e57*).
  - **Trajectory**: Trajectory data (*.traj,*.pos,*.txt,*.csv,*.asc,*.xyz,*.pts,*.out).
  - **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(*.csw),Riegl MMS File(*.csw),LIDARUSA SNOOPY File(*.txt),OrbitPos(*,txt *.csw)] and the image file storage location is accurate.
  - **vector**: vector data (Shape Files(*shp)*, *Geojson Files(*geojson), Kml File(*.kml)*, *Dxf File(*dxf).
- 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 (Shape Files(*shp)*, *Geojson Files(*geojson), Kml File(*.kml)*, *Dxf File(*dxf).

LiDAR360 MLS software crashes after startup?

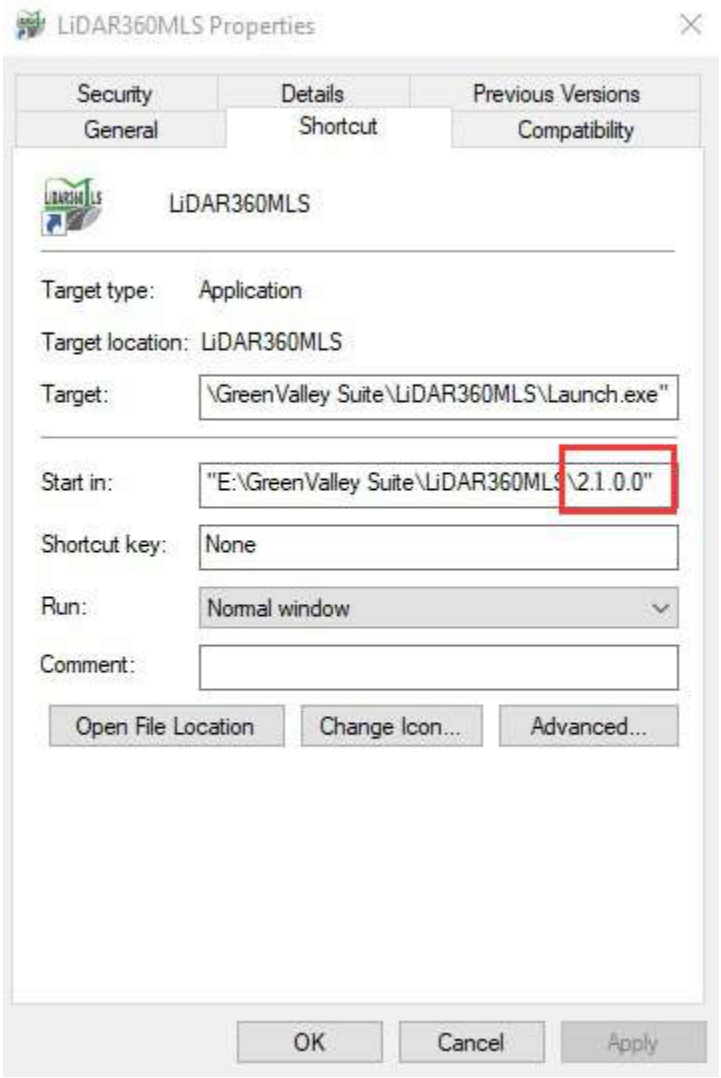
- Please check whether the computer screen is plugged in correctly. Normally, it should be plugged into the position of the graphics card connector of the host computer.

The high-resolution screen computer software cannot be started or the software is stuck?

- Method 1: In the software installation directory (eg: D/LiDAR360MLS/2.1.0.0/), press and hold the Shift 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.

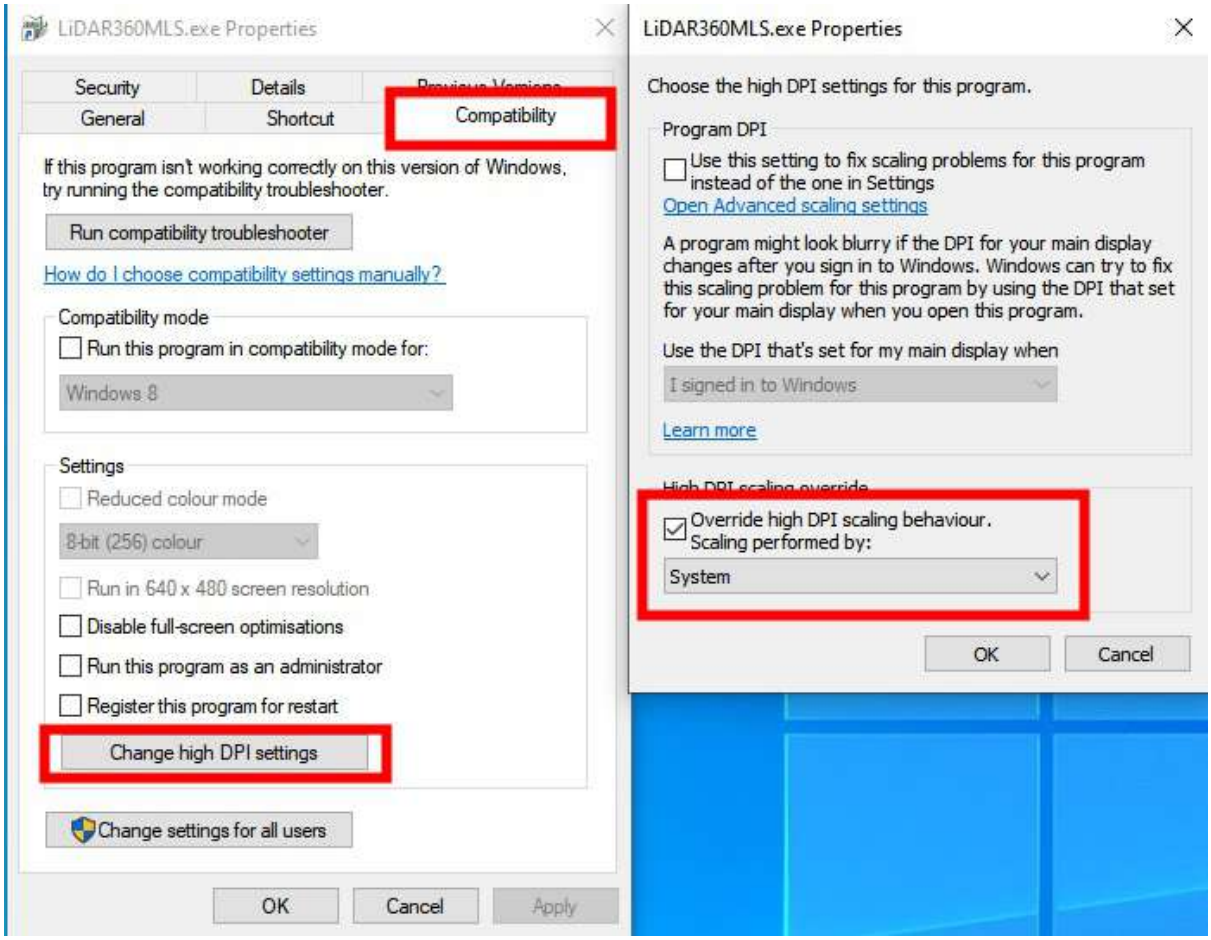


- Method 2: In the software installation directory (eg: D/LiDAR360MLS/2.1.0.0/), send the LiDAR360MLS.exe in the directory to the desktop shortcut, then right-click the shortcut, open the properties, and at the end of the target. Add **-autoDPR**, then double-click the program to start the software normally.



On a high-resolution screen, the software startup displays abnormally, with icons appearing unusually large.

- Right-click on the LiDAR360MLS startup program and click on "Properties."
- Switch to the "Compatibility" panel.
- Click on "Change high DPI settings."
- In the "High DPI scaling override" section, check the scaling behavior and modify it to "System."
- After applying the changes, launch the software, and the display should be normal.



After using the software for a long time, double-clicking to run it starts slowly？

- Prepare the software activation code
- In the ProgramData folder on the C drive, find the BitAnswer folder (C:\ProgramData\BitAnswer)

> Note: The ProgramData folder is a hidden folder, you need to check the "Hidden Items" box in the view section of the folder to see the folder.

- Delete the contents of the BitAnswer folder and re-run the software to re-activate it to solve the problem of slow start-up.

After running the software, some of the displays were found to be in the wrong place, such as the misplaced hints in the 3D window of the element extraction function

This usually happens when there are multiple graphics cards on the computer and the monitor is plugged into the integrated display.

- You can first check if the monitor socket is plugged into a unique graphics card (Nvidia is recommended), and it is recommended that the monitor is plugged into a unique graphics card.
- If you do not want to change the position of the monitor socket, you can refer to Adjust High-Performance Graphics Mode to run the software in high performance mode.

The point cloud does not display in the new project software.

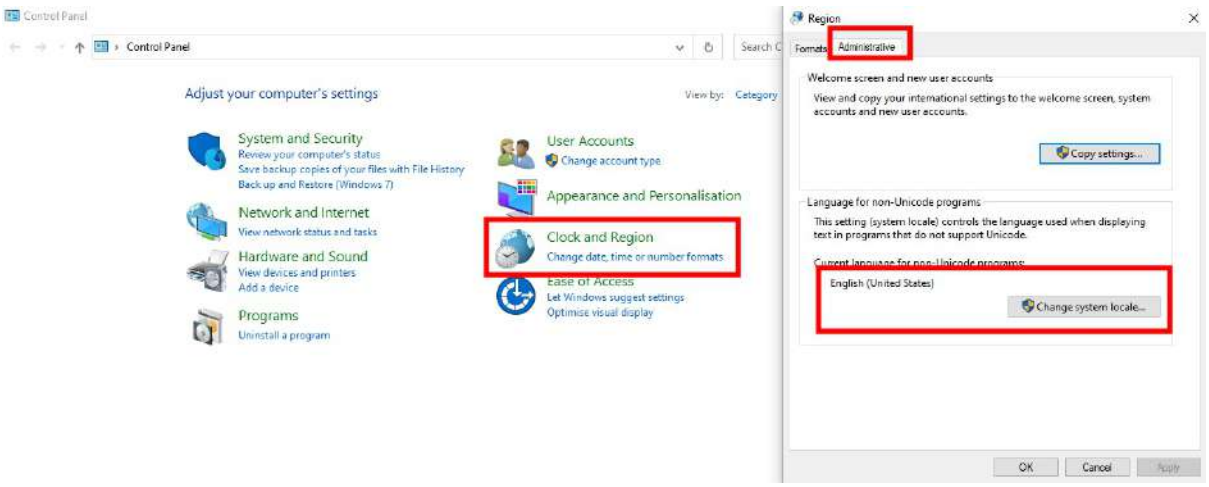
This situation usually occurs when there are more than one official NVIDIA panel, whose version is wrong. Or there is a software setting problem.

- Download the corresponding version of NVIDIA Control Panel and refer to Adjustment of High-Performance Graphics Mode to run the software in high-performance mode. software to run in high-performance mode

After the software is launched, the interface displays garbled characters

This issue commonly arises due to non-Unicode program language settings on the computer.

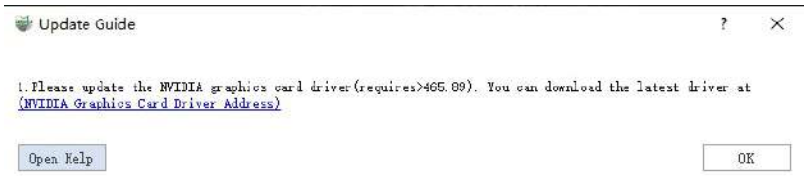
- Open the computer control panel.
- Locate the "Clock and Region" option.
- Click on "Change date, time, or number formats."
- Switch to the "Administrative" tab.
- In the "Language for non-Unicode programs," ensure that the language used by the current system is consistent with the setting here. For English Windows systems, the default setting is usually appropriate. If language packs for other languages, such as Chinese language packs, are installed, set this option to the corresponding language pack.



When using Custom Deep Learning Classification, If prompted to "Please go to (service pack address) to download the complete service package," please click the link to download and install it before launching the feature.

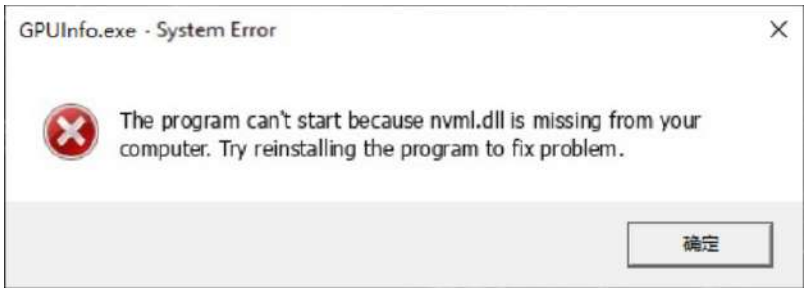
- Please click the link to download and install the complete service pack before launching the feature.

When using Custom Deep Learning Classification, if prompted "Please update the graphics card driver (version required > 465.89), you can download the latest driver from (graphics card driver address)".



- Please click on the graphics card driver address to download and install the latest version of the graphics card driver. This will resolve the issue.

When using Custom Deep Learning Classification, if you get "The code cannot continue because nvml.dll was not found. Reinstalling the program may solve this problem".



- Please first check if the graphics card driver is installed. If you have confirmed that you have installed the latest version of the graphics card driver and the issue still persists, you can add the path **C:\Program Files\NVIDIA Corporation\NVSMI** (the path may vary based on your graphics card driver installation) to the system environment variable Path.

When running the training module, I receive the following message: "GPU driver version is too old. Please update to version 465.89 or above."

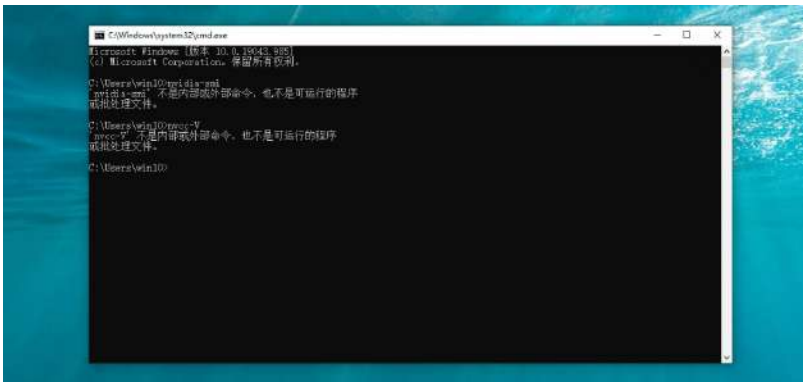
- Please upgrade your graphics card driver to at least version 465.89.

When running the training module, I get the message "Nvidia GPU not found or driver not installed."

- Please make sure you have an Nvidia GPU installed and that the graphics card driver is properly installed

When using deep learning related functions, if the following prompt is printed in the log window, but the computer has an NVIDIA graphics card

- To check if there is any problem related to NVIDIA graphics card driver, you can use win+r key combination to hit the run screen of Windows and type cmd to open the command prompt window. In the Command Prompt window, type nvidia-smi and press Enter. If the interface prompts "nvidia-smi is not an internal or external command, nor is it a program that can be run", it proves that the current computer has an NVIDIA graphics card, but the graphics card driver is abnormal, so it is recommended to download NVIDIA drivers from NVIDIA Driver Download page to download the corresponding driver.



When labeling training data, should the number of the same category be uniform across all the data?

- The number of the same category must be the same, otherwise a good training model cannot be obtained. If there are data with different categories, you can use the classification function in LiDar360 MLS software to number all the data with the same categories and then train them.

If I already have a trained model and now need to continue training it with new data, what should I do?

- It is possible to use both old and new data as training sets and then retrain or use the existing model as a pre-trained model to continue training.

When using the Custom Deep Learning Classification module, I choose to train 100 rounds but why does it ends the training early without reaching 100 rounds?

- The module has an early stop function to save training time by stopping training when the mIoU of the validation data is not improved after 10 consecutive rounds of training.

When using the Custom Deep Learning Classification module and I terminate the training early, how do I get a report on the training sets?

- There is a button to print the report in the training screen, click Print Report to print after terminating the training.

When performing the inference process with a trained model, where can I find the data after inference?

- After inference is finished, the source data will be overwritten. If the source data has category information, it will be replaced. So it is recommended to backup the inferred data before inference.

LiDAR360MLS Release Notes

V7.2.0

- New Features:
  - Supported for LiGrip and LiBackpack data preprocessing
    - Improved the success rate of SLAM calculation.
    - Enhanced the efficiency of SLAM calculation.
    - Ensured consistency between the data collection end and the post-processing end.
    - Optimized point cloud colorization
  - Added Floor Plan Module
    - Added automatic vectorisation function for floor plan.
    - Added DXF export function for floor plan vectors.
    - Added one-click export function for blue print.
    - Supported intersecting and perpendicular line drawing models.
    - Supported line intersection、line merging、line breaking and moving.
    - Supported vertices editing.
    - Supported length、area、angle measurement.
  - Facade Survey Module
    - Added point cloud polygonal section unrolling function.
    - Added point cloud curved section unrolling function.
    - Added point cloud curved section unrolling function.
  - Platform
    - Point Cloud Display
      - Added point cloud X-ray rendering.
      - Added point cloud surface enhancement rendering.
    - Added Blur Plates and Faces function for panoramic/planar images.
    - Added EXIF update function for panoramic/planar images.
    - Added brightness contrast export functionality for point clouds.
    - Added pre-trained classification model for railway scenes.
    - Added section move step adjustment settings.
    - Support Traditional Chinese language pack.
  - Vector Editing
    - Added line intersection function
    - Added auto-close mode for right-click on line drawing function.
    - Added precision setting function for attribute field.
  - Urban Forestry Module
    - Added the function of extracting seed points globally.
- Enhancement：
  - Platform
    - Fixed the problem of losing DXF layer styles when exporting.
    - Fixed the failure of exporting vectors from point layers imported via TXT.
  - Element Extraction Module
    - Optimised the function of railway track detection, support category filtering
    - Optimised power line detection function, support category filtering
  - Facade Survey Module

        - Fixed the problem of exporting DXF layer name anomaly.
    - Road Analysis Module
        - Optimised the calculation results of road longitudinal section parameters
        - Optimised road damage detection
    - Fixed other bugs

V7.1.1

- Enhancement:
    - Optimized orthophoto density map export.
    - Optimized the import shp from table, supported direct import into points, lines and surfaces, and supported coordinate conversion.
    - Optimized track jump detection and repair function
    - Fixed the issue of manually editing the label settings in the profiling module.
    - Fixed AMD graphics card display issues.
    - Fixed imagelist update problem
    - Fixed track version reading problem
    - Fixed node editing crash problem.
    - Fixed other bugs in the software

V7.1.0

- New Features:
    - Platform
        - Added point cloud brightness and constrast adjustment.
        - Added screenshot position setting.
        - Added project organization functionality.
        - Added import point layers from tables.
        - Added image file update functionality.
        - Added data transformation functionality.
    - Preprocessing
        - Added detection and correction for lateral jumps in trajectories
    - Feature Extraction
        - Added railway detection feature.
    - Vector Editing
        - dded functionality to add XYZ to the attribute table.
        - Added the ability to export point layers in CSV format.
        - Added layer styling with support for styling based on attributes.
    - Urban Forestry
        - Added functionality for individual tree orientation measurement.
        - Added functionality for exporting individual tree reports.
- Enhancement:
    - Platform
        - Optimized volume measurement algorithm, supporting category filtering and calculation based on selected.
        - Fixed the issue of not being able to copy when exporting to DXF.

        - Optimized software interface, distinguishing between Chinese and English prompts.
        - Optimized feature reports, differentiating between Chinese and English reports.
    - Preprocessing
        - Fixed the issue of updating imagery data in aerial strip stitching.
        - Optimized control point correction function, supporting tie points on RGB information.
        - Optimized point cloud coloring algorithm.
    - Feature Extraction
        - Optimized functionality for extracting linear features.
    - Vector Editing
        - Optimized operations for functions like line-to-point conversion, point-to-line conversion, and surface construction.
        - Enhanced interactive operations for functions such as move, parallel copy, etc.
        - Expanded the line-to-point conversion function to support converting all nodes of a line vector to a point layer
    - Urban Forestry
        - Optimized the algorithm and process for individual tree segmentation.
        - Enhanced the functionality for creating thematic maps in ecological landscape analysis, now supporting the independent generation of thematic maps for multiple files.
    - Fixed other bugs in the software

V7.0.0

- New Featunres:
    - Platform
        - Added point cloud data format batch conversion
        - Added configurable project templates
        - Added export functionality for geojson data format
        - Added orthophoto window, supporting display, measurement, and mapping of orthophotos.
        - Added multi-window display modes for point cloud data based on different attributes.
        - Added Addeditional attributes for trajectories.
        - Added a tool for selecting within surfaces.
        - Added volume measurement reporting.
        - Added volume measurement functionality for two phases.
        - Added point cloud reprocessing functionality.
        - Added functionality for Addeditional operations on point cloud attributes.
        - Added Point Cloud Tool Module
        - Added SOR filtering.
        - Added noise filtering.
        - Added point cloud resampling functionality.
        - Added point cloud merging functionality.
        - Added functionality to normalize based on ground points.
        - Added de-normalization functionality.
        - Added point cloud normal calculation functionality.
        - Added GPS time conversion functionality.
        - Added projection definition functionality.
        - Added coordinate transformation functionality.
        - Added functionality to extract based on Addeditional attributes.
    - Preprocessing

        - Added colorization functionality for mobile laser scanning point clouds.
    - Point Cloud Classification and Extraction
        - Added classification pre-training models for indoor, outdoor, underground parking garagescene point cloud data obtained by SLAM device.
        - Added custom deep learning classification functionality.
        - Added plane detection.
        - Added cylinder detection.
        - Added classify by cover.
        - Added region growing functionality.
    - Feature Extraction
        - Supported automatic extraction of all types of road markings
        - Supported merge to lane
        - Added Pole Seed Point Transformation Functionality.
        - Added Pole Seed Point Update Functionality.
        - Added Pole Seed Point Editing Functionality.
        - Added individual segmentation of cars and buildings functionality.
    - Vector Editing
        - Added image (panoramic, planar, orthophoto) based vector drawing and editing.
        - Added radius circle and diameter circle drawing function.
        - Added Bezier and B-spline curve drawing modes.
        - Added tracking drawing mode.
        - Added point-to-line conversion functionality.
        - Added line-to-point conversion functionality.
        - Added polygon-to-line conversion functionality.
        - Added line-to-polygon conversion functionality.
        - Added layer interrupt layer function
        - Added construction surface function
        - Added multi-part to single-part feature
        - Added topological edge editing
        - Added auxiliary line mode .
        - Added field calculator.
        - Added object photo database storage .
        - Added attribute quality check tool.
        - Added geometry quality check tool.
    - Road Analysis
        - Added custom clearance analysis.
        - Added intersecting lines clearance analysis .
        - Added bridge height limit analysis and analysis report.
        - Added urban power distribution line analysis and analysis report .
        - Added road damage detection.
        - Added pavement condition index(PCI) calculation and report generation.
        - Added longitudinal section analysis.
        - Added model key point extraction.
        - Added road TIN generation.
        - Added vehicle collision simulation analysis .
        - Added Urban Forestry Module
        - Added seed point transformation functionality.
        - Added seed point update functionality.
        - Added seed point editing functionality.

        - Added tree height and diameter calculation based on seed points functionality.
        - Added measurement of urban tree parameters.
        - Added point cloud extraction based on TreeID.
        - Added measurement of parameters including DBH, Height, Crown Width, Crown Length, Crown Diameter, Crown Diameter SN, Crown Diameter EW ,CBH, Angle, Area, etc.
        - Added ecological landscape thematic map , supporting green viewing ratio(GVR), green volume, sky view factor(SVF), etc
        - Added generation of ecological landscape reports.
        - Added ecological landscape simulation analysis functionality.
- Enhancement:
    - Supported project creation wizard
    - Optimized on-surface and under-surface selection tools.
    - Optimized vector smoothing functionality.
    - Optimized node editing functionality.
    - Optimize tree segmentation functionality, supporting classification into trunk and crown categories.
    - Expanded orthophoto export based on intensity, height and RGB attributes
    - Optimized image and vector linkage setting
    - Fixed other bugs in the software

V2.0.4

- Added Modules and Features
    - Optimized node editing function, Supported “Z value” and “move to” function for quick modification by picking Z
    - Supported various vector selection tools to change the line direction, and Optimized shortcut keys and tips
    - Supported Riegl MMS panoramic image file import and Optimized Orbit format
    - Supported for South-North crown diameter (SN), East-West crown diameter (EW), crown length (CL) and crown width (CW) in the calculation of individual tree parameters
    - Updated trajectory data
    - Fixed the problem of Ligeo and LiFuser-BP project opening failure
    - Fixed the problem of strip adjustment
    - Fixed the problem of invalid horizontal section dragging
    - Fixed other bugs in the software

V2.0.3

- Added Modules and Features
    - New 2D mode
    - Optimised horizontal section function
    - Optimise edit vertices function
    - Optimise drape to ground function
    - Optimize vector objects and the corresponding panoramic image window linkage jumping
    - Fix other bugs in the software

V2.0.2

- Added Modules and Features
  - Add new project from template
  - Add import gpkg function
  - Support layer renaming
  - Add road damage report export function
  - Add clearance analysis report export function
  - Add DXF and KML import and export functions
  - Add dashed line batch drawing mode
  - Supports lane line and road edge detection with shortcut keys to switch between semi-automatic and manual modes
  - Add custom template drawing method for cross walk and parking spot
  - Add line smoothing function
  - Support vector objects and the corresponding panoramic image window linkage jumping
  - Fix other bugs in the software

V2.0.1

- Added Modules and Features
  - Fix the problem of database storage
  - Optimize the field selection limit when import trajectory
  - Optimize the right click menu of the directory tree
  - Optimize the interface of vector export function
  - Optimize the emptying tips of the directory tree layers

V2.0.0

- Added Modules and Features
  - New project
    - Supports drag and drop point cloud to create a new project, and save as to the specified path
    - Supports for importing original projects of LiFuser-BP and LiGeo
    - Supports point cloud and panoramic data entering software for more than 90% of mobile measurement devices on the market in formats such as Trimble MX9, Leica Pegasus, Orbit Pos, etc.
  - Database
    - Supports database for vector storage
    - Supports adding, deleting, modifying and querying layers in the database
  - Basic platform
    - Added point cloud and vector projection conversion function
    - Supports point cloud format conversion
    - Supports basic tools such as point cloud selection and cropping
    - Supports vector data format conversion
    - Supports category display 0-255 categories
    - Supports cross-sectional export of color orthophotos
    - Supports for mobile measurement device travel direction arrows and default view settings
  - Vector editing
    - The right-click menu has been added to the movement function, which supports moving in a specified distance and direction

        - Added arbitrary polygon, circle selection, ball selection, random selection, three-dimensional selection, and cylinder selection
        - Added polygon combination and split functions
        - Added one-to-many interrupt function
    - Symbol library and callout labels
        - Added feature symbolization function
            - Supports domestic standard road facility symbol library
            - Support for common symbol library
            - Supports symbol number storage
            - Support to set symbols for collected point features
            - Support for modification of symbol
        - Added annotation function
            - Support for adding annotation layers
            - Support for adding annotations to independent objects
            - Support for adding annotations to object nodes and edges
            - Support for adding annotations to 3D, elevation annotation
            - Supports annotation style modification
        - Added layer label function
            - Supports all layer properties for label display
            - Configurable styles for labels
            - Facilitate operations such as attribute entry
        - New vector line object direction display
    - Road feature extraction module
        - Added the modularization extraction function of strip features
        - Added monomerization segmentation function
        - Added monomerization editing function
        - Added monomerization parameter extraction function
        - Added AI-based road surface marking recognition function
        - Added batch processing function for road surface marking recognition
        - Added AI-based batch extraction of road facilities
    - Point cloud classification and extraction
        - Point cloud classification
            - Supports point cloud deep learning classification
            - Supports polygon-based point cloud classification
            - Supports point cloud classification based on vector lines
            - Supports air noise classification
            - Supports subsurface classification
            - Support above ground point classification
            - Support for separating lows
            - Supports outlier classification
            - Supports proximity point classification
            - Supports ground point classification
            - Supports classification by attribute
            - Supports classify by cluster size
        - Point cloud extraction
            - Supports extraction by category
            - Supports extraction by elevation
            - Supports extraction by intensity
            - Supports extraction by GPS time

        - Supports extraction by return number
    - Planar camera browsing measurement
        - Added planar camera data and point cloud overlay browsing display
        - Added planar camera measurement
        - Added planar camera calibration
        - Added planar camera undistort
    - Vehicle point cloud preprocessing
        - Supports trajectory segmentation
        - Supports laser placement inspection and calibration function
        - Supports trajectory quality check and repair
        - Added control point correction function
        - Added vehicle trajectory splicing function
    - Road analysis module
        - Added road damage detection function
        - Added headroom analysis function
        - Added visual field analysis function
        - Added road section analysis function
        - Added road cross section parameter extraction function

V1.2 - 10/03/2022

- New modules and features:
    - Added Facade Measurement Module
        - Supports vertical elevation
        - Supports horizontal reference line drawing
        - Supports range line drawing
        - Supports taking a section based on a reference/extent line and making adjustments
        - Supports the drawing of the basic elements of the facade
        - Supports for construction line drawing
        - Supports array drawing
        - Supports translation and rotation
        - Supports for exporting DXF and orthophotos
        - Added shortcut key configuration function
        - Added layer manager function
        - Added combined display function
        - Added cross drawing and vertical drawing functions
        - Added capture module
- Optimization function
    - Optimize the drawing method of rectangular traffic signs

V1.1 - 10/03/2022

- New modules and features:
    - Added data block module
        - Supports blocking by scale
        - Supports track segmentation
        - Supports area block node editing

- Supports area block merging
- Supports point cloud segmentation based on block
- Supports area block selective display
- Supports display and hide of area blocks and their labels
    - Added more road feature template symbol library
- Supports all types of arrow reticle drawing
- Supports for manhole covers, parking spaces, and drainage grate to be drawn according to templates
- Supports traffic signs to be drawn according to templates
- Supports Chinese and English character drawing
- Supports non-motor vehicle signs, no U-turn signs are drawn according to the template
    - Added template matching function
- Supports automatic vectorization matching based on default templates
- Supports custom feature templates and auto-vectorization or manual vectorization
    - Added multi-type vector editing function
- Supports for modifying the shape of line features
- Supports tracing other vectors when drawing vectors
- Supports point break line, line break line, line break surface
- Supports vector affixed to ground
- Supports precise selection when vector glands
    - Added attribute table calculation and predefined functions
- Added attribute batch filling function
- Added line feature length calculation function
- Added character replacement function
    - Added node editing function
- Support for individually modifying node coordinate values
- Support batch modification of node Z value
    - Added the function of modifying the project
    - Added layer vector copy function
- Supports copying of elements on the same layer
- Support vector geometry copy between different layers