Manual Revision

SATLAB GEOSOLUTIONS AB

Revision Date	Revision Level	Description
Dec.2025	V3.0	Sat-LiDAR User Manual

Sat-LiDAR User Manual

Contents

Software Introduction 10
1.1 Software Overview 10
1.2 Technical Parameters 11
1.3 Installation and Uninstallation 12
1.3.1 Software Installation 12
1.3.2 Uninstalling the software 13
1.3.3 Software Startup 13
Sat-LiDAR Quick Instruction 15
2.1 Software Interface 15
2.1.1 Welcome Interface 15
2.1.2 Main Interface 15
2.1.3 Interface elements 16
2.2 Standard Operation Procedure 28
2.2.1 New project 29
2.2.2 Original data Check 29
2.2.3 Strip edit 29
2.2.4 Flight route inspection 30
2.2.5 Project and point cloud combine setting 31
2.2.6 Data combine and optimization 32
2.2.7 Point cloud quality check 32
2.2.8 Section view check stratification 35
2.2.9 Trajectory Rectify 35
2.2.10 Precision check 37
2.2.11 Point cloud coloring 38
2.2.12 Point Cloud Processing 39
2.2.10 Generate terrain products 43
Description of the main functions 46
3.1 Files 46

3.1.1 New Project 46
3.1.2 Open Project 48
3.1.3 Close Project 49
3.1.4 Save Project 49
3.1.5 Open Files 49
3.1.6 Opening Workspace 50
3.1.7 Save Workspace 50
3.1.8 Save Workspace As 50
3.1.9 Close 50
3.1.10 System Settings 50
3.1.11 About 52
3.1.12 Exit 53
3.2 View 54
3.2.1 View Browsing 54
3.2.2 Rendering 56
3.2.3 Other 61
3.3 Aircraft 67
3.3.1 Auto Process 67
3.3.2 Trajectory Solving 76
3.3.3 Combine Setting 77
3.3.4 Begin Combine 82
3.3.5 Stop Combine 82
3.3.6 Strip Rectify 82
3.3.7 Point Cloud Smoothing 84
3.3.8 Point Cloud Colorize 84
3.3.9 Point Cloud Export 87
3.3.10 Related Control Points 88
3.3.11 Accuracy Inspection 89
3.3.12 Point Moving 94
3.3.13 Original Data Check 94

3.3.14 Flight Route Quality Check 95
3.3.15 Strip Overlap Analysis 96
3.3.16 Height Different Quality Check 97
3.3.17 Density Quality Check 98
3.3.18 Coverage Quality Check 99
3.3.19 Automatic Quality Check 100
3.3.20 Point Cloud Processing 101
3.3.21 Generate DOM 101
3.4 Tools 103
3.4.1 Measurement tools 103
3.4.2 Selection tools 107
3.4.3 Point selection tools 111
3.4.4 Other tools 114
3.5 Process 116
3.5.1 Section View 116
3.5.2 Point Cloud Tiling 116
3.5.3 Select Tiling Block 117
3.5.4 Automatic Classification 117
3.5.5 Reset Classify 118
3.5.6 Noise Filter 118
3.5.7 Classify Ground 118
3.5.8 Median Ground Point 119
3.5.9 Reset Classify Parameters 120
3.5.10 Partial Classification 120
3.5.11 Execute 120
3.5.12 Saving Classification Parameter 121
3.5.13 Manage Classification Template 122
3.5.14 Manual Registration 122
3.6 Terrain 123
3.6.1 Generate DEM 123

3.6.2 Generate DSM 124
3.6.3 Generate contour lines 124
3.6.4 Generate Elevation Points 125
3.6.6 Earthwork Calculations 127
3.7 Layout 129
3.7.1 Layer Management 129
3.7.2 Properties Panel 129
3.7.3 Toolbox 129
3.7.4 Information Output 129
3.7.5 Welcome Interface 129
3.8 Profile Selection 130
3.8.1 Section Display 130
3.8.2 Moving the profile down 131
3.8.3 Move Profile Up 131
3.8.4 Move Profile 131
3.8.5 Rotate Profile 131
3.8.6 Extend profile 132
3.8.7 TIN 132
3.8.8 Hide TIN 133
3.8.9 TIN Modes 133
3.8.10 TIN Rendering 133
3.8.11 Light-off 133
3.8.12 Manual Classification 133
3.9 Toolbox 137
3.9.1 Data Processing 137
3.9.2 Format Transform 142
3.10 Engineering Application 147
3.10.1 One-Click Processing 147
3.10.2 Reference Centerline 147
3.10.3 Generate Reference Horizontal Section 148

3.10.4 Generate Measured Vertical Section 149
3.10.5 Generate Measured Horizontal Section 149
3.10.6 Start Editing 151
3.10.7 Export Sections 151
3.10.8 Generate Section View 152
3.10.9 Delete 153
3.10.10 Section Method Earthwork Calculation 153
3.10.11 Multi-Period section Comparison 154
3.10.12 Lower left section tree control 155
3.10.13 Calculation of tunnel overshoot and undershoot 155
3.10.14 Calculation of cross-sectional flow area 156
3.10.15 Cross-section sedimentation analysis 158
3.11 Draw 159
3.11.1 View the properties 160
3.11.2 Add Point 160
3.11.3 Add Line 160
3.11.4 Add Rectangle 160
3.11.5 Add Arbitrary Rectangle 160
3.11.6 Add Polygon 161
3.11.7 Add Circle 161
3.11.8 Delete 161
3.11.9 Line Break 161
3.11.10 Two-Point interruption 161
3.11.11 Connect 161
3.11.12 Copy 162
3.11.13 Extend 162
3.11.14 Smooth 162
3.11.15 Add Arc 162
3.12 Architectural 162
3.12.1 Point Cloud Slice 162

3.12.2 Export 163
3.13 SLAM 164
3.13.1 SLAM Processing 164
3.13.2 Sampling Points Align 166
3.13.3 Coloring 167
3.13.4 Point Cloud Denoising 167
3.13.5 Manual denoising 168
3.13.6 Collection Point 168
3.13.7 Point Cloud Export 169
3.13.8 Related Control Points 170
3.13.9 Accuracy Inspection 171
3.13.10 Earthwork Calculations 177
3.14 3DGS 180

Chapter 1

Software Introduction

This chapter contains:

- Software overview
- Technical parameters
- Installation and uninstallation
- About

Software Introduction

1.1 Software Overview

This software is Sat-LiDAR point cloud pre-processing and post-processing software, mainly for SatLab LiDAR solution, the function is to utilize the POS data solved by the combined navigation module of the mobile measurement system to convert the scanned point coordinates based on the equipment coordinate system in the original scanning file of the three-dimensional laser scanner to the geodetic coordinate system, and the output results include: the target point cloud file (Output formats include hlz, las, and xyz).

Sat-LiDAR software supports the accuracy verification function of point cloud data for users.

The software also provides comprehensive data quality checking functions: including original data checking, route quality checking, air strip overlap analysis, height difference quality checking, density checking, range checking, and support for exporting pdf format quality checking report.

At the same time, the software also supports point cloud tiling, point cloud classification, DEM generation, contour line generation, elevation point generation and other functions.

1.2 Technical Parameters

Operating system	Microsoft Windows10 (64-bit) Microsoft Windows11 (64-bit)
CPU	Intel@Core i7 or above; 6-core processor
Memory	Not less than 16GB
Graphics Card	recommended NVIDIA discrete graphics card, video memory not less than 2GB
Monitor resolution	1920*1080 and above
NET Environment	NET Framework Version 3.5

1.3 Installation and Uninstallation

1.3.1 Software Installation

1. Click on the Sat-LiDAR software installation package to bring up the installation screen shown below.

Figure 1-1 Software Installation Interface

2. Click Next, set the installation path (usually keep the default)
3. Click Next to install the software.
4. Click Finish to complete the software installation.

Figure 1-2 Successful Software Installation Interface

5. The program is successfully installed and the Sat-LiDAR program icon appears on the desktop.

Figure 1-3 Sat-LiDAR icon

Note: Turn off your firewall and antivirus software during installation to prevent the installation process from being interrupted by antivirus software;

1.3.2 Uninstalling the software

Method 1: Click the computer start menu, find Sat-LiDAR, click "Uninstall Sat-LiDAR", a dialog box will pop up whether to uninstall or not, click Next --- Finish to uninstall.

Method 2: Go to Control Panel -> Programs and Features, right-click to uninstall it.

1.3.3 Software Startup

After the software is installed, if the software is not authorized, the dialog box shown below will pop up when the software starts. After closing the dialog box, the software will exit automatically.

1. Software Dongle
2. License file

Copy the serial number from the Register ID then get license file from SatLab technical team.

Figure 1-4 Software License Register

Chapter 2

Sat-LiDAR quick instruction

This chapter contains:

- Software interface
- Simple workflow

Sat-LiDAR Quick Instruction

2.1 Software Interface

2.1.1 Welcome Interface

After starting the software, a welcome interface will pop up, which mainly introduces the new features and main functions of this version of the software.

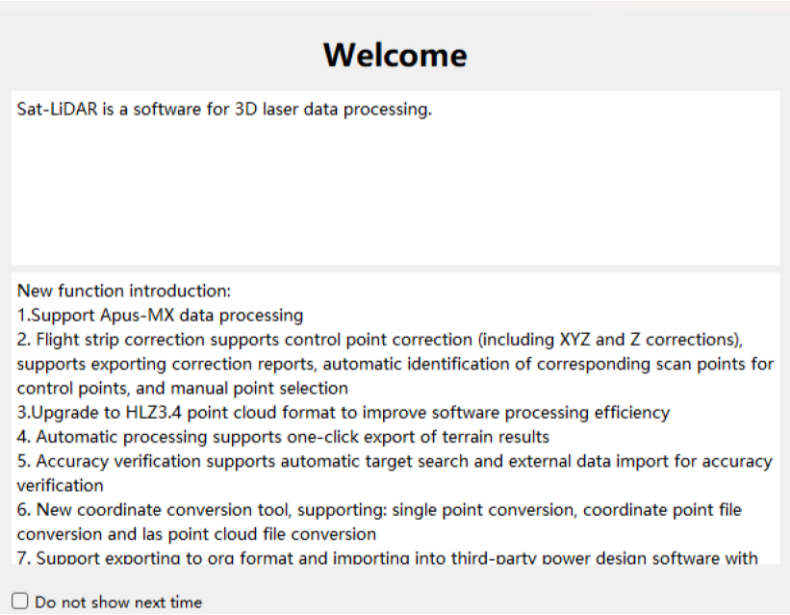

Figure 2-1 Welcome Interface

2.1.2 Main Interface

The Sat-LiDAR software mainly consists of a menu bar, toolbar, workspace, normal toolbar, data view, log output and view control area.

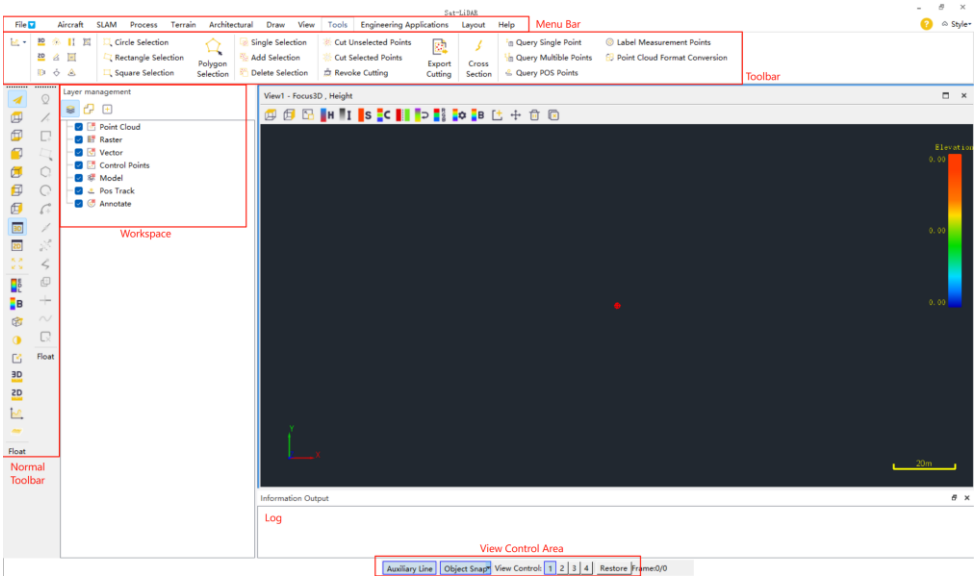

Figure 2-2 Sat-LiDAR Main Interface

2.1.3 Interface elements

Sat-LiDAR software is mainly divided into twelve menu modules:

<File>, <Aircraft>, <SLAM>, <Process>, <Terrain>, <Architectural>,<Draw>, <View>, <Tools>, <Engineering Applications>, <Layout> and <Help>.

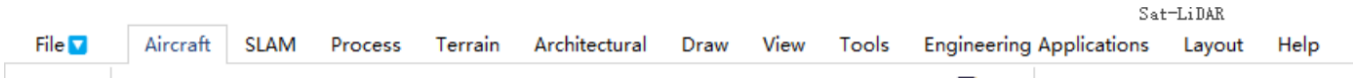

Figure 2-3 Sat-LiDAR Menu bar

2.1.3.1 File

The <File> menu mainly consists of the following options:

- New Project
- Open Project
- Save Project
- Close Project
- Open Files
- Open Work Area
- Save Work Area
- Save Work Area As
- Close Work Area
- Check the update
- System Setting
- About
- Exit
- List of the most recently opened files

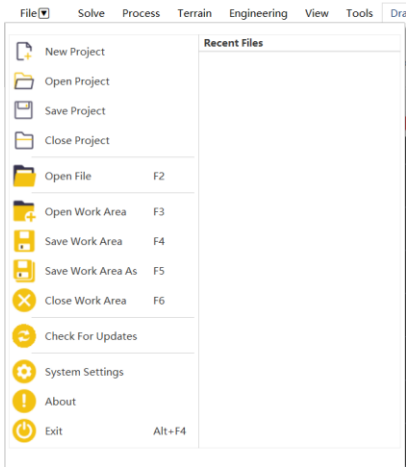

Figure 2-4 Sat-LiDAR File menu

2.1.3.2 View

The <View> menu controls the main view browsing display, including browsing and rendering options.

1. Browsing Functions:

- Panoramic browsing
- Flying browsing
- Setting the center of rotation
- Sync to view
- Roll-up
- Track Roaming
- View linkage
- View screenshots
- Setting the view to show all
- Full screen display
- Top view
- Bottom view
- Left view
- Right view
- Front view
- Back view
- 2D view
- 3D view

2. Rendering Functions:

- Set different rendering methods for the point cloud
- Adjust transparency display
- Adjust point size

3. Rendering Methods:

- Elevation rendering
- Strength rendering
- Monochromatic rendering
- Category rendering
- Airbelt rendering
- Echo rendering
- RGB rendering
- RGB rendering

- Circulating elevation rendering
- EDL rendering
- Mixed rendering

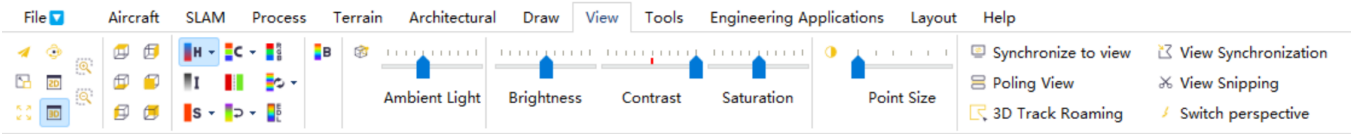

Figure 2-5 Sat-LiDAR View menu

2.1.3.3 Aircraft

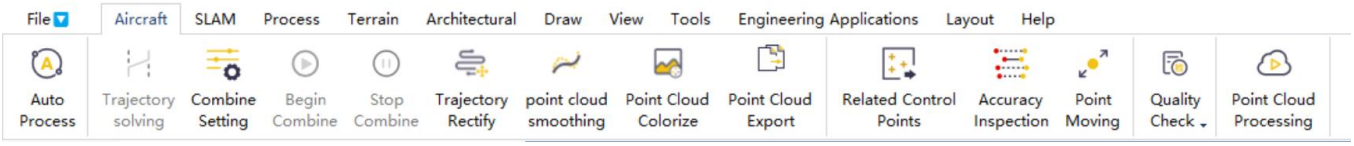

Figure 2-6 Sat-LiDAR Aircraft menu

1. Data Combine:

- Combine setting and data combining for selected POS projects or point cloud files
- Start/stop combining for selected point clouds through the Start Combine/Stop Combine menu item

2. Trajectory Rectify:

- One-click strip adjustment

3. Point Cloud Smoothing:

- Optimize point cloud

4. Point Cloud Coloring:

- Perform coloring operations on the selected point cloud

5. Accuracy Inspection:

- Verify the data absolute accuracy via imported control points.

6. Quality Check:

- Data integrity check
- Route quality check
- Strip overlap check
- Height difference quality check
- Density check
- Coverage Quality check
- Automatic quality checks

2.1.3.4 Tool

The <Tool> menu includes point cloud measurement, point cloud cutting and export, query, and format conversion.

1. Point Cloud Measurement:
   - Measure 3D distance
   - Measure 2D distance
   - Measure density
   - Measure slope
   - Measure angle
   - Measure height
   - Measure area
   - Measure volume
   - Measure surface area
2. Point Cloud Selection:
   - Circle selection
   - Rectangle selection
   - Square selection
   - Polygon selection
3. Point Cloud Cutting and Export:
   - Cut Unselected Points
   - Cut Selected Points
   - Revoke Cutting
   - Export Cutting
4. Cross section
5. Query
   - Single-point query
   - multi-point query
   - POS point query
   - measurement point annotation operation
6. Point Cloud Format Conversion:
   - Convert point cloud format and version number according to user needs

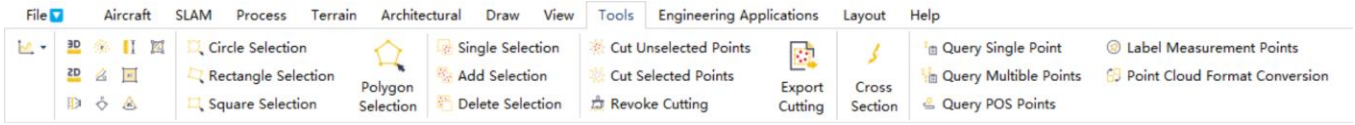

Figure 2-7 Sat-LiDAR Tool menu

2.1.3.5 Process

The <Process> menu includes point cloud split, point cloud registration and point cloud classification functions.

1. Point Cloud Split:
   - Split point clouds by customized block size.
   - Split by the customized number of blocks.
   - Split according to the scale of the plot.
   - Split according to vector formatting.
   - Choose output format and version as needed.
2. Point Cloud Registration:
   - Manual Registration.
   - Sketch Stitching.
3. Point Cloud Classification:
   - Reset classification.
   - Noise filtering.
   - Ground point classification.
   - Middle ground point classification.
   - Save and manage classification templates.

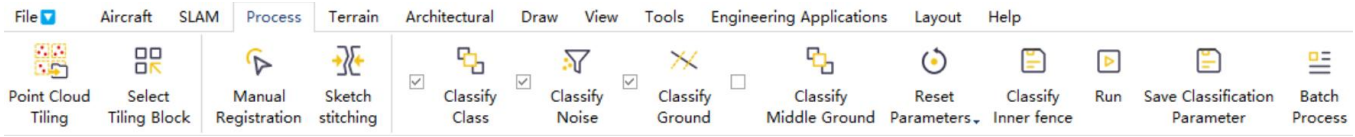

Figure 2-8 Sat-LiDAR Process menu

2.1.3.6 Terrain

The <Terrain> menu includes the following functions:

1. Generate DEM (Digital Elevation Model):
   - Generate .tif and .img format DEMs for engineering LAS point clouds by grid size.
2. Generate DSM (Digital Surface Model):
   - Generate .tif and .img format DSMs for engineering LAS point clouds by grid size.
3. Generate Contour Lines:
   - Generate contour lines in .dat, .txt, and .xyz formats for project LAS point clouds by grid size.
4. Generate Elevation Points:
   - Generate elevation points in .dat, .txt, and .xyz formats for engineering LAS point clouds by grid size.
5. Generate Power Point
   - Support the generation of engineering HLZ and LAS point cloud according to the DEM sampling interval and section height difference. Power point file in the org format.

6. Earthwork Calculation:

- Generate single-phase or multi-phase fill and excavation data for project LAS point clouds by grid size.

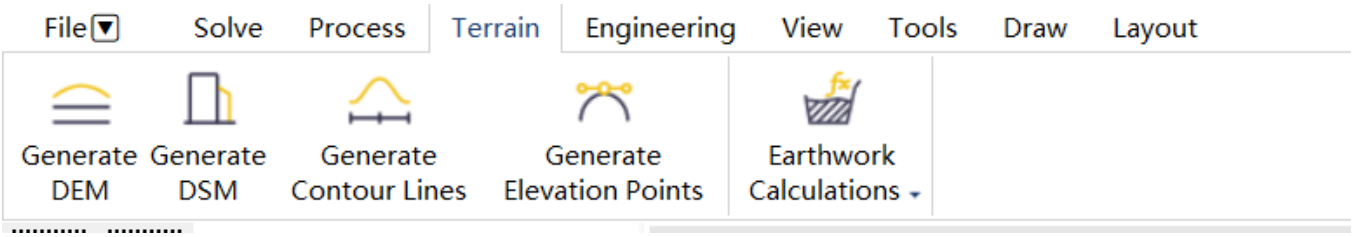

Figure 2-9 Sat-LiDAR Terrain menu

2.1.3.7 Layout

The <Layout> menu is used to control the showing/hiding of the panels, views and elements in the views. The panels included in the software are:

- Layer Management
- Properties Panel
- Toolbox
- Information Output (Console)
- Show/Hide of Welcome Screen

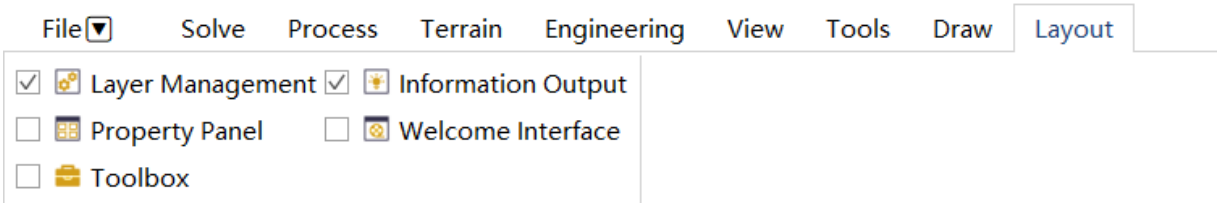

Figure 2-10 Sat-LiDAR Layout menu

2.1.3.8 Engineering Applications

- Generates reference horizontal section
- Generate vertical section
- Generate horizontal section
- Measured longitudinal section
- Edit the sections
- Export the sections
- Generate section view
- Support the section method for earthwork calculation
- Support for multi-period section comparison

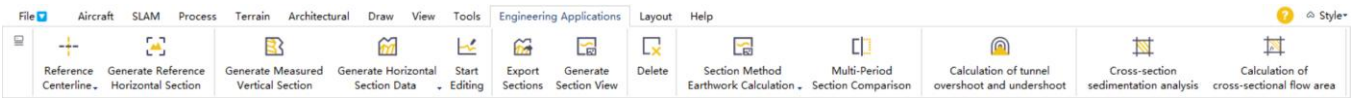

Figure 2-11 Sat-LiDAR Engineering menu

2.1.3.9 Draw

Single used to add all kinds of points, lines, and surface entities:

- Delete the entity
- Line interrupt
- Two points interrupt
- Connect
- Copy
- Extend
- Smooth
- Select layers and encoding properties
- View properties

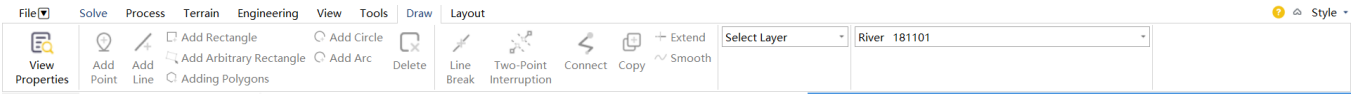

Figure 2-12 Sat-LiDAR Draw menu

2.1.3.10 Common toolbar

To facilitate user operations, commonly used tools are placed in the frequently used toolbar. By default, the toolbar is in a docked state.

- Float: Click the <Float> button to adjust the toolbar to a floating state.
- Restore: Close the floating window to restore the toolbar to the docked state.

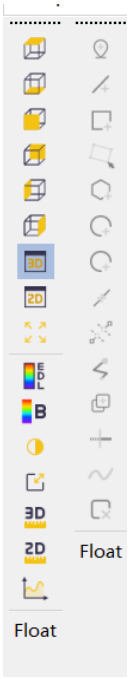

Figure 2-13 Common Toolbar in Docked State

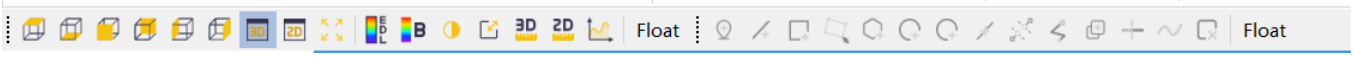

Figure 2-14 Common toolbars in the floating state

Table 2-1 List of Common Toolbar Functions

	Bottom view
	Front view
	Back view
	Left view
	3D view
	2D view
	Full screen
	EDL rendering
	Turn on the elevation + intensity mixed rendering
	Display Transparency
	Intensity strength
	Measure 3D distance

2D	Measure 2D distance
	Section view

2.1.3.11 Workspace

The workspace catalog is categorized according to layers, including:

- Point cloud
- Raster
- Vector
- Control point
- Model
- POS trajectory
- Annotation layers

When loading data, it will be linked to the corresponding layer based on the data type. For example, loaded LAS data will be linked to the point cloud layer.

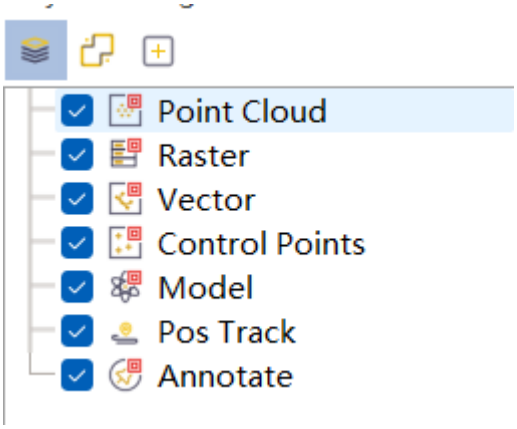

Figure 2 -15 Workspace

2.1.3.12 Data View

The data view is the main area of the software. It includes:

- Multi view
- Profile view
- 2D view

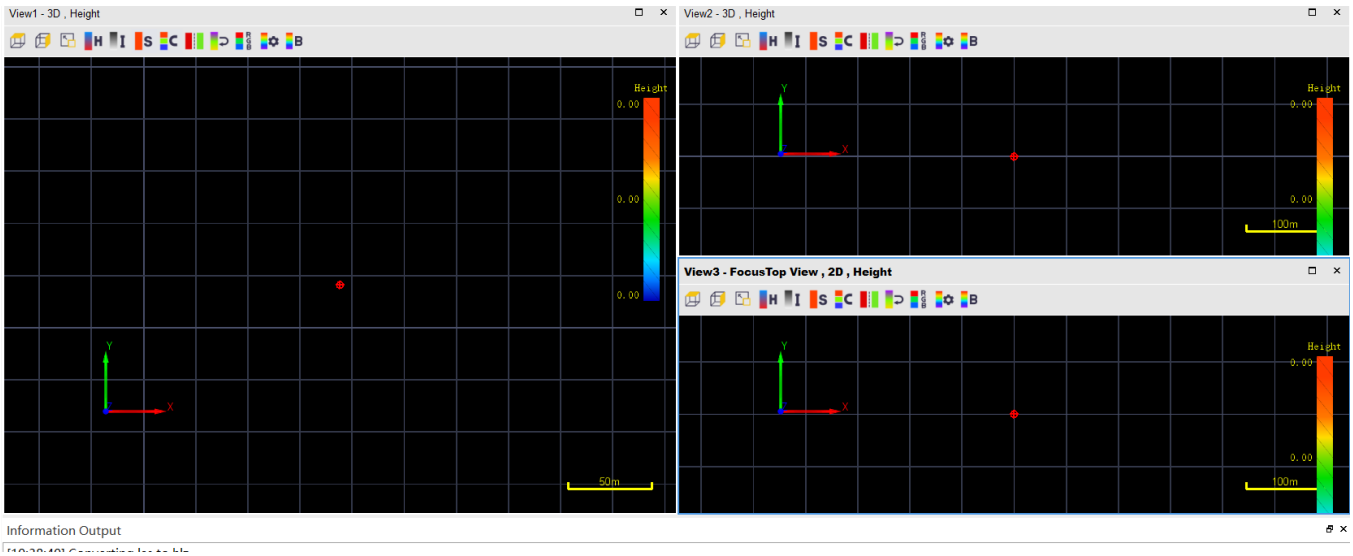

Figure 2-16 Multi-view and sectional view

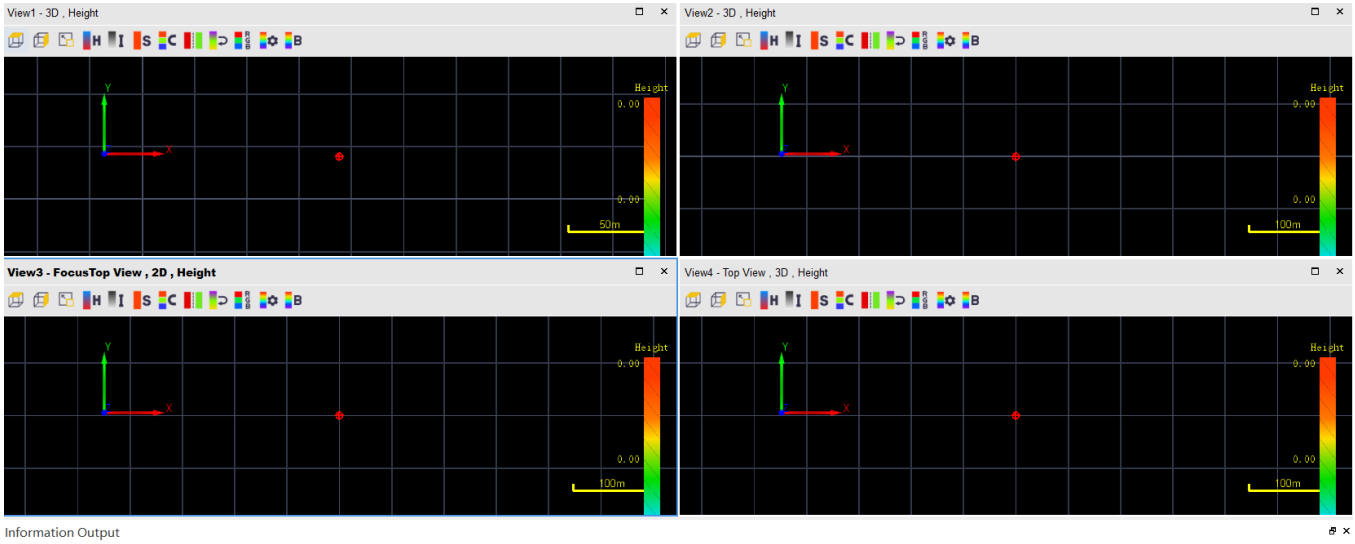

Figure 2-17 Multi-view control bar

The title bar of each multi-view window consists of view basic attribute information and common tools.

1. Attribute Information:

- View name (e.g., view 1)
- View angle (e.g., top view)
- 2D and 3D display (e.g., 3D view)
- View rendering method (e.g., elevation rendering)

2. Common Tools:

Refer to Table 2-2 for a detailed list of commonly used tools.

Figure 2-18 Multi-View Window Title Bar

Table 2-2 List of Tool Functions in Multi-View Title Bar

	Top view
	Right view
	Zoom Full View
	Elevation Rendering
	Intensity Rendering
	Monochrome Rendering
	Classification Rendering
	Zonal Rendering
	Echo Rendering
	RGB Rendering
	Classify Render Settings

3. Attribute Panel:

Point cloud data information including:

- File location
- Number of points
- Enclosing box size
- Maximum and minimum value points
- Transform parameters
- Format version
- Generation time

Property Panel

Property Name	Property Value
File Location	F:\Satlidar-\Power-Demo\Color-powerdemo.hlz
Count of Points	30397909
Bounding Box(d...	(223.824982, 268.220993, 50.842995)
Mininum(x,y,z)	x: 675159.910004 y: 1187201.209000 z: -14.288002
Maxinum(x,y,z)	x: 675383.734985 y: 1187469.429993 z: 36.554993
7 Parameters	Pitch: 0 Roll: 0 Yaw: 0 x: 675000.000000 y: 1187000.000000 z: -100.000000 Scale: 1
Format Version	HLZ 3.4
Creation Date	2024,10,22,

Figure 2 -19 Attribute Panel - Point Cloud Data

For raster data, including:

- File location
- Wavelength
- Base elevation
- Elevation range
- Pixel size

Property Panel

Property Name	Property Value
File Location	E:/APUS-data/APUS-testdata/1color_transf...
Wavelength	1
Basic Elevation	0
Elevation Ran...	Minimum: 635.939 Maximum: 709.148
Pixel Size	Width: 474 Height: 479

Figure 2-20 Attribute Panel - Raster Data

2.1.3.13 Toolbox

The toolbox includes some common tools, mainly including point cloud format conversion, point cloud to image, extraction, point cloud amplitude, data processing and other functions.

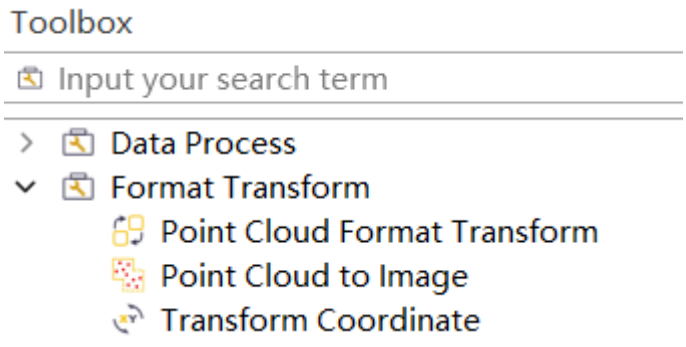

Figure 2-21 Toolbox

2.1.3.14 Information Output

The message output is used to display information about key steps in the data processing process, error messages and warning messages.

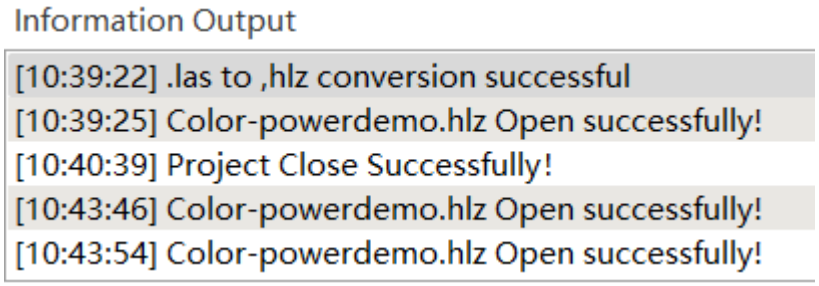

Figure 2-22 Console Output Panel

2.1.3.15 Status Bar

The main purpose of the status bar is to display alert messages, such as the progress of task execution. The process can be canceled at any time by clicking the <Cancel> button.

Figure 2-23 Progress Bar

2.2 Standard Operation Procedure

The following is a standard workflow for data process. For more detailed instructions, please refer to each chapter of the manual. This procedure is a solution we provide to the user. Once you become proficient with the software, you do not need to strictly follow this procedure. The usual process involves:

- Preparation: Ensure that all necessary data and files are ready and accessible.
- Project Setup: Create and configure a new project according to your needs.
- Data Import: Import the required data such as point clouds, raster images, or vector files into the workspace.
- Data Processing: Use the tools and functions available in the software to process the data, such as point cloud combining, classification, and rendering.
- Analysis: Perform analysis and quality checks on the processed data, including integrity checks,

accuracy checks, and elevation quality checks.

- Visualization: Adjust the view settings and rendering methods to visualize the data effectively.
- Export: Export the processed data and results in the desired formats and configurations.

2.2.1 New project

Creat new project <File> -> <New project>, input projecet name and select project folder.

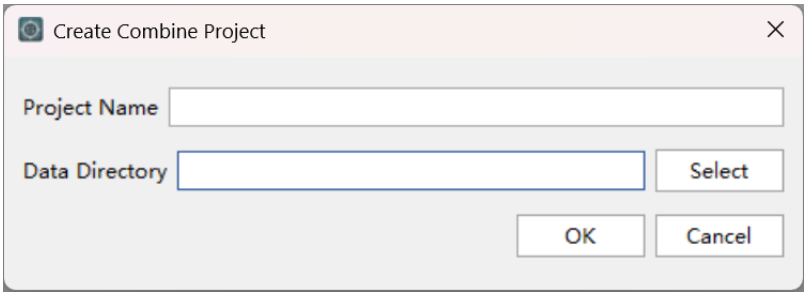

Figure 2-24 New Project

2.2.2 Original data Check

Click <Original data Check> under the <Quality Check> menu on the main interface. The original data quality check contains four items:

- Data integrity check
- Scanning data check
- Image data check
- POS data check

Abnormal items will be listed in the notes and reports for reference.

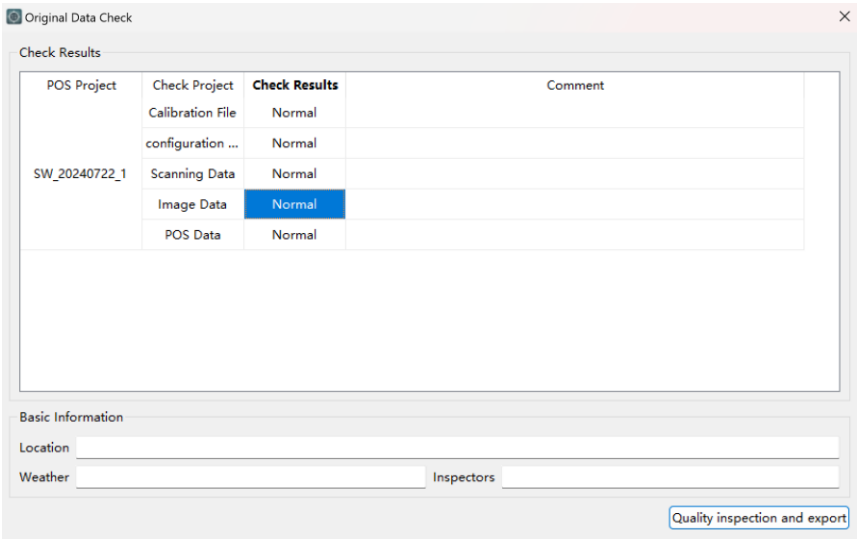

Figure 2-25 Original data Quality Check

2.2.3 Strip edit

Its main functions include

1. Drawing the POS cutting area
2. Cutting the POS track
3. Adding / deleting the cutting navigation belt
4. Cutting the POS track.

Figure 2-26 Airbelt Edit Panel

2.2.4 Flight route inspection

1. Click <Quality Inspection> menu under <Route Quality Inspection>.
2. Set the parameters according to the project requirements:
   - Design height
   - Design speed
3. Click Quality Inspection after completing the setup.

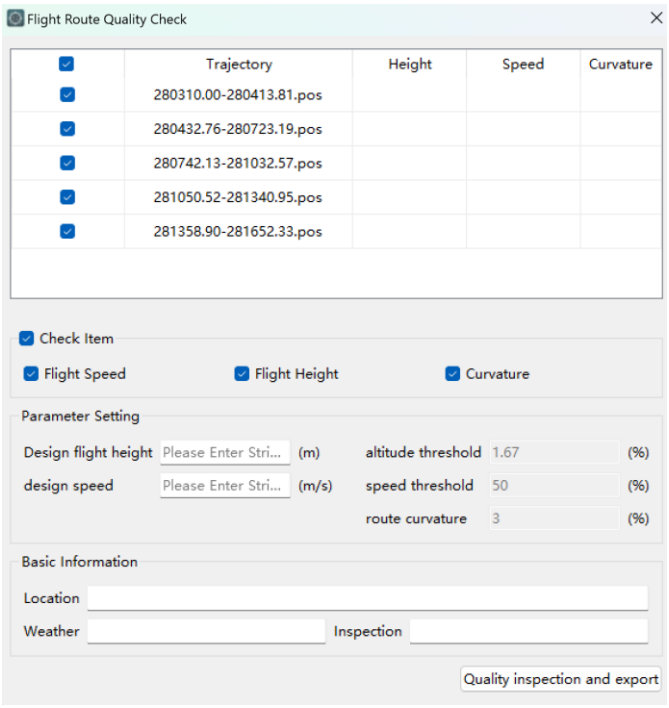

Figure 2-27 Flight Route Quality Check

2.2.5 Project and point cloud combine setting

1. Combine project setting

The setup parameters are applied to the entire POS project. In this dialog box, users can:

- Check the format and version of the output point cloud.
- Set the coordinate conversion parameters.
- Denoise (select according to the scene whether has power line or not)
- Point cloud optimization

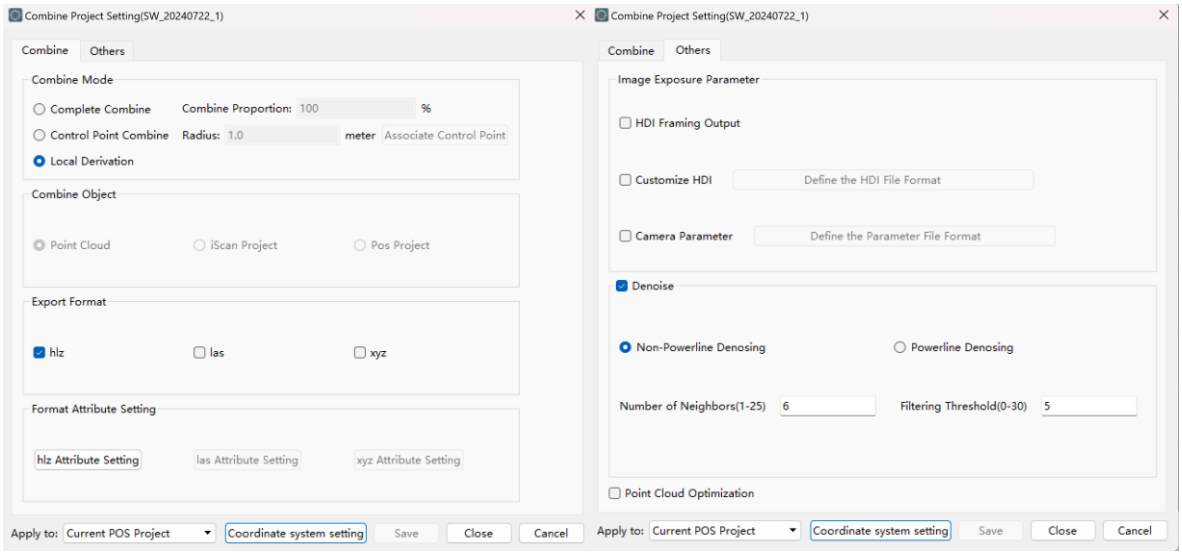

Figure 2-28 Project Combine Settings

2. Point cloud combine setting

Select the FOV of LiDAR and advance setting of filtering parameter.

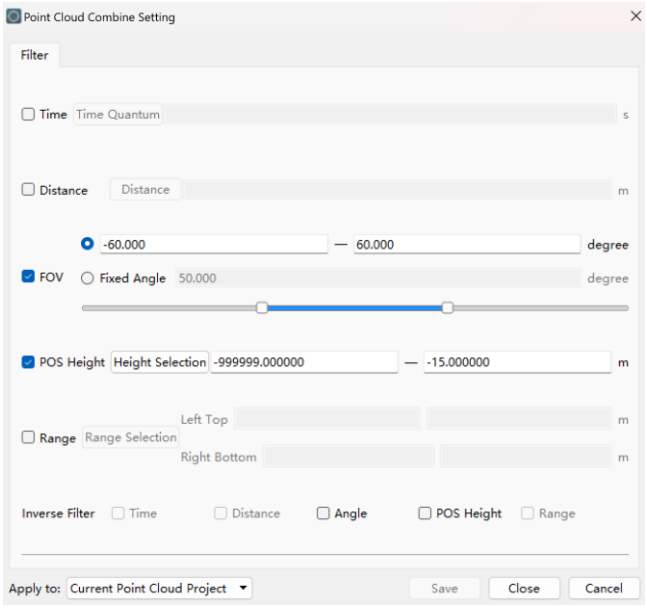

Figure 2-29 Point Cloud Combine Settings

2.2.6 Data combine and optimization

1. Begin combine
2. Trajectory rectify
   - One click auto strip adjustment processing to refine data.
3. Point cloud smoothing
   - Set parameter to smooth point cloud.

2.2.7 Point cloud quality check

After you get the point cloud, the quality check of the point cloud can be carried out if necessary.

2.2.7.1 Strip overlap check

1. The project management panel opens the fusion project and loads the navigation band
2. Click the "navigation band overlap check" under the menu on the main interface
3. The system will automatically recommend the grid size and overlap rate threshold parameters according to the data, and the user can set the parameters according to the project requirements
4. Click to start checking after the setting is complete

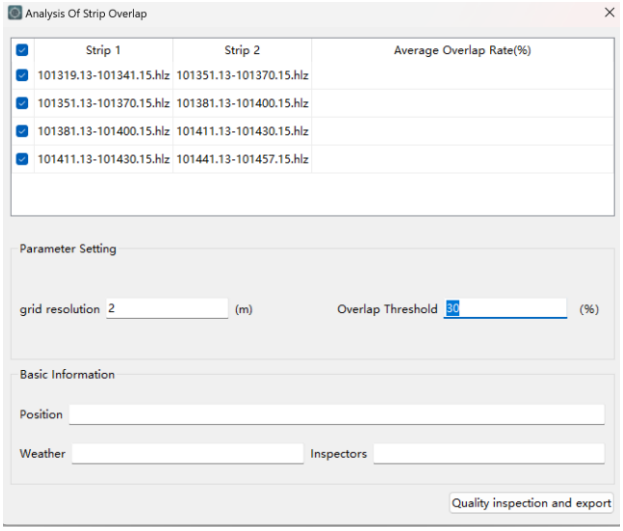

Figure 2-30 Overlap Inspection

2.2.7.2 Height difference Quality Check

1. The Engineering management panel opens on the fusion project
2. Load the display point cloud file
3. Click the menu on the main interface, and the "high difference quality check" dialog box pop up. By default, the size of the grid is 2 meters, and the critical threshold is 0.5 meters. Fill in the location of the test area, weather conditions and inspectors
4. Click the button [quality check and export], the program will check the high difference quality and export the report

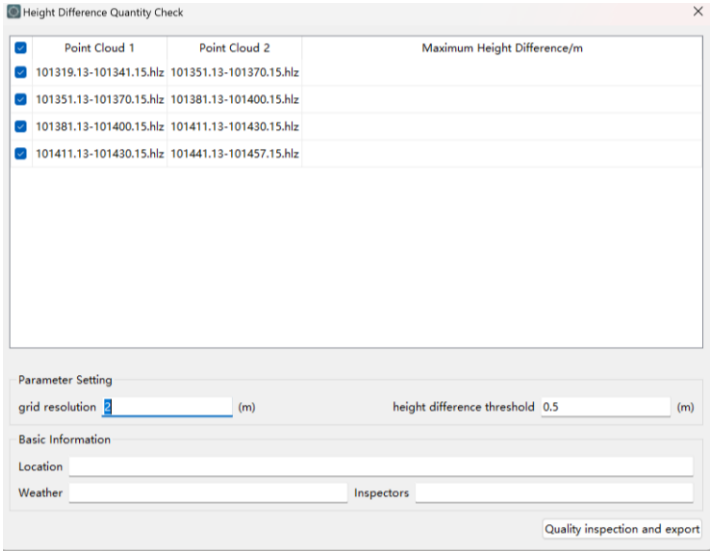

Figure 2-31 Height Difference Quality check

2.2.7.3 Density Quality Check:

1. The project management panel opens the fusion project and loads the display point cloud file

2. Click "Density Quality Check" under the "Quality Inspection> menu on the main interface, and the" Density Quality Check " dialog box pop up. By default, the size of the grid is 2 meters, and the average density threshold is 10 points / square meter. Point cloud files can be added and removed according to the project requirements

3. Click the quality check and export, the program will check the density quality and export the corresponding report

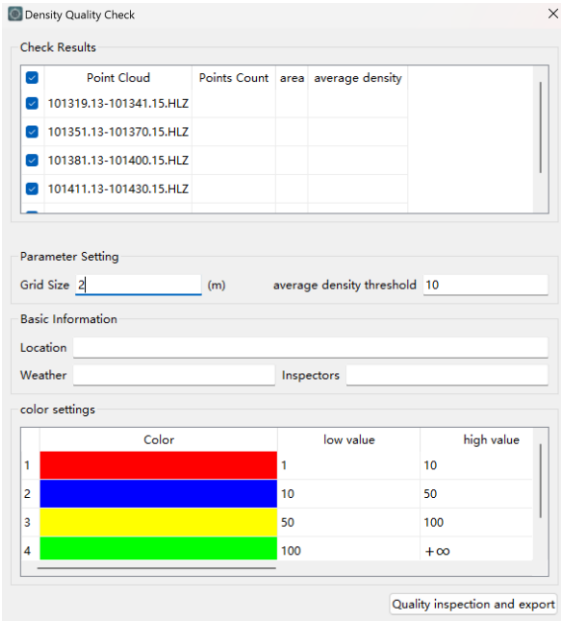

Figure 2-32 Density Quality Check

2.2.7.4 Coverage inspection

1.Open Project

In the Project Management panel, open the fusion project and load the point cloud.

2.Start Coverage Check

Click Aircraft> Quality Check > Coverage Check on the main menu. The Coverage Check window will appear.

3.Set Parameters

Choose a grid size (e.g., 1m x 1m). Import the boundary file (.shp, .dxf, or .dwg) that outlines the survey area.

4. (Optional) Add Info, You can fill in optional fields:

Survey location, Weather conditions, Inspector name, Run and Export

5.Click Run Check and Export.

The software will check the coverage and create a PDF report. After it's done, the folder with the report will open automatically.

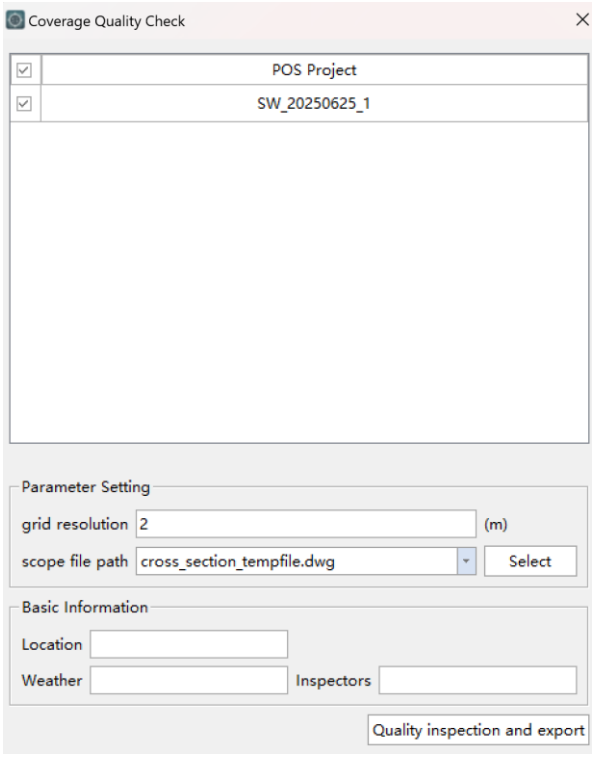

Figure 2-33 Completion rate inspection

2.2.8 Section view check stratification

1. On the main interface, click the menu <Tool> ->Vertical section

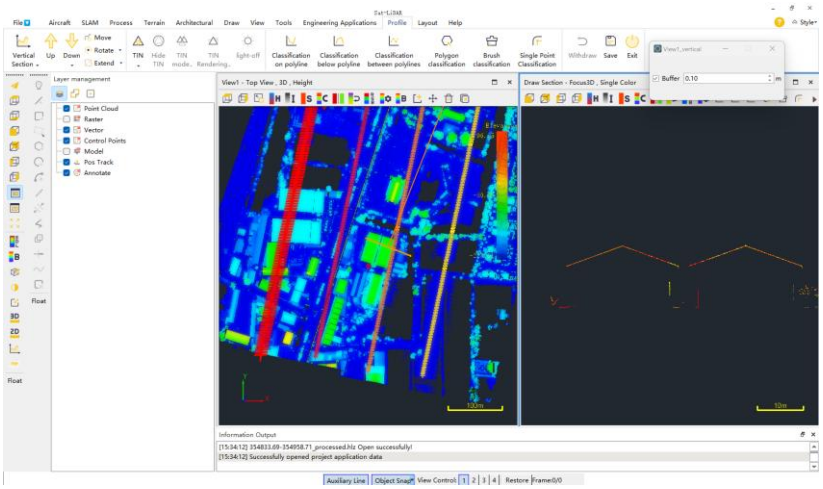

Figure 2-34 Section view

2.2.9 Trajectory Rectify

This process refines the trajectory (POS data) to improve the consistency and overlap of point cloud data across sorties.

1. Open Aircraft Project

Launch the Project Management Panel, open the relevant fusion project, and load the display point

cloud files.

2.Access Trajectory Rectification Tool

Go to the main menu and select Aircraft > Trajectory Rectify. The Trajectory Rectify dialog box will appear.

3.Select Sorties and Input Files

Check the sorties that require correction. For each sortie, specify the corresponding POS file and raw point cloud file.

4. Import Ground Control Points (Optional)

If target control points are available, import them. Enable Automatic Target Matching to automatically detect and match control targets in the scene.

5. Define Checkpoints (Optional)

In the Control Point List, right-click any point to mark it as a Checkpoint, which will be excluded from the rectification calculation but used for accuracy assessment.

6. Execute Rectification

Click OK to start the trajectory correction process.

7. Review Outputs

Once rectification is completed, the adjusted point cloud is auto-loaded in the viewer. A detailed Trajectory Rectification Report is generated in the point cloud output directory. The report includes:

Overview of rectified point cloud quality

Minimum error, maximum error, and RMS (Root Mean Square) deviation

If XYZ control points are used, the report additionally includes:

Ground control coordinates and scan point coordinates (before and after rectification)

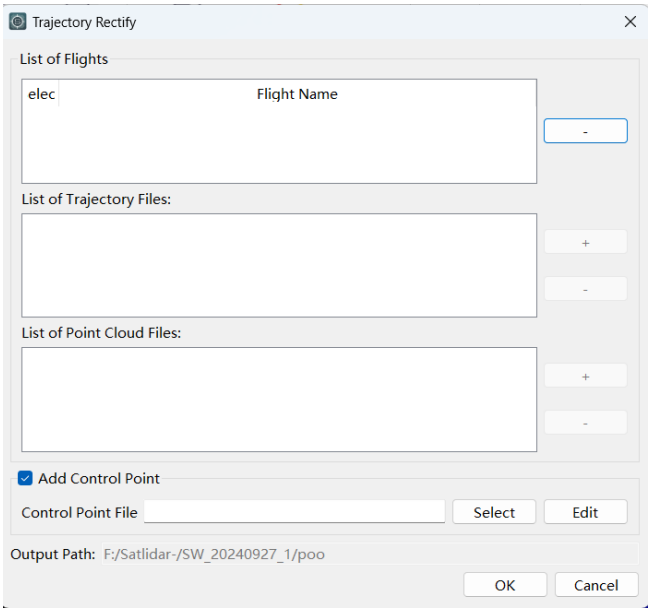

Figure 2-35 Trajectroy Rectify Interface

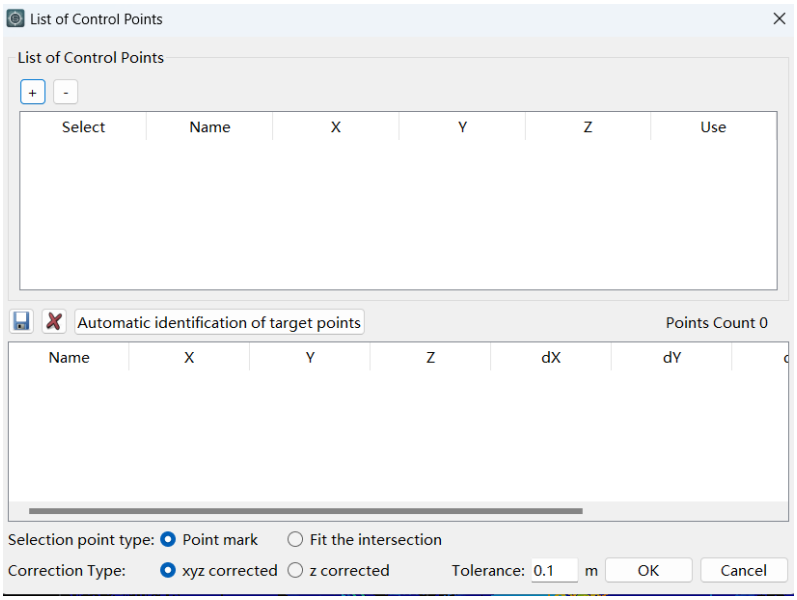

Figure 2-36 Automatically identify targets

2.2.10 Precision check

On the premise that the point cloud has been loaded and the point cloud control points are associated, enter the fast accuracy verification dialog box. Precision check provides two ways of manual point selection and automatic search for point selection.

1. Manually select points:

By selecting the scanning point to check with the control point, the precision check is obtained. Click fast accuracy verification, the page will call up the magnifying glass view for use. In each control point corresponding to match the scanning point, left click to confirm the selection point, scanning point, after success, control points in the list of the right point will fill the corresponding scanning point, if not satisfied with the point, can be relocated to the corresponding scanning point position, select the scanning point, or can manually modify the scanning point coordinates, according to the return confirmation. At this time, the selection of all scanning points is completed, click the accuracy check, and the system will automatically calculate the accuracy results of the current data according to the corresponding relationship between the control point and the scanning point.

2. Automatic search:

when click automatic search, the program first find the targets near the control point, the current support size is 50cm * 50cm target square board, through the algorithm automatically identify the target center position automatically identify the target center scanning point, such as no target punctuation is automatic search function is generally only applicable to the elevation type of point. Click the automatic search, the program will automatically detect the coordinate position of the scanning point

corresponding to the control point.

2.2.11 Point cloud coloring

Two method to colorize:

- Single Image: Click <Point Cloud Coloring> through the menu bar, the default is the single image coloring TAB page, and the system will automatically load the point cloud path and output path. Check the color point cloud, click OK, will be checked the point cloud for single image color
- Ortho Image：Orthoactive coloring: click <Point Cloud Coloring> through the menu bar, default to the single image coloring TAB page, switch to the orthoactive coloring module, the system will automatically load the point cloud path and other data paths. If it is inconsistent with the requirements, it can be manually selected. Check the selected colored point cloud, select the image TIF file, and select the output path, and the checked point cloud will be positively colored.

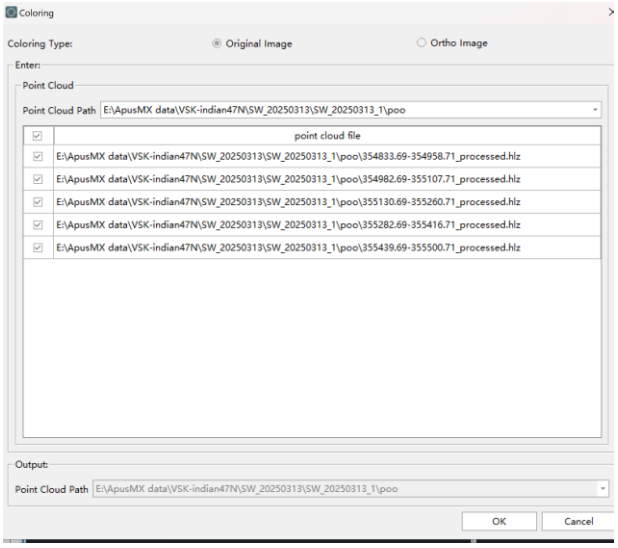

Figure 2-37 Single Image

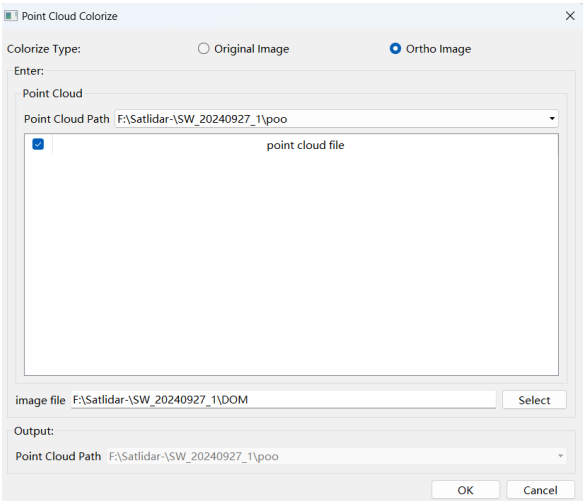

Figure 2-38 Ortho Image

2.2.12 Point Cloud Processing

Click <Point Cloud Processing> and the point cloud data will be loaded to the <Process> module.

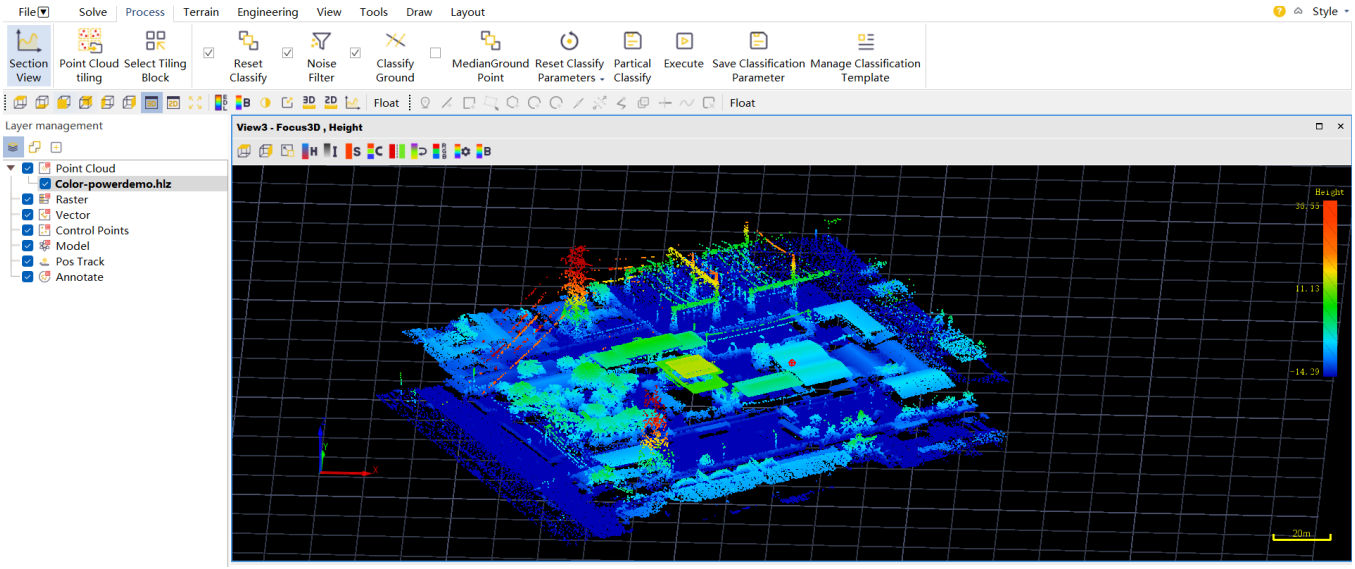

Figure 2-39 Point Cloud Processing

2.2.12.1 Point Cloud tiling

Split the point cloud according to specific criteria, generates tile vector and extracts each small piece of the point cloud for separate processing. This will speed up the processing efficiency.

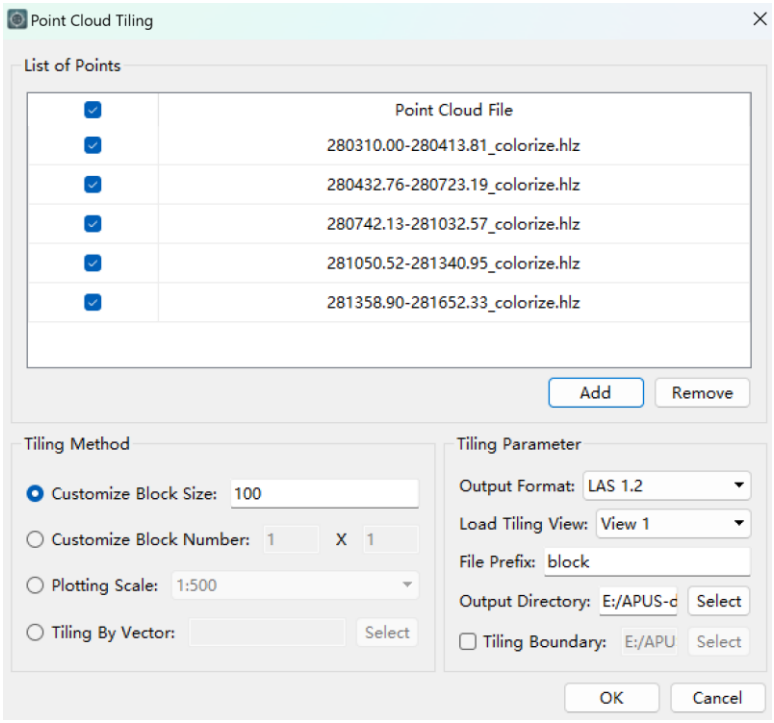

Figure 2-40 Point Cloud Tiling

2.2.12.2 Point cloud Classify

1. Reset Classify

Classify by class, the default is all classes classify to <1-Unclassify>.

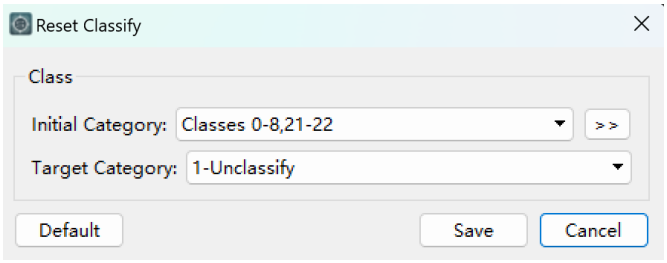

Figure 2-41 Reset Classify

2. Noise Filter

Classify noise points including SOR filter, Outlier filter and low points fiter.

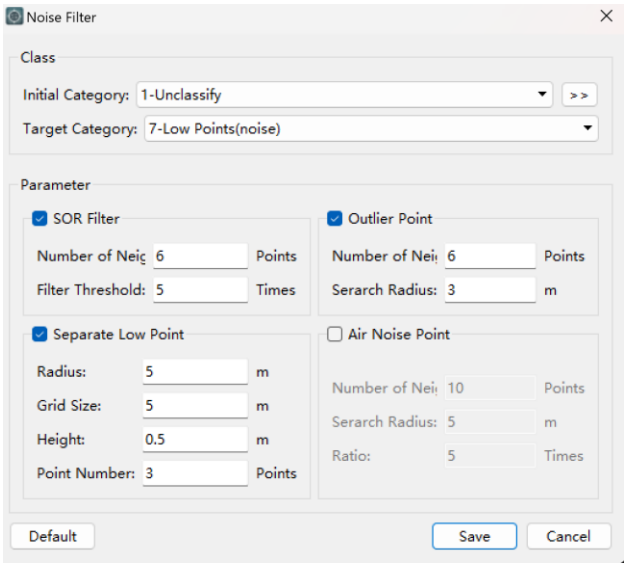

Figure 2-42 Noise Filter

3. Classify Ground

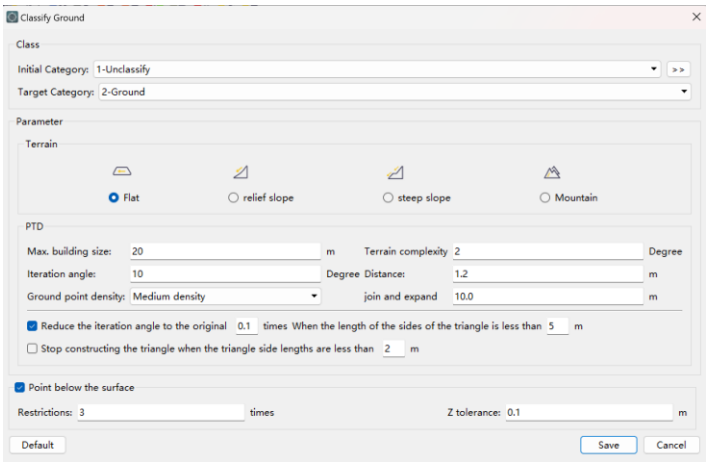

Figure 2-43 Ground point classification

4. Median point of the ground

Ground Median Point is not checked by default.

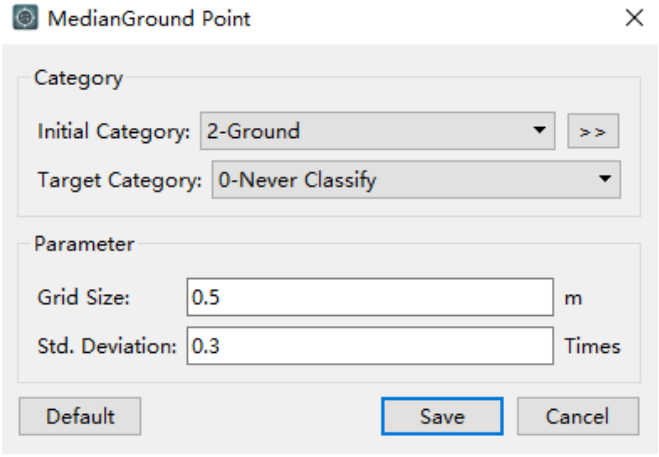

Figure 2-44 Classify Median Ground point

5. Execute classification

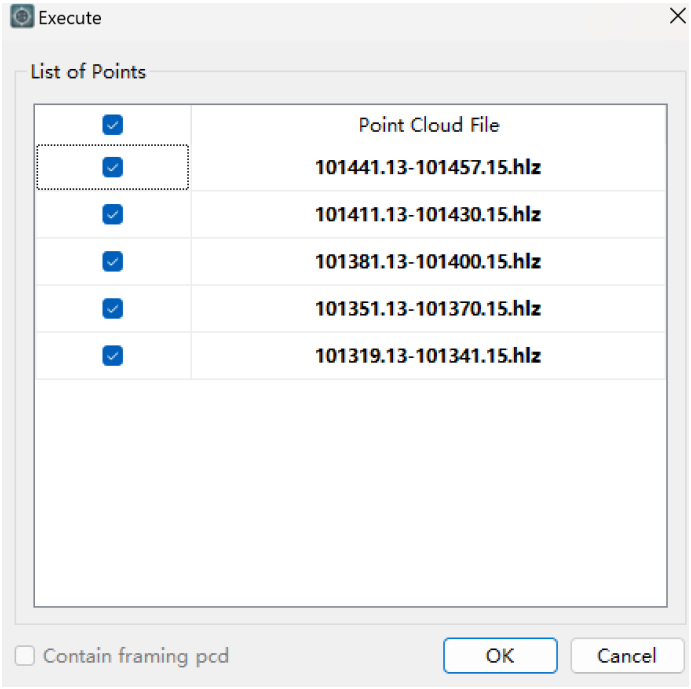

Figure 2-45 Median point of the ground

6. Partical Classify

Select specific area for ground point classification.

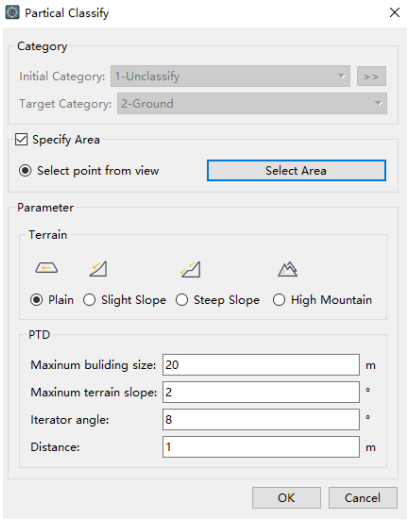

Figure 2-46 Partical Classify

7. Manual Classify

In the profile view, you can manually classify the points. Manual classification involves editing and modifying the categories to which the point cloud belongs. This includes:

- Classification above the folding Line: Classifying points above the drawn line.
- Classification under the folding Line: Classifying points below a drawn line.
- Classification between the folding Lines: Classifying points between two drawn lines.
- Classification of polygons: Classifying points within a defined polygonal area.
- Classification of brushes: Using a brush tool to classify points in a specific area.
- Classification of single Points: Classifying individual points manually.

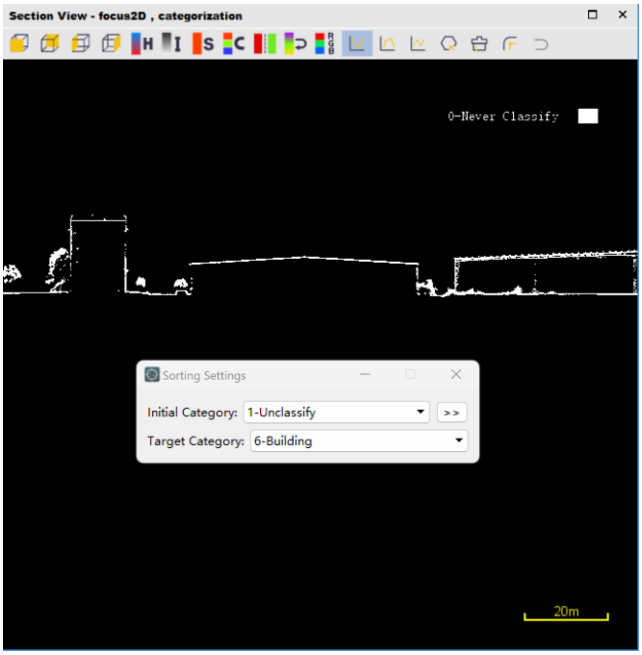

Figure 2-47 Manual Classify

2.2.10 Generate terrain products

1. Generate DEM：This function is mainly used to generate the relevant DEM model files locally.

- On the main screen, click the menu <Terrain> <Generate DEM>
- The dialog box can select the point cloud data that needs to generate the DEM model file
- Set the ground category and grid size of the generated model, where the resolution size is divided into three types: 1:500,1:1000 and 1:2000, corresponding to different grid sizes
- Set the format of export files, mainly tif and img
- Since this function can batch DEM model files for multiple point clouds, it is necessary to click merge to generate, select the folder to save the batch files, and click [OK] to start generation

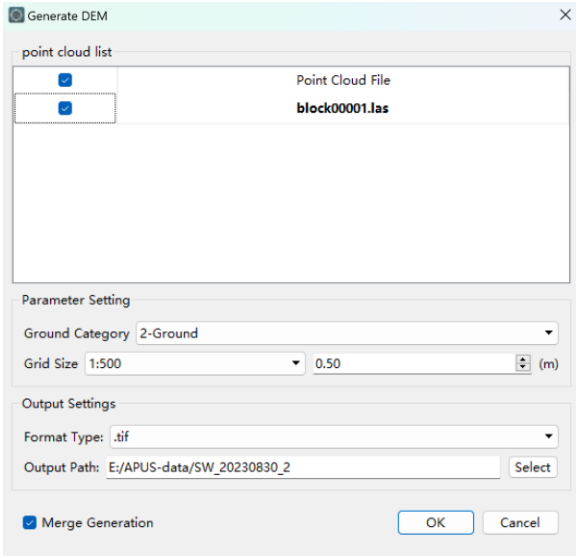

Figure 2-48 Generate DEM

2. Generate contour lines: This function is mainly used to generate relevant contour model files locally

- On the main screen, click menu <Terrain> <Build contour>
- In the dialog box, you can select the point cloud data required to generate contour model files, and select the terrain type that the point cloud data belongs to
- Set the ground point category, contour ratio and contour levels of the generated model, and the grid contour ratio is 1:500,1:1000 and 1:2000 respectively, corresponding to different contour levels
- Set the format of the export file, mainly in the dxf format. Since this function can batch generate contour model files for multiple point clouds, it is necessary to select the folder that saves the batch files
- Click [OK] to start the generation

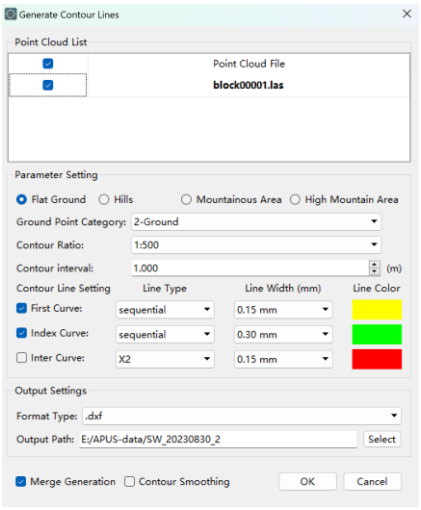

Figure 2-49 Generate contour lines

3. Generate Elevation Points: This function is mainly used to generate the relevant elevation point information text file locally, mainly including the numbering, x coordinate, y coordinate and the corresponding elevation value

- On the main screen, click menu <Terrain> →<Generate elevation Point>
- The dialog box selects the point cloud data required to generate the elevation point file
- Set the ground category and the grid size of the generated elevation point. The resolution size of the grid is divided into three types: 1:500,1:1000 and 1:2000, corresponding to different grid sizes
- Set the format of the export files, mainly in three text file formats: dat (Southern Cass), txt (NXYZ) and xyz (NXYZ). Since this function can batch generate files on multiple point clouds, you finally need to select the folder that saves the batch files
- Click [OK] to start the generation

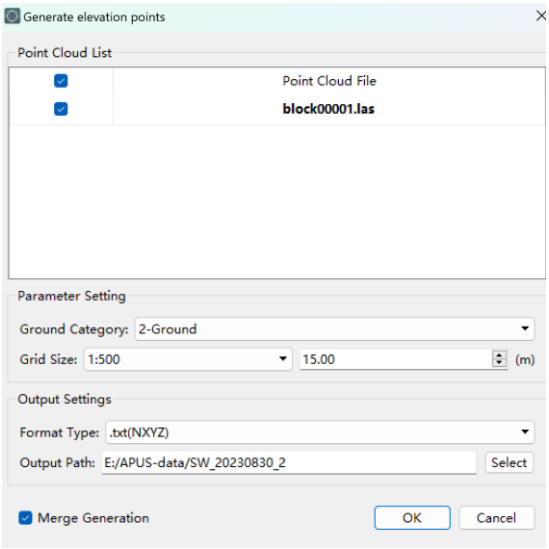

Figure 2-50 Generate Elevation Points

Chapter 3

Description of the main functions

Description of the main functions

3.1 Files

The project file (*.hdw) is a snapshot of the current workspace. When saved, it allows for quick reopening to recover the files associated with the current project data. Saving a project file only saves the file path and the data in the workspace, but not the point cloud and image data.

File Menu Options:

1. Open Data: Load new data into the workspace.
2. Open Workspace: Open a previously saved workspace (*.hdw).
3. Save Workspace: Save the current workspace state to a project file (*.hdw).
4. Save Workspace As: Save the current workspace state to a new project file (*.hdw).
5. Close Workspace: Close the current workspace.
6. System Settings: Adjust system settings for the software.
7. About: View information about the software.
8. Exit: Close the software.
9. Recent Files List: Access a list of recently opened files for quick reopening.

3.1.1 New Project

1. Open the New Project Menu:
   - In the main interface, click <File> → <New Project>.
2. Enter Project Name:
   - In the "Project Name" field (see Figure 3-1), enter the name for the new fusion project.
3. Select Data Catalog:
   - Click the [Select] button next to the "Data Catalog" item.
   - Choose the data directory that contains all the necessary data (typically a folder containing several iScan projects).
4. Create the Project:
   - Click the <OK> button.
   - A new fusion project will be created, and the project information will be saved in the specified data directory.

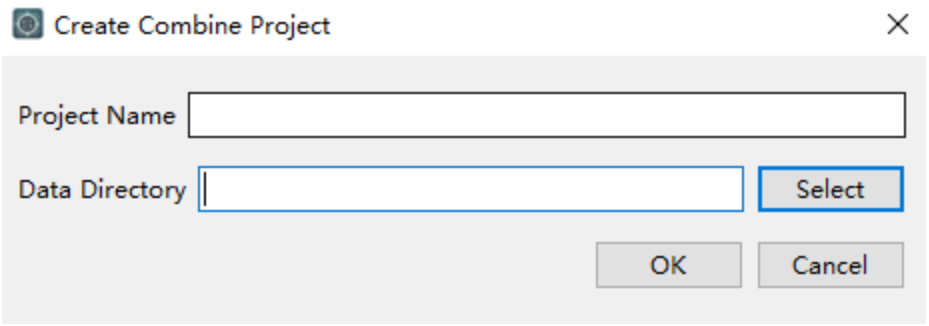

Figure 3-1 New Project dialog box

point out:

1. The Project directory shall contain all the scan projects. This example contains the scan data of "HD099_20230814_9". The project directory shall select a single scan directory or multiple scan directory, and the project file "1. hdc" is stored under "Scan directory".

2. The Project Catalogue should be the scanned project catalogue or its parent catalogue. This example fusion project contains only one scanning project, HD099_20230814_9 is both fusion project directory and scanning project directory, remember that "Data directory" should not be "below" <D: \ data \ S2 \ xingneng \ HI \ HD099_20230814_9> (must not select data directory as "001_Auto_01" level).

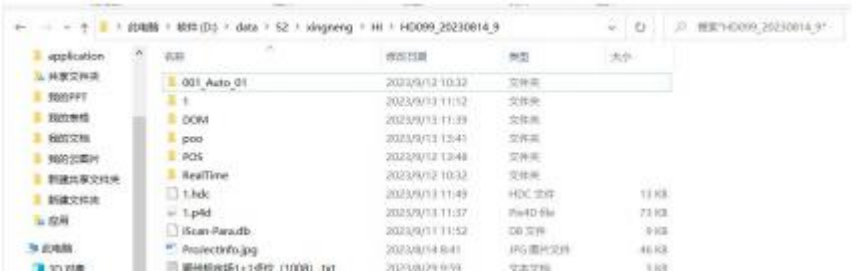

Figure 3-2 Data Catalogue

After the new project is completed, the program will automatically search for all qualified iScan projects in the selected directory, which is displayed in the tree list of the left file view.

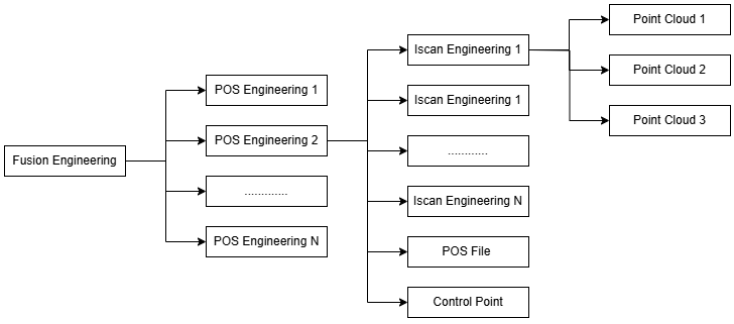

Figure 3-3 Schematic diagram of the fusion engineering structure

- The fusion project contains several POS projects, POS, which contains all the data collected by one section of POS
- POS, engineering contains several iScan, engineering, a POS file and a control point file (if required)

- The iScan project contains several point cloud data (generally no more than 3 data data)
- There are icon identification types before each node, including folders, point clouds, files and other types
- Each node is preceded by checkbox for each node and can be checked for fusion processing
- Each node can be right-button pop up the corresponding menu, the corresponding processing

Frequently asked questions:

1. The prompt information in the following figure is popped up, indicating that the trajectory configuration file iScan-Route.config format is not correct. Therefore, we need to check the parameters of the file, whether there are any assigned parameters or wrong parameters.

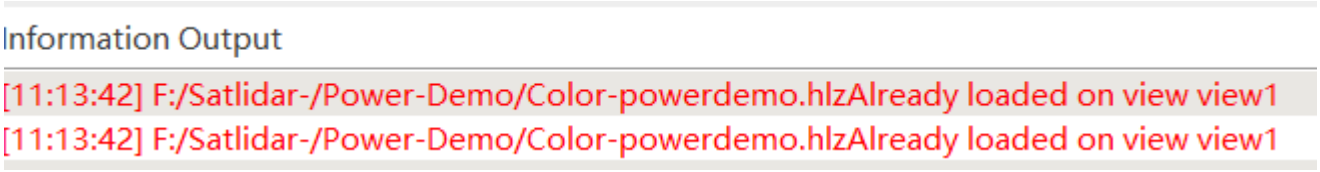

Figure 3-4 Error prompt box

3.1.2 Open Project

1. Open the Project Menu
   - In the main interface, click <File> → <Open Project>.
2. Select Project File
   - A dialogue box (as shown in Figure 3-5) will pop up.
   - Select the desired project file.
3. Open the Project
   - Click the <Open> button.
   - Once the combine project file is opened, the Project Manager's tree list will automatically display all the data associated with the combine project.

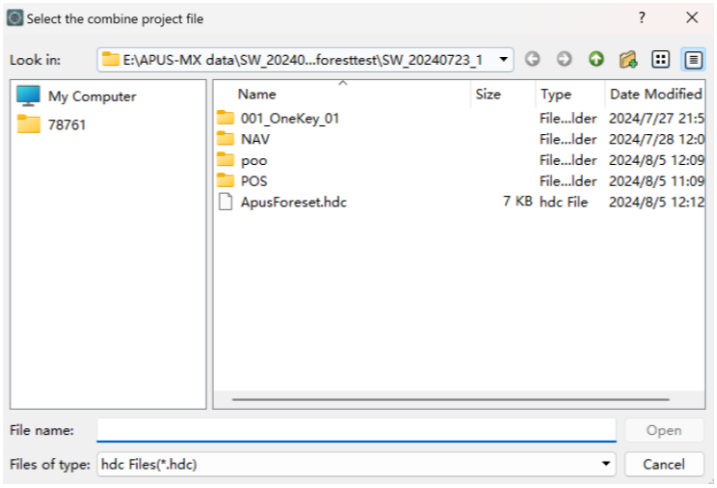

Figure 3-5 Open project dialog box

3.1.3 Close Project

1. Close the Project
   - In the main interface, click <File> → <Close Project> to close the current combine project.
2. Clear Display Data
   - After closing, the display data in each view will be cleared.
3. Save Project Confirmation
   - A dialog box (Figure 3-6) will pop up, prompting whether to save the project file.

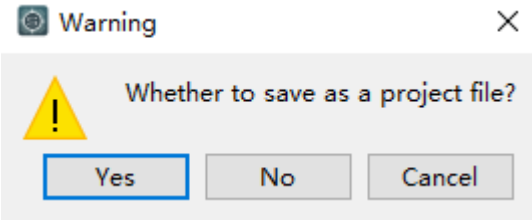

Figure 3-6 Close Project dialog box

3.1.4 Save Project

1. Save the Project
   - When the combine project changes (e.g., modifying the combine parameters through the combine setting dialog box), click <File> → <Save Project> in the main interface.
2. Confirm Save
   - The current state of the combine project, including any changes made, will be saved.

3.1.5 Open Files

Opening Files

1. Open the File Menu:
   - Click <File> → <Open Files> in the main interface.
2. Supported File Types:
   - The supported data types are: hlz, las, laz, xyz, txt, dxf, shp, tif, img, cpt, pos, ply.
3. Select and Open File:
   - Select the file that needs to be opened.
   - Double-click the file or click the <Open> button.
4. Load File:
   - The program will load the file under the corresponding layer node based on its format.
   - If a view is selected when loading the file, it will be loaded in that view.
   - If no view is selected, the system will automatically open a view and load the file.

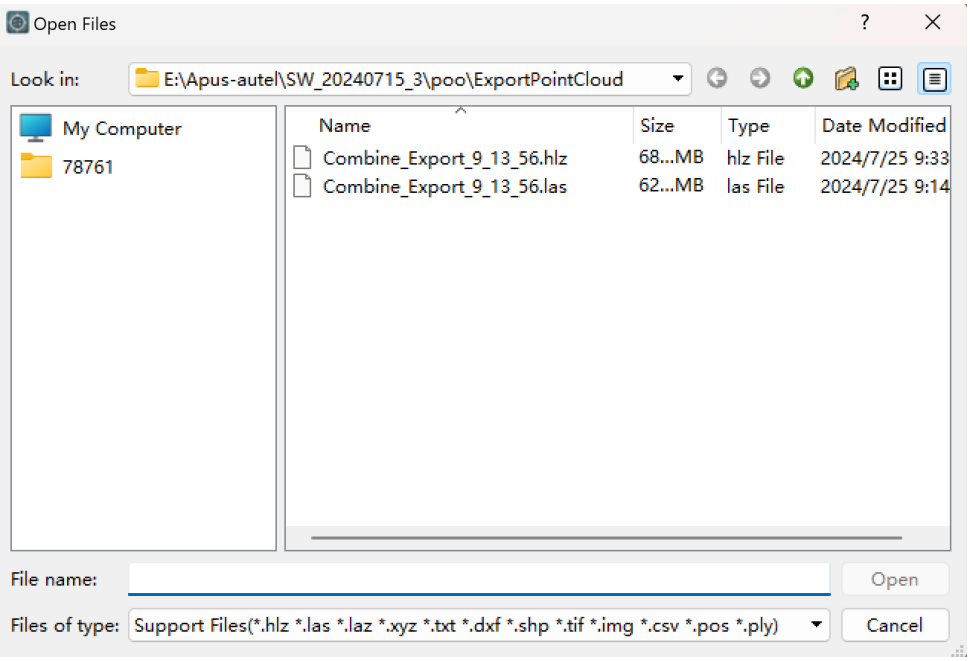

Figure 3-7 Open Files

3.1.6 Opening Workspace

Click <Open Workspace> to load an existing processing project file (*.hdw).

3.1.7 Save Workspace

By clicking <Save Workspace>, the software saves all the data of the current operation to the workspace.

3.1.8 Save Workspace As

Click <Save Workspace As>, the software will save the current workspace to another workspace file.

3.1.9 Close

By clicking <Close>, the software will close the current workspace, including the view display.

3.1.10 System Settings

1. Shortcut

- Defining keyboard shortcuts for common functions.

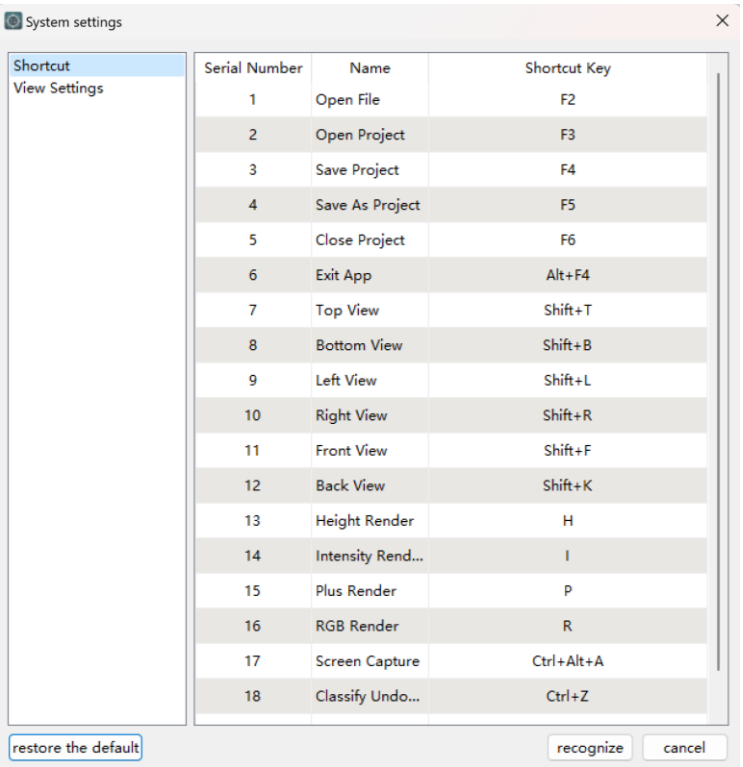

Figure 3-8 Shortcuts setting

2. Cloud platform account

In the case of networking, enter the user name and password, and click log in. After logging in to the cloud platform account, the user can check the cloud base station for the cloud base station calculation in the trajectory calculation interface. The cloud base station calculation supports the data of nearly three months for the calculation.

3. View Setting

The view settings are primarily for adjusting the main view interface elements and measurement tool properties. This is illustrated in Figure 3-10.

(1) Main View Interface Elements:

- Background Color: Modify the background color of the main view.
- Display Controls: Control the display and hiding of:
  - Grid
  - Axes
  - Legend
  - Scale

(2) Measurement Tool Attributes:

- Selection Colors:
  - Modify the selection point color.
  - Modify the selection line color.

- Line Width: Set the line width of the measurement line.
- Measurement Colors:
  - Modify the color of the measurement line.
  - Modify the color of the measurement surface.

(3) Object Capture Mode:

- Enable object capture mode for key points in the mapping tool.
- Automatically capture endpoints, intersection point, nearest point, and extension line in the plotted line.

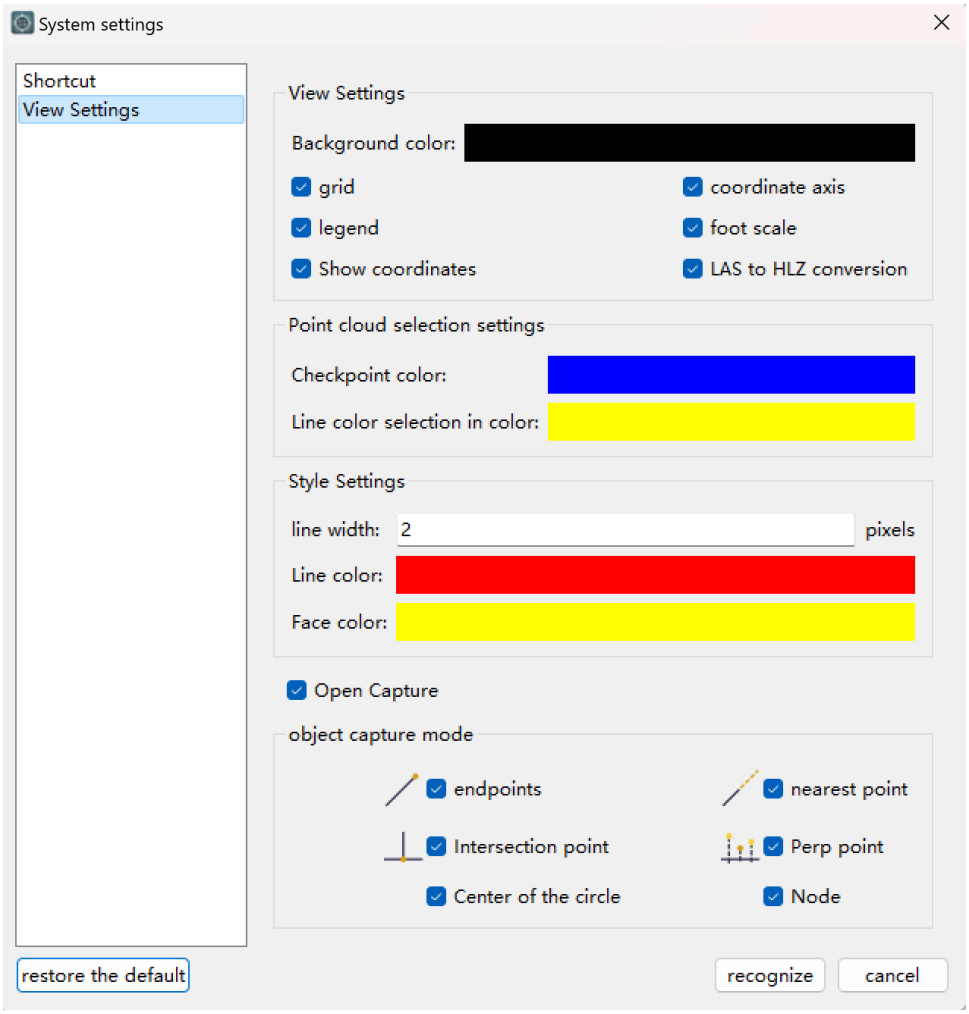

Figure 3-9 View Setting

3.1.11 About

Clicking the <About> button brings up a dialog box displaying the following information about the software:

- Software name
- Version number

- SDK version
- Build date
- Other necessary details

This information helps users identify the specific version and build of the software they are using.

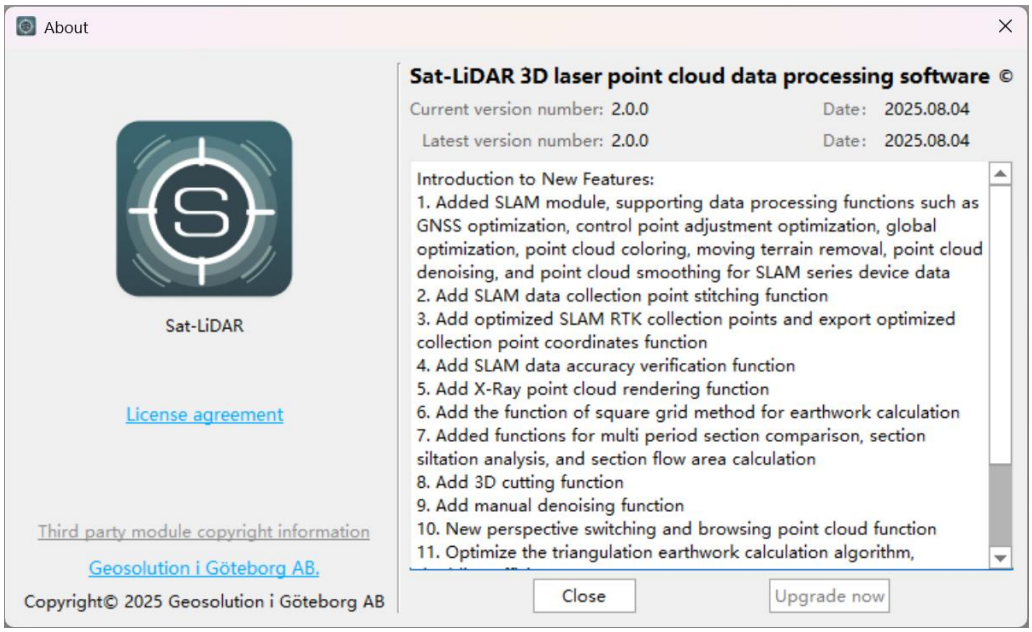

Figure 3-10 About

3.1.12 Exit

Clicking the <Exit> button will close the software.

3.2 View

The <View> menu controls the main view browsing display, including browsing, rendering, and other display functions.

3.2.1 View Browsing

1. Fly View

Click the <View>→<Fly view> to browse the point cloud in the main 3D view.

- View Rotation: hold down the left mouse button and move the mouse to rotate the view.
- Zoom In/Out: Slide the mouse wheel to zoom in and out of the view.
- View Panning: Hold down the right mouse button and move the mouse to drag the view .

The following tools are only available in fly view mode:

<Zoom screen>, <Top View>, <Bottom View>, <Left View>, <Right View>, <Front View>, <Back View>.

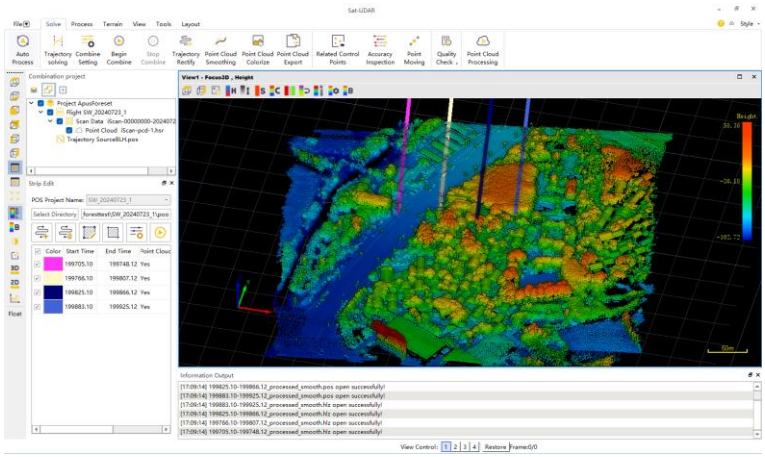

Figure 3-11 Fly View

2. Zoom Screen

Cick the <View>→<Zoom screen>, or click the button on the title bar of the view.

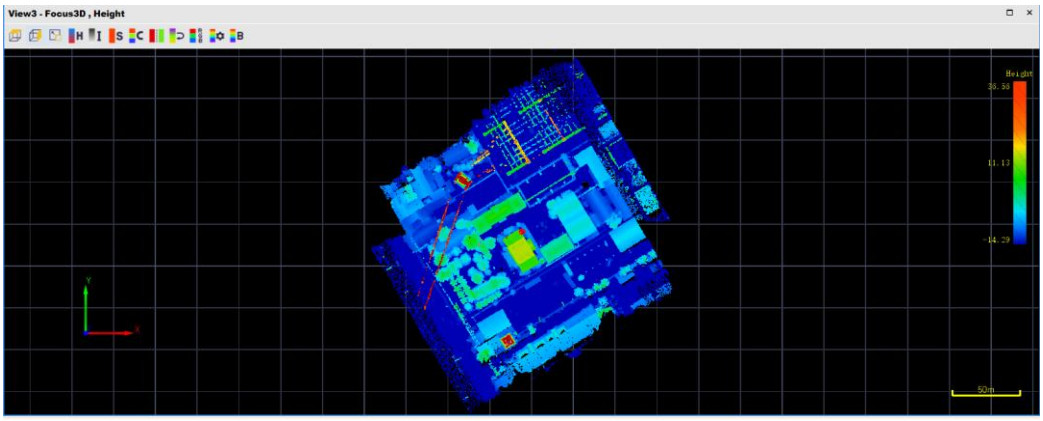

Figure 3-12 Zoom Screen

3. Full Screen

Click the <View>→<Full Screen>, or the button in the common toolbar, press the ESC key to exit the full-screen display.

4. Rotate Center

Click the <View>→<Rotation Center>, select the location of the rotation center in the point cloud view.

5. 2D View

Click the <View>→<2D View>, or the button in the toolbar to lock the point cloud view in a 2D state.

6. 3D View

Click the <View>→<3D View>, or the button in the common toolbar. The point cloud view will be locked in 3D.

7. Zoom In

Click the <View>→<Zoom In>, press the left mouse button and draw the rectangle, the view inside the rectangle will be enlarged.

8. Zoom Out

Click the <View>→<Zoom Out>, press the left mouse button and draw a rectangle, the view will be zoomed out centered on the rectangle.

9. Top View

Click the <View>→<Top View>, or click the button in the common toolbar.

10. Bottom View

Click the <View>→<Bottom View>, or click the button in the common toolbar, the point cloud will be displayed in bottom view.

11. Left View

Click the <View>→<Left View> , or click the button in the toolbar, the point cloud will be displayed in the left view.

12. Right View

Click the <View>→<Right View>, or click the button in the title bar of the view window. The point cloud will be displayed in the right view.

13. Front View

Click the <View>→<Front View> , or click the button in the toolbar. The point cloud will be displayed in front view.

14. Back View

Click the <View>→<Back View>, or click the button in the toolbar. The point cloud will be displayed in the back view.

3.2.2 Rendering

1. Height Rendering

Click the <View> → <Height Rendering> or the button in the title bar to display on Elevation. By select the drop down menu to change the color band of this rendering mode.

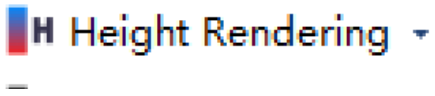

Figure 3-13 Height Rendering

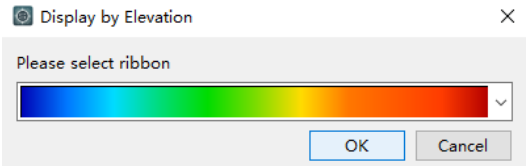

Figure 3-14 Elevation Rendering setting

2. Intensity Rendering

Click the <View>→<Intensity Rendering>, or the button in the title bar of the view window, the point cloud will be displayed as a rendering of the intensity value.

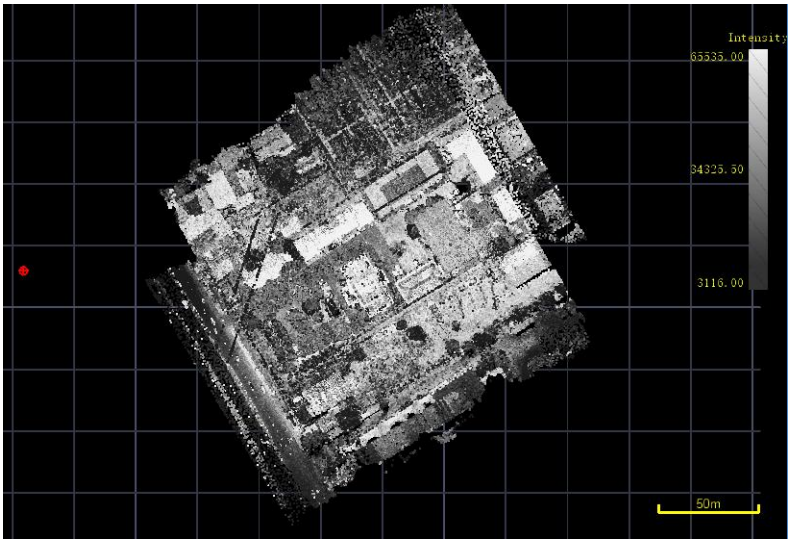

Figure 3-15 Intensity Rendering

3. Single Color Rendering

The menu <View>→<Single Color Rendering> button on the main interface is divided into two parts: arrow and icon.

(1) Click on the arrow part, the drop-down menu will pop up (Figure 3-16).

- <Single Color Rendering Settings>: set the Single Color rendering color corresponding to each file loaded in the current software (Figure 3-17).
- <Apply>: update the rendering color.
- <Reset Color>: restore all the color settings to the default.

(2) Click on the icon section, or the button in the title bar of the view window to single color render the point cloud in the currently selected view according to the set color.

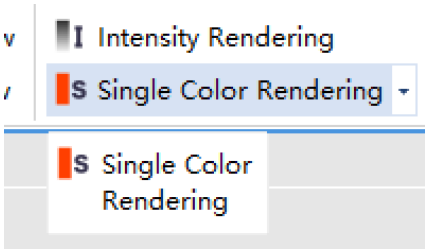

Figure 3-16 Single Color Rendering

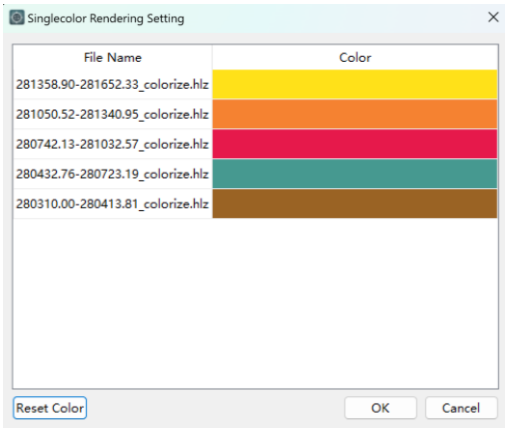

Figure 3-17 Single Color Rendering setting

4. Classify Rendering

The <View>→<Classify Rendering> button on the main interface has two parts: an arrow and an icon.

Clicking the arrow reveals a drop-down menu (Figure 3-18).

(1) Selecting <Category Rendering Settings> opens a dialog box (Figure 3-19) with 25 point cloud categories to set colors and toggle rendering via checkboxes.

(2) Click <OK> to apply the settings, instantly updating all relevant views. <Reset Color> restores default colors.

(3) Clicking the icon or the button in the view window's title bar renders the point cloud in the current view based on the selected category's colors.

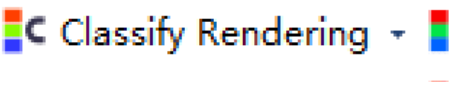

Figure 3-18 Classify Rendering

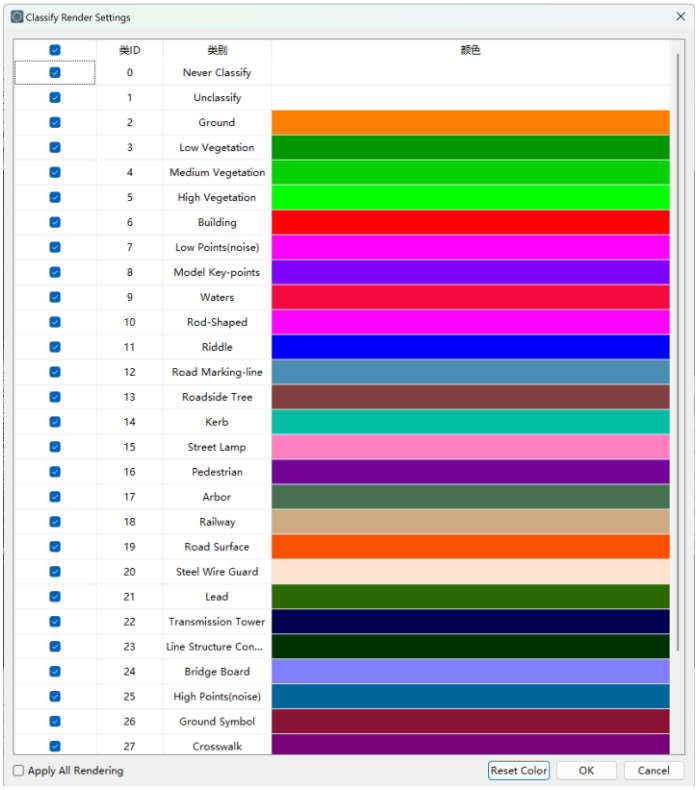

Figure 3-19 Classify Rendering setting

5. Airstrips rendering

Cick the menu <View>→<Airstrips Rendering>, or the button in the title bar of the view window, the point cloud in the currently selected view will be displayed as rendered by the aerotape (Figure 3-20).

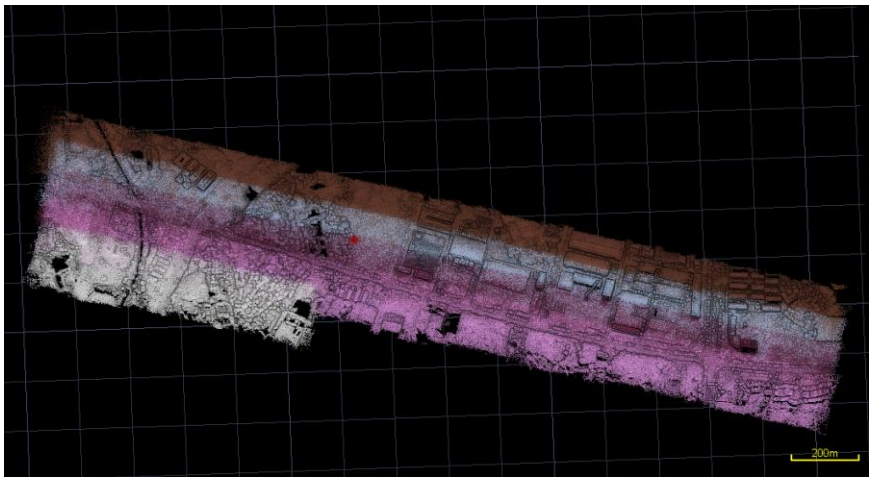

Figure 3-20 Airstrips Rendering

6. Echo Rendering

The <View>→<Echo Rendering> button on the main interface consists of an arrow and an icon.

(1) Clicking the arrow opens a drop-down menu (Figure 3-21).

(2) Selecting <Echo Colorstrips Setting> brings up a dialog box (Figure 3-22) where you can set colors for single, first, middle, and last echoes.

(3) Click <OK> to apply the changes, which will immediately update the echo rendering color if the current view is using echo rendering.

(4) Click the icon or the button in the view window's title bar renders the point cloud in the current view according to the set echo colors. Each view in a multi-view setup uses its own echo rendering settings.

Figure 3-21 Echo Rendering

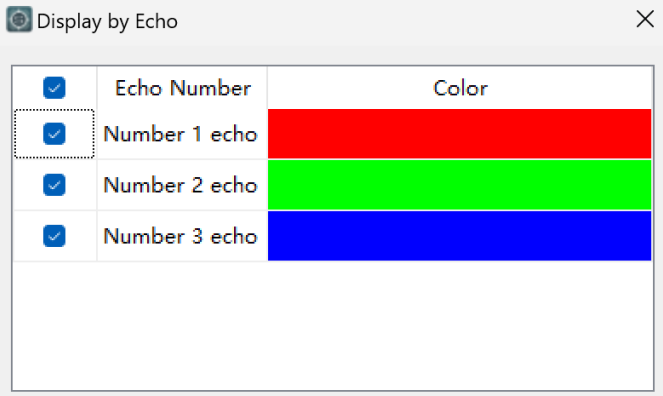

Figure 3-22 Echo Rendering setting

7. RGB Rendering

Click the menu <View>→<RGB Rendering>, or the button in the title bar of the view window, the point cloud will be displayed as rendered in true color (Figure 3-23).

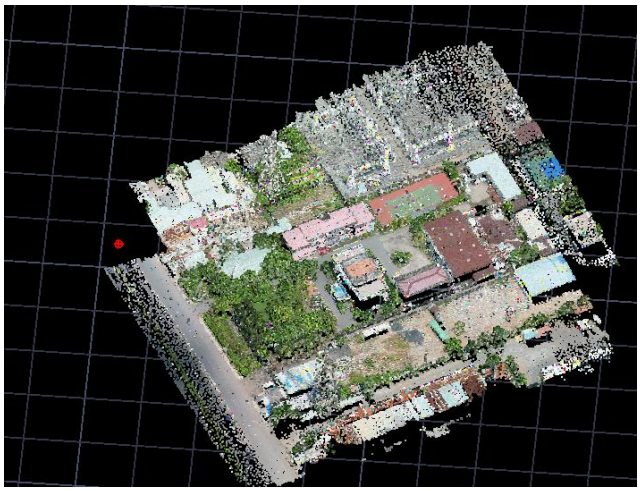

Figure 3-23 RGB Rendering

8.Cyclic Height Rendering

Cick the menu <View>→<Cyclic Height Rendering>, the point cloud will be displayed through the cyclic elevation to render (Figure 3-24).

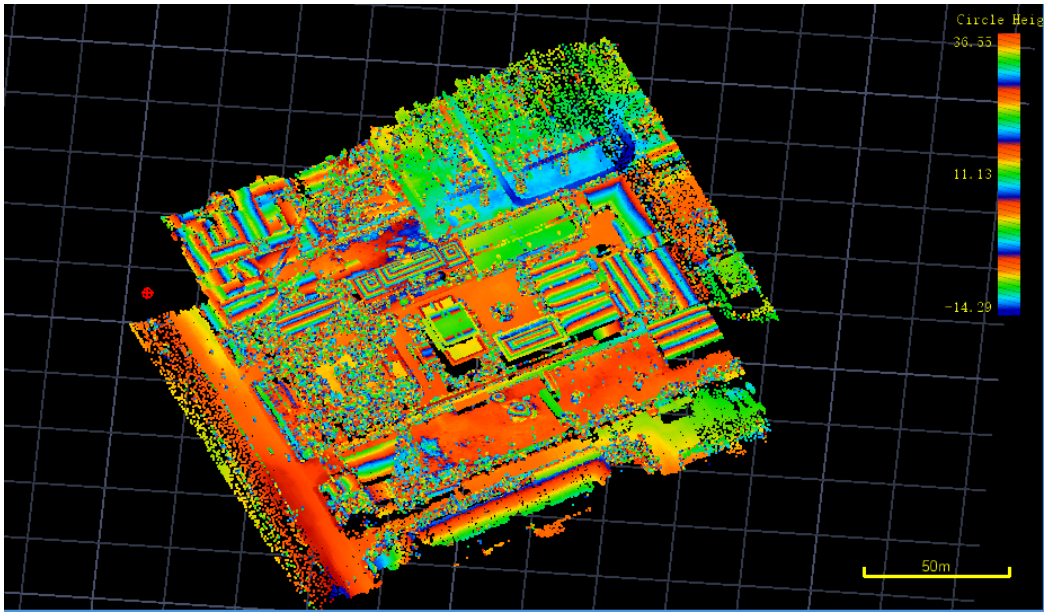

Figure 3-24 Cyclic Height Rendering

9.EDL Rendering

Click the menu <View>→<EDL Rendering>, or the button in the common toolbar, the point cloud is rendered when the contour visualization enhancement (Figure 3-25).

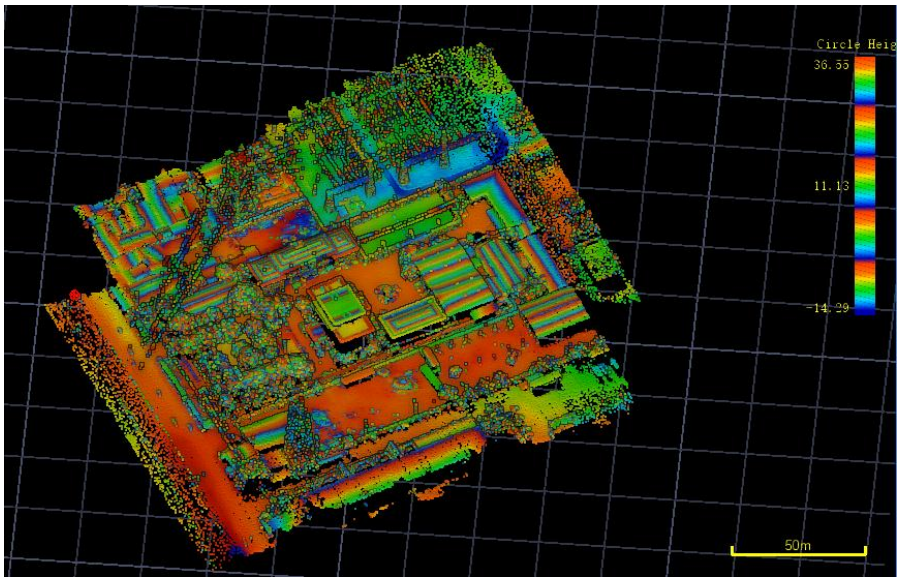

Figure 3-25 EDL Rendering Contour Enhancement

10 Mixed rendering

On the main screen, click the menu <View> Mixed Render, or the common toolbar push-button,

conduct the elevation + intensity rendering when rendering the point clouds.

3.2.3 Other

1. Show Transparency

Click the menu <View>→<Show Transparency>, or the button in the toolbar, the current view will show the transparency of the point cloud (Figure 3-26).

Cancel: click the menu <View>→<Show Transparency> again (Figure 3-27).

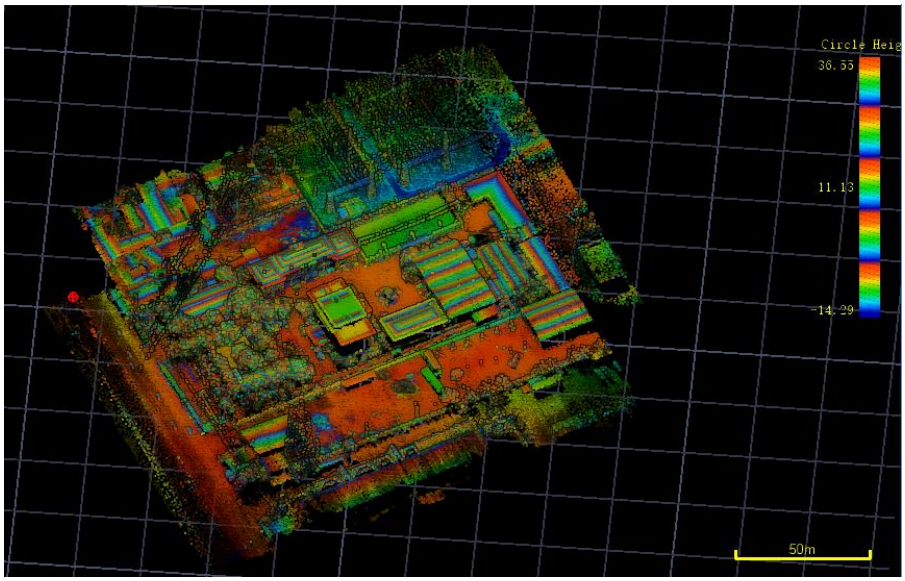

Figure 3-26 Show transparency

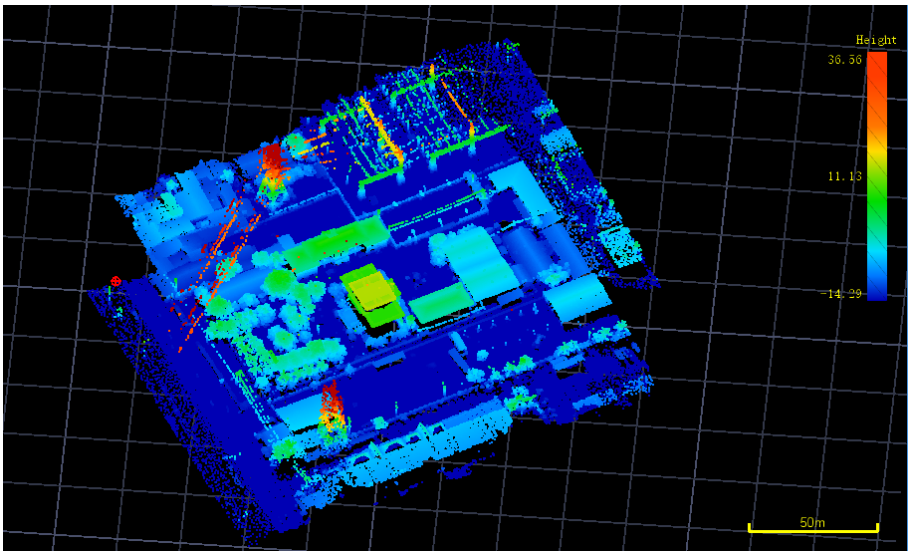

Figure 3-27 Cancel display transparency

2. Point size

Slide the menu <View>→<Point Size> slider on the main interface, the size of the point cloud in the main view changes with the movement of the slider (Figure 3-28).

- The slider corresponds to the point cloud from left to right from small to large, such as shown in Figure 3-29 for the point cloud is the smallest, such as shown in Figure 3-30 for the point cloud is the largest.

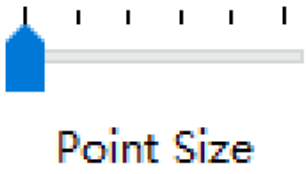

Figure 3-28 Point Size slider

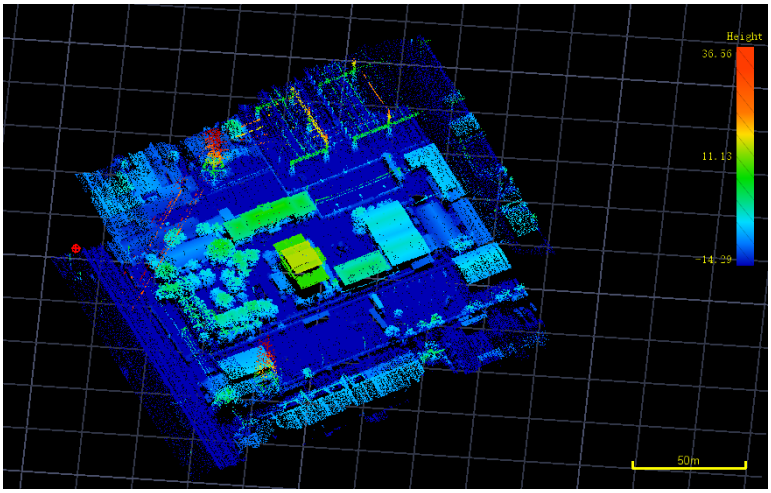

Figure 3-29 Point size set to minimum

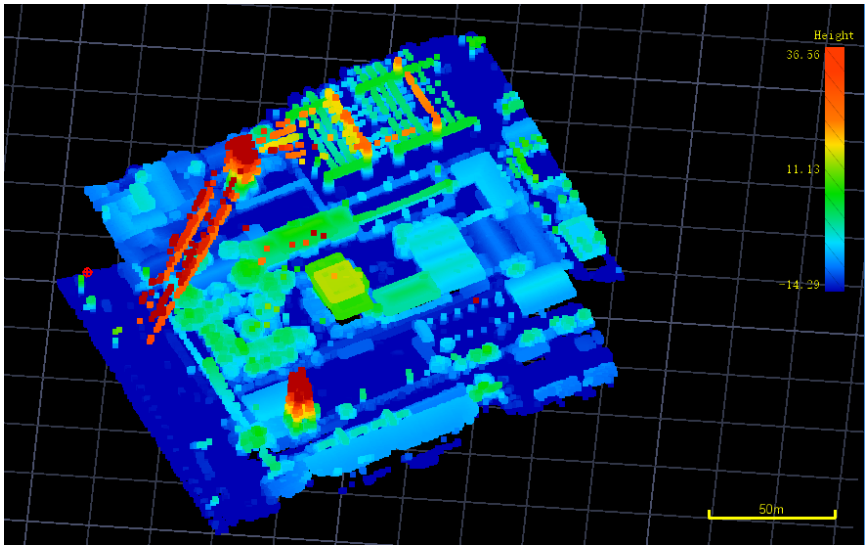

Figure 3-30 Point size set to maximum

3. Synchronize to View

Click the menu <View>→<Synchronize to View> to open the dialog box (Figure 3-31).

- When multiple point clouds are loaded in the main view, select the view, rendering type, and point clouds to be synchronized in the dialog box, and then click Apply to synchronize.
- If the rendering method selected is Category Rendering, you can adjust category visibility.

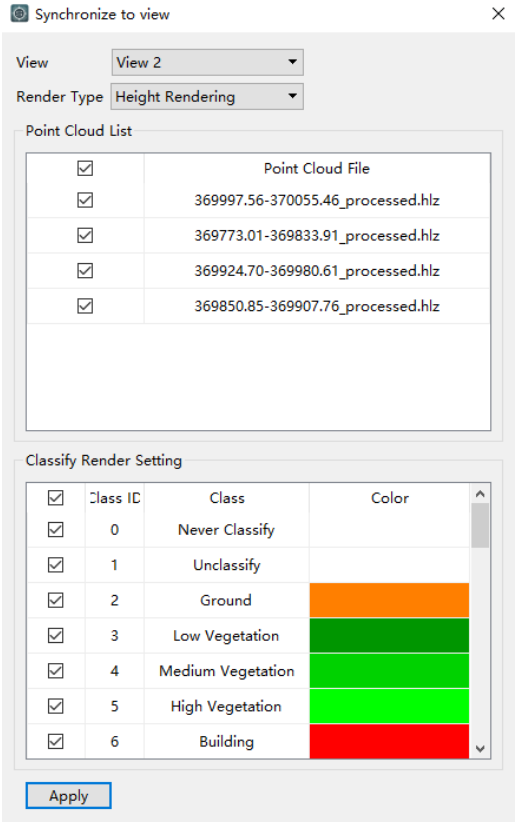

Figure 3-31 Synchronize to View

4.Poling View

Click the menu <View>→<Poling View> , hold down the Shift key, the left mouse button drag, draw a rectangular box, release the mouse on the drawing of the region obscured (Figure 3-32).

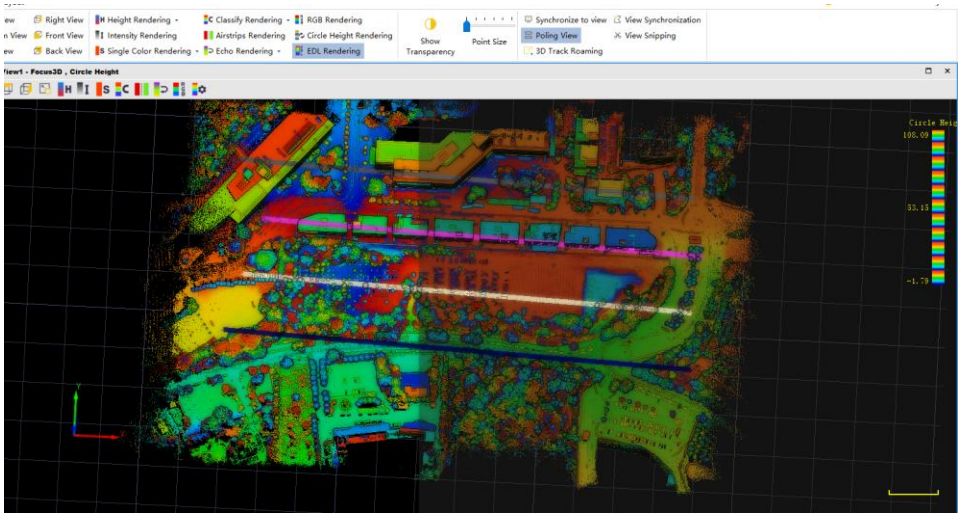

Figure 3-32 Poling View

5.Track Roaming

(1) Click the menu <View>→<3D Track Roaming> , pop-up roaming settings dialog box (Figure 3-33).

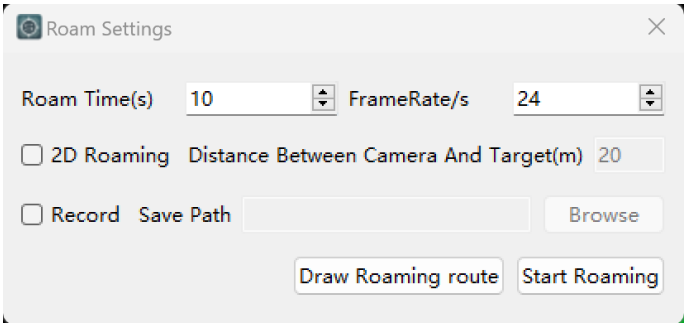

Figure 3-33 3D Roaming Settings

(2) Set the time and frame rate, choose whether to record the screen and the path to save the video, click <Draw Roaming Route>, and then click the left mouse button in the view to select the roaming path node (Figure 3-34).

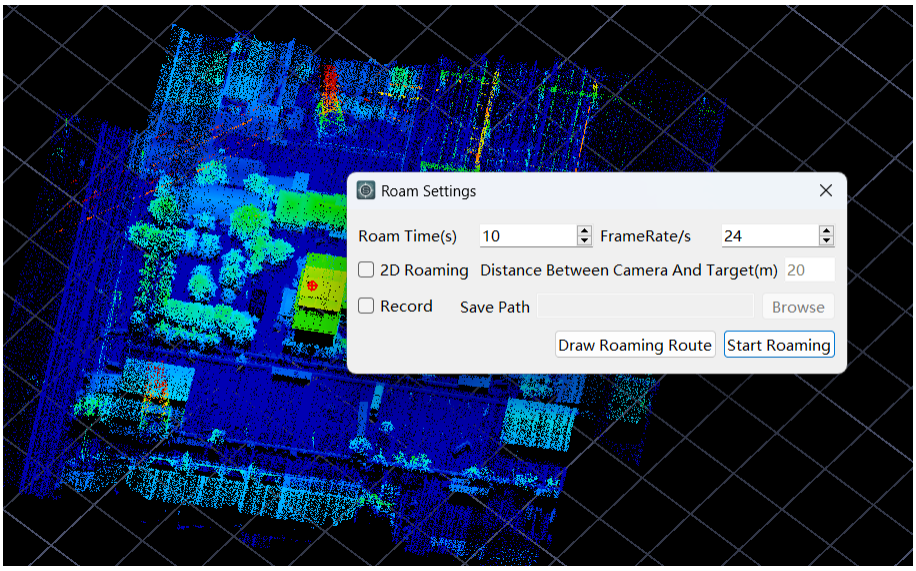

Figure 3-34 Select roaming path

(3) Click <Start Roaming>, it will display the roaming in full screen (Figure 3-35).

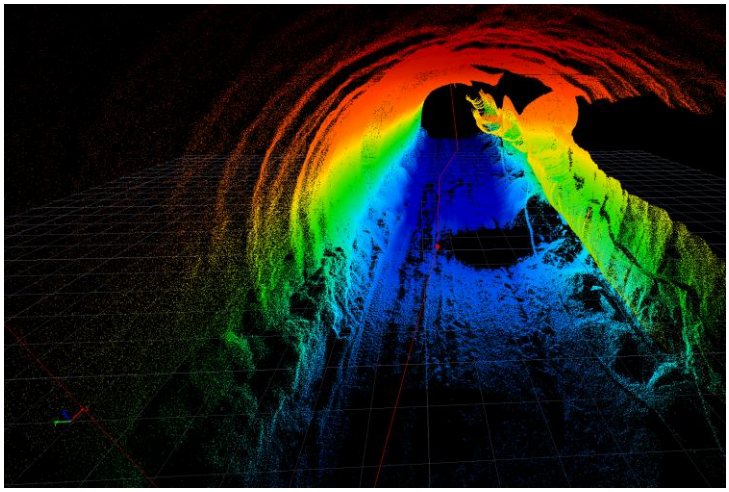

Figure 3-35 3D roaming

(4) At the end of the 3D roaming, the video is saved in the selected path.

6. Synchronize to view

(1) Click the menu <View>→<Synchronize to view > .

(2) By checking the need to link the view, click <OK>. Checked view windows can be synchronized in the browsing process (Figure 3-36).

(3) Cancel the synchronization: click <View Synchronization> again.

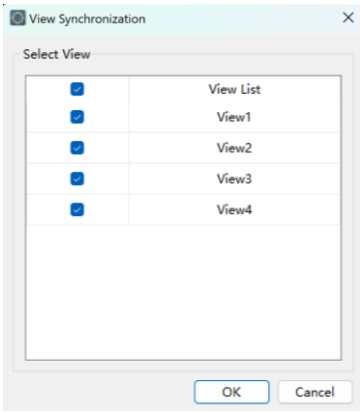

Figure 3-36 View synchronization settings

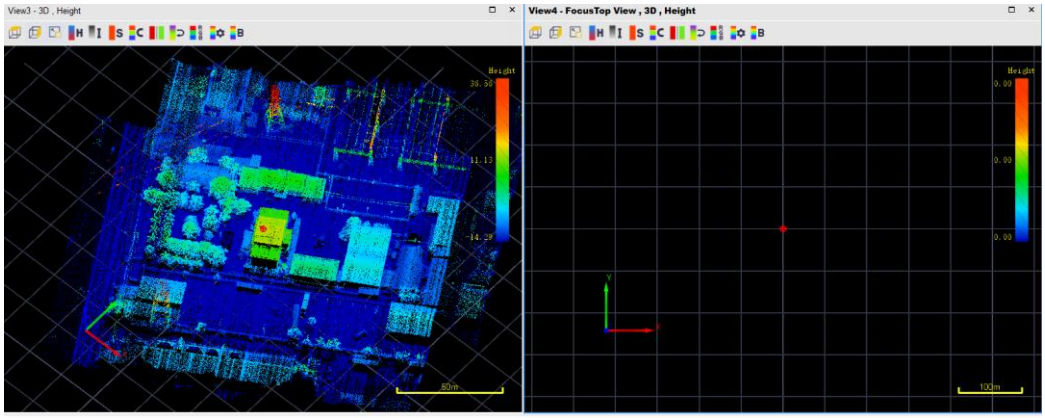

Figure 3-37 View synchronization

7. View Snipping

(1) Click the menu <View>→<View Snipping> .

(2) Press the left mouse button and move anywhere on the screen, draw a rectangular box,.

(3) Click the <Save As> button to take a screenshot to save (Figure 3-38).

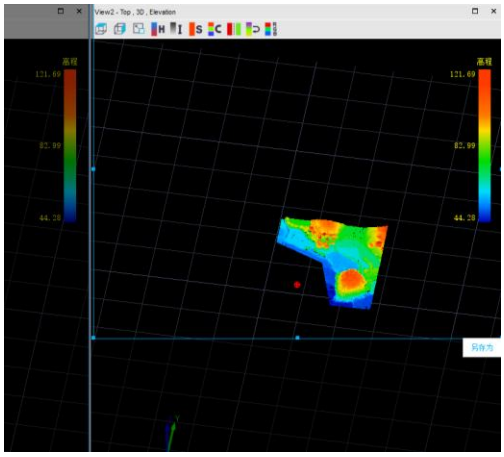

Figure 3-38 View Snipp

3.3 Aircraft

The <Aircraft> menu mainly contains Auto Process, Trajectory solving, Trajectory Rectify, Quality Check, Data combine, Point Cloud Colorize, Point cloud smooth, and Accuracy Inspection.

3.3.1 Auto Process

Automatic processing is a one-stop, fast and simple data processing function, which facilitates customers to quickly and conveniently output point cloud data results from the original data, one-click base station data import, trajectory solution, pos cutting, point cloud fusion, redundant photo removal, point cloud coloring, and terrain results export.

Steps:

(1) Copy the original data collected by the equipment to the computer.

(2) Create a new combine project.

(3) Click <Aircraft> → <Auto Process> to enter the automatic processing interface (Figure 3-39).

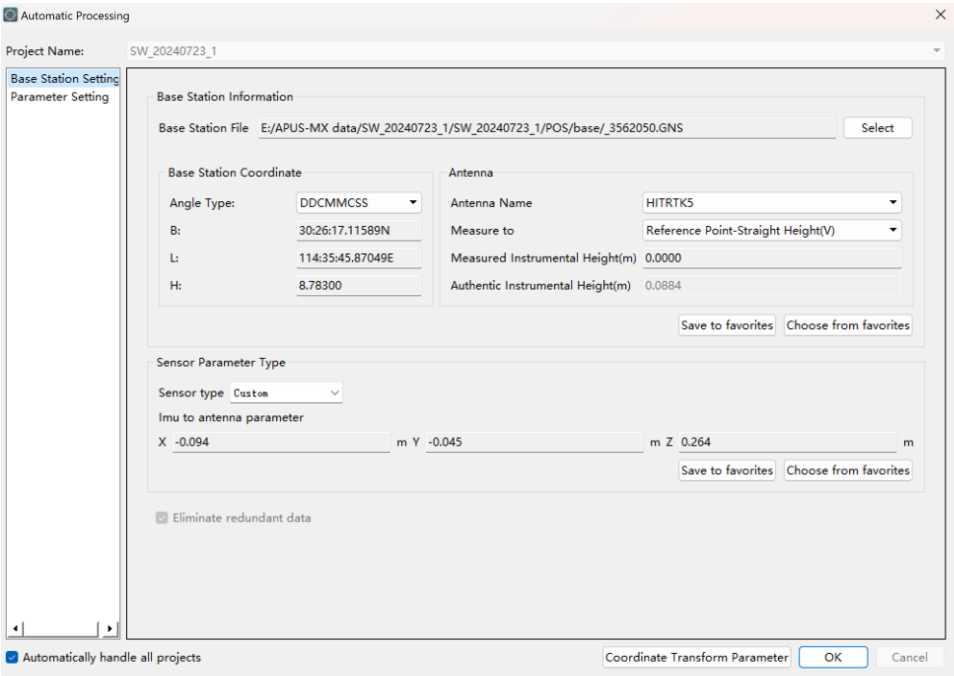

Figure 3-39 Automatic processing dialog box

1. Base Station Settings

(1) Base station setting includes base station file selection, base station coordinate setting, choose base station information from favorites, save to favorites, output coordinate type, and eliminating redundant data.

(2) Coordinate points can be saved to favorites, set the name of the base station, set the base station name and click <OK> (Figure 3-40).

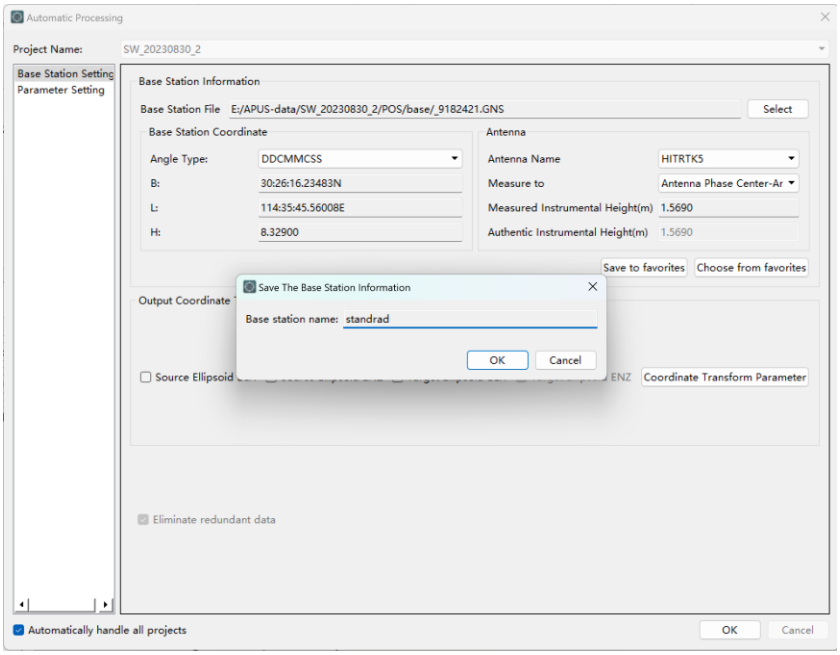

Figure 3-40 Save to Favorites

(3) Favorites in the existing base station coordinates, click < Choose from Favorites > to enter the Favorites interface to select the base station coordinates need to be replaced.

(4) Click < OK > to complete the replacement of base station coordinates (Figure 3-41).

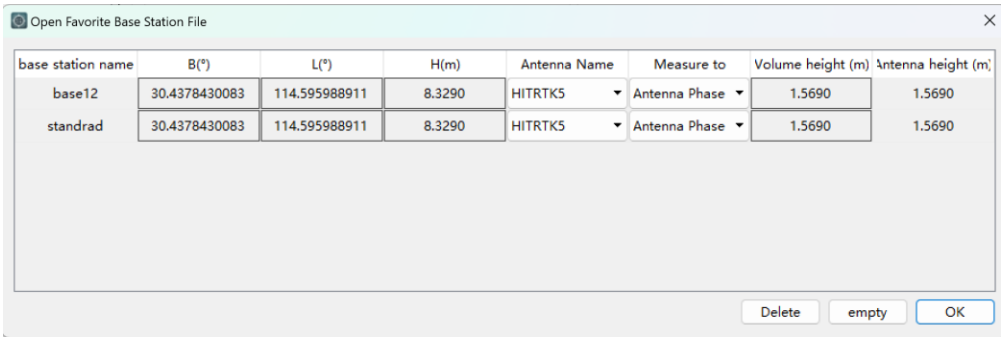

Figure 3-41 Select from Favorites

2. Parameter setting

The parameter settings include raw data check, point cloud data resolution, quality inspection, navigation belt correction, point cloud coloring, point cloud output, and topographic results export; point cloud data resolution is checked by default, and the detailed introduction of other capabilities can be viewed in 3.3.10-3.3.20 in this document.

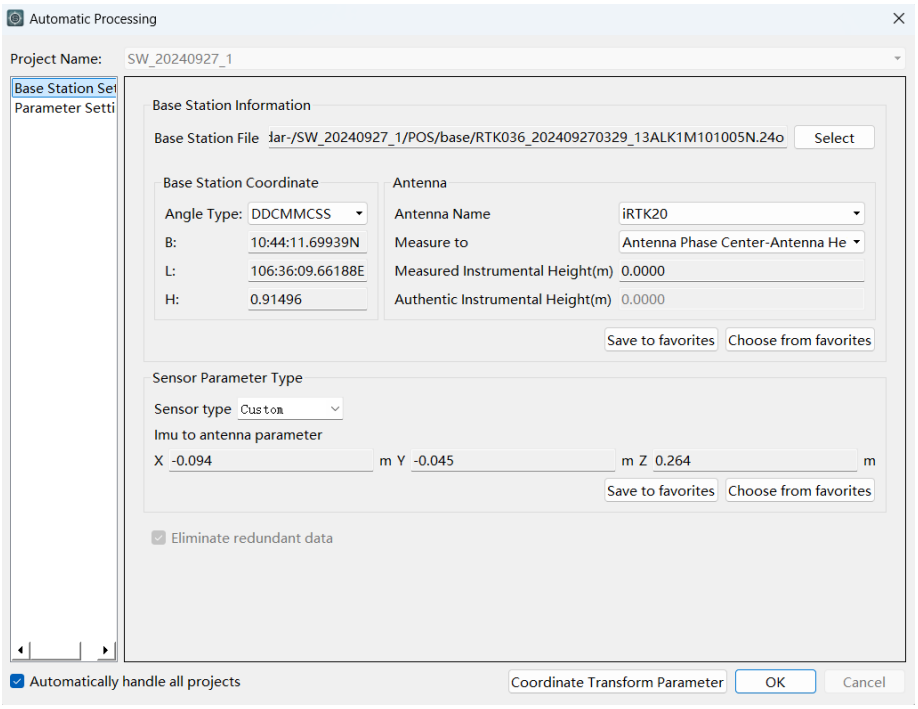

Figure 3-42 Parameter settings

After the configuration is completed, click [OK] and the software will automatically process the engineering data. The processing progress is displayed in the right information output bar and the progress bar in the lower left corner.

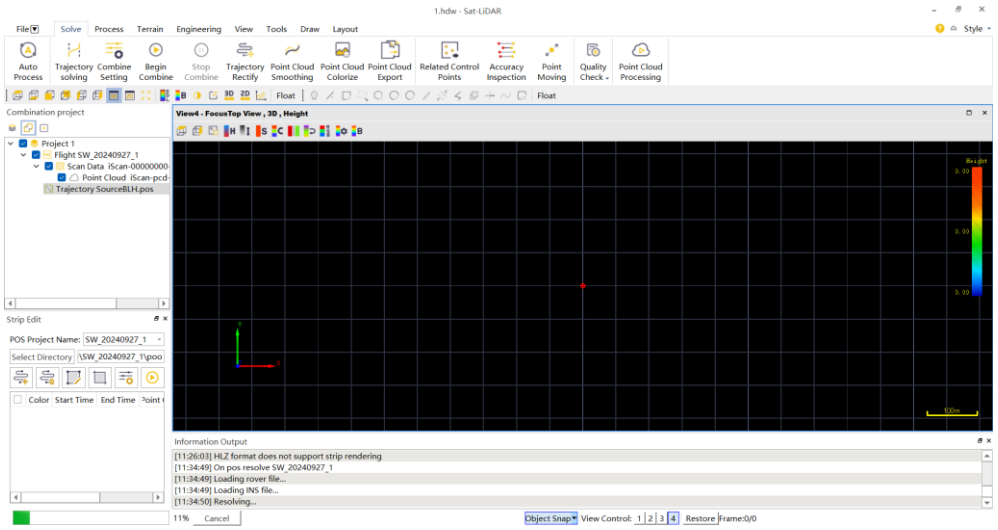

Figure 3-43 Automatic processing

After the data is processed, the processed point cloud will be automatically loaded into the navigation editing panel.

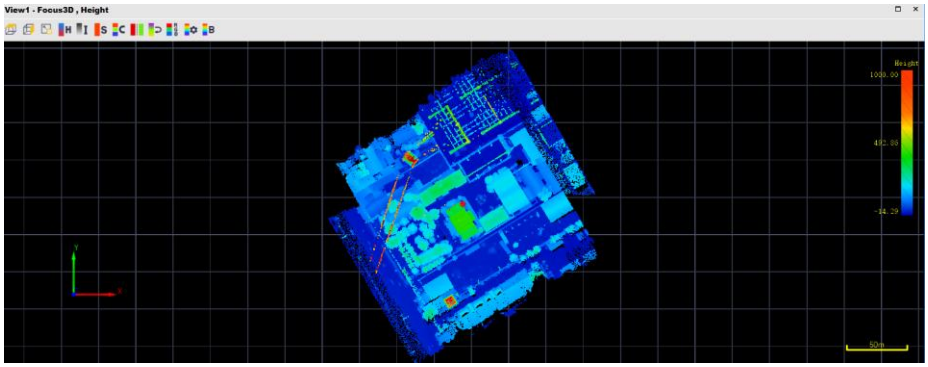

Figure 3-44 Point cloud loading

3. Coordinate system setting

Coordinate system settings include projection, datum, ellipsoid transformation, plane transformation, height fitting, grid, translation parameter, option, parameter calculation, import, and coordinate system management (Figure 3-45).

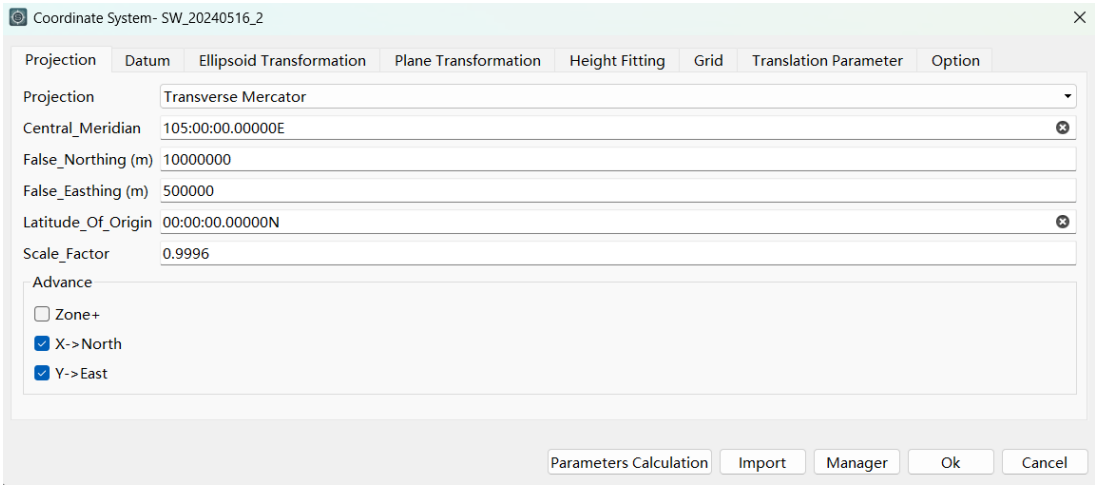

Figure 3-45 Coordinate system settings

(1) Projection

The "Projection" tab is used to transform the projection and set the related parameters (Figure 3-45).

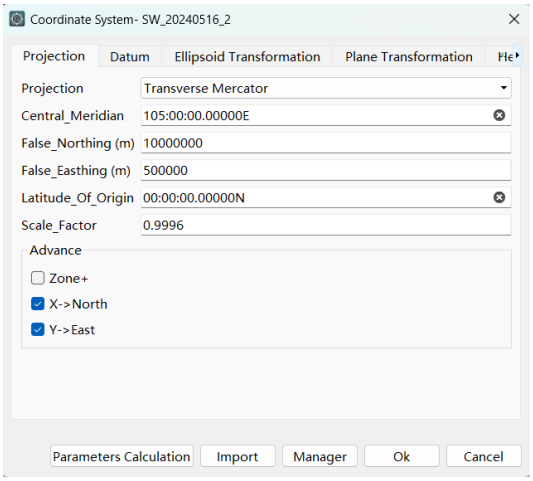

Figure 3-46 Projection

(2) Datum

Users can select or set the corresponding source ellipsoid and target ellipsoid name (if the ellipsoid is pre-built in the software, the ellipsoid parameter under the corresponding target ellipsoid name will be loaded automatically and the ellipsoid parameter is not editable).

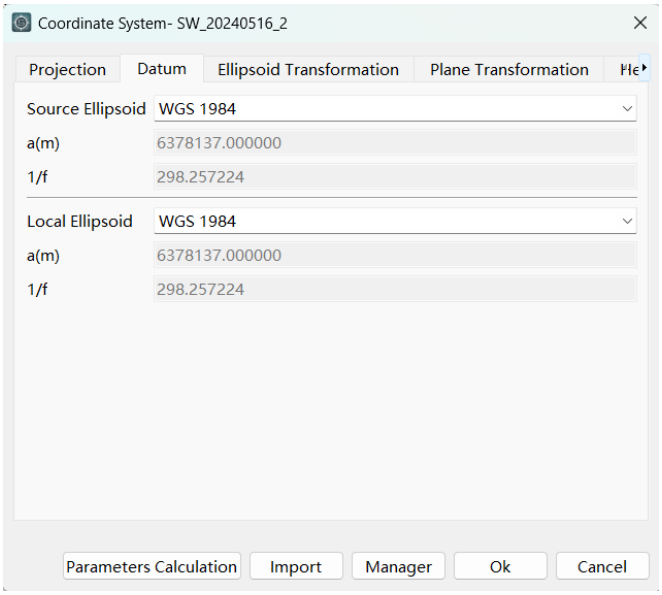

Figure 3-47 Datum

(3) Ellipsoid transformation

The transformation models are as shown in Figure 3-48.

- Bursa-Wolf Transformation, Molodensky, One-touch can be obtained after parameterization, while the rest require inputting the corresponding values.

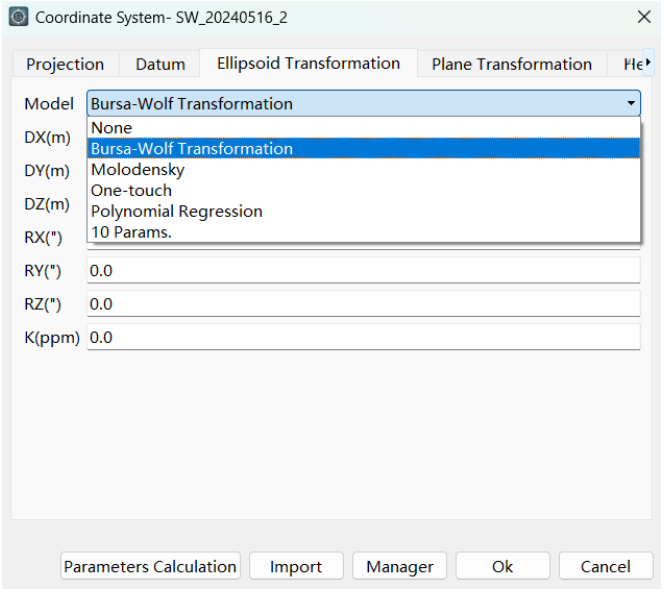

Figure 3-48 Ellipsoid transformation

(4) Plane transformation

Different parameters can be modified by selecting different transformation models (Figure 3-49).

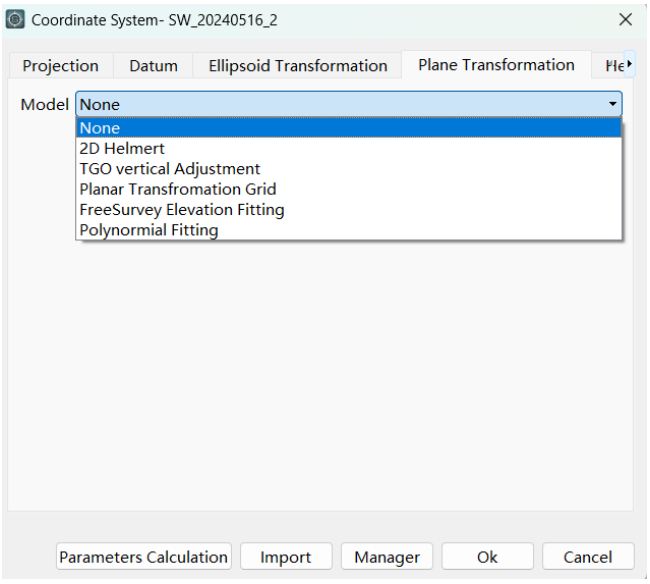

Figure 3-49 Ellipsoid transformation

(5) Height fitting

The transformed models are as shown in Figure 3-50.

- The FreeSurvey Elevation Fitting obtains the elevation fit parameters from the results of the parameter calculation, height fitting can be done by selecting the required model and filling in the relevant parameters.

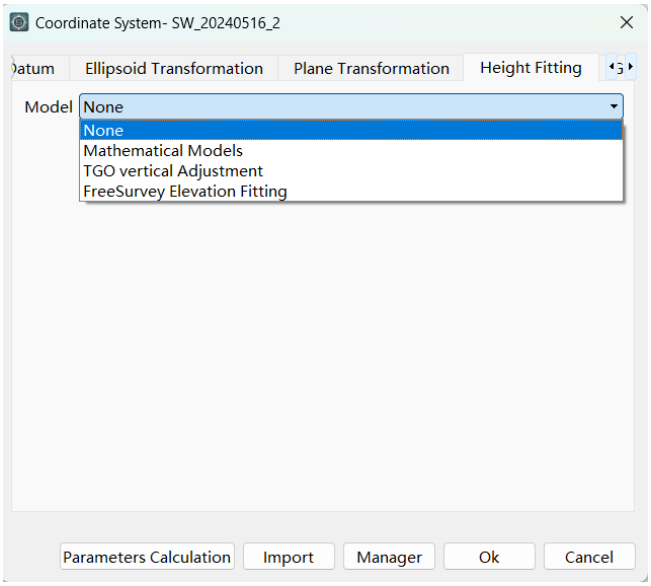

Figure 3-50 Height fitting

(6) Grid correction

The supported grid types are as shown in Figure 3-51.

- Users can select the corresponding plane correction grid type.

- The grid switch is off by default, and the grid file drop-down box on the right side is not selectable, when the switch is turned on, the imported grid file can be selected from the grid file drop-down box on the right side.

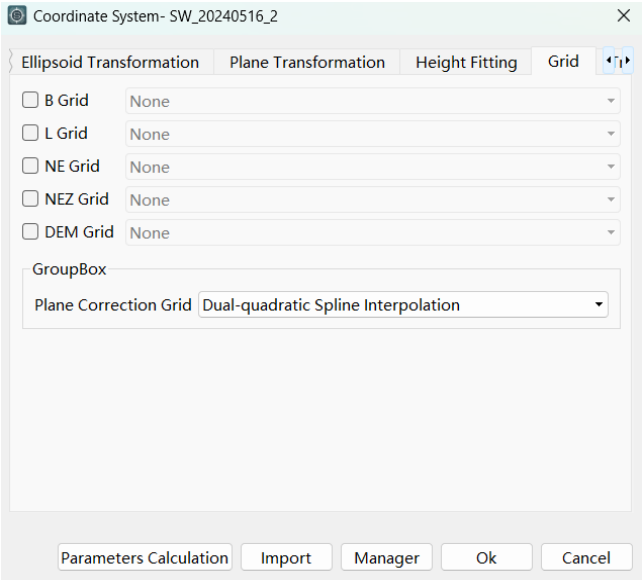

Figure 3-51 Grid

(7) Translation parameter

The translation parameter is mainly used for the point panning function of Sat-Survey or SGS RTK module (Figure 3-52).

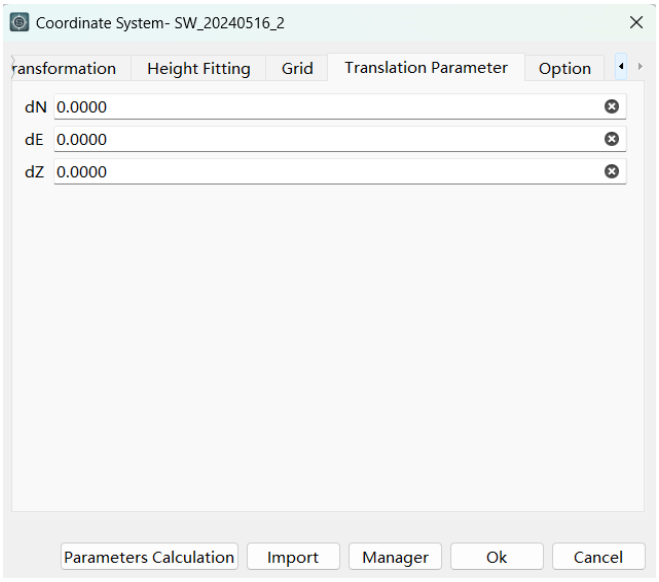

Figure 3-52 Translation parameter

(8) Option parameters

The adjustable parameters are shown in Figure 3-53.

Whether the file is encrypted or not:

- if the dam body parameter file (upgraded, version number is greater than or equal to 17) is encrypted, you can check whether the coordinate system date is outdated, and you can't edit the ellipsoid parameters;
- If the dam parameter file (not upgraded, version number less than 17) is encrypted, the ellipsoid parameters can be edited.

When creating a new coordinate system, the latitude and longitude range configuration item is turned off by default, and the user can set the maximum and minimum range of the coordinate system latitude and longitude.

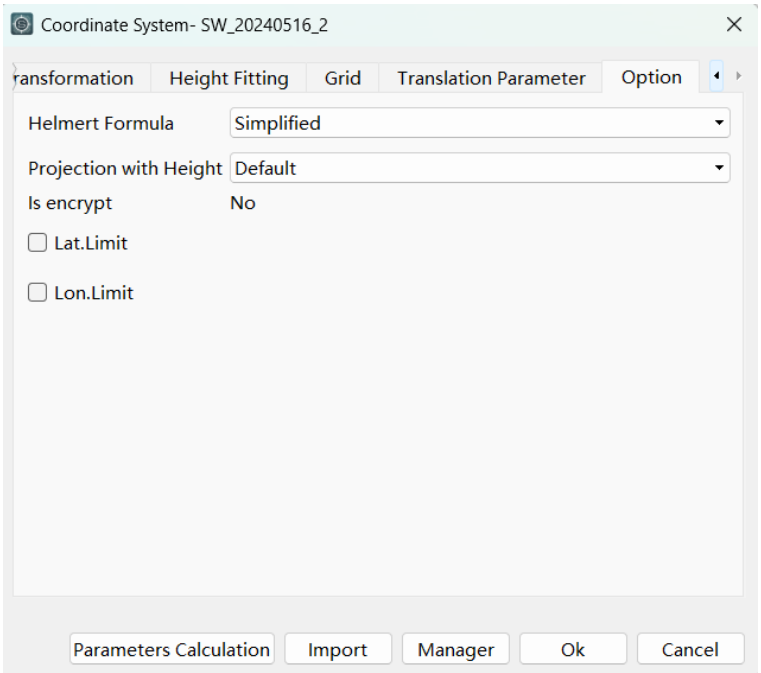

Figure 3-53 Option

(9) Parameter calculation

a) Click the [Parameter Calculation] button to enter the parameter calculation interface, ensure the projection and datum information are correct before calculation.

b) Click the [Open] button to select a parameter calculation file. Supported formats: parameter calculation point pair file (*.txt), Carlson localized parameter file (*.loc), custom (*.txt).

c) Select the file type and corresponding file, then click OK to import and display the point pairs in the list.

d) Click [Add] to manually enter and confirm point pair coordinates, which will be displayed in the list.

e) Choose a calculation type and click [Calculation] to display the results based on the input point pairs and calculation type.

f) Click [Save] to save the custom parameter point pairs and click [Apply] to automatically fill in the parameters on the corresponding tab page.

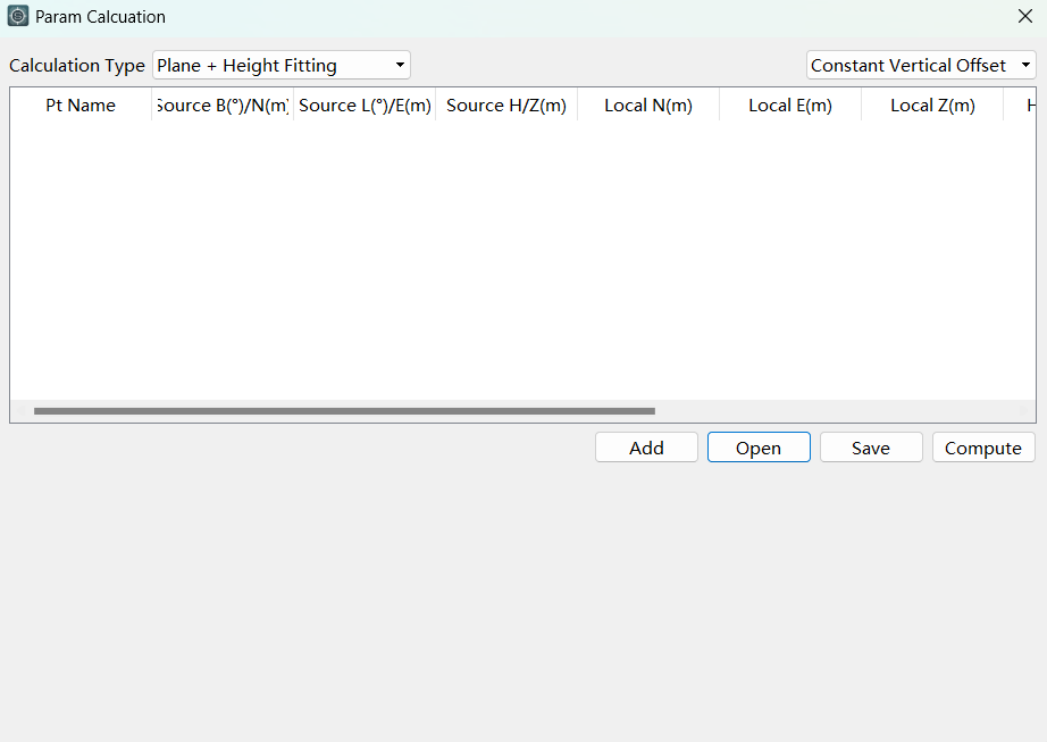

Figure 3-54 Parameter calculation

(10) Import

Click the [Import] button to open the file selection window, elect the corresponding coordinate parameter file (DAM or DC format), click OK. The successfully read coordinate system file will update the parameters in the projection, datum, and plane transformation configurations in the coordinate system setting interface.

(11) Coordinate system management.

a) Click [Coordinate System Management] to enter the interface.
b) Click [Predefined] radio box to select a continent to show corresponding countries and their coordinate systems.
c) Click the [User Defined] box to display the custom coordinate system list, click a coordinate system to see its details on the right, but the name cannot be edited.
d) Select a custom coordinate system file and click [Delete] to remove it.
e) Click [Download] to download predefined coordinate system and grid files.
f) Click [Update Built-in Ellipsoid] to automatically update ellipsoid parameters.
g) Click [New] to clear the coordinate system name, enter a new name, edit parameters, and click [Save] to save.
h) Click [Apply] to apply the selected coordinate system, and then click [Cancel] to close the interface.

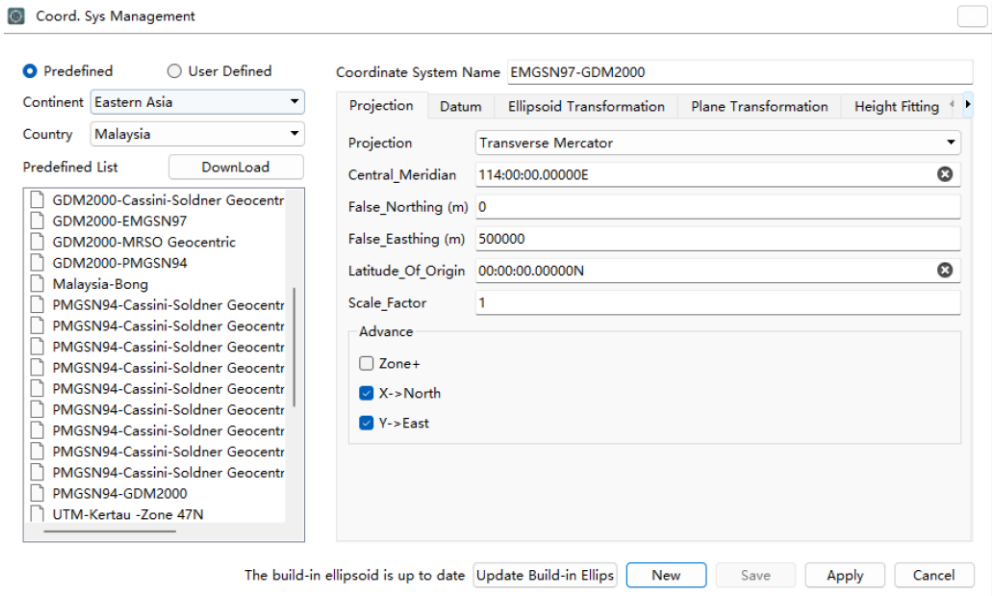

Figure 3-55 Coordinate system management

3.3.2 Trajectory Solving

1. Click menu <Solve>→<Trajectory Solving> to open the solving interface (Figure 3-56);

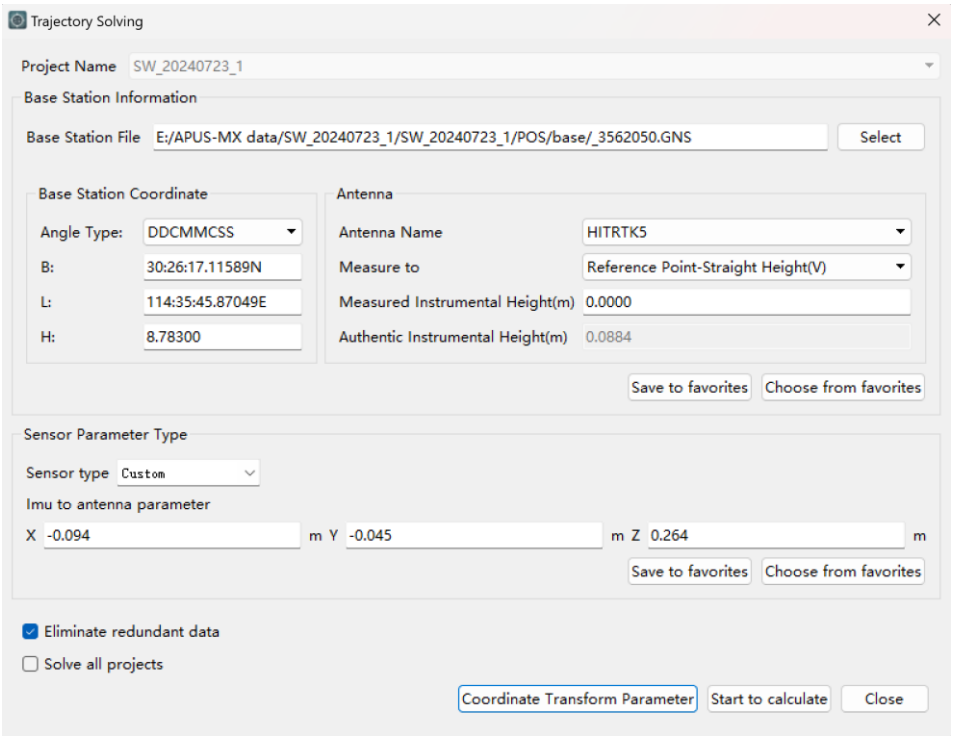

Figure 3-56 Base station information

2. For details of local trajectory solving function, please refer to the base station setting in 3.3.1.
3. If you need to generate complete trajectory pos:

   (1) uncheck the "Eliminate redundant data ";

   (2) check "Solve all Projects" .

4. It will solve all pos trajectorys under the current project with the currently input base station coordinates as the base station.
5. After the configuration is completed, click <Start to calculate> and then the software starts to solve the trajectory pos, after the solving is completed, the solved pos trajectory is loaded in the main view and loaded into the editing column of the aerial tape (Figure 3-57).

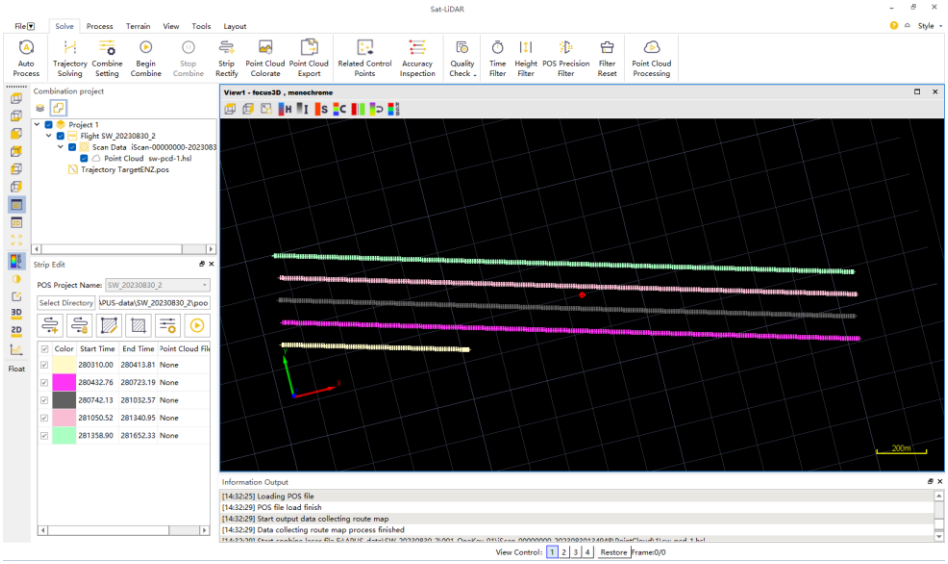

Figure 3-57 Local trajectory solving

3.3.3 Combine Setting

Selecting the POS project and point cloud file, set the combine parameters for the POS project and point cloud file respectively.

The combine function mainly utilizes the POS data to transfer the scanned point coordinates in the original scan file to the geodetic coordinate system, and the output results include: target point cloud file, POS track file (lin), where the lin file is created by default.

You need to set some parameters before fusion, right-click on the project node or point cloud node in the file view and Click the "Combine Settings" menu to make the relevant settings.

Note:

(1) The Combine Settings dialog box is a non-modal dialog box. After the Combine Settings dialog box pops up, left-click different tree nodes on the workspace list, the Combine Settings dialog box will update the settings accordingly.

(2) After setting the parameters in the Combine Settings dialog box of a point cloud node, you can choose to apply them to the point cloud file, the current project or the whole combine project in one of the three ways.

- Start: click <Solve>→<Begin Combine>, you can open the log output box to view the log information.

- Terminate: click <Solve>→<Stop Combine> to stop the combine process.

1.POS Project Combine Setting

Select the current project in the project management panel, right-click the menu and click <Combine Setting>, or click the tool button on the toolbar to bring up the POS Project Combine Setting dialog box (Figure 3-58).

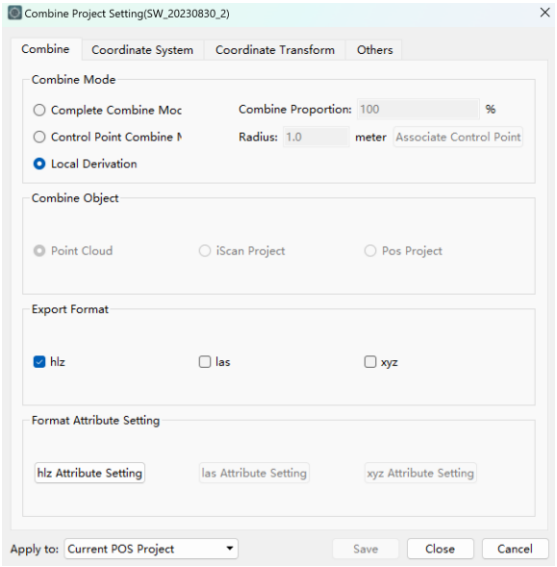

Figure 3-58 Project Combine Settings

(1) Combine

The "Combine" Tab Page is divided into 4 sections from top to bottom (Figure 3-59).

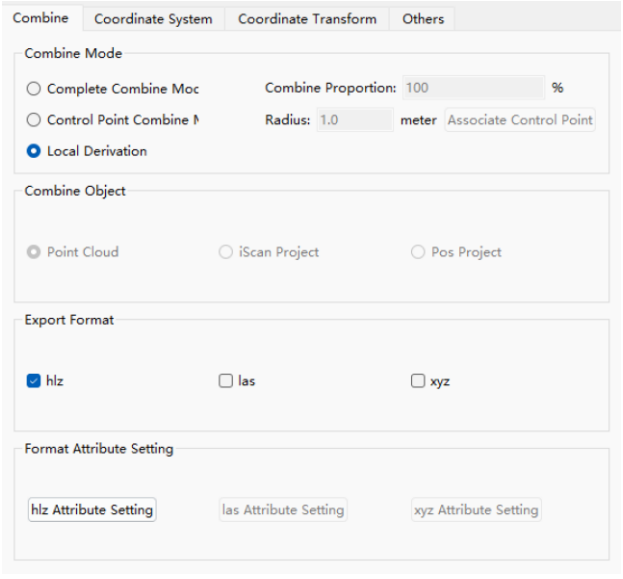

Figure 3-59 Combine Tab Page

i) Combine Mode:

- Complete Combine Mode : Combine all data which can change the proportion.

- Control Point Combine Mode: Indicates that the combine is performed according to the radius distance of the imported control point file;
- Local Derivation: the on-board data fuse and output the POS file selected by the local box, and the onboard data fuse the drawing POS cutting area.

ii) Combine object:

Combine output is performed using a single point cloud file, iScan project, or POS project as the basic unit. For example, if the combine mode is complete combine mode and the combine object is POS project, all the point clouds in the whole POS project will be merged into one point cloud file for output.

iii) Export Format: you can choose one or more of the three target point cloud formats: hlz, las, xyz.

iv) Format attribute settings: including hlz attribute settings, las attribute settings and xyz attribute settings.

a) hlz attribute settings include hlz3.2 and hlz3.3 two versions;

b) las attribute settings:

- las attribute settings include las version, time type, and whether or not to export by volume.las format currently supports the following versions: 1.0-1.4, which 1.0-1.3 version scanning angle value is -90 to 90 degrees, 1.4 scanning angle value is -180 to 180 degrees.
- The scanning direction with a 1 indicates that the left to right, 0 indicates that the right to left, the bottom of the left for the negative, the bottom of the right for the positive;
- The time type currently supports the GPS standard time and the GPS seconds of the week;
- las format support for the sub-volume export function, the support of the size of the sub-volume are: 512M. 1024M, 2048M,4096M.

c) the xyz format supports customizing the content and order of the fields, mainly including: coordinates X, Y, Z, intensity, time, echo sequence number, echo number and other fields.

(2) Other

The "Others" tab is shown in Figure 3-60.

i) Custom hdi file: After checking the box, another hdi file will be created in the directory of the standard hdi file, the file name is "iScan-Image-custom.hdi", and the exposure time in the file will be in weeks and seconds using GPS, and the contents and order of the fields can be customized.

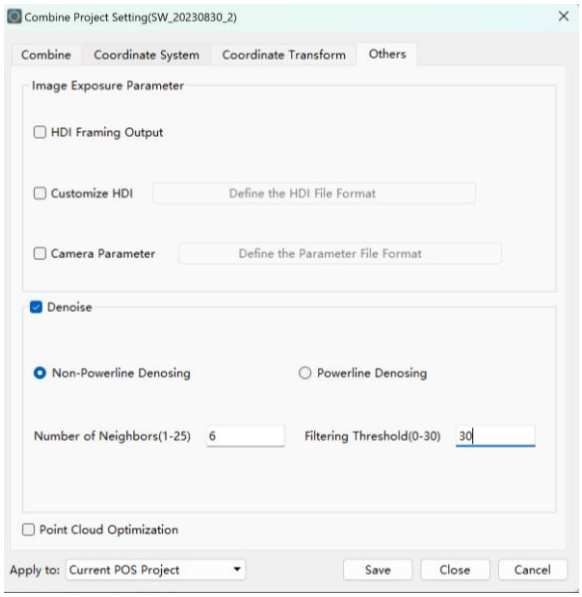

Figure 3-60 Other setting

ii) Camera parameter: After checking the box, when combine outputs the hdi (POS attitude parameter of the camera at the moment of exposure) file, it will output the camera's external parameter file at the same time, with the same output path, and the name of the file is "iScan-Image-external.txt", and the contents and the order of fields in the camera's external parameter file can be customized.

iii) Denoising: denoising removes noise by spatial features, which is used to select according to whether the current fused point cloud needs to retain the power line features or not, if the collected data does not contain power line data, then select non-power line denoising, if the collected data contains power line data and needs to retain the complete power line point cloud, then select power line denoising.

- Number of Near Neighbors: indicates the number of points near the target point used for counting when calculating the distance between the target point and the point cloud. The larger the value is, the more points are involved in the calculation, the larger the search range is, the filtering threshold calculated applies to more points, and the denoising effort is smaller, and vice versa the denoising effort is larger.
- Filtering threshold: a coefficient when calculating the filtering threshold, the larger the value is, the larger the calculated filtering threshold is, the smaller the denoising strength is, and vice versa, the larger the denoising strength is.

2.Point cloud file combine settings

Project management panel select the point cloud file, right-click menu click <Combine Settings>, the point cloud file combine settings dialog box pops up (Figure 3-60), from top to bottom is divided into 6 parts.

(1) Time: After checking the box, click the button <Time Period>, in which you can enter multiple

time periods or pick up the time from the POS track.

(2) Distance: The distance represents the distance of the scanning point from the scanner, after checking the box, click <Distance Segment>, in which you can set multiple distance segments for filtering.

(3) Field of view: including non-fixed angle and fixed angle, fixed angle will be automatically hidden in onboard mode; non-fixed angle represents the scanner's angle of rotation when scanning (multi-line scanners need to determine the scanning direction of rotation), the range of the field of view is -180 degrees to 180 degrees. In onboard mode, the field of view is checked by default and ranges from -45 degrees to 45 degrees. Fixed angle range of 0-360 degrees, the default is 50 degrees, equivalent to non-fixed angle input -25 degrees to 25 degrees.

(4) POS Height: This height indicates the height of the scanning point relative to the POS center.

(5) Range: Define the range of the rectangle area by setting the coordinates of the upper-left and lower-right corners of the rectangle, the coordinate system to which the coordinates belong is the coordinate system defined in the "Coordinate System" page.

(6) Reverse Filtering:

- Check the corresponding filtering indicator to indicate that "the point cloud with the indicator in the specified range will be filtered out and will not be fused to the output".
- Unchecked to indicate that "the point cloud with the indicator not in the specified range will be filtered out and will not be fused to the output".

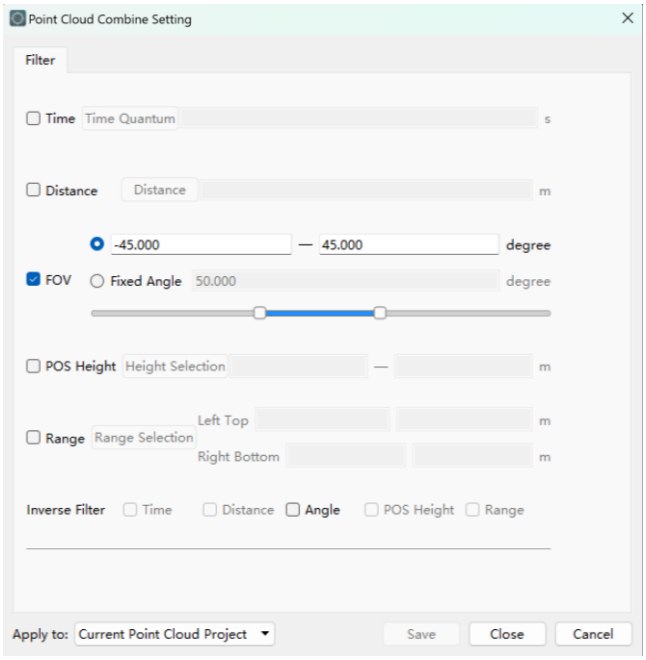

Figure 3-61 Filtering Setting

3.3.4 Begin Combine

1. Open the combine project, project management panel check the need to fuse the project.
2. Click the menu <Solve>→<Begin Combine>, or click the toolbar tool button 【 】, to start fusing the data, the status bar displays the progress of combine and combine of the information output in the log view (Figure 3-62).
3. Combine is complete, the log view will be "combine success" message prompt.

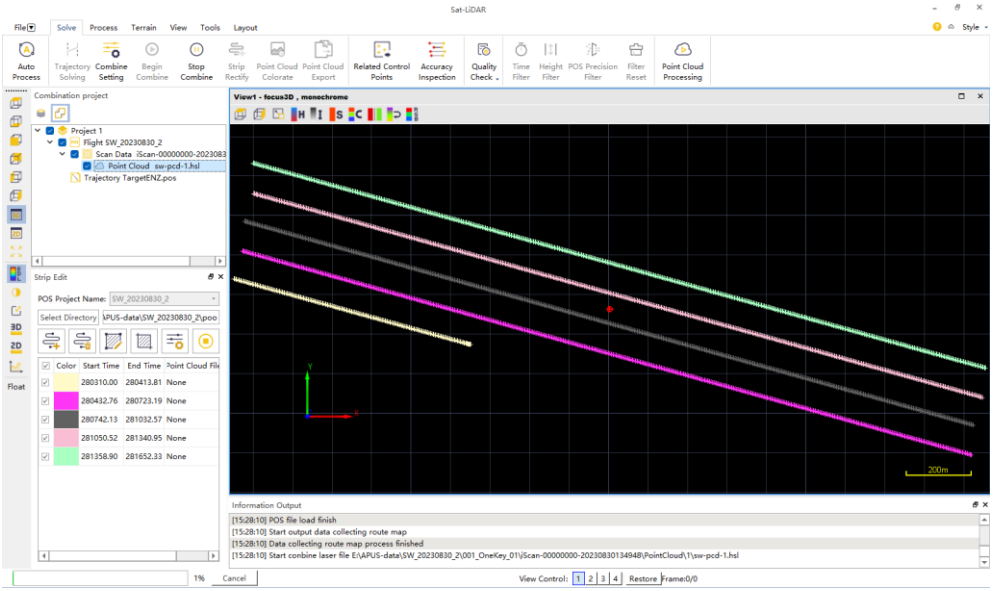

Figure 3-62 Start combine

3.3.5 Stop Combine

In the process of data combine, you can click menu <Solve>→<Stop Combine> to stop the current combine.

3.3.6 Strip Rectify

This function mainly deals with the non-overlapping point clouds of neighboring air strips through the algorithm, so that the air strips overlap to the maximum extent.

1. Open the combine project in the project management panel, load and display the point cloud files, click <Solve>→ < Trajectory Rectify >, and then "Trajectory Rectify" box pops up (Figure 3-62).

2. Select the flight that needs to perform aerial tape correction, select the corresponding POS file and point cloud file.

3. Add target control points (if any) Select the target points for automatic target point identification operation, click OK to start the Trajectory rectification.

4. After the completion of the rectification, the corrected point cloud will be automatically loaded into the view, and the Trajectory rectification report will be output in the Point Cloud Output Catalog, which displays the corrected point cloud, as well as the minimum error, maximum error, and RMS of the

Trajectory rectified, if the control points are used for xyz rectification, the coordinates of the control points and scan points before and after calibration are additionally displayed, as well as the error.

- List of Flights: support to create new flight, add flight file (.hdc), remove flight file; add duplicate flight, there will be an error message;
- List of Trajectory Files: supports adding POS data and removing POS data for the currently selected flights.;
- List of Point cloud file: support adding point cloud data and removing point cloud data for the current selection; use hlz format point cloud correction, and the corrected hlz point cloud format is hlz3.4 format.
- Add control point: support selecting control point to xml file, support editing control point for target identification corresponding to the scanning point.

(1) Control point list: you can change the type of one or more control points to control point or check by right clicking.

- Control point: participate in Trajectory rectify correction,
- Check: not participate in correction, participate in the output of the accuracy report after correction.

(2) Auto Identify Target Points: Click the button of Auto Identify Target Points on the dialog box of Control Points List, the program will automatically search for square checkerboard targets in the Control Points Attachment, and 50cm*50cm targets are supported at present. There is an option to choose xyz correction or z correction on the dialog box.

- z correction: only the z coordinate of the control point will be used, and the final result is that only the z coordinate of the point cloud will be involved in the correction;
- xyz correction: the xyz coordinate of the point cloud will be used, and the final result is that the xyz coordinate of the point cloud may be modified.

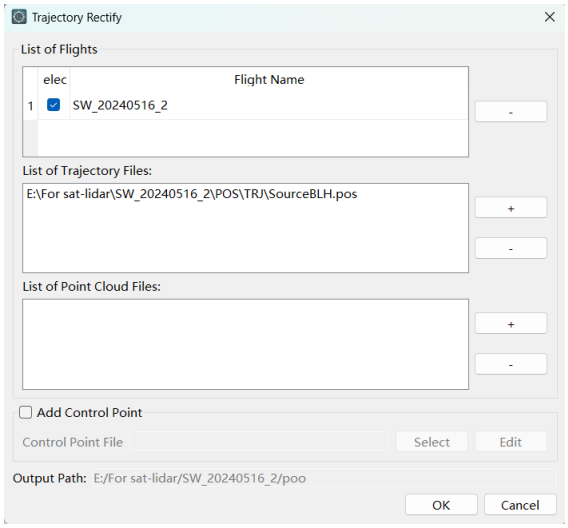

Figure 3-63 Strip ajustment dialog box

3.3.7 Point Cloud Smoothing

1. Click the menu <Solve>→<Point Cloud Smoothing>.
2. The default point cloud displayed in the point cloud file list is the point cloud checked and displayed in the air strip editing panel, check the point cloud that needs to be smoothed by the point cloud, and set the radius of search, which is 0.2 by default.
3. Click [OK], point cloud smoothing generates point cloud files with a "processed" suffix.
4. If you use the hlz format point cloud for point cloud smoothing, the finished point cloud will be in the hlz3.4 format.

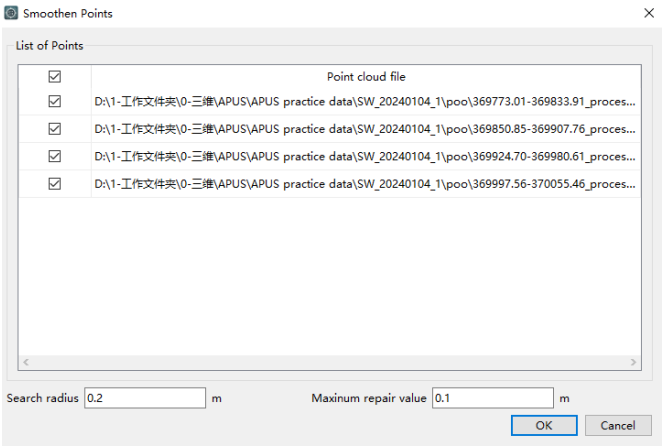

Figure 3-64 Point Cloud Smoothing dialog box

3.3.8 Point Cloud Colorize

Airborne point cloud color assignment includes two aspects: Original Image and ortho image.

1. Original Image colorize

Click "Point Cloud Colorize" to open the point cloud color assignment dialog box.

(1) if the corresponding combine project has been opened, the corresponding point cloud will be loaded automatically, if the fused point cloud in the cut tape of the tape editing panel will also be loaded into the interface automatically.

(2) If use the hlz format point cloud for original image colorize, the point cloud after colorize will be in hlz3.4 format.

- Enter point cloud catalog: default is POO catalog.
- Output point cloud catalog: the output point cloud file name is automatically named as the original file name plus "_colorize".

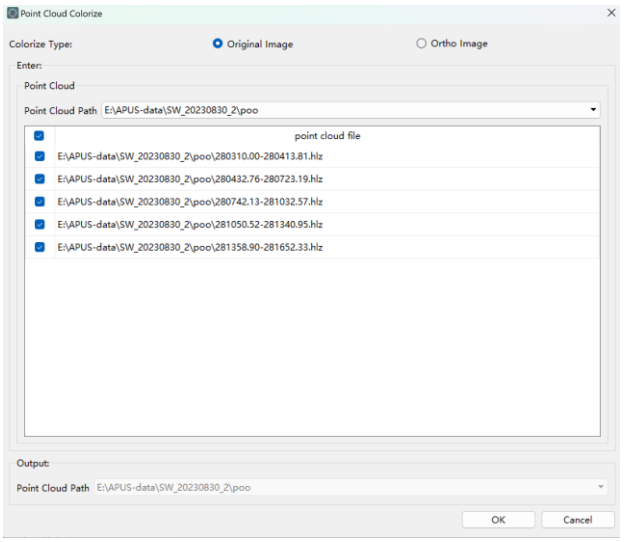

Figure 3-65 Point cloud colorize dialog box

The effect of color assignment is shown in Figure 3-66 and Figure 3-67:

Figure 3-66 Overall point cloud Original Image colorize effect

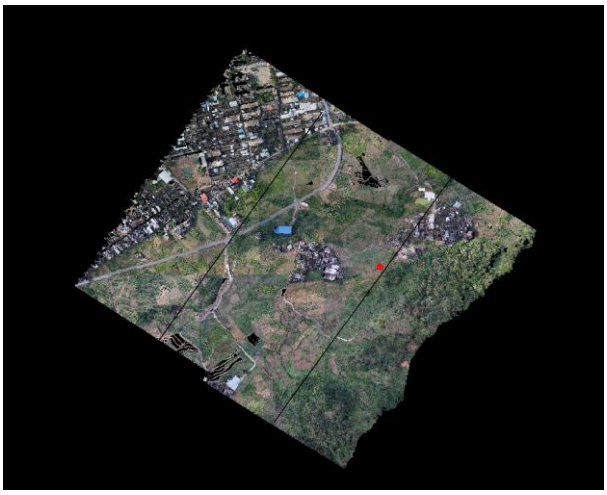

Figure 3-67 Original Image colorize

2.Ortho Image colorize

(1) If a folder named DOM exists in the project directory, and the folder holds orthophotos in TIF format, it will be loaded into the interface automatically;

(2) If the DOM folder does not exist, you need to manually select the path of the tif orthophotos.

(3) If use the hlz format point cloud for Ortho Image colorize, the point cloud after colorize will be in hlz3.4 format.

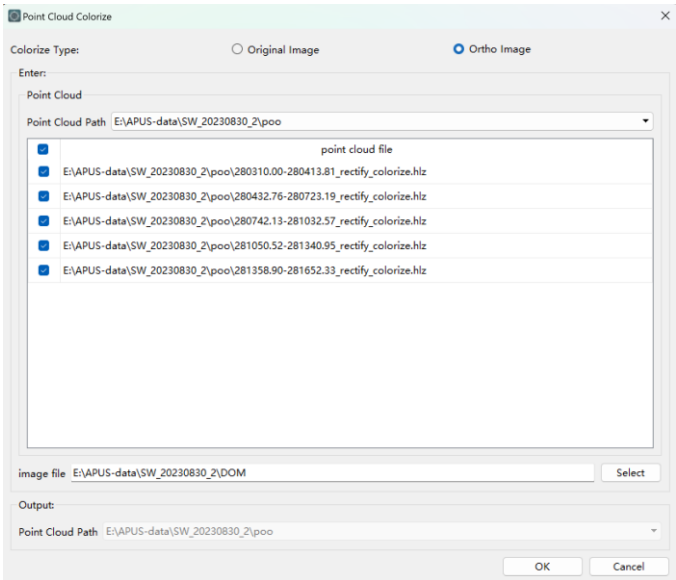

Figure 3-68 Ortho Image colorize Dialog Box

(4) Select a project in the project management panel, right-click <Point Cloud Colorize> to assign color to a single project, or click <Solve> → <Point Cloud Colorize> to assign color to all the checked projects.

(5) After the color assignment is completed, it will automatically reload the successfully assigned point cloud and display the color assignment effect in true color rendering (Figure 3-69).

Figure 3-69 Ortho Image colorize effect

3.3.9 Point Cloud Export

Click menu <Solve>→<Point Cloud Export> to bring up a dialog box (Figure 3-70). Users can select the desired export options, output format, point cloud attributes to customize the export.

Export options:

- Regular Export: complete export.
- Segmented Export: each original scan file is exported to multiple target point cloud files according to GNSS time segments.
- Rectangle Framing: each original scanned file is exported to multiple point cloud files according to the polygons in the polygon file, and the polygons involved in the export can be set by the field values (Figure 3-71).
- Volume Export: the point cloud data can be exported to multiple files according to the set size (this setting is only valid for las format point cloud).
- Output formats: include hlz, las and xyz, and the export format can be multi-selected.

Click the Export button to export the selected point cloud.

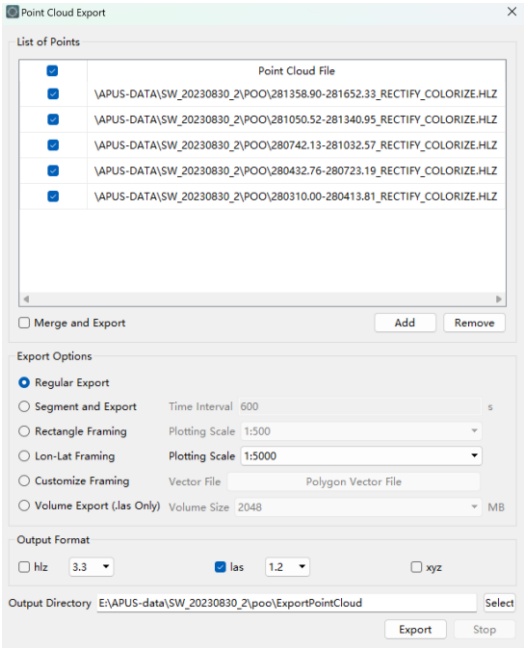

Figure 3-70 Main interface of point cloud export

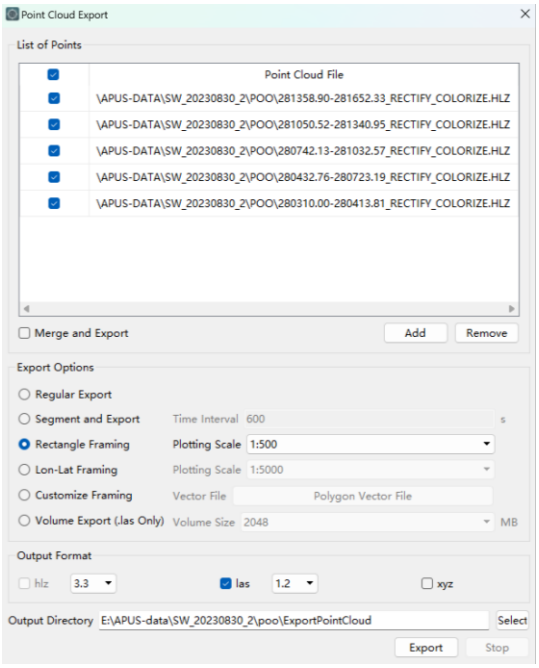

Figure 3-71 Split export settings

3.3.10 Related Control Points

1. Click the menu <Solve>→<Related Control Points> to support the associated txt format and csv format control points.
2. After selecting the control point file, click OK to open the ASCII file window (Figure 3-72), supports the setting of how many lines in front of the control point is ignored;
3. You can also set the control point of the specific meaning of each column of data (control point name, x, y, z), set up, click on the application, the control point is associated with the current point cloud.Support associated control points in txt format and cpt format.

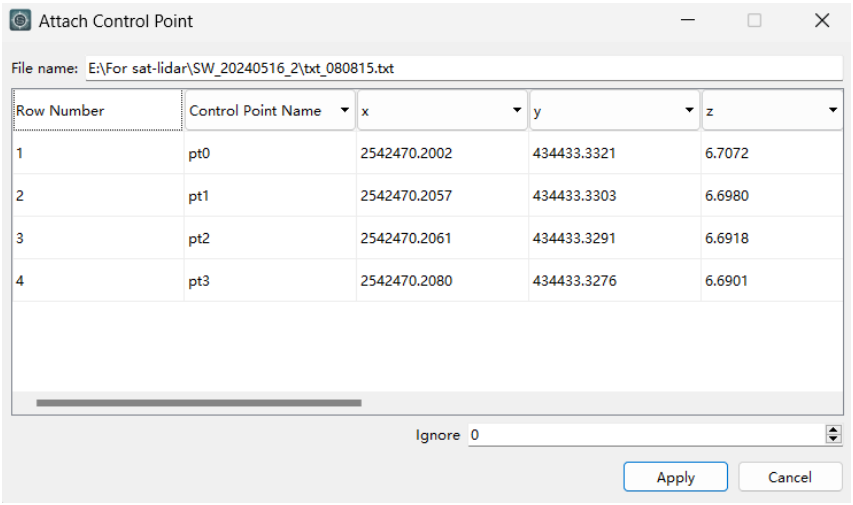

Figure 3-72 Related Control Points

3.3.11 Accuracy Inspection

Load the point cloud and associate it with the control points of the point cloud, click <Solve>→ <Accuracy Inspection> (Figure 3-73).

Accuracy check provides two ways to select points manually and automatically.

1. Manual point selection: Select the scanning point to check the accuracy with the control point to get the result of accuracy checking.
   (1) Click "Accuracy Check" to open the magnifying glass view for point selection. Left-click to select the matching scanning point for each control point.
   (2) After the scanning point is successfully selected, the right side of the control point in the control point list will be filled with the corresponding scanning point (Figure 3-74).
   (3) If you are not satisfied with the selection, re-locate or manually adjust the scanning point coordinates and press Enter to confirm.
   (4) After selecting all scanning points, click "Display Results" to calculate data accuracy (Figure 3-75).
   (5) Click "Export Report" to save the report and jump to the report saving interface (Figure 3-76).
2. Auto Search: Accuracy checking is carried out by means of automatic searching of elevation points by the program, and the auto search function is generally applicable to the points of elevation type.
   (1) Clicking Auto Search, the program will automatically detect the coordinate position of the scanning point corresponding to the control point (Figure 3-77), the other operations are consistent with Quick Point Selection.

The center error σ_x in the z-direction is calculated by the formula:

$$\sigma_x = \sqrt{\frac{\sum_{i=1}^{n} X_i^2}{n}}$$

The center error σ_y in the z-direction is calculated by the formula:

$$\sigma_y = \sqrt{\frac{\sum_{i=1}^{n} Y_i^2}{n}}$$

The center error σ_z in the z-direction is calculated by the formula:

$$\sigma_z = \sqrt{\frac{\sum_{i=1}^{n} Z_i^2}{n}}$$

The formula for calculating the center error σ_{plane} in the plane is:

$$\sigma_{plane} = \sqrt{\sigma_x^2 + \sigma_y^2}$$

where X_i、 Y_i、 Z_i indicate the coordinate error values of the scanning point and the control point in X, Y and Z directions respectively.

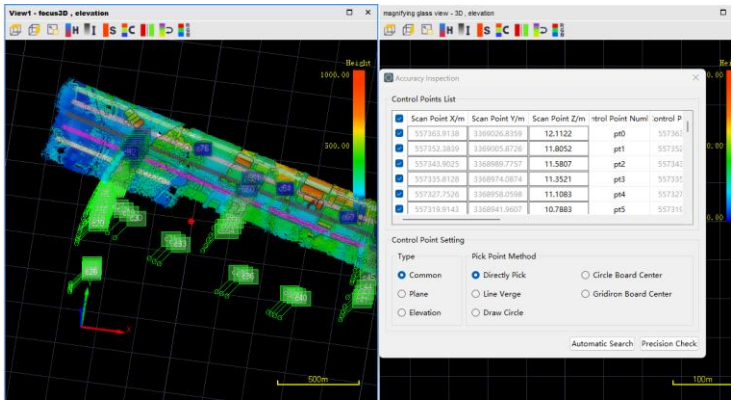

Figure 3-73 Accuracy Inspection main interface

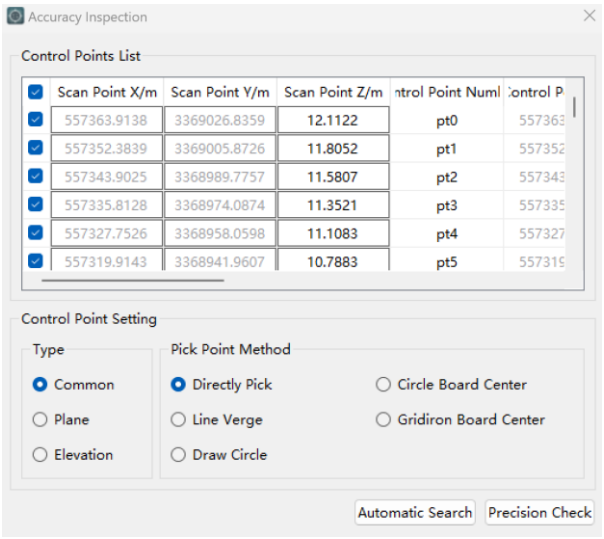

Figure 3-74 Starting point selection

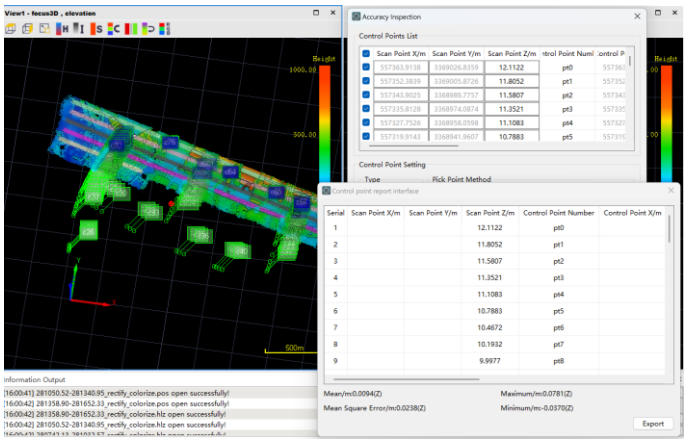

Figure 3-75 Accuracy Inspection

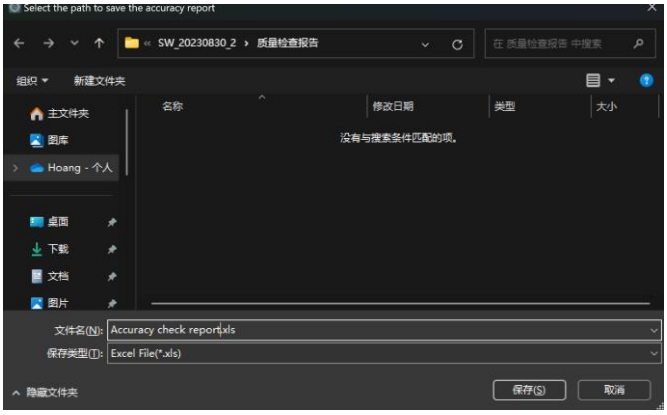

Figure 3-76 Export Report

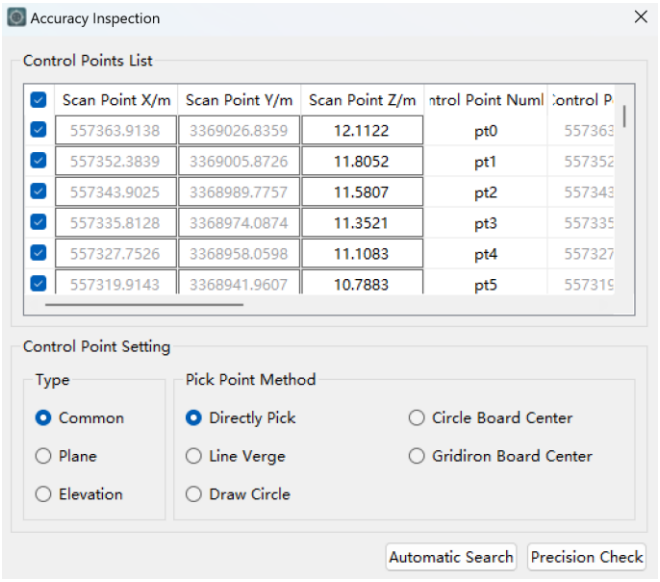

Figure 3-77 Accuracy Inspection interface after auto search

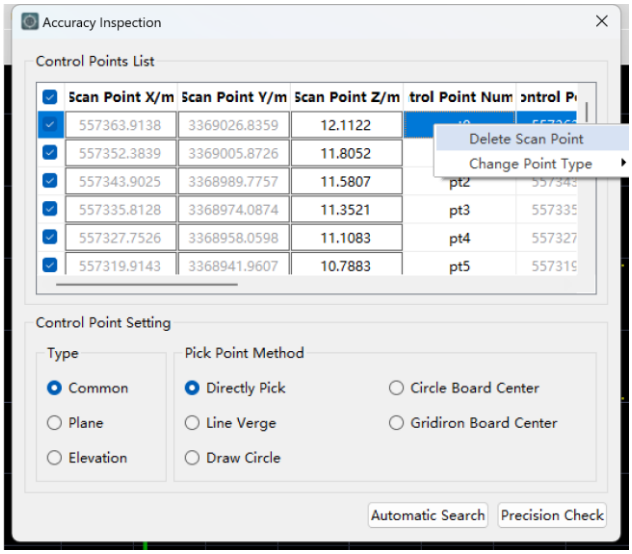

Figure 3-78 Delete Scan Points

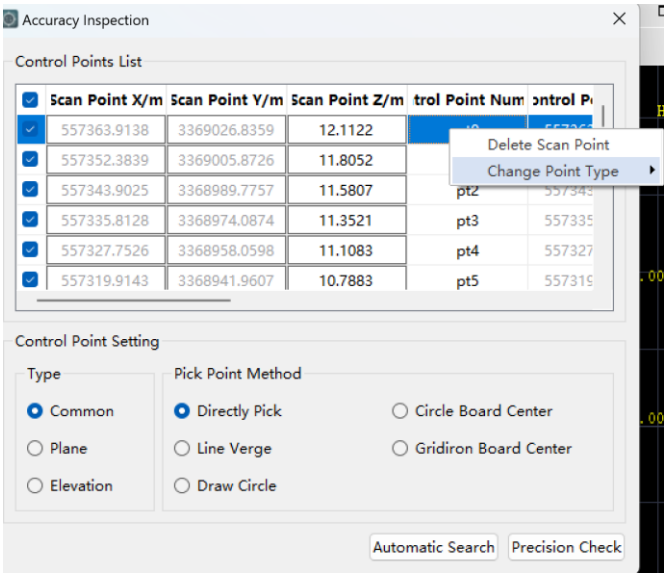

Figure 3-79 Changing point type

Types of control points: including ordinary, plane and elevation control points, plane control points are only valid for X and Y coordinates, elevation control points are only valid for Z coordinates.

Point selection methods: ordinary point selection, fitting intersection, fitting center of circle, fitting circular target, fitting checkerboard target.

- Ordinary point selection: directly in the magnifying glass view of the left click to select the point; fit intersection is based on the identification of the intersection of two lines to determine the point of selection (Figure 3-80).
- Fit the center of the circle: click on the magnifying glass view to select two points to determine the diameter of the circle, move the mouse to determine the shape of the circle, at this time the center of the circle is to determine the point (Figure 3-81).
- Fitting a circular target: based on the selected point cloud, the center of the target is automatically fitted (Figure 3-82).
- Fitting a checkerboard target: select the center point position for the checkerboard grid points (Figure 3-83).

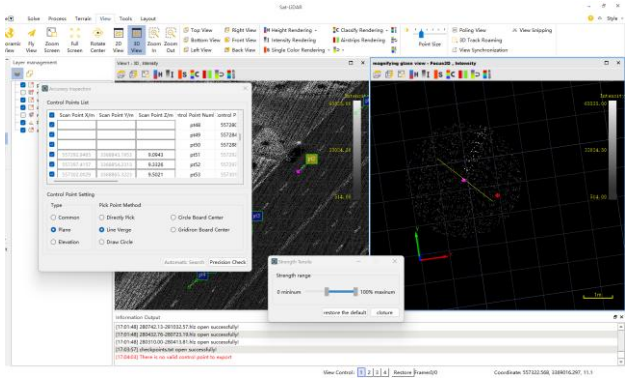

Figure 3-80 Fitting the intersection

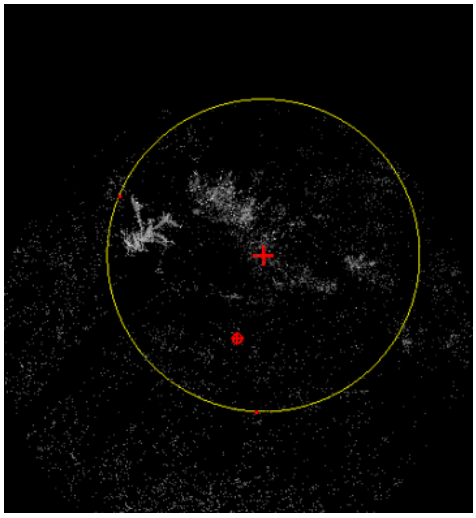

Figure 3-81 Fitting the center of a circle

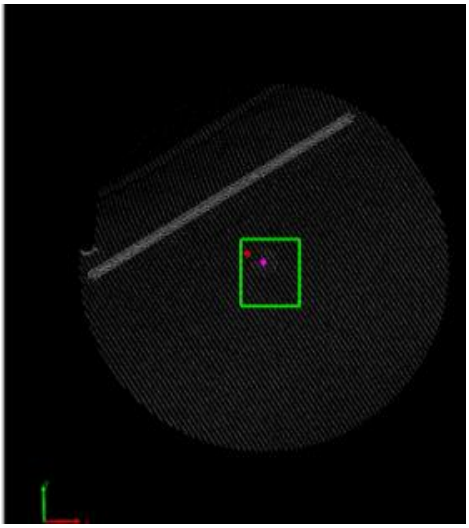

Figure 3-82 Fitting Circular Targets

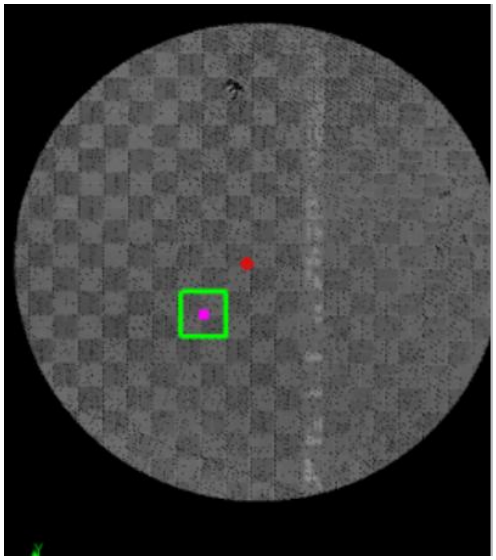

Figure 3-83 Fitting a checkerboard target

3.3.12 Point Moving

Load the relevant point cloud data, click <Solve>→<Point Moving>. Point Panning provides the setting of xyz three-direction panning amount, enter the value, click "OK", automatically loaded into the view when done.

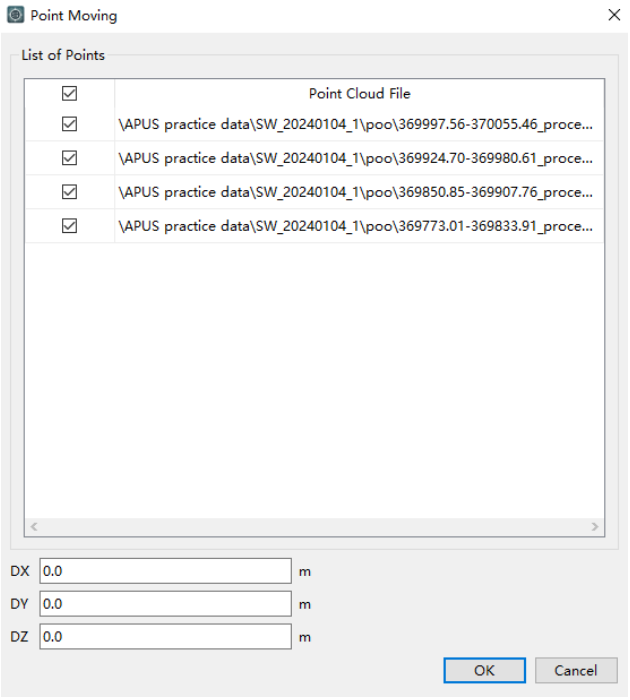

Figure 3-84 Point Moving dialog box

3.3.13 Original Data Check

1. Open the combine project in the project management panel and click <Solve>→<Quality Check> → <Original Data Check> to open the "Original Data Check" box (Figure 3-85).

2. The original data quality check includes four items: data integrity check, scan data check, image data check and pos data check. Select the check items to be checked by checking the boxes.

- Data Integrity: Check whether there is any missing scan data, image data and pos data.
- Scan data: Checks the size of the original scan data.
- Image data: Check if the image and syn match.
- POS data: Check if there is any abnormality in pos data. This is done by setting a change threshold for the POS anomaly check item, above which it will be detected and reported as an anomaly.

3. Optionally fill in the survey area, weather, inspectors, and related information.

4. Click "Quality Inspection and Export" to check the raw data.

5. Once completed, the PDF report is automatically exported, and the folder opens.

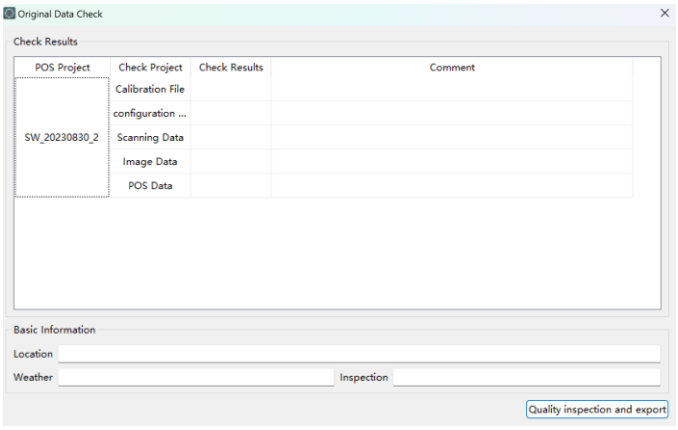

Figure 3-85 Original data check screen

3.3.14 Flight Route Quality Check

Open the combine project in the project management panel, load the display point cloud file.

1. Click <Solve> → <Quality Check> → < Flight Route Quality Check> (Figure 3-86).
2. Click button 【⊕】 to add the cropped POS track in the track list.
3. Click button 【⊖】 to delete the selected POS track from the track list.
4. Fill in the location of the survey area, weather conditions, and inspectors.

The system will automatically recommend the altitude, speed and curvature parameters according to the data, and the user can set the parameters according to the project requirements.

After the setting is completed:

1. Click ''Start Inspection'', the route quality will be inspected, the results of altitude, speed and curvature will be displayed in the Route Quality Inspection interface (Figure 3-86).
2. Optionally fill in the measurement area, weather, and inspector details.
3. Click "Quality Check and Export" to begin the route inspection.

The report will be automatically generated in PDF format, and the folder will open upon completion.

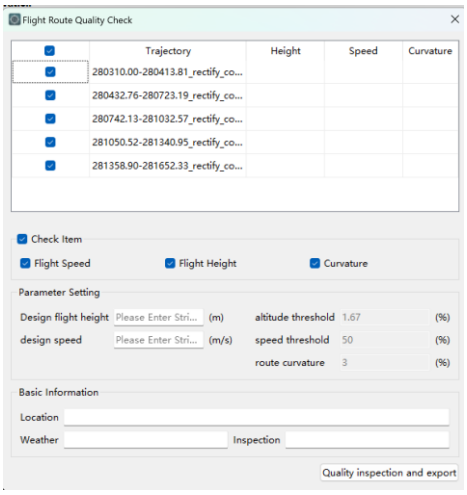

Figure 3-86 Flight Route Quality Check dialog box

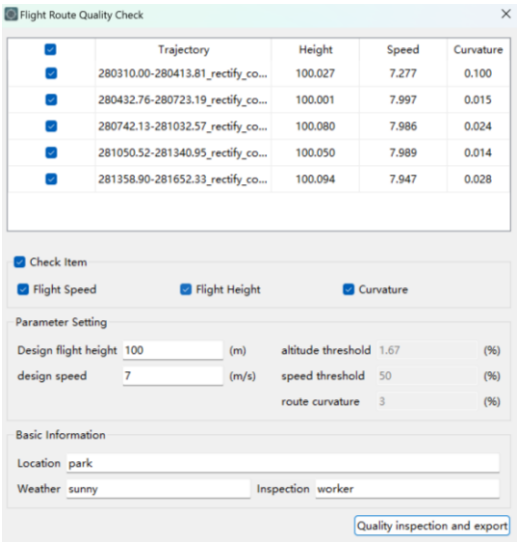

Figure 3-87 Flight Route Quality Check Results

(1) Altitude Analysis: The report contains the name of each air strip, the minimum altitude, maximum altitude, average difference between the maximum and minimum altitude, altitude variance, average altitude of the air strip, and whether the air strip is qualified or not according to the design altitude and altitude tolerance set by the user.

(2) Speed Analysis: The report contains the name of each air strip, the minimum air speed, maximum air speed, average difference of maximum and minimum air speed, air speed variance, average air speed, as well as the speed consistency parameter set by the user to judge whether the air strip is qualified or not.

(3) Flight Attitude Analysis: The report contains the name, maximum roll angle, average roll angle, maximum pitch angle, average pitch angle, course curvature (%) of each air strip, and whether the air strip is qualified or not according to the course curvature threshold set by the user.

3.3.15 Strip Overlap Analysis

1. Open the combine project in the project management panel, load the display point cloud file.
2. Click the menu <Solve> → <Quality Check> → <Strip Overlap analysis> (Figure 3-88).
3. The default grid size is set to 2 meters, and the overlap threshold is 30%. After clicking the Quality Check button, the program will check the overlap rate of the zones (Figure 3-89).
4. You can selectively fill in the location of the survey area, weather conditions, inspectors and related information.
5. Click on the "quality check and export ", the program begins to carry out the overlap analysis of the zones.

After the completion of the inspection, the report is automatically exported (pdf format), the report automatically opens the folder after the completion of the folder.

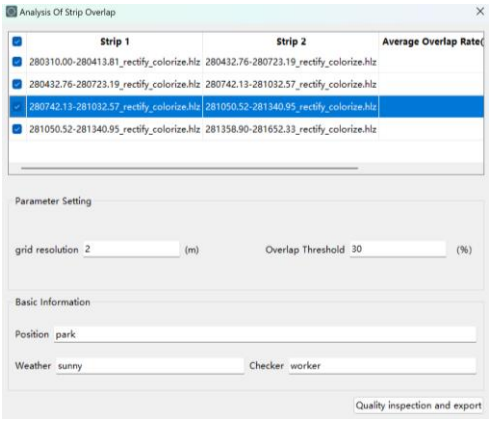

Figure 3-88 Analysis Of Strip Overlap

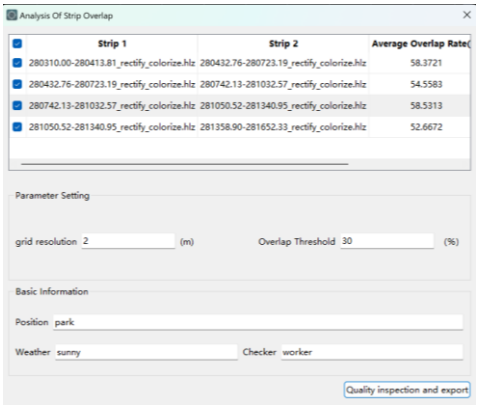

Figure 3-89 Strip Overlap Analysis Results

3.3.16 Height Different Quality Check

Open the combine project in the project management panel, then load and display the point cloud file. In the main interface, click <Solve> → <Quality Check> → <Height Difference Quality Check> to open the dialog box (Figure 3-90).

- The default grid size is 2 meters, and the critical threshold is 0.5 meters. Click the <Quality Check> button to start the check with a 30% threshold.
- Optionally, fill in the survey area location, weather conditions, and inspector details. Once you click the quality check button, the program will perform the height difference quality check, with results displayed in Figure 3-91.

After the check, you can optionally provide the survey area location, weather conditions, and inspector details, then click "Quality Check and Export." The program will perform the height difference quality check again and automatically export the report in PDF format, which will open automatically once complete.

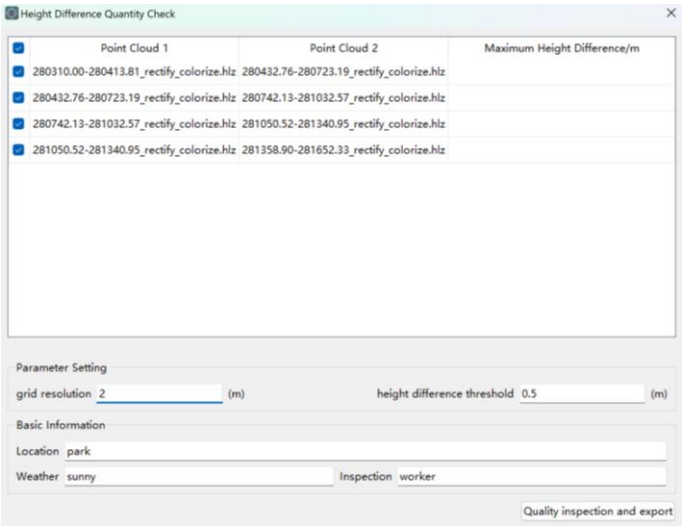

Figure 3-90 Height Different quality check dialog box

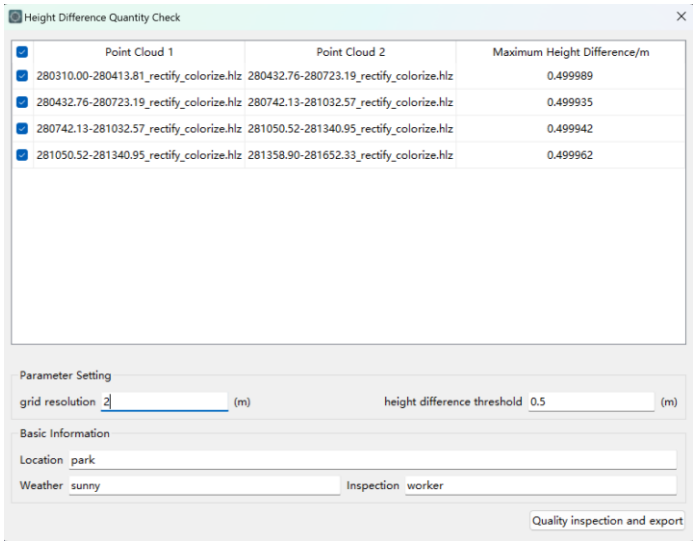

Figure 3-91 Result of height Different quality check

3.3.17 Density Quality Check

1. Open the combine project in the project management panel, load and display the point cloud files.
2. Click <Solve> → <Quality Check> → <Density Quality Check> in the main interface to pop up the dialog box (Figure 3-92).
3. The default grid size is 2 meters, and the average density threshold is 10 points per square meter. You can add or remove point cloud files as needed. Click "Start Check" to begin the density quality check, and the results will be displayed (Figure 3-93).
4. You can optionally fill in the survey area location, weather conditions, and inspector information.
5. Click "Quality Check and Export" to start the density quality check.

Once the check is complete, the program will automatically export the report in PDF format, which will open automatically when finished.

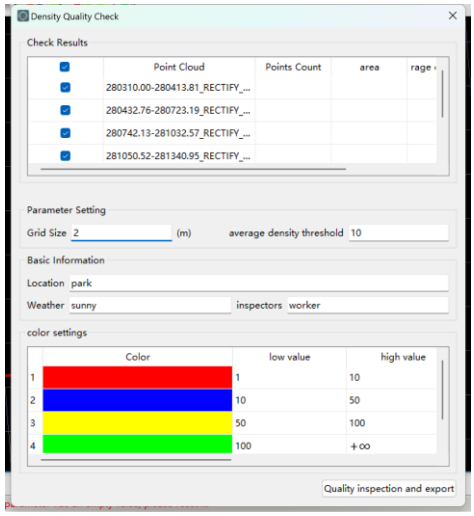

Figure 3-92 Density Quality Check dialog box

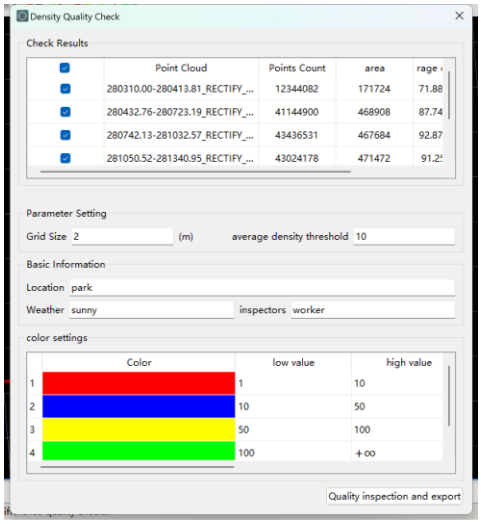

Figure 3-93 Density quality check results

3.3.18 Coverage Quality Check

1. Open the combine project in the project management panel, load and display the point cloud files.
2. Click <Solve> → <Quality Check> → <Coverage Quality Check> in the main interface to pop up the dialog box (Figure 3-94).
3. Set the grid size parameters as needed, and import the range line files from the top view of the cloud (.shp/.dxf/.dwg).
4. Optionally, fill in the survey area location, weather conditions, and inspector information.
5. Click "Quality Check and Export" to start the completion rate check.

Once the check is finished, the program will automatically export the report in PDF format and open the folder containing the report.

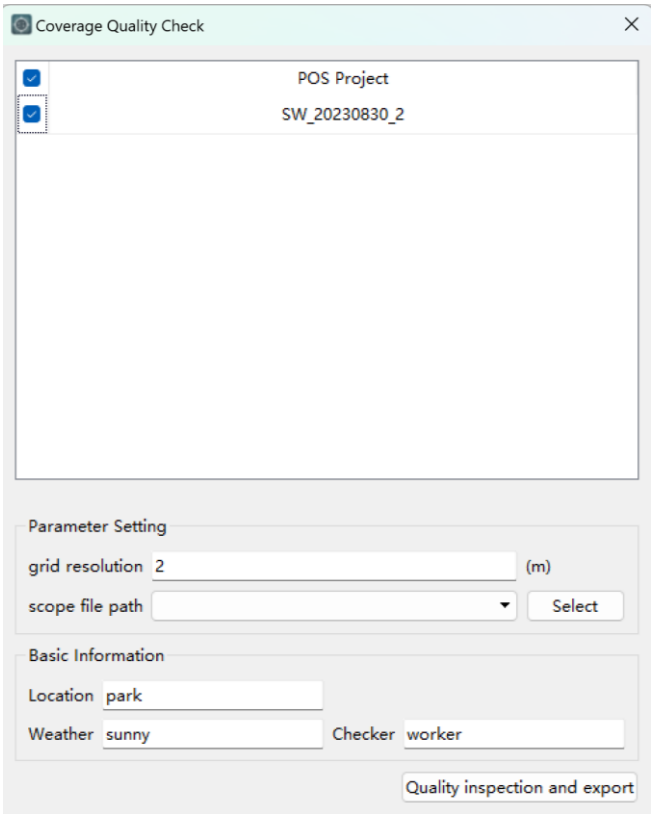

Figure 3-94 Point cloud range check dialog

3.3.19 Automatic Quality Check

This tool allows users to run multiple quality checks in one click and export a report.

Steps:

1. **Open Project**

In the **Project Management** panel, open the fusion project and load the point cloud.

2. **Start Auto QC**

Click **Solution > Quality Check > Auto Quality Check**.

The **Auto QC** window will open.

3. **Select Checks & Set Parameters**
   - Select the quality check items you need (all are selected by default).
   - Input the required parameters or files for each check.
   - (Optional) Fill in survey location, weather, and inspector name.
4. **Run and Export**

Click **Run Check and Export**.

The software will process the selected checks and generate reports.

Reports are saved in the **"Quality Check"** folder under the POS project directory.

Notes:

- If **Flight Line Check** is selected, you must input **designed flight height** and **designed speed**. Otherwise, an error will be shown, but other checks will still run.
- If **Coverage Check** is selected, the boundary file must be **fully covered by the point cloud top view**. If not, an error will be shown, but other checks will still run.
- If **no checks** are selected, the software will show an error and stop.

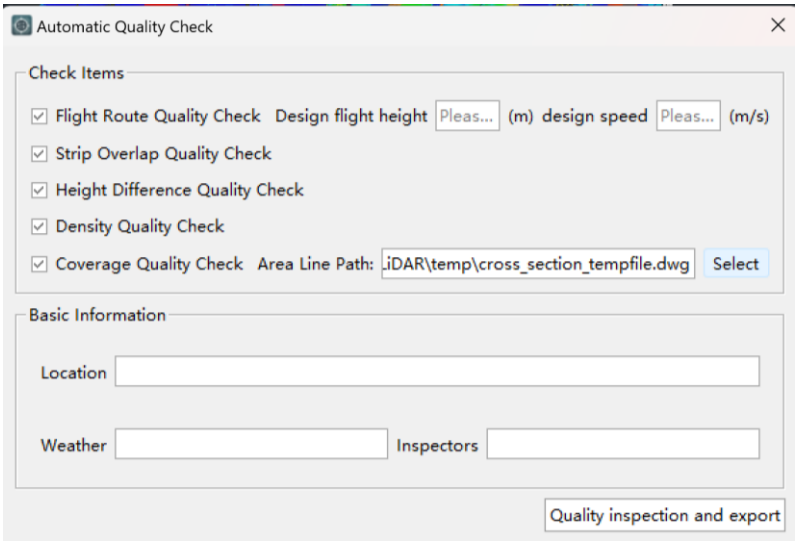

Figure 3-95 Automatic Quality Check dialog box

3.3.20 Point Cloud Processing

Click menu <Solve>→ <Point Cloud Processing>, it will close the current project, uninstall the project, and load the hlz point cloud in the layer panel.

- If there is a colorized point cloud, i.e., the file name has "449480.99-449507.01_colorize.hlz", it will prioritize to load colorized point cloud.
- If there is no colorized point cloud, it will still load the fused point cloud.

3.3.21 Generate DOM

When generating an orthophoto (DOM) from point clouds, please note that this feature has minimum hardware requirements. An NVIDIA graphics card is required, with a model RTX 2060 or above, and at least 4 GB of VRAM. The computer must also have CUDA version 10 or higher installed.

If the system does not meet these requirements, the software will display a warning indicating that the computer configuration is insufficient.

When the hardware requirements are met, the airborne dataset to be used for DOM generation has been loaded, and the trajectory has been processed and trimmed, clicking Generate DOM will open the configuration dialog. Two processing options are available: Quick Mode and Professional Mode.

Figure 3-95 Generate DOM

Quick Mode:The DOM is generated directly within Hi-Lidar. Once the log indicates that the DOM generation is complete, you can find the resulting TIF file in the project data directory.

3.4 Tools

3.4.1 Measurement tools

1. Measure 3D Distances

(1) Click the menu <Tools> → <Measure 3D Distances>.

(2) Click the left mouse button to draw, the measured distance data and line will be displayed in the view.

(3) Double-click on the left mouse button to end the measurement line drawing (Figure 3-96).

(4) Cancel: click <Measure 3D Distance> again.

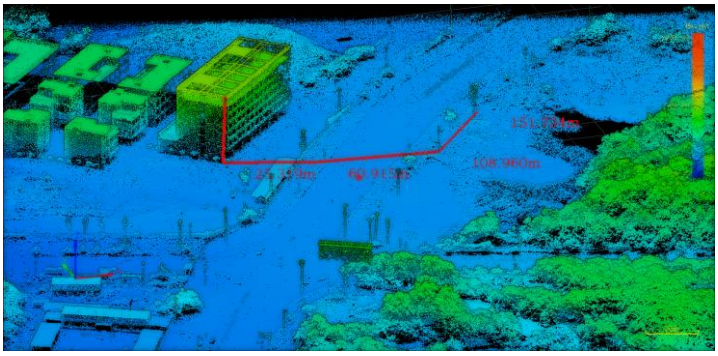

Figure 3-96 Measure 3D distances

2. Measure 2D Distance

(1) Click the menu <Tools>→<Measure 2D Distance>.

(2) Click the left mouse button to select the node in the two-dimensional view of the measurement line drawing, the measured distance data and line will be displayed in the view.

(3) Double-click on the left mouse button can be the end of the measurement line drawing (Figure 3-97).

(4) Cancel: click <Measure 2D Distance> again.

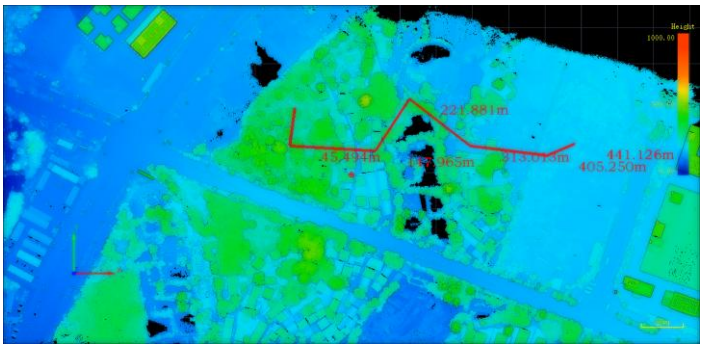

Figure 3-97 Measure 2D distance

3. Measure Density

(1) Click the menu <Tools> → <Measure Density>.

(2) Press the left mouse button to select the scope of the density, lift the left mouse button, the

density of the box within the scope of the point data will be displayed (Figure 3-98).

(3) Cancel: click <Measure Density> again.

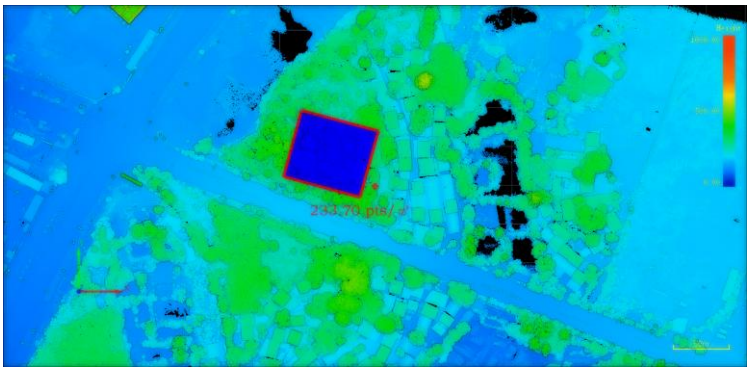

Figure 3-98 Measure Density

4. Measuring Slope

(1) Cick the menu <Tools> → <Measure Slope>.

(2) Click the left mouse button to select the two points that need to calculate the slope, draw the slope line (Figure 3-99), the drawn slope value is displayed in real time on the view.

(3) Cancel: click <Measure Slope> again.

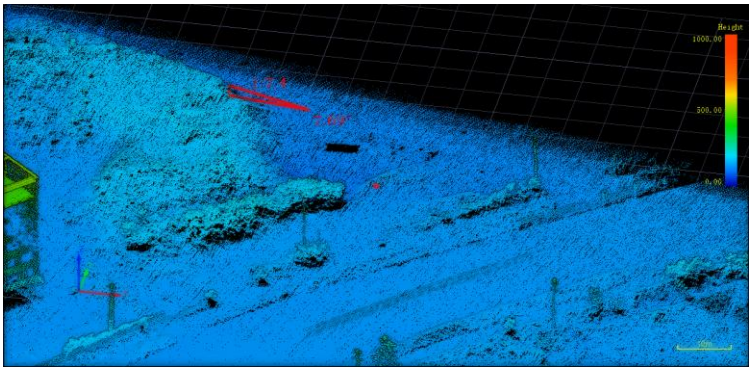

Figure 3-99 Measure the slope

5. Measure Angle

(1) Click the menu <Tools> → < Measure Angle >.

(2) Click the left mouse button in the view to select the two points where the angle needs to be calculated, and then draw an angle line (Figure 3-100), the drawn angle value is displayed in real time on the view.

(3) Cancel: click < Measure Angle > again.

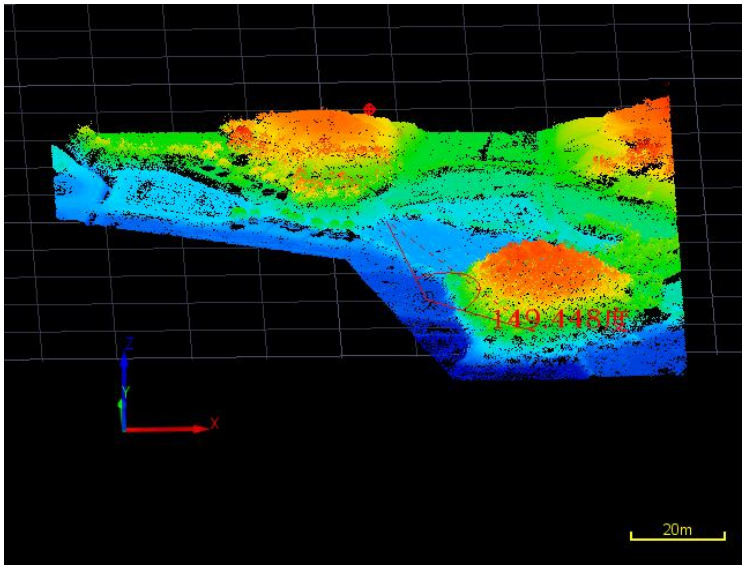

Figure 3-100 Measuring Angle

6. Measure Height

(1) Click the menu <Tools> → <Measure Height>.

(2) Click the left mouse button to select the two points that need to calculate the height, and draw a height line (Figure 3-101), the height value is displayed in real time on the view.

(3) Cancel: click <Measure Height> again.

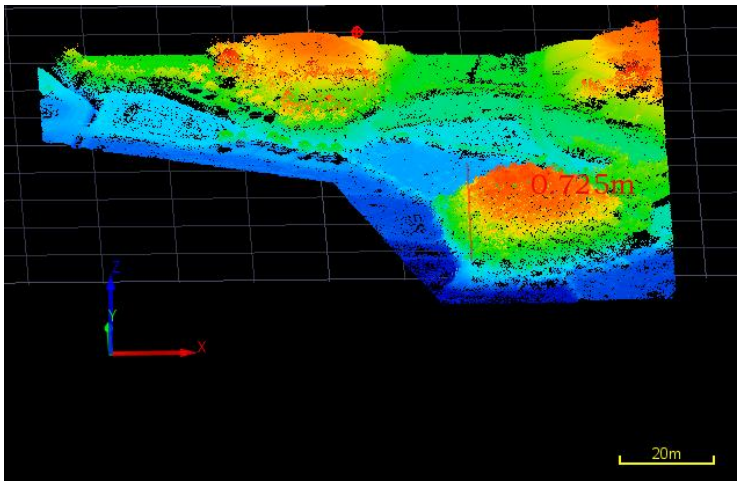

Figure 3-101 Measure Height

7. Measure Area

(1) Click the menu <Tools> → <Measure Area>.

(2) Click the left mouse button to draw polygons at different locations in the view (Figure 3-102).

(3) Double-click the left mouse button to end the drawing, and the value of the area within the polygon will be displayed in real time on the view.

(4) Cancel: click <Measure Area> again.

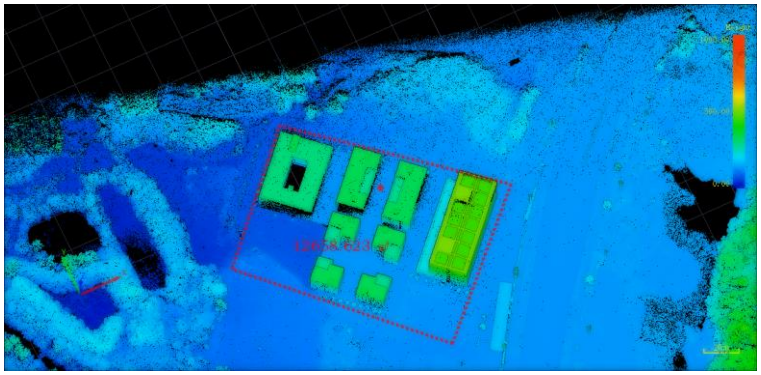

Figure 3-102 Measure Area

8. Measure Volume

(1) Click the menu <Tools> → <Measure Volume>.

(2) Click the left mouse button to draw polygons in different locations in the view, double-click the left mouse button to end the drawing.

(3) The Measure Volume Settings dialog box pops up (Figure 3-103), where relevant parameters can be set. Click the <OK> button, the volume value within the polygon is displayed on the view (Figure 3-104).

(4) Cancel: click <Measure Volume> again.

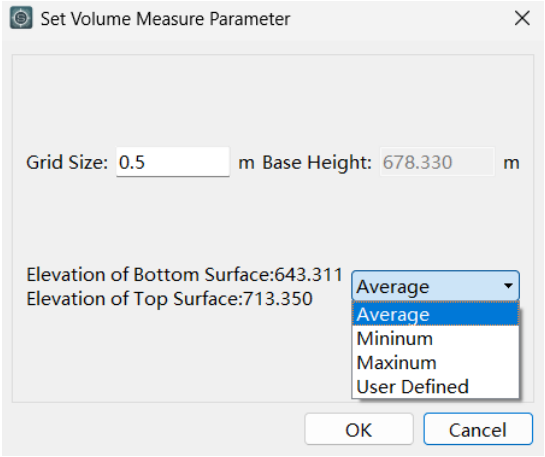

Figure 3-103 Measure Volume setting

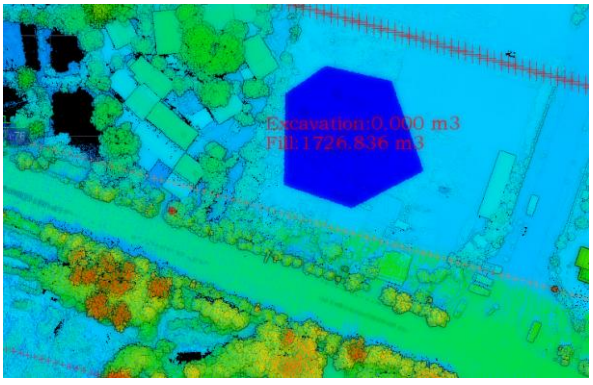

Figure 3-104 Measure Volume

9. Measure Surface Area

(1) Click the menu <Tools> → <Measure Surface Area>.

(2) Click the left mouse button to draw polygons at different locations in the view (Figure 3-105), double-click the left mouse button to end the drawing, set the size of the grid, click OK, and the surface area value within the polygon will be displayed in real time on the view.

(3) Cancel: click <Measure Surface Area> again.

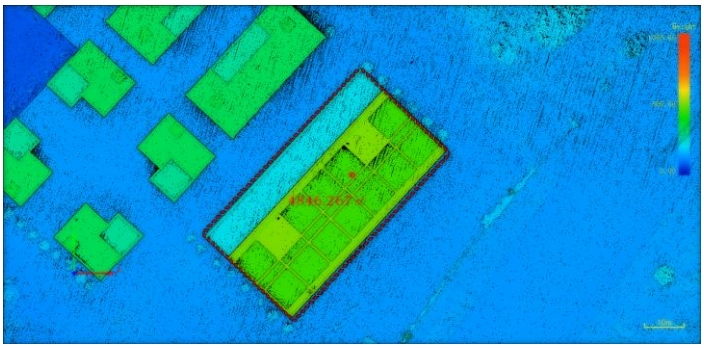

Figure 3-105 Measure Surface Area

3.4.2 Selection tools

1. Polygon Selection

(1) Click the menu <Tools> → <Polygon Selection>.

(2) Click the left mouse button to draw polygons in different locations in the view, double-click the left mouse button to end the drawing, the points within the polygon will be in the selected state and change the color (Figure 3-106).

(3) The selected state of the point cloud color can be set in <File> → <System Settings> → <Selected Points Color>.

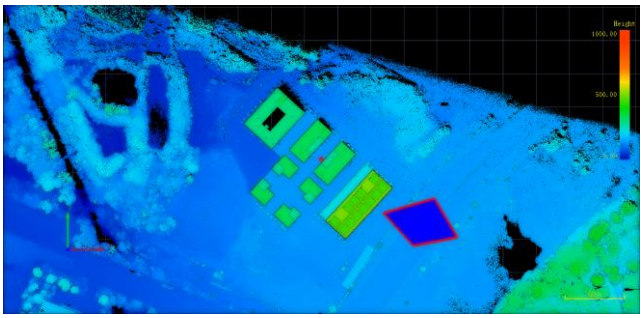

Figure 3-106 Polygon Selection

2. Rectangle Selection

(1) Click the menu <Tools> → <Rectangle Selection>.

(2) Click the left mouse button to draw a line, and then click the left mouse button again to draw a rectangular box, the points within the rectangle will be in the selected state and change the color (Figure 3-107).

(3) The selected state of the color of the cloud of points can be set up in the <Documents> → <System Settings> → <Selected Points Color>.

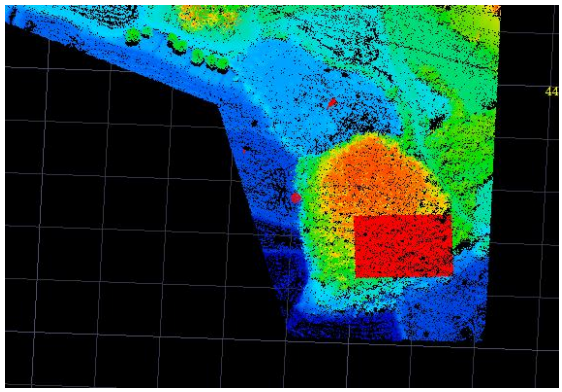

Figure 3-107 Rectangle Selection

3. Square Selection

(1) Click the menu <Tools> → <Square Selection>.

(2) In the view, click the left mouse button to draw a line, and then click the left mouse button again to draw a rectangular box, the points within the rectangle will be in the selected state and change the color (Figure 3-108).

(3) The selected state of the color of the cloud of points can be set up in the <File> → <System Settings> → <Selected Points Color>.

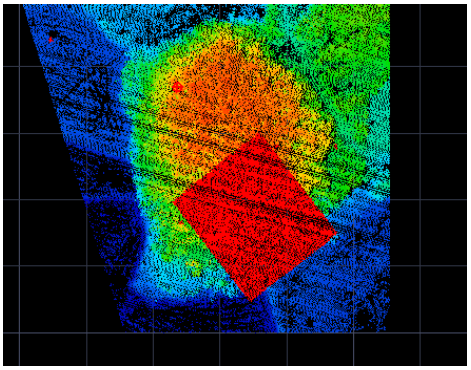

Figure 3-108 Square Selection

4.Circle Selection

(1) Click the menu <Tools> → <Circle Selection>.

(2) In the view, click the left mouse button to determine the location of the center of the circle, move the mouse to the appropriate radius and then click the left mouse button again to draw a circular box, the points within the circle will be in the selected state and change the color (Figure 3-109).

(3) The selected state of the point cloud color can be set up in the <Document> → <System Settings> → <Selected Points Color>.

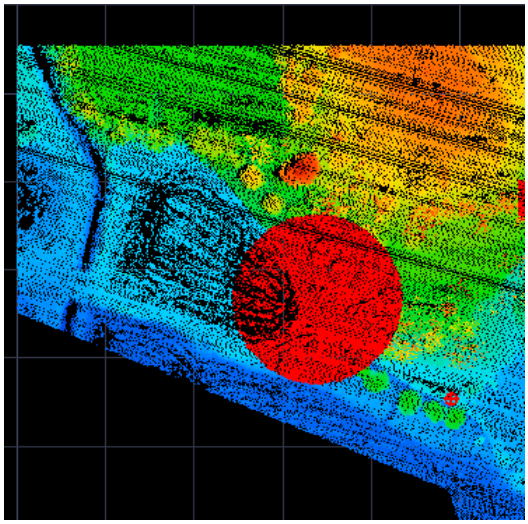

Figure 3-109 Circle Selection

5. Single Selection

(1) Click the menu <Tools>→<Single Selection>.

(2) The point cloud in the view is in the state of selection only at this time, and each time you select a point cloud it will cancel the last selected point cloud.

6. Add Selection

(1) Click menu <Tools>→<Add Selection>.

(2) The selected point cloud in the view is in add selection point cloud state, and all the selected point clouds are set to selected state.

7. Delete Selection

(1) Click menu <Tools>→<Delete Selection>.

(2) The point cloud selected in the view is decrease selection point cloud state, if there is a point cloud that is already in the selected state, cancel its selected state.

8. Cut Unselected Points

(1) Click menu <Tools>→<Cut Unselected Points>.

(2) Only the point cloud with selected status will be retained in the current view (Figure 3-110).

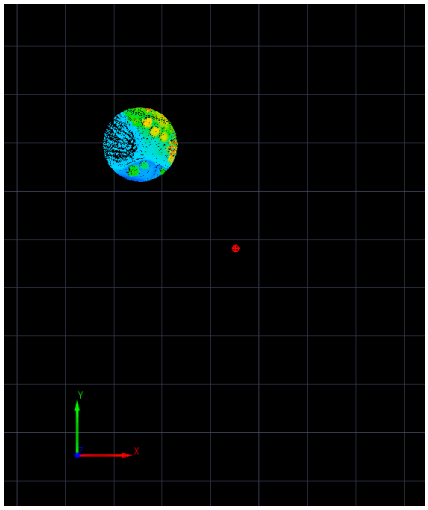

Figure 3-110 Cut Unselected Point

9. Cut Selected Points

(1) Click the menu <Tools>→<Cut Selected Point>.

(2) Only the point cloud with non-selected status will be retained in the current view (Figure 3-111).

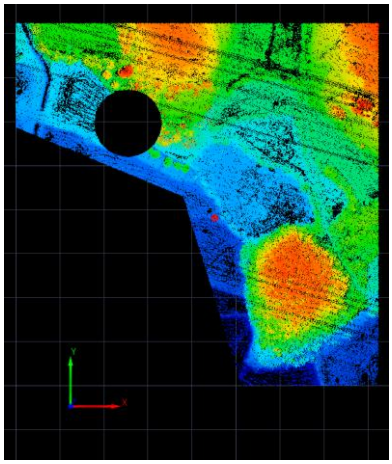

Figure 3-111 Cut Selected Points

10. Revoke Cutting

Click menu <Tools>→<Revoke Cutting> to cancel the Cut Selected Point and Cut Unselected Point and display all point clouds.

11. Export Cutting

The function of Export Crop is to export the cropped point cloud to a separate file.

(1) Click <Tools>→<Export Cutting > to bring up the dialog box (Figure 3-112), set the category of the exported point cloud, the format of the exported point cloud, the version of the exported format, and the path of the exported point cloud.

(2) Click the <>>> button and <Properties> button to bring up the dialog box (Figure 3-113),

select and set the export category and export properties respectively.

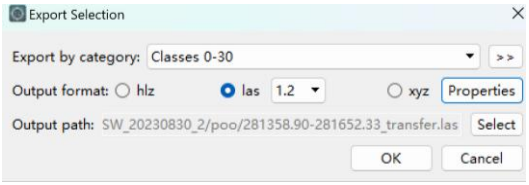

Figure -112 Export Cutting

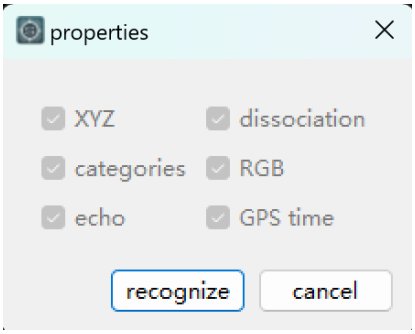

Figure 3-113 Exporting categories and properties

3.4.3 Point selection tools

1.Query Single Point

(1) Click menu <Tools> → <Query Single Point>.

(2) Click the point cloud that needs to be queried in the point cloud view, the pop-up dialog box (Figure 3-114) displays the detailed information of the point cloud, the point information can be copied to the clipboard by clicking <Copy>.

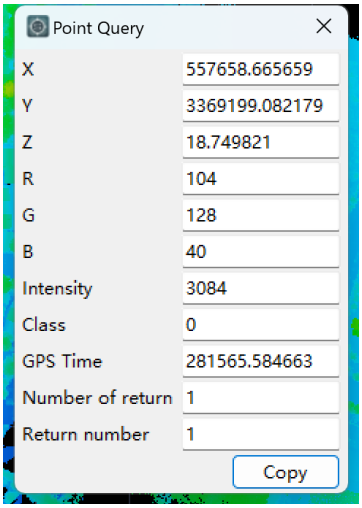

Figure 3-114 Query Single Point details

2. Query Multible Points

(1) Click menu <Tools> → <Query Multible Points>.

(2) A dialog box will be popped up (Figure 3-115), and then click the point cloud that needs to be queried in the point cloud view.

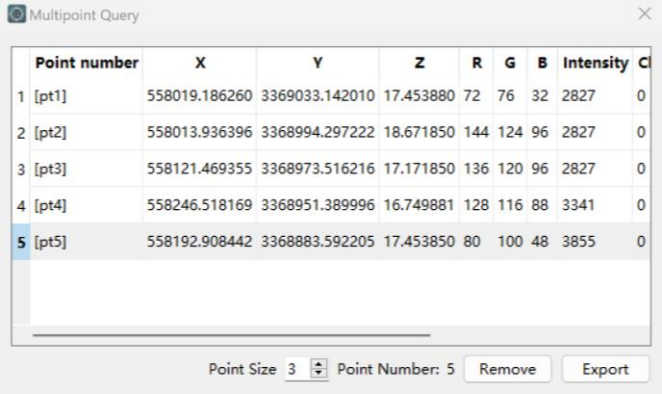

Figure 3-115 Multi-points query pop-up window

(3) Click the point cloud in the view, the basic attribute information of the queried point cloud will be displayed in the multi-point query pop-up window, including: 3D coordinates, RGB, intensity, category and GPS time.

(4) Add labels under the point cloud nodes in the workspace, click the labels in the workspace or the label line in the multi-point query pop-up window, the queried point cloud in the view is in the highlighted state (Figure 3-116), and the point pt1 is displayed in red.

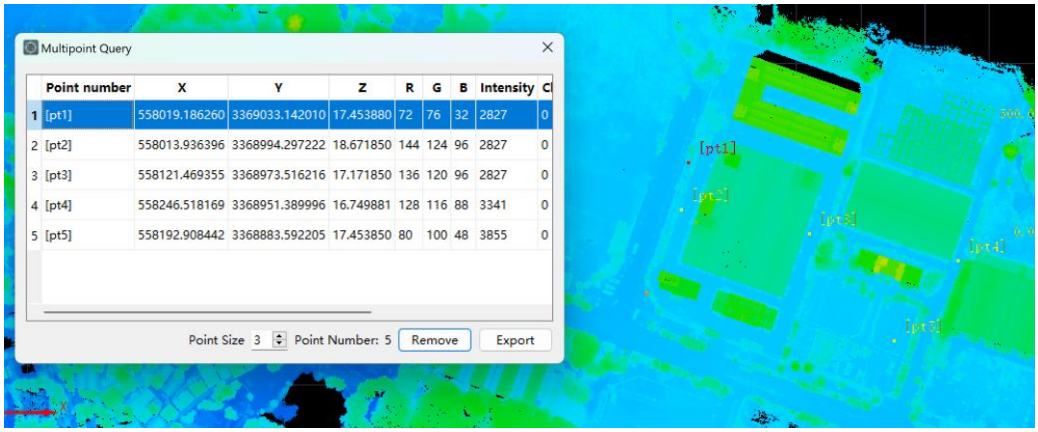

Figure 3-116 Multi-points query display

3. Query POS Points

(1) Load the POS track file. Click menu <Tools>→ < POS point query>, the "Query POS point" tool is activated, the cursor turns into the state of selecting a point 【 】.

(2) Then click on the POS point that needs to be queried in the POS track (Figure 3-117), which displays detailed information of the queried POS point.

(3) You can copy the point information to the clipboard by clicking the button 【Copy】.

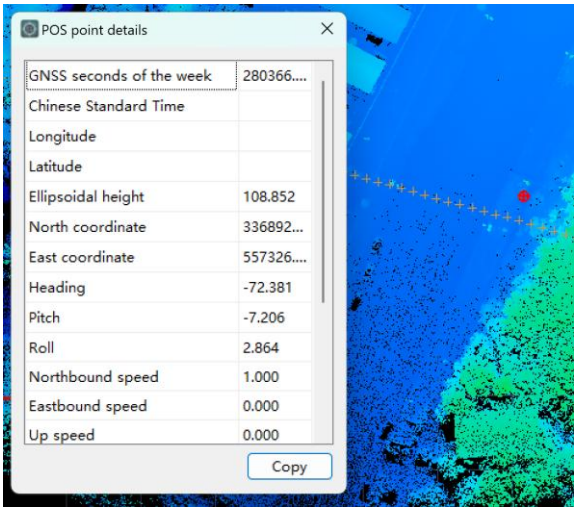

Figure 3-117 Point Details dialog box

4. Label Measurement Points

(1) Click the menu <Tools>→ < Label Measurement Points >.

(2) Select a feature point in the 3D view, click the left mouse button, will pop up the dialog box of the new annotation point (Figure 3-118), which already has a point of information, you can manually name the notes.

(3) Click <Add Annotation> to complete the selection of annotation points (Figure 3-119).

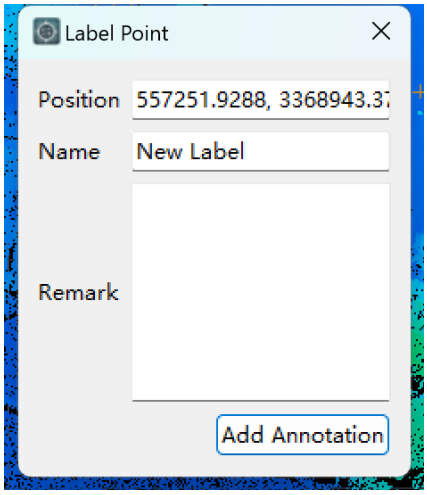

Figure 3-118 The New Annotation Point dialog box

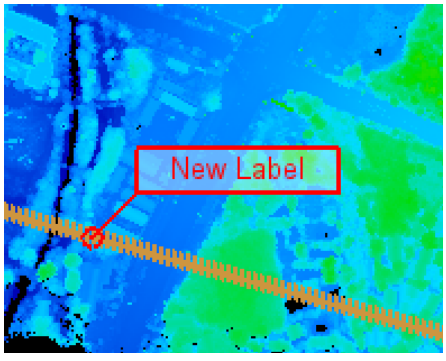

Figure 3-119 Point labeling

(4) After adding a markup point it will be automatically loaded into the markup layer in the workspace (Figure 3-120).

(5) Right-click on the annotation point name to bring up a dialog box and click Remove to remove the annotation point name from the workspace and the annotation point from the view.

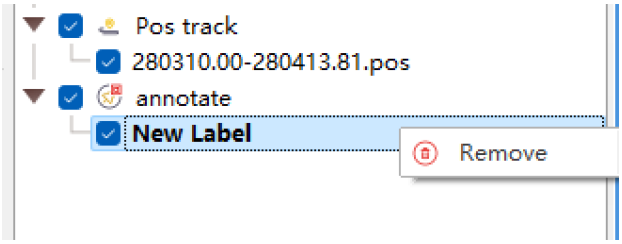

Figure 3-120 Labeling Points in Layers

(6) Click the right mouse button on the labeled node, the pop-up menu Figure 3-121, select <Save Data> in the menu, you can save all the labeled point information under the current labeled layer to a file.

(7) Click <Import Data> in the menu to reload the saved annotation point information into the view and workspace.

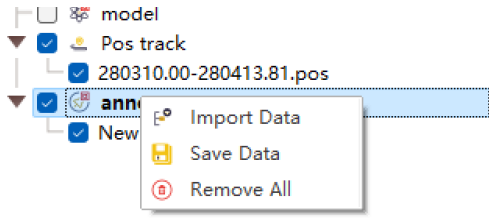

Figure 3-121 Saving and Importing Marking Points

3.4.4 Other tools

1. Point cloud format conversion

(1) Click menu <Tools> → <Point Cloud Format Conversion>.

(2) The point cloud list lists the point cloud files loaded in the current software, check the files that need to be formatted,

(3) The <Merge and Export> check box can be set to whether to merge multiple input files for export, and you can select the format of the exported files, the format version and the path of the exported files below.

(4) Click the <Properties> button, you can select the properties of the exported file.

(5) Click <OK> to complete the export (Figure 3-123).

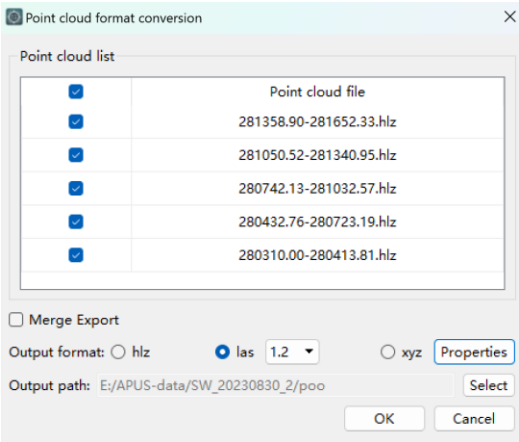

Figure 3-122 Point cloud format conversion

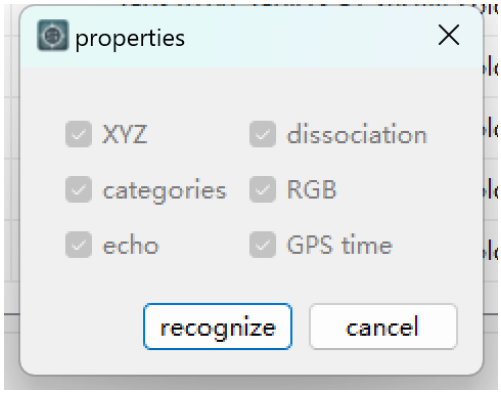

Figure 3-123 Point cloud format conversion properties

2. Remove Strip Edge Point

(1) Click <Tools>→< Remove Edge Strip Point >.

(2) The point cloud files are listed in the list of point cloud files loaded in the current software, check the files that need to be removed from the tape edge points, and then click <Start Execution>.

(3) After the removal is completed, the pop-up dialog box will show "Whether to load the point cloud after cropping? After the removal is finished, a popup will appear, "Do you want to load the removal point cloud?", click "Yes" to load the removal point cloud into the view.

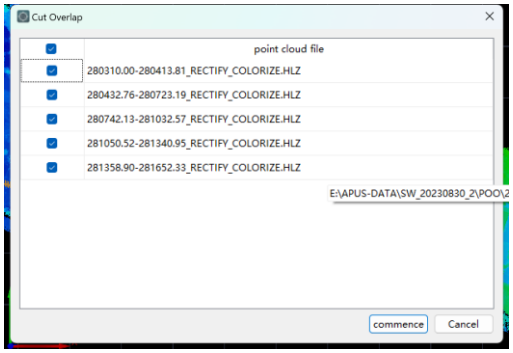

Figure 3-124 Remove Strip Edge Point

3.5 Process

3.5.1 Section View

Click menu <Process> → <Section View> on the main interface, a new Profile View window is added to the main view, and a new Profile Options menu is added to the menu, the contents of the Profile menu will be introduced in subsection 3.8.

3.5.2 Point Cloud Tiling

The main purpose of the point cloud staging is to staging the point cloud according to a certain standard, generate the staging vector information, and then extract each small piece of point cloud through <Process>→<Select Tiling Block> for individual operation.

1. Click menu <Process> →<Point Cloud tiling> to open the dialog box (Figure 3-125). The window lists the point cloud files currently loaded by the software, check the files that need to be staged. When more than one point cloud file is selected, the software will first merge multiple point clouds and then perform the staging.
2. After selecting the file, you need to choose the splitting method and set the splitting parameters, when setting the customized block size and the number of blocks, users can refer to the box size information prompted in the window to make reasonable settings.
3. Click <OK> when the settings are complete, the software will generate the vector file after the splitting and load it to the vector layer in the workspace and the view selected by the user in the splitting parameters (Figure 3-126).

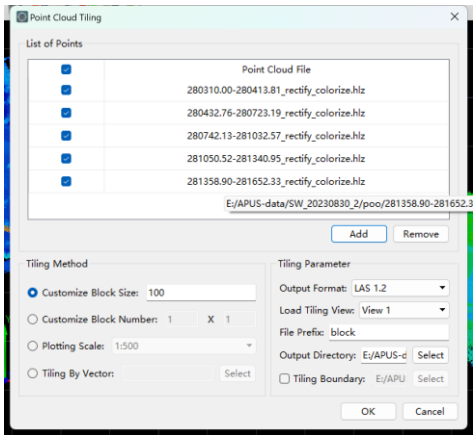

Figure 3-125 Point Cloud Tiling Settings

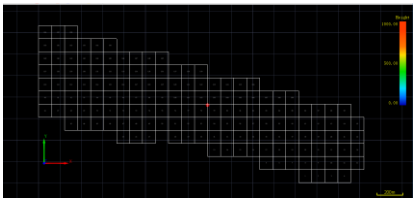

Figure 3-126 Split Vector Generation

3.5.3 Select Tiling Block

1. After generating split vector information through <Process>→<Point Cloud Split>, click menu <Process> → <Select Split Block> to activate the button, and then click on any split block in the Split Vector view to extract the point cloud in that block.
2. After clicking on the 16th split block, the view will only show the point cloud in the block (Figure 3-126), and at the same time, the software will automatically extract the part of the point cloud and generate a point cloud file to be loaded into the workspace, so that the user can operate on the part of the point cloud.
3. If you want to extract the point cloud in more than one block, just click <Process> → <Select Split Block> button again and repeat the above operation.

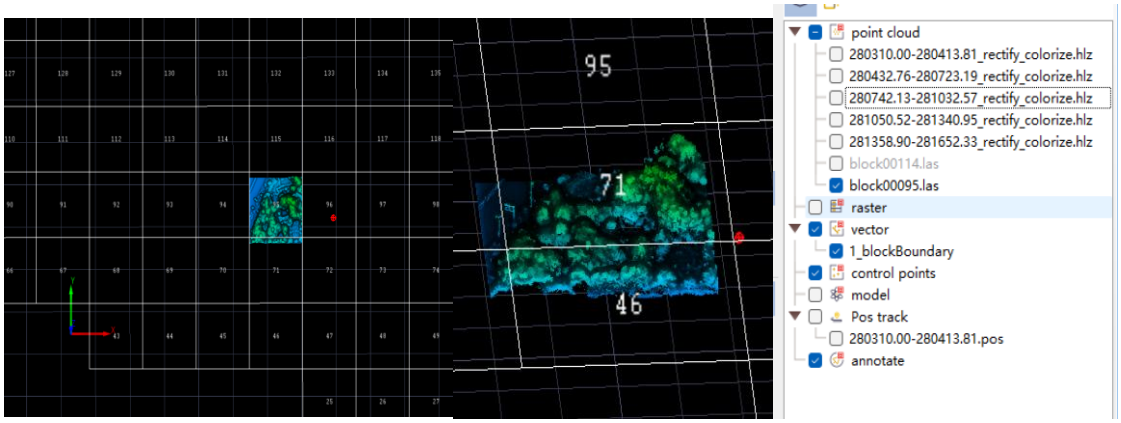

Figure 3-127 Select Split Block

3.5.4 Automatic Classification

1. This version of the software mainly has several classification methods of reset classification, noise filtering and ground point classification, which can realize batch combination operation.
2. In <Process> menu bar, each classification method corresponds to <Reset Classification>, <Noise Filter>, <Ground Spot Classification> buttons respectively.
3. After clicking the buttons, you can set the parameters of the corresponding classification methods, and the set parameters can be saved as templates and managed in <Process> → <Save Classification Parameters> and <Process> → <Manage Classification Templates>, and there is a selection box in front of each button to set whether or not the classification method is used.
4. When the point cloud is categorized：
   - If the point cloud format is hlz3.4, the category will be modified directly on the original file.
   - If the point cloud format is a low version of hlz format, it needs to be automatically converted to hlz 3.4 format for categorization.
   - After the conversion is completed, it will automatically overwrite the original low version of the point cloud data.
5. When using las data classification:

- If there is a hlz3.4 with the same name in the same directory, then directly use this hlz3.4 for classification.
- If there is a hlz3.2 or 3.3 with the same name in the same directory, then the program will automatically convert a hlz3.4 and overwrite the previous hlz to classify the point cloud.
- If there is no hlz with the same name in the same directory, then the program will automatically convert a hlz3.4, and use this converted point cloud to classify the point cloud.

3.5.5 Reset Classify

1. Click menu <Process> → <Reset Classify> on the main interface to bring up the dialog box (Figure 3-128) in which you can set the original category and target category for reset.
2. Click <Save> to save the parameters to <Save Classification Parameters>.

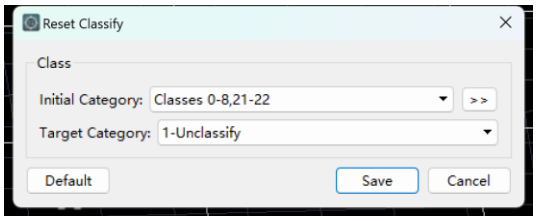

Figure 3-128 Reset classification

3.5.6 Noise Filter

1. Click menu <Process> → <Noise Filter> on the main interface to bring up the dialog box (Figure 3-128), in which you can set the noise detection method, search radius and the minimum number of points within the search radius.
2. Click <Save> to save the parameters to <Save Classification Parameters>.

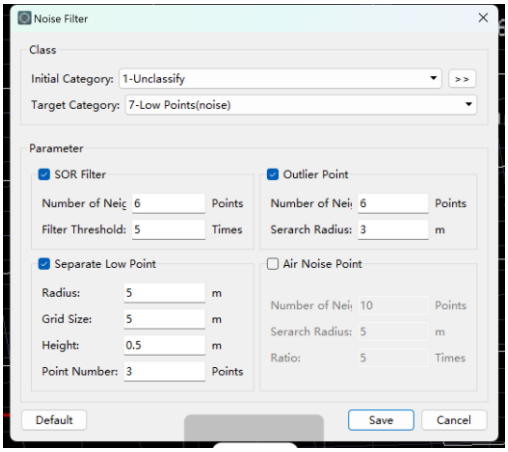

Figure 3-129 Noise Filter

3.5.7 Classify Ground

Click menu <Process> → <Classify Ground>, ground point classification is mainly divided into two ways: fabric algorithm and morphological filtering, and the parameters that need to be set up are shown in Figure 3-130.

When clicking the <Save> button, whichever Classify Ground way the current page is in will be adopted, and the parameters of the way will be saved to <Save Classification Parameters>.

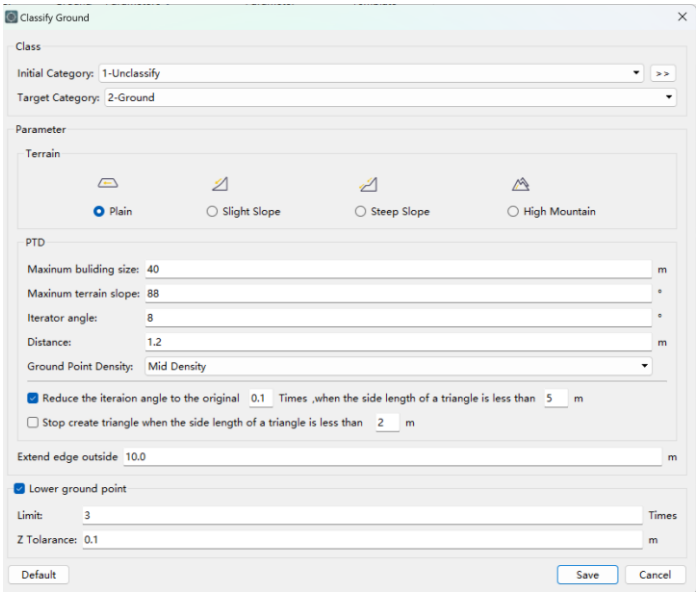

Figure 3-130 Classify Ground

3.5.8 Median Ground Point

Click menu <Process>→< Median Ground Point >, Median Ground Point Classification processes the points in the ground point category in the point cloud, extracts the layer in the z-direction which is in the middle, changes the category information to Median Ground Point, and writes it to the category attribute of the las point cloud.

Median Ground Point can be set up with the grid size and standard deviation multiplier (Figure 3-131).

- The bigger the standard deviation multiplier set up, the more ground median points will be categorized.
- The smaller the grid size is set, the better the ground median point categorization effect is, but the longer the time will be; the ground median point is unchecked by default.

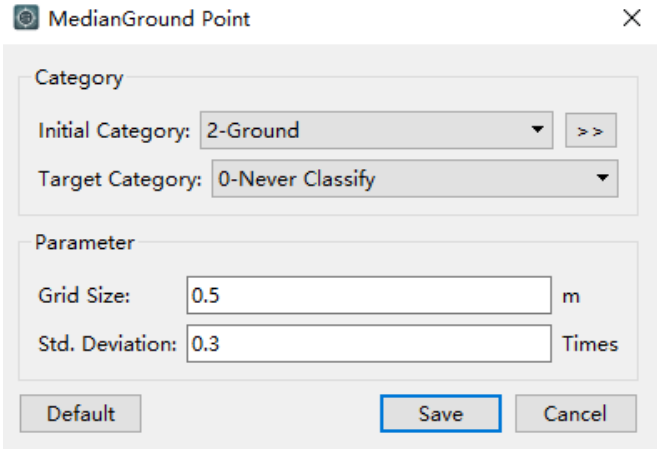

Figure 3-131 Median Ground Point

3.5.9 Reset Classify Parameters

Click the menu <Process>→< Reset Classify Parameters> button (when the saved category parameter templates, the name of the button is the name of the selected template), a drop-down menu will pop up (Figure 3-132), the options in the menu include Reset Category Parameters and all the saved category parameter templates, and the name of the button is the name of the currently selected template.

Click <Reset Classify Parameters> in the menu, you can reset the parameters of the current classification to the default, click a template can be set as selected.

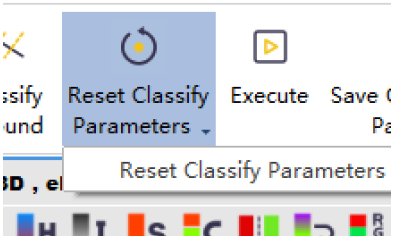

Figure 3-132 Reset Classify Parameters

3.5.10 Partial Classification

Click menu <Process> → <Partial Classification> to bring up the interface of local classification, check the specified range, click the box to select the range, draw the range in the view, fill in the parameters that need to be set up after completion, click Processing, and then classify the ground points in the boxed range again.

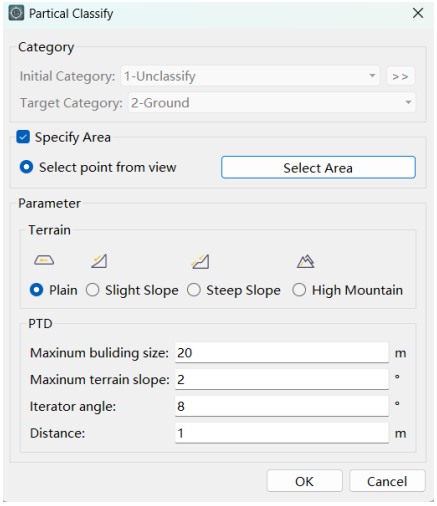

Figure 3-133 Partial Classification

3.5.11 Execute

Click menu <Process> → < Execute > button, the interface shown in Figure 3-134 will pop up, which lists the list of data files loaded in the current software, you can select the files that need to perform the combination of classification.

After the selection is completed, click <OK>, the software will be in accordance with the parameters of the currently selected classification templates for the batch combination of the selected point cloud files for classification Process, and the classification effect will be displayed in the view.

After the Process is completed, the classification effect will be displayed in each view.

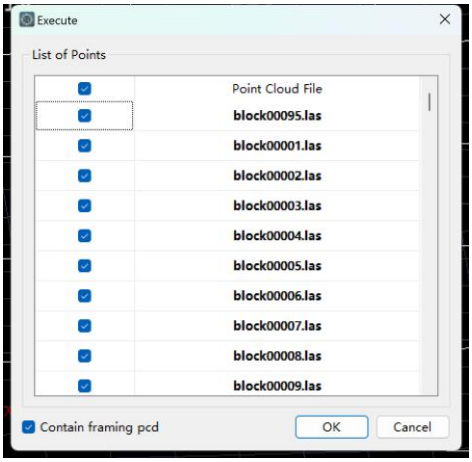

Figure 3-134 Execution windows

3.5.12 Saving Classification Parameter

After setting the parameters of each classification, click menu <Process>→<Save Classification Parameters> on the main interface, the information of each classification parameter is recorded in the interface (Figure 3-135), and when the font of the parameter information is grayed out, it means that the classification method is not selected.

After entering a name in Template Name and clicking the <OK> button, you can save the current setting as a template.

The newly saved template will be automatically added in the drop-down menu of <Reset Classification Parameters> and <Manage Classification Templates>, and at the same time, the <Reset Classification Parameters> button name will be synchronized with the name of the current template.

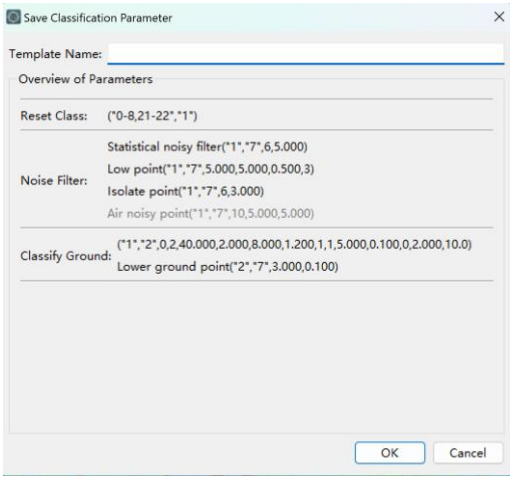

Figure 3-135 Saving Classification Parameter

3.5.13 Manage Classification Template

Click <Process>→< Manage Classification Template > button, the dialog box will be popped up (Figure 3-136), which lists all the category templates saved by the user, and when you click on one of the templates, the detailed parameter information of the template will be displayed underneath.

- Click <Delete> to delete the currently selected template.
- Click <Import> to import the template file from outside.
- Click <Export> to save the currently selected template information as a template file.

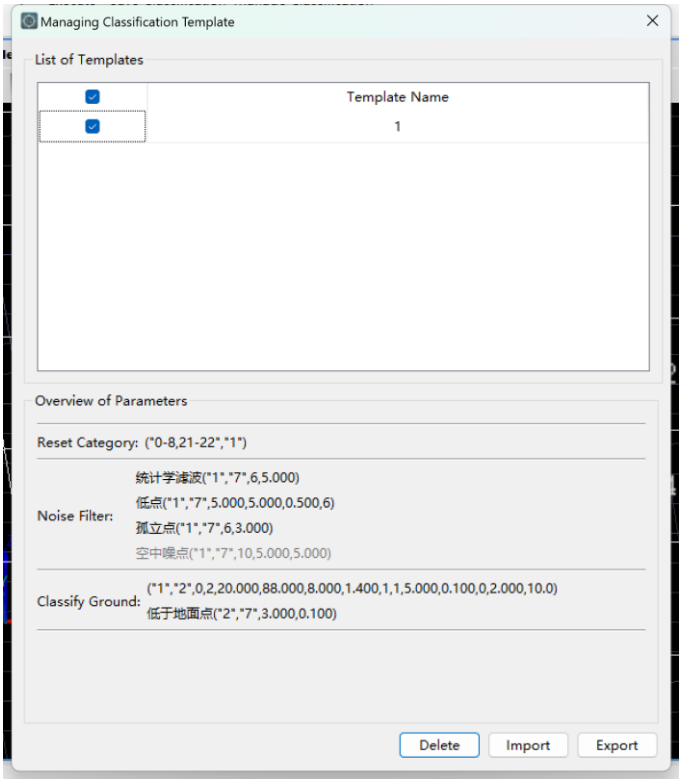

Figure 3-136 Manage Classification Templates

3.5.14 Manual Registration

This function is mainly used for manual registration of SLAM point clouds. Registration is performed based on the same-name point pairs. Load the point cloud file to be registered, click the menu <Process> → <Manual Registration>, enter the registration interface, the interface displays the base view, target view, and result view, select the point cloud file to be paired in the manual registration interface on the right, select the same-name point pairs in the base view and target view, the registration interface will display the stitching error value, and the result view will display the registration effect. If the current registration effect is satisfactory, click the Apply button to close the manual registration interface and return to View 1. View 1 displays the registration effect. Users can click <Airborne> → <Point Cloud Export> to export the registered point cloud file as needed.

3.6 Terrain

3.6.1 Generate DEM

This function is mainly used to generate relevant DEM model files locally. Click menu <Terrain>→ <Generate DEM>, the dialog box will pop up (Figure 3-137).

1. You can select the point cloud data needed to generate the DEM model file, set the ground category and grid size of the generated model, the resolution size is divided into 1:500, 1:1000 and 1:2000, corresponding to different grid sizes.
2. Next, you need to set the format of the exported file, which is mainly tif and img.
3. Since this function can batch generate DEM model files for multiple point clouds, you need to select the folder to save the batch files and click <OK> to start generating.
4. After the generation of files need to be manually loaded in the software (Figure 3-138).

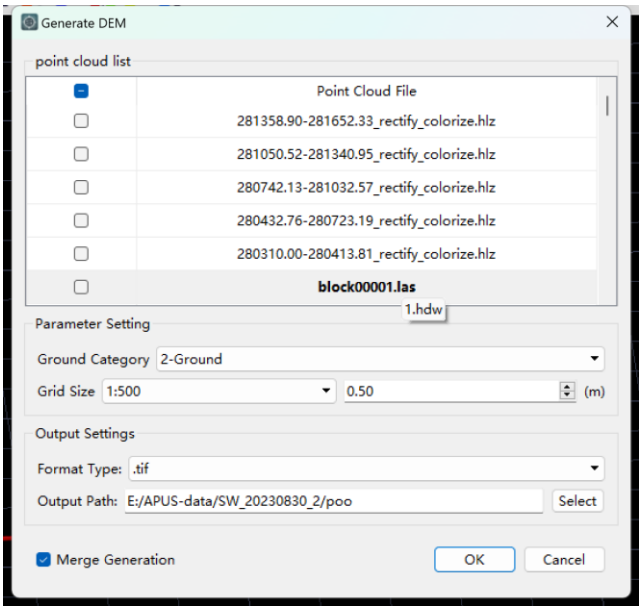

Figure 3-137 Generate DEM

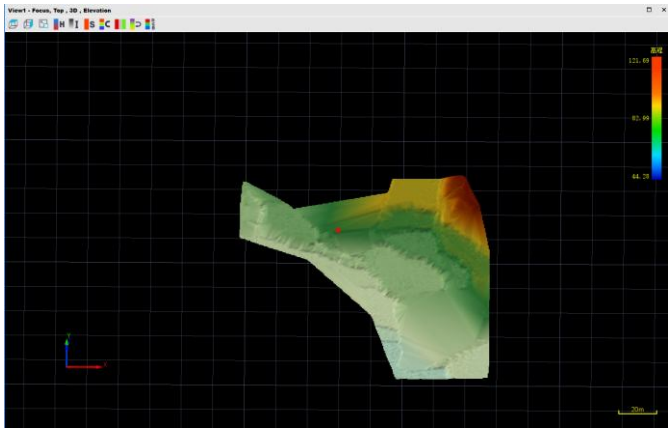

Figure 3-138 Generated DEM model file

3.6.2 Generate DSM

This function is mainly used to generate relevant DSM model files locally.

1. Click the menu <Terrain>→<Generate DSM>, the dialog box will pop up (Figure 3-139).
2. You can select the point cloud data that need to be generated to the DSM model file, and set the ground category and grid size of the generated model.
   - The resolution size is divided into three kinds: 1:500, 1:1000, and 1:2000, which corresponds to different grid sizes, respectively.
3. Next, you need to set the format of the exported file, which is mainly tif and img. As this function can batch generate DSM model files for multiple point clouds, you need to select the folder to save the batch files and click <OK> to start generating. The generated files need to be loaded manually in the software.

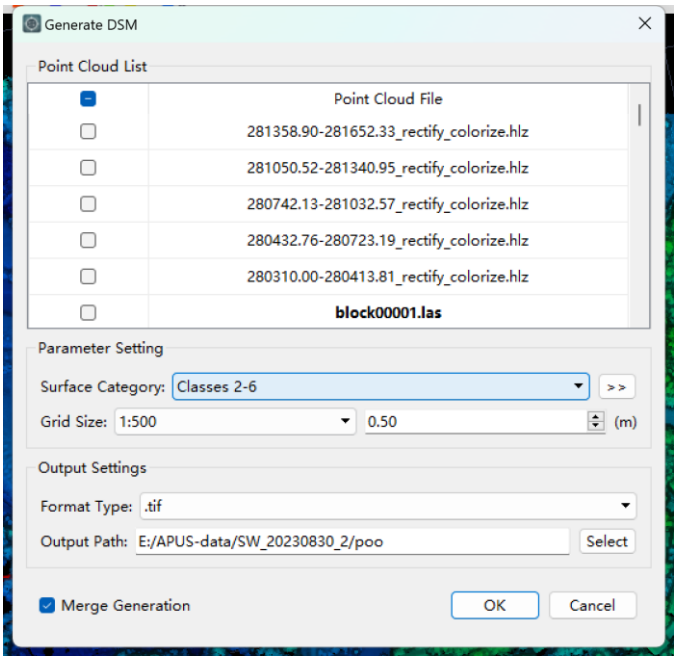

Figure 3-139 Generate DSM

3.6.3 Generate contour lines

This function is mainly used to generate the related contour model file locally.

1. Click <Terrain>→<Generate contour lines>, the dialog box will pop up (Figure 3-140).
2. You can select the point cloud data that you need to generate the contour line model file, select the terrain type to which the point cloud data belongs, and set up the ground type, the starting value of the contour line, and
   - The grid size for the generated model is divided into three kinds of grid resolution sizes: 1:500, 1:1000 and 1:2000, corresponding to different grid sizes respectively.
   - The grid resolution size is divided into three types: 1:500, 1:1000 and 1:2000, which

correspond to different grid sizes.

3. Next, you need to set the format of the export file, which is mainly shp file. Since this function can generate contour model files for multiple point clouds in batch, you need to select the folder to save the batch files and click <OK> to start generating. After the generation of files need to be manually loaded in the software (Figure 3-141).

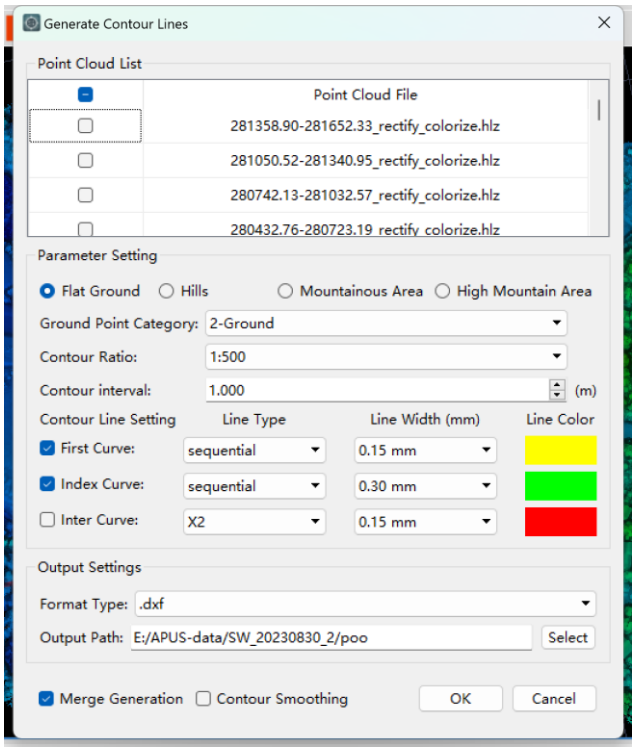

Figure 3-140 Generate contour lines

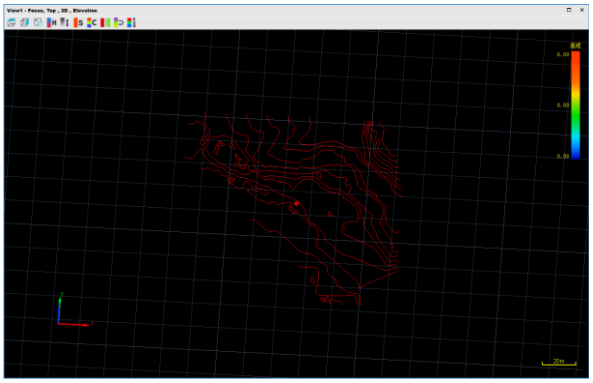

Figure 3-141 Generated contour model file

3.6.4 Generate Elevation Points

This function is mainly used to generate the related elevation point information text file locally, mainly including the number, x-coordinate, y-coordinate and corresponding elevation value.

1. Click menu <Terrain>→<Generate Elevation Points>, the dialog box will pop up (Figure 3-142). you can select the point cloud data that you need to generate the elevation point file,

set the ground category and the grid size for generating the elevation points.

- The grid resolution is divided into 3 kinds: 1:500, 1:1000, and 1:2000, which correspond to different grid sizes respectively.

2. Next, you need to set the format of the export file, which is mainly dat, txt (NXYZ) and xyz (NXYZ) text file format. As this function can generate files for multiple point clouds in batch, you need to select the folder to save the batch files and click <OK> to start generating, the generated elevation point information file is shown in Figure 3-143.

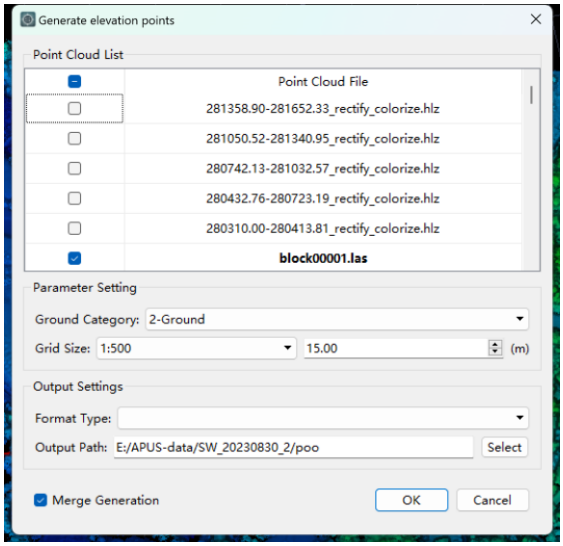

Figure 3-142 Generating Elevation Points

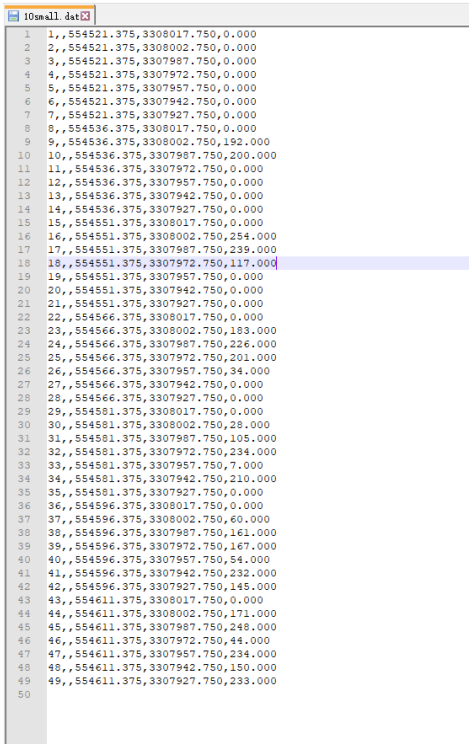

Figure 3-143 Generated elevation point information file

3.6.6 Earthwork Calculations

Under <Terrain> module, <Earthwork Calculation>, you can perform earthwork calculation, which mainly contains single-phase earthwork calculation and multi-phase earthwork comparison. The calculation format supports HLZ, LAS, Tif can choose the base elevation, set the range of calculation.

1. Single-period calculation

Single-period calculation is shown in Figure 3-145.

For single-phase earthwork calculation, you need to select a point cloud file, which supports (HLZ,LAS.TIF) format. Down can be selected view imported files, the right side <...> can be exported from the local computer a point cloud file. The datum elevation can be set by the right side pull box, including minimum, maximum, average, self-selected points, customized.

- The self-selected point can be clicked directly from the view to get a datum elevation point;
- The customized one can be manually filled in an elevation value as the datum elevation.
- The grid size can be set by default, the grid size determines the density of thinning, the larger the value, the thinner the point cloud thinning, the lower the calculation accuracy, the faster the speed, when the calculation of the point cloud data is very large, you can increase the size of the parameter in order to get a faster calculation speed.
- The specified range can be unspecified or specified. The specified range can be read from a file, such as .shp, .dxf, .txt files, or the range can be selected from the view.

Click Calculate and wait for the calculation result to pop up, as shown in Figure 3-146 After the calculation, you can export the results to (pdf, html, csv, xls, txt) format.

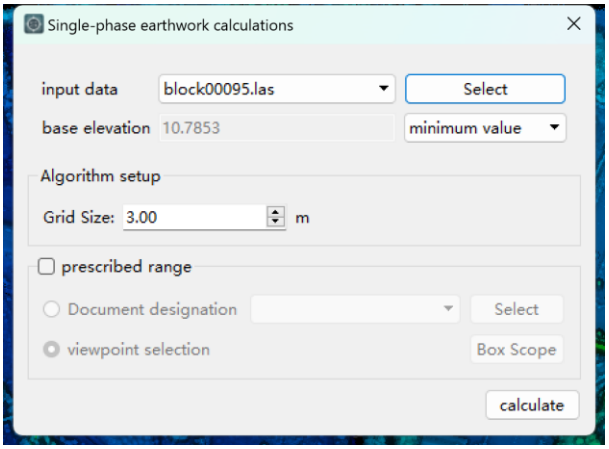

Figure 3-145 Single-period calculation

Single-period calculate result

2023-11-14 18:00:46 周二

Parameter Setting

一 peroid point cloud	block00095.las
二 peroid point cloud	

Grid Size	3.000	m	Base height	10.785	m

Result

一 peroid maxinum height difference	17.915	m	一 peroid maxinum diameter	312.058	m
二 peroid maxinum height difference	0.000	m	二 peroid maxinum diameter	0.000	m
2D perimeter	396.000	m	3D perimeter	433.292	m
2D projected area	9801.000	m^2	3D surface area	12559.792	m^2
Cut area	9801.000	m^2	Fill area	0.000	m^2
Cut surface area	12559.792	m^2	Fill surface area	0.000	m^2
Cut volume	61150.289	m^3	Fill volume	0.000	m^3
一 peroid mininum height	11.474	m	一 peroid maxinum height	29.389	m
二 peroid mininum height	0.000	m	二 peroid maxinum height	0.000	m

Figure 3-146 Earthwork calculation results

2. Multi-phase earthwork calculations

Multi-phase earthwork calculations is shown in Figure 3-147.

Multi-phase earthwork calculations at least join two periods of data for calculation.

- The right side of the list "+" can be added to the point cloud to the list, the default list of data behind the reference to the previous list of data, through the up and down arrows to adjust their reference logic. "-" can remove the selected data.
- Grid size sets the size of the extraction thinning, the same as the single-phase data, the default value is 5. The range designation is also consistent with the single-phase calculation.
- Click Calculate and wait for the calculation to pop up. After the calculation, you can export the results, support (pdf, html, csv, xls, txt) format.

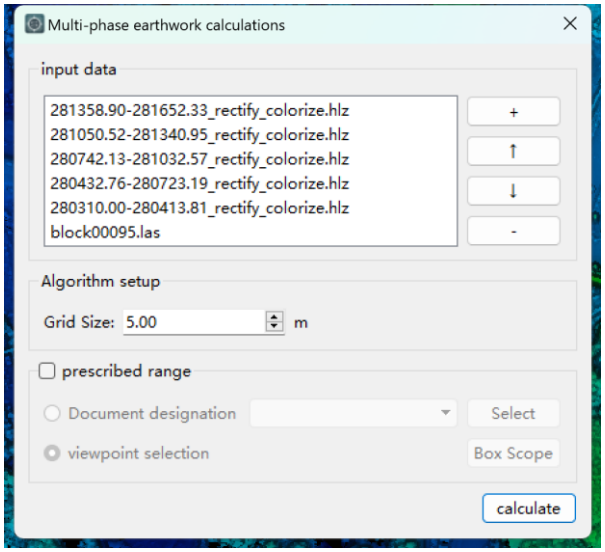

Figure 3-147 Multi-phase earthwork calculations

3.7 Layout

3.7.1 Layer Management

Users can show/hide the layer management panel according to their needs, checking means show and unchecking means hide.

3.7.2 Properties Panel

Users can show/hide the property panel according to their needs, checking means show, unchecking means hide.

3.7.3 Toolbox

Users can show/hide the Toolbox panel according to their needs, checking means show, unchecking means hide.

3.7.4 Information Output

Users can show/hide the message output panel according to their needs, checking the box means show, unchecking it means hide.

3.7.5 Welcome Interface

Users can display the welcome screen according to their needs, clicking the checkbox will bring up the welcome screen.

3.8 Profile Selection

Profile menu is hidden by default, Click the menu <Process>→<Profile Selection> to automatically display the profile editing operation menu (Figure 3-148).

At the same time, the Profile View window and the Profile Buffer Settings window are displayed (Figure 3-149), and the buffer size can be fixed by checking <Buffer> and setting the size in the window. Click the arrow part of the <Profile View> button, and then click <Buffer Setting> in the pop-up drop-down menu to bring up the same window.

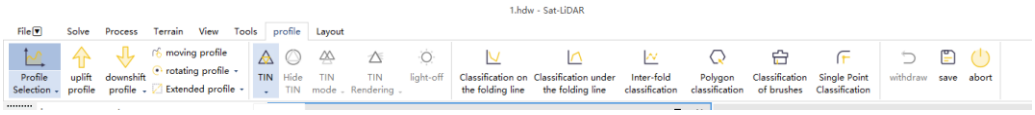

Figure 3-148 Profile Edit Menu

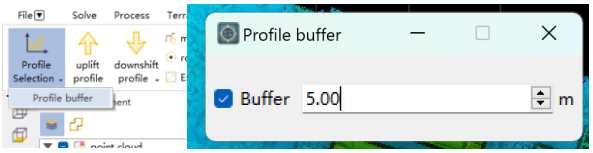

Figure 3-149 Setting the profile buffer

3.8.1 Section Display

Click the menu <Profile> → <Profile View>, enter the profile editing menu in the point cloud view to determine the profile line, and then determine the buffer size of the profile, as shown in Figure 3-150, the profile operation is completed in two steps:

1. Left mouse click on the start point and end point to determine the profile line (profile identification). If a fixed size buffer has been set, then at this point the software will automatically generate a buffer along the profile line, indicated by the yellow dashed box, without the need to carry out step 2;

2. Not set a fixed size buffer, move the mouse perpendicular to the profile line, click the left mouse button to determine the size of the profile buffer.After the drawing is completed, the resulting profile view will be displayed in the Profile View window (Figure 3-151).

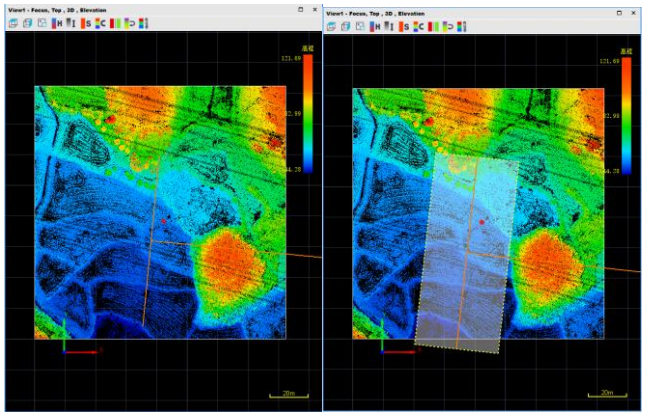

Figure 3-150 View draws profile lines and buffers

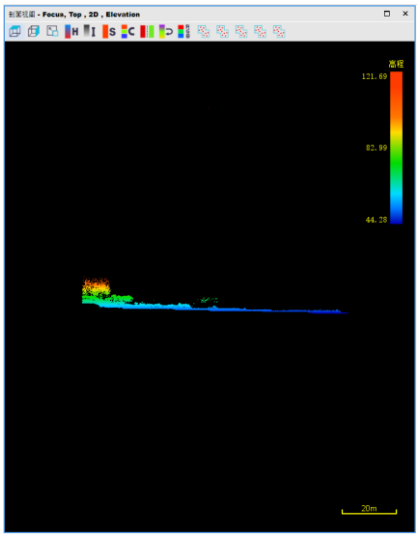

Figure 3-151 Profile view display

3.8.2 Moving the profile down

Click the menu <Profile>→<Downshift Profile> icon part, the profile is downshifted according to the given step size, and the profile view is automatically updated after the downshift operation.

Click menu <Profile>→<Downshift Profile> arrow part, click <Step Setting> in the drop-down menu, and set the size of the fixed step in the pop-up window (Figure 3-152).

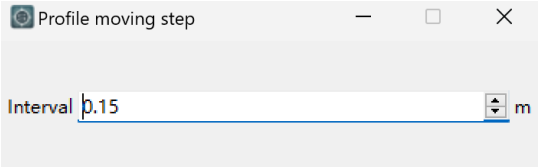

Figure 3-152 Step Size Setting

3.8.3 Move Profile Up

The Move Profile Up operation is the same as the Move Profile Down operation, and the operation structure is the opposite of the Move Down operation.

3.8.4 Move Profile

Click menu <Profile>→<Move Profile> to move the previously drawn profile markers in the main point cloud view with the mouse, and the profile editing view will be updated automatically after the mouse is released.

3.8.5 Rotate Profile

Click menu <Profile>→<Rotate Profile> arrow part, click <Rotate Setting> in the pop-up menu, a window like Figure 3-153 will pop up to set the rotation angle of the current profile.

After closing the window and clicking menu <Profile>→<Rotate Profile> icon part, the profile view

will be rotated in accordance with the set angle, and the profile editing view will be updated to show the new profile view automatically.

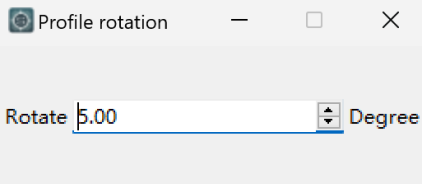

Figure 3-153 Profile rotation settings

3.8.6 Extend profile

Click menu <Profile>→<Extend Profile> arrow part, click <Extend Settings> in the pop-up menu (Figure 3-154), set the extension length of the profile, close the window and click menu <Profile>→ <Extend Profile> icon part, the profile view will be extended in accordance with the set length, and the profile editing view will be updated to the new profile view automatically.

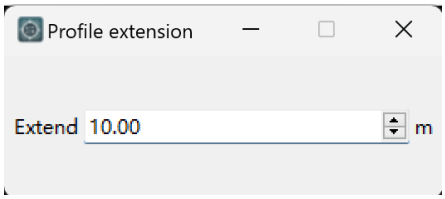

Figure 3-154 Profile Extension Settings

3.8.7 TIN

1. TIN Construction

TIN Construction function targets the ground points in the currently displayed point cloud, and it can realize the browsing of TIN and the synchronous editing of point cloud and TIN.

Load the chunked point cloud, click <TIN Construction>, check the point cloud to build TIN, click OK, wait for some time, the TIN will be built, and then you can browse the TIN in the same way as the point cloud.

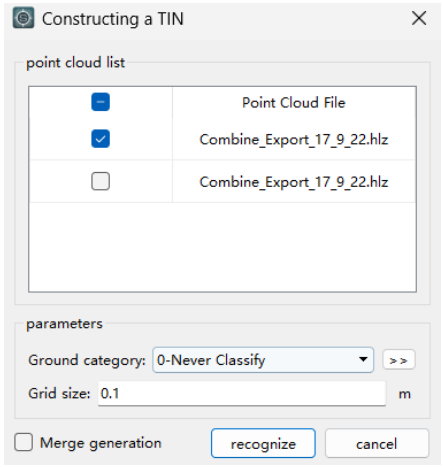

Figure 3-155 Constructing a TIN

2. Clear TIN

You can click <Clear TIN> to clear the constructed TIN.

3.8.8 Hide TIN

You can hide TINs that are currently constructed and displayed.

3.8.9 TIN Modes

Three modes of rendering are supported: triangle mode, point mode, and solid mode.

3.8.10 TIN Rendering

Supports two rendering modes: monochrome rendering, elevation rendering

3.8.11 Light-off

Manually set the lighting on/off effect.

3.8.12 Manual Classification

Manual classification is mainly used to edit and modify the category of the point cloud by manual selection, which mainly includes: on-folding line classification, under-folding line classification, between-folding line classification, polygon classification, brush classification, and single-point classification.

1. Classification on the folding lines

(1) Click menu <Profile>→<Classification on the folding lines> to bring up the Category Setting dialog box (Figure 3-156).

(2) You need to set the initial category and target category in the dialog box, click the <>>> button, and you can make multiple selections for the initial category in the Select Category dialog box (Figure 3-157).

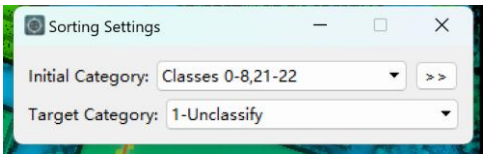

Figure 3-156 Sorting Settings

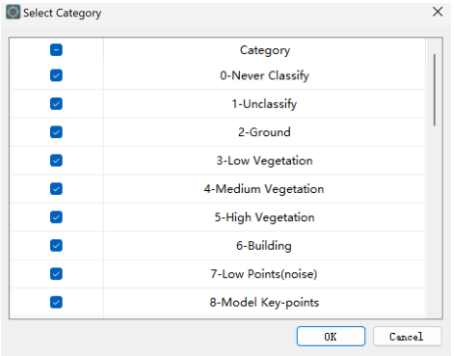

Figure 3-157 Select Category

(3) When the selection is complete, close the Classification Settings dialog box. Click the left mouse button anywhere on the profile view and drag the mouse to draw a folded box (Figure 3-158).

(4) During the drawing process, click the right mouse button to redraw, double-click the left mouse button to end the folding line drawing, located in the folding box inside the point cloud, all belong to the initial category of the point cloud's category will be set to the target category.

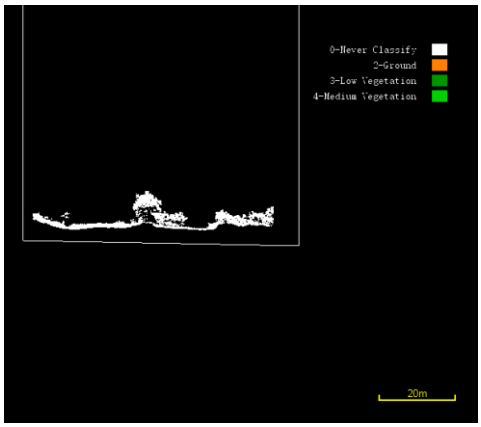

Figure 3-158 Classification on Folding Lines

2. Classification under folding Line

The procedure is the same as < Classification on the folding lines >, and the direction of the drawn folding line box is downward (Figure 3-159).

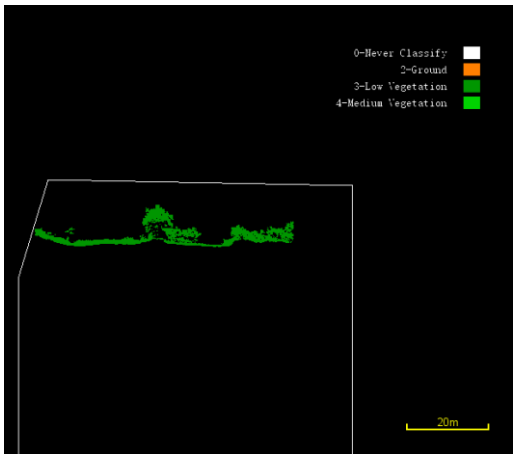

Figure 3-159 Classification under folding Line

3. Inter-fold classification

The operation procedure is the same as that of <Categorize on Folding Line>, and the upper and lower tolerance of the folding box should be set in the category setting window, and the wider the tolerance is, the wider the folding box will be drawn.

In the point cloud located between two fold lines, all the point clouds belonging to the initial category will be set as the target category (Figure 3-161).

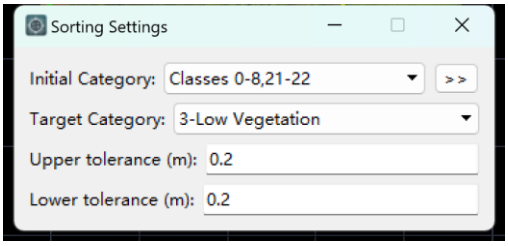

Figure 3-160 Inter-fold classification settings

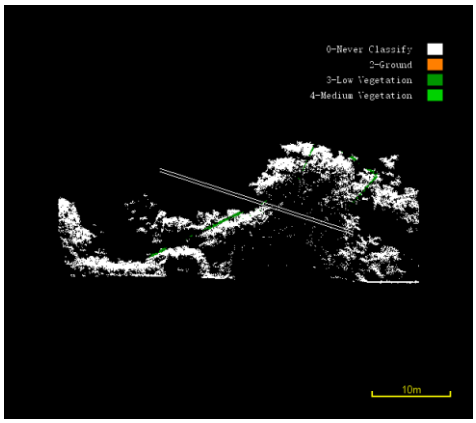

Figure 3-161 Inter-fold classification

4.Polygon classification

The operation steps are the same as <Categorize on Folding Lines >. The category of all the point clouds located inside the fold box that belong to the initial category will be set to the target category (Figure 3-162).

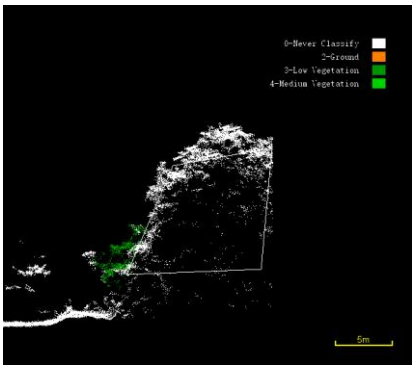

Figure 3-162 Polygon classification

5.Classification of brushes

(1) Click menu <Profile> → < Classification of Brush >, in the pop-up dialog box, in addition to setting the initial category and target category, you also need to set the brush radius (Figure 3-163).

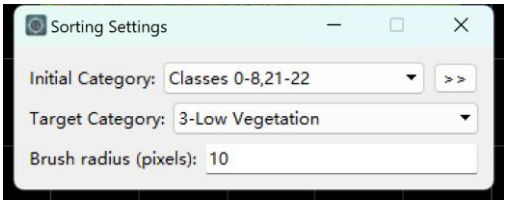

Figure 3-163 Brush Classification Settings

(2) Close the Brush Classification dialog box, press the left mouse button anywhere in the profile view, then the mouse cursor will be changed to a circle consistent with the brush radius (Figure 3-164), press and hold the left mouse button, move the mouse and then lift the left mouse button.

(3) At this time, the cursor circle swept through the point cloud data, all the initial category of point clouds belonging to the category of the target category will be set.

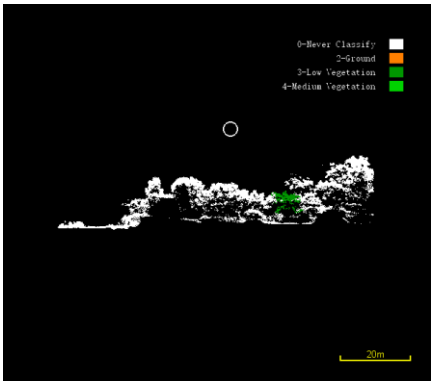

Figure 3-164 Classification of Brush

6. Single Point Classification

(1) Click the menu <Profile>→<Single Point Classification>, after selecting any point in the profile view, the selected point will be marked by the red crosshair cursor, and the Single Point Classification dialog box will be popped up (Figure 3-165).

(2) In the dialog box, you can modify the attributes of the point, including coordinates, color, intensity value, category and GPS time. After clicking the <Modify> button, the modified effect will be updated in the view immediately.

Figure 3-165 Single Point Classification

7. Undo Editing

Categories that have been manually categorized and not saved can be undone by clicking the undo button.

8. Save

Click the menu <Profile>→<Save >, and click <Yes> in the pop-up dialog box to save the point cloud data after processing the functions in the Profile menu.

9. Abort

Click menu <Profile>→< Abort > to exit profile editing and hide the profile editing menu.

3.9 Toolbox

3.9.1 Data Processing

1. Extract

The Extract tool allows you to filter the data and generate a new data file based on a number of conditions.

(1) Extract Category

The extract category tool can extract all the point clouds of the category according to the category selected for extraction by the user, and check the merge export to save the data in one file. Multiple point cloud files can be selected for operation.

- Click <Toolbox>→<Data Process>→<Extract>→< Class Export >, and a dialog box will pop up (Figure 3-166).
- The point cloud files are listed in the list of point cloud files loaded in the current software, check the point cloud files that need to be extracted by category, support to add point cloud files, remove point cloud files; check Merge Export to merge the extracted point clouds into a single file for exporting; select the point cloud files to export, support to select multiple categories; select multiple categories.
- Check Merge Export to save the data in one file. Select the category to be exported, support selecting multiple categories; select the required output format (hlz, las, xyz).
- The output path is stored in the POO path by default, support modifying it to a customized path, click <OK> to start the category extraction, and there will be a reminder message in the information output after the extraction is completed.

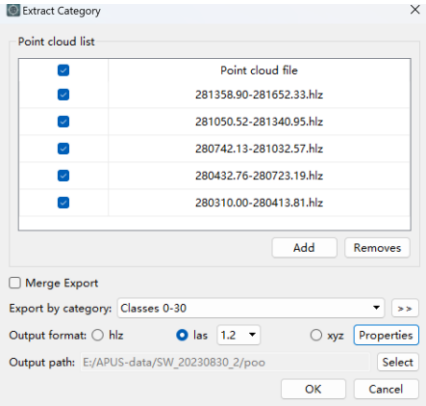

Figure 3-166 Category Extraction

(2) Extract Echo

The echo extraction tool can extract all the point cloud data of the echo to be saved in a file according to the type of echo selected by the user.

- Click <Toolbox>→<Data Process>→<Extract>→< Echo Export>, a dialog box will pop up (Figure 3-167).
- The point cloud files are listed in the list of point cloud files loaded in the current software, check the point cloud files that need to be extracted for echo extraction, and it supports to add and remove point cloud files.
- Check Merge and Export to merge the extracted point clouds into a single file for exporting.
- Select the point cloud types to be exported, and it supports to select multiple categories.
- Select the desired output format (hlz, hlz, hlz); and it supports to select the desired output format (hlz, hlz).
- Select the type of echo to be exported, support to select multiple categories.
- Select the required output format (hlz, las, xyz); the output path is stored in the POO path by default, support to modify the output path to a customized path.
- Click <OK> to start the echo extraction, there will be a reminder message in the information output after the extraction is completed.

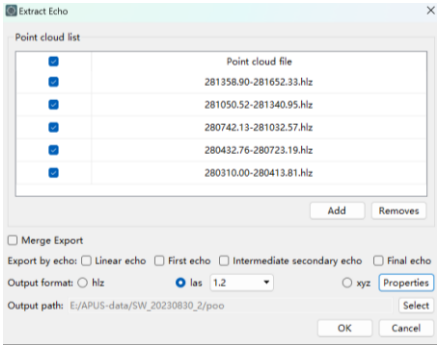

Figure 2-167 Echo Extraction

(3) Elevation Export

The Elevation Export tool can extract all the point cloud data within the range according to the elevation range input by the user and save them to a file.

- Click <Toolbox>→<Data Process>→<Extract>→< Elevation Export>, a dialog box will pop up (Figure 3-168).
- List the point cloud files in the list of point cloud files loaded by the software, check the point cloud files that need to extract the elevation, support adding and deleting point cloud files;
- Check Merge and Export to merge the extracted point clouds into a single file for export;
- Input the range of elevation values to be exported;
- Select the desired output format (hlz, las, las);
- select the output format (hlz, las, las, las) to save the Save the data as a file.
- Input the range of elevation values to be exported;
- select the desired output format (hlz, las, xyz);
- The output path is stored in the POO path by default, and it supports to modify it to a customized path.
- Click <OK> to start the elevation extraction, and there will be a reminder message in the output of the information after the extraction is completed.

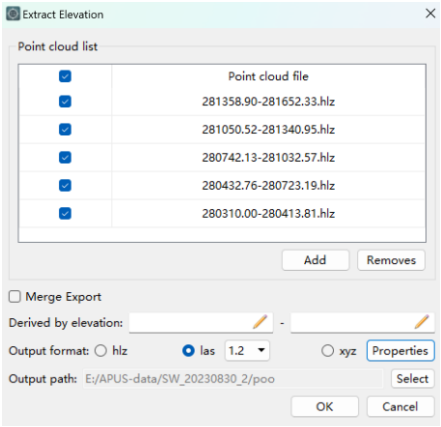

Figure 3-168 Elevation Extraction

2. Process

(1) Remove the edge points of the air strip

- Click <Toolbox> → <Data Process> → <Process> → < Remove Strip Edge Points > on the main interface, a dialog box will pop up (Figure 3-169).
- The point cloud files loaded in the software are listed in the Point Cloud File List, check the files that need to remove the tape edge points, and then click <Start Execution>.
- After the culling is finished, a popup will appear, "Do you want to load the culled point cloud", click "Yes", and the culled point cloud will be loaded into the view.

If hlz data is processed, the point cloud generated after processing is hlz version 3.4.

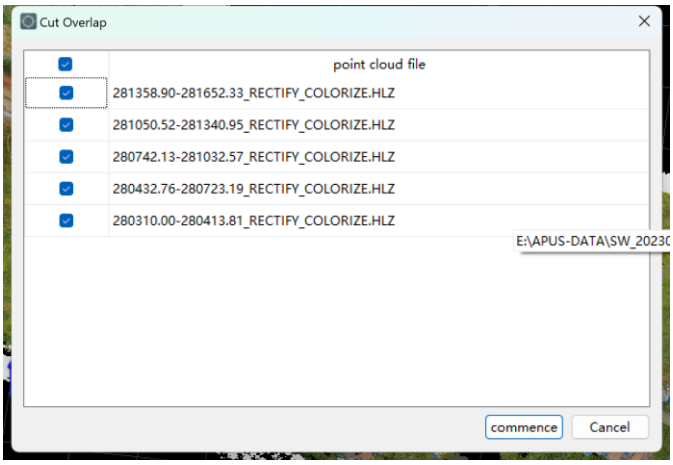

Figure 3-169 Remove the edge points of the air strip

(2) Remove redundant data

- Click <Toolbox> → <Data Process> → <Process> → < Remova redundancy data> on the main interface, and a dialog box will pop up (Figure 3-170).
- Select the original photo POS file, store the original photo POS directory;
- Set the point threshold and angle threshold; click <Crop Preview> to check the effect of the POS crop, and then click <Data Removal> to remove invalid data in the POS of the photo.
- If the cropping is wrong, click the <Data Reject> button to reject the invalid data in the POS, and at the same time, save the rejected invalid photos to a new folder under the Image folder.
- For the "Remove Invalid Photos and POS" tool, if the cropping is not as expected, you can adjust the Point Threshold and Angle Threshold parameters and preview again until it meets the expectation.

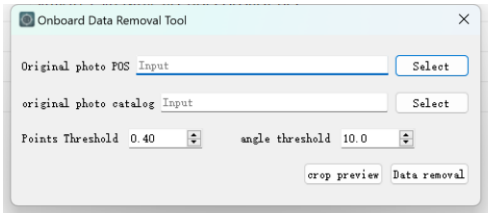

Figure 3-170 Remove redundancy data

Figure 3-171 Crop Preview

(3) Point Cloud Optimize

- Click <Toolbox> → <Data Process> → <Process> → <Point Cloud Optimize> on the main interface, and a dialog box will pop up (Figure 3-172).
- The point cloud file list lists the point cloud files loaded in the current software, check the files that need to be optimized for the point cloud, support to add point cloud files, remove point cloud files;
- Set the number of nearest neighbors, the default is 20, the default path is stored in the poo path, support to modify to a custom path, after the optimization is completed, click "Yes", the optimized point cloud will be loaded into the view.
- Click <Start Execution>, after the optimization is finished, a popup will appear, "Whether to load the optimized point cloud into the view", click "Yes" to load the optimized point cloud into the view.

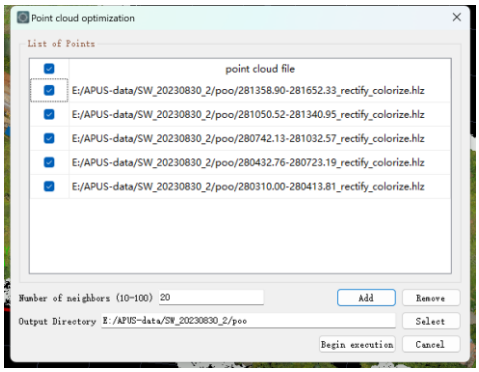

Figure 3-172 Point Cloud Optimize

3. Tiling

(1) Point Cloud Framing

The main purpose of the point cloud binning is to bin the point cloud according to certain criteria, generate the binned vector information, and then extract each small piece of point cloud for individual operation by <Process> → <Select Tiling Blocks>.

- Click <Toolbox> → <Data Processing> → <Tiling> → <Point Cloud Framing>, the pop-up window will list the point cloud files loaded by the software (Figure 3-173).
- Check the files that need to be split. When more than one point cloud file is selected, the software will first merge multiple point clouds and then perform the splitting operation.
- After selecting the file, you need to choose the splitting method and set the splitting parameters.
- When setting the customized block size and the number of blocks, users can refer to the information in the window prompted by the size of the enclosing box to make reasonable settings.
- Click <OK> when the settings are completed, the software will generate the vector file after

the splitting and load it to the vector layer in the workspace and the view selected by the user in the splitting parameters.

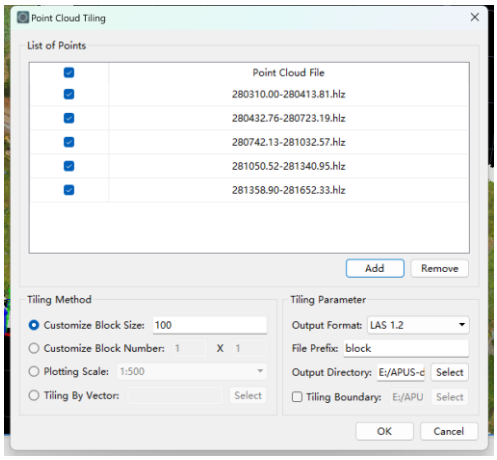

Figure 3-173 Point Cloud Framing

3.9.2 Format Transform

1. Point cloud format transform

Point cloud format conversion tool provides conversion between point cloud and other file formats.

- Click <Toolbox> → <Format Transform> → <Point Cloud Format Transform> on the main interface, and a dialog box will pop up (Figure 3-174).
- The point cloud list lists the point cloud files loaded in the current software, check the files that need to be converted to other file formats.
- The <Merge Export> checkbox can be used to set whether or not to merge multiple input files for export, and the following can be used to select the format of the exported file, the format version, and the path of the exported file.
- Click the <Properties> button, a dialog box will pop up (Figure 3-175), you can select the properties of the exported file, and click <OK> to complete the export.

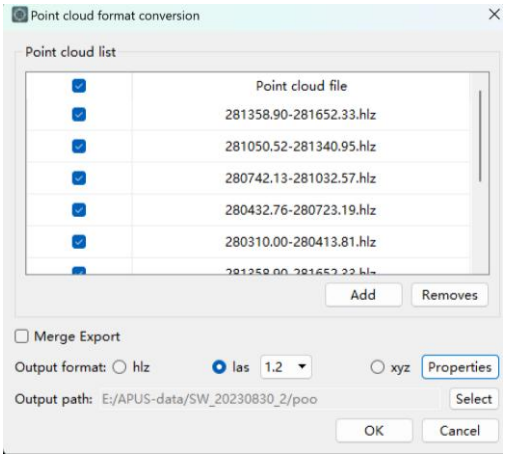

Figure 3-174 Point cloud format transform

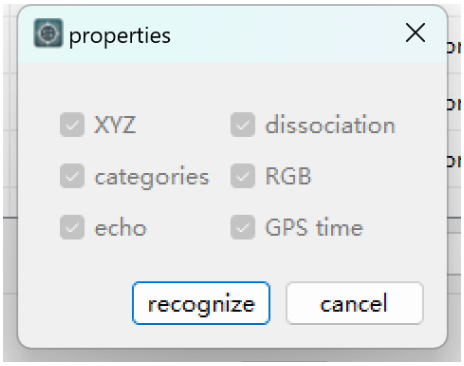

Figure 3-175 Point cloud format transform attributes

2. Point Cloud to Image

The Point Cloud to Image tool saves the point cloud in the view as a .png image.

- Click <Toolbox> → < Format Transform > → <Point Cloud to Image> on the main interface (Figure 3-176), the dialog box of Select Save As Path, select the folder to save the image.
- Click <Select Folder>, and the message output prompts: "Point Cloud to Image Conversion Succeeded".

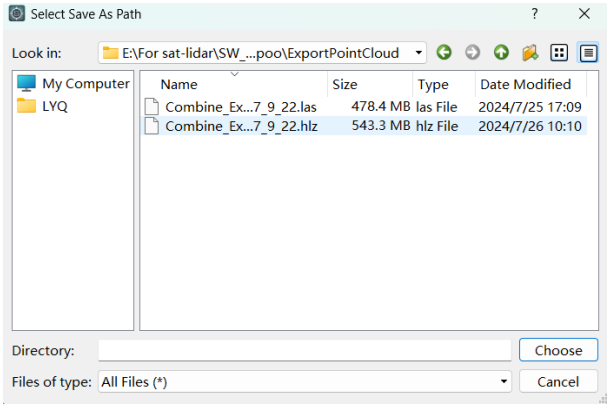

Figure 3-176 Point Cloud to Image

3. Transform Coordinate

(1) Single point transform

Support the conversion between the geodetic, local and spatial coordinates of the source ellipsoid and the geodetic, local and spatial coordinates of the target ellipsoid.

- Click <Toolbox>→<Format Transform >→<Transform Coordinate> on the main interface, a dialog box will pop up (Figure 3-177).
- Click [Single Point] to enter the single-point coordinate conversion interface;
- Click the [Coordinate System] button to enter the coordinate system setup interface and set the parameters of the coordinate system;
- Click [Confirmation] to return to the single-point coordinate conversion interface; support to set the source ellipsoid or target ellipsoid geodetic coordinate, local coordinate, spatial

coordinate and the target ellipsoid geodetic coordinate.

- After setting the coordinate system parameters, click [Cancel] to exit the coordinate system setting interface, the set coordinate system parameters will not take effect;
- Support manually inputting the coordinates of the target ellipsoid;
- Click [Reverse Calculation] to calculate the coordinates under the target ellipsoid;
- Click [Reverse Calculation] to calculate the coordinates under the source ellipsoid.
- The software can calculate the coordinates under the source ellipsoid.
- In the coordinate conversion, there will be some precision loss, and the value of coordinates will be changed when we keep on doing the forward and backward calculation.

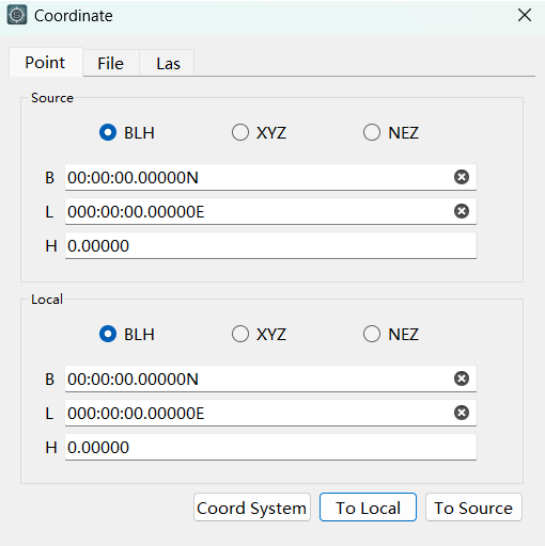

Figure 3-177 Transform Coordinate

(2) Coordinate point file transformation

Support the conversion between the geodetic, local and spatial coordinates of the source ellipsoid and the geodetic, local and spatial coordinates of the target ellipsoid with the imported coordinate point files and coordinate system parameters.

- Click <Toolbox>→<Format Transform>→<Transform Coordinate> on the main interface, and then click [Coordinate Point File] to enter the interface of coordinate point file conversion (Figure 3-178).
- Click [Coordinate System] button to enter the interface of coordinate system setting, and set the parameters of coordinate system.
- Click [Confirmation], the interface of coordinate system setting will be closed, and then return to the interface of [Conversion of Coordinate File]; it supports the selection of BLH XYZ or NEZ coordinate type.

- Click [...] to enter the [Custom Format Settings] interface; support the selection of options to be imported in the optional field column, the options are determined by the coordinate type selected in the [Coordinate Point File Conversion] interface, and the options selected for import will be displayed in the order of the content column.
- Click [OK] to enter the [File Directory] interface, select the files to be imported, and then click [OK] to return to the [Coordinate Point File Conversion] interface.
- Click [OK] button to enter the [File Directory] interface, select the file to be imported, and then click [OK] button, the software reads the file content and returns to the [Coordinate Point File Conversion] interface and displays it in the file column.
- After selecting the BLH, XYZ or NEZ option in the conversion result column, click the [Calculate] button or the [Reverse Calculation] button, the software converts the list of results to display the results of the conversion, and the [Export] button displays the results of the conversion as available.
- Click the [Export] button to enter the [Customize Format] interface, the user in the optional fields in order to select the options need to export, click [OK] button, the software returns to the [coordinate point file conversion] interface and prompts the export success.
- The software returns to the [Coordinate Point File Conversion] interface and prompts for successful export.

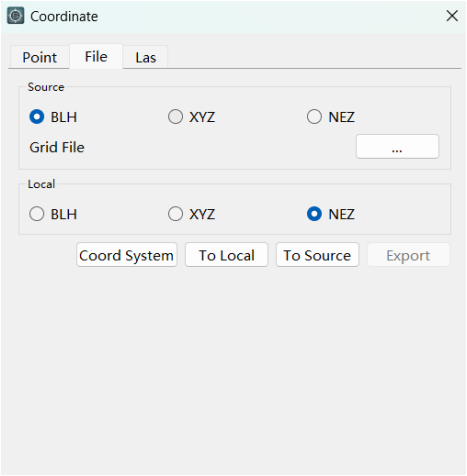

Figure 3-178 Coordinate point file conversion

(3) Las point cloud file conversion

Support to read the point data from the imported las file, convert the coordinates from source ellipsoid to target ellipsoid and then save the converted las point cloud data.

- Click <Toolbox>→<Format Transform>→<Transform Coordinate> on the main interface, and then click [Las Point Cloud File] → [Coordinate System] button to enter the coordinate system setting interface, set the parameters of the coordinate system.
- After setting the parameters of the coordinate system, click [Confirmation], and the parameters of the coordinate system will be used for returning to the las point cloud file conversion interface;
- Click [Add] to bring up the file selection pop-up; and then save the converted las point cloud data.
- Click [Add] to bring up the file selector, select the corresponding las point cloud file, click [OK] to load the point cloud file to the point cloud list;
- Manually input or click [Select] to select the storage path of the converted las point cloud file in the file directory selector; click [OK], the software will save the converted las point cloud file under the output path;
- Click [Cancel], the selected point cloud file will not be loaded to the point cloud list;
- Click [Remove], the added point cloud file will be removed;

The coordinates in the converted las point cloud files are the source ellipsoid coordinates by default, and the coordinates in the converted las files are the target ellipsoid coordinates.

The coordinate system corresponding to the coordinates in the added point cloud file must be consistent with the set coordinate system parameters, otherwise the las file saved after conversion will be converted abnormally, which will lead to abnormal file contents and cannot be opened successfully.

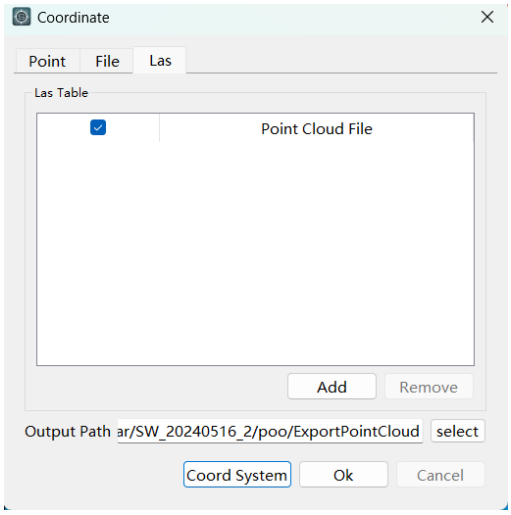

Figure 3-179 Transform Coordinate

3.10 Engineering Application

<Engineering Application> menu for section generation, by import the reference center line or view drawing reference center line, generate reference cross section, measured cross section, measured longitudinal section, and support, the measured longitudinal, transverse section editing, export, generate section, support section method for earthwork calculation, support multiple section contrast.

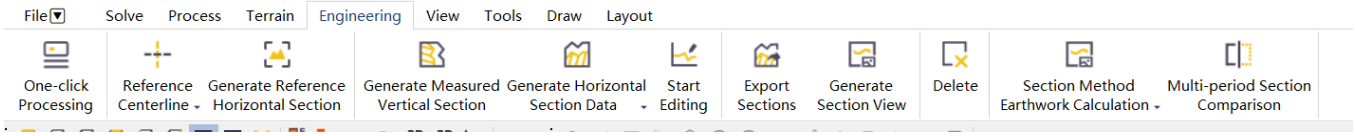

Figure 3-180 Engineering Application

3.10.1 One-Click Processing

One-Click Processing is used for the export of terrain results, according to the imported point cloud, you can export the terrain results with one click, and support Ground Point classification, Contour Lines generation, DEM generation, Elevation Points generation, and support according to Terrain and Actual Result Requirements to perform advanced settings for the generated Terrain Results.

3.10.2 Reference Centerline

Reference center line provides two ways to generate: support view drawing reference center line and support importing reference center line

3.10.2.1. View drawing

After clicking [View Drawing], the mouse switches to the drawing vector state, selects the drawn position through the left click, confirms the drawn vector through the right click, and the view displays the vector file, and displays the drawn reference line in the lower left tree node.

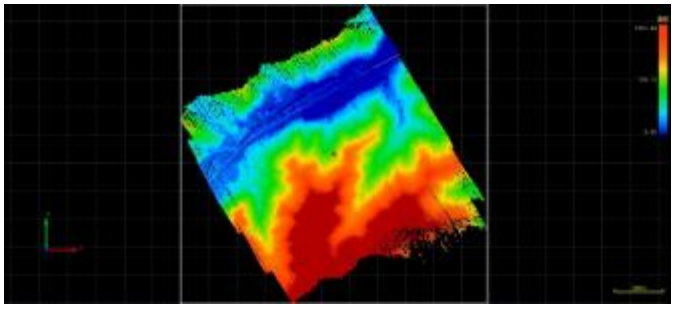

Figure 3-181 views are drawn

3.10.2.2. Import

Click on Reference Centerline ->Import to select Import Centerline, Cross Section Line, or Auto (Vertical, Cross Section). The selected vector file will be imported into the view and displayed in the lower left tree node.

3.10.3 Generate Reference Horizontal Section

After drawing or importing the reference center line, the reference section can be generated through the center line, providing the setting parameters to automatically generate the reference section line, and generate the reference section line manually.

(1) Automatically generate reference cross-section lines

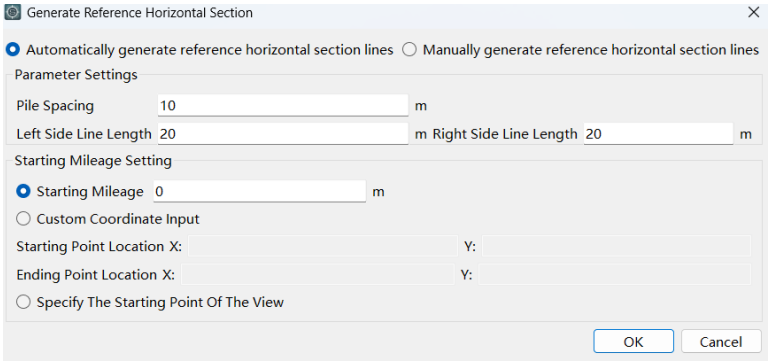

Figure 3-182 automatically generate reference cross sections

Parameter setting: support setting the pile number spacing, left and right line length. Starting mileage setting: support the starting mileage to input the starting mileage value, support the custom coordinate input start and end XY value setting to generate the range of reference section, support the starting point of three ways to set the starting mileage. If you select a view, then click OK to select the point on the reference centerline with the left mouse button to determine the starting point for generating the reference section.

(2) Manually generate reference cross-section lines

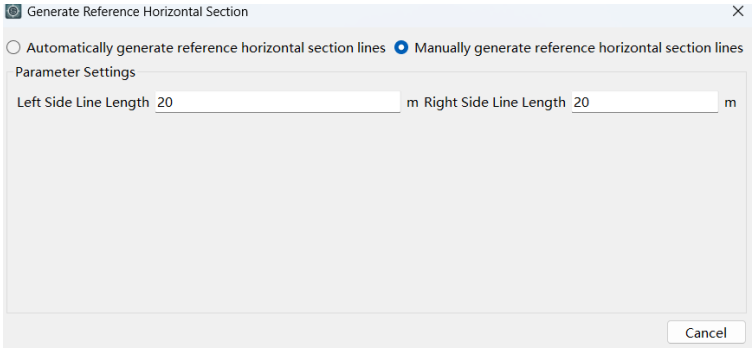

Figure 3-183 manually generated reference cross-sections

It is supported to set the left and right lengths. After selecting the reference section line manually, it is necessary to select the points on the reference centerline through the left mouse button to determine the location to generate the reference section.

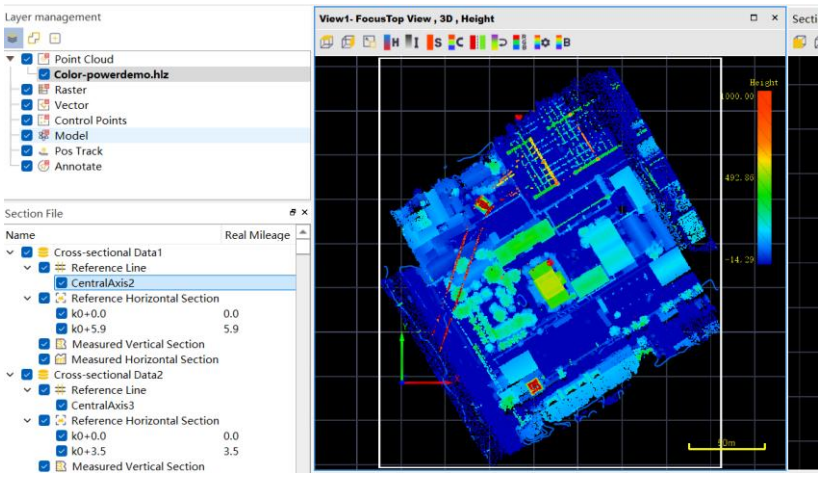

Figure 3-184 automatically generates reference cross-section effects

3.10.4 Generate Measured Vertical Section

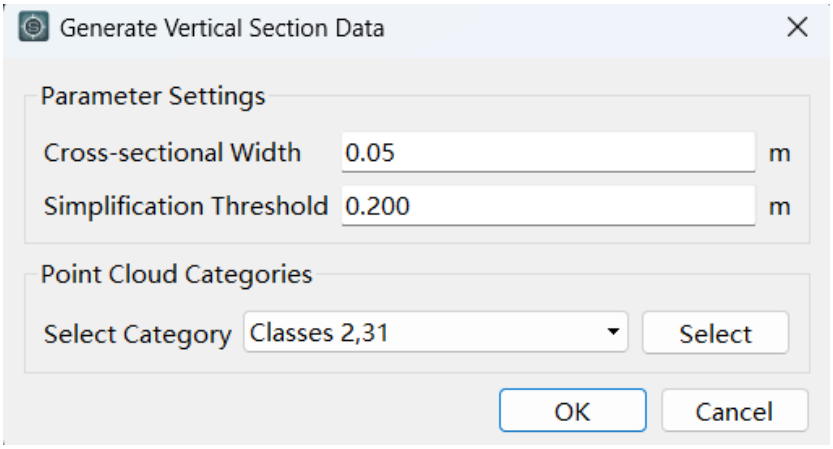

Figure 3-185 Generate Measured Vertical Section interface

Support to set section width, simplify threshold, select point cloud category (default is ground point + ground site) to generate the measured profile.

(1) Simplified threshold: mainly function to ignore the profile points with the height difference of adjacent points less than this parameter.

(2) Section width: generate the width value of the section.

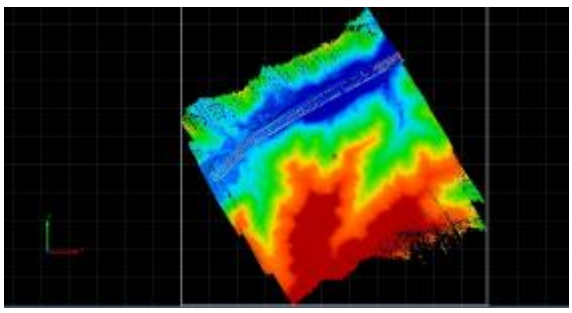

Figure 3-186 Generate Measured Vertical Section effect

3.10.5 Generate Measured Horizontal Section

1 Horizontal Section

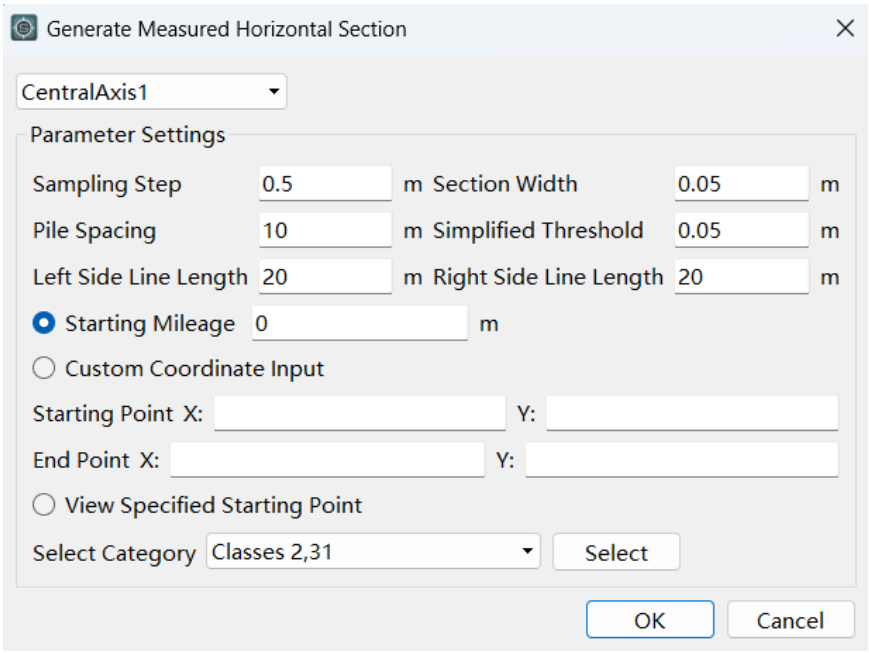

Figure 3-187 Generate Measured Horizontal Section interface

The measured cross section is generated in many ways:

(1) By Reference Centerline-> Generate Measured Horizontal Section: can set the sampling step, section width, simplified threshold, pile spacing, left and right line length, provide starting range setting: support starting mileage input starting mileage value, support custom coordinates input start and end XY value set to generate reference section, support view specify the starting point of three ways to set the starting range

(2) By Reference Centerline -> Generate Reference Horizontal Section -> Generate Measured Horizontal Section: sampling step length, section width and simplified threshold can be set. Other settings inherit the settings that generate the reference section.

(3) By Reference Centerline -> Generate Measured Vertical Section -> Generate Measured Horizontal Section.

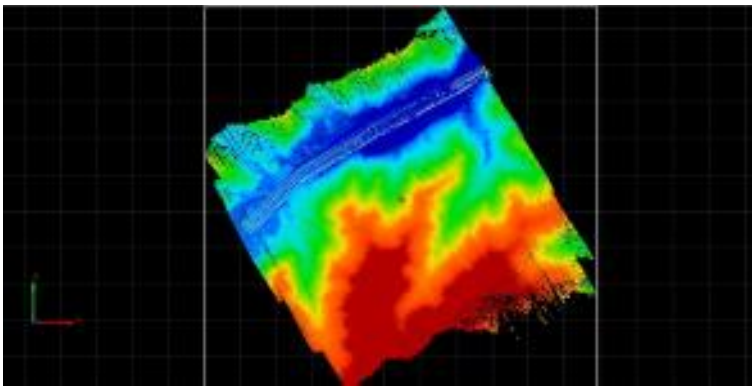

Figure 3-188 Generate Measured Horizontal Section effect

2. Tunnel Horizontal Section

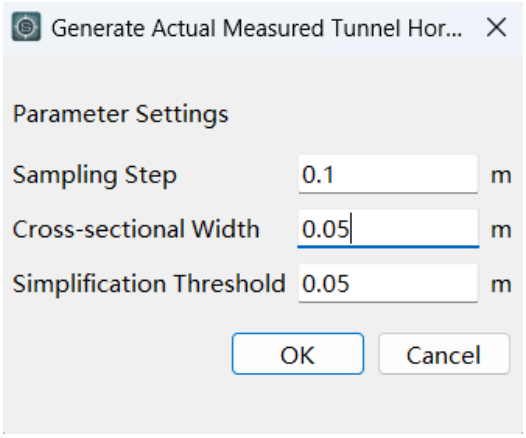

Figure 3-189 Generate Actual Measured Tunnel Horizontal Section interface

By setting the sampling step size, the section width and the simplified threshold, actual measured tunnel horizontal section can be generated and loaded into the view at the same time.

3.10.6 Start Editing

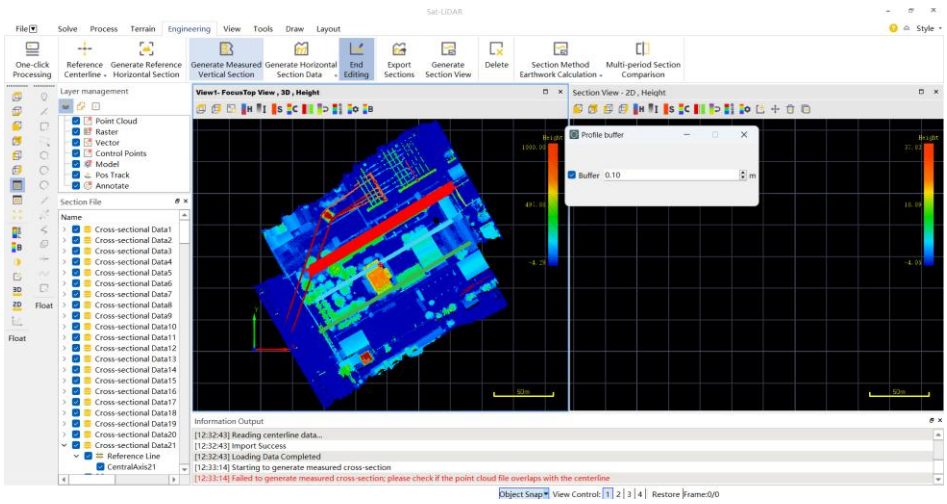

Figure 3-190 Start Editing interface

Click the Start Editing button, and you can click the lower left tree node or the vector line to be edited on the view to edit the currently generated measured longitudinal and cross sections. The software view 1 highlights the node of the current longitudinal section line, automatically opens the section view, and displays the currently selected longitudinal section line and longitudinal section point cloud in the section view. Click the End Editing button to end the longitudinal section editing function and close the section view.

The section view provides four editing functions: add node, delete node, move node, and delete nodes in batches.

3.10.7 Export Sections

Export the measured longitudinal and cross sections generated by the current data.

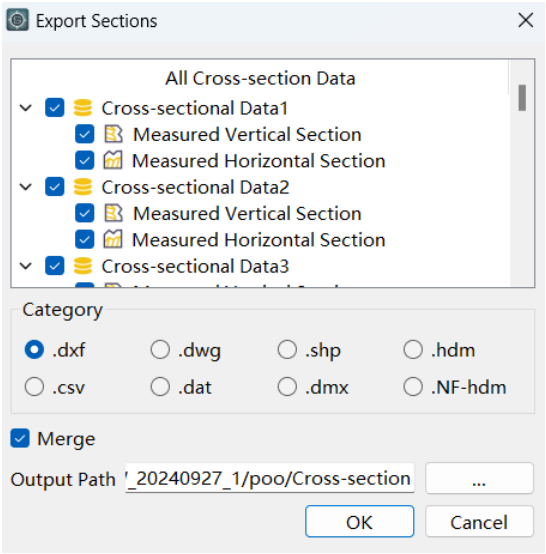

Figure 3-191 Export section interface

When the user selects the mileage file dmx, hdm, NFhdm, the [Output mileage file format] can be set, but the [Merge] checkbox cannot be grayed out and cannot be set. .hdm and .NF-hdm output cross-section data. .dmx outputs longitudinal section data.

When selecting dxf, dwg, shp, dat, csv, the [Merge] can be set, but the [Output mileage file format] is grayed out and cannot be set.

3.10.8 Generate Section View

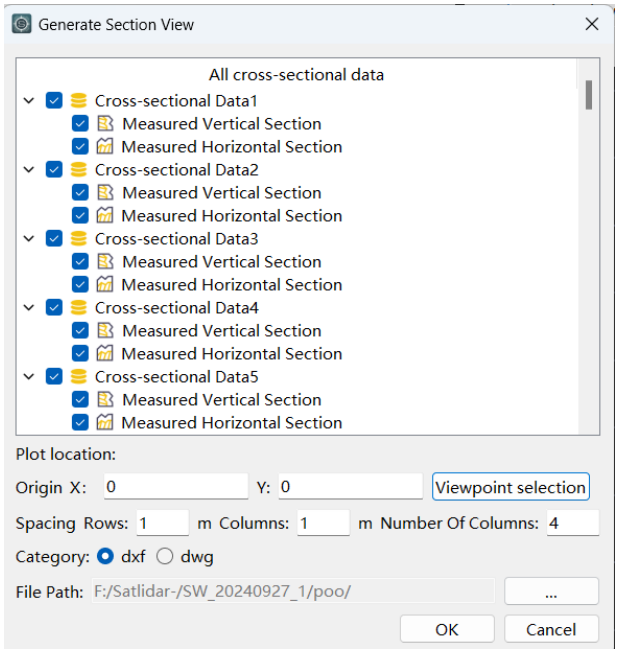

Figure 3-192 Generate Section View interface

The measured longitudinal and transverse sections will be generated for section map generation, and the position of the section can be displayed on the view and selected. The rows, column spacing and the number of columns generated by the section can be set. Support for the generation of section maps in both DXF and DWG formats.

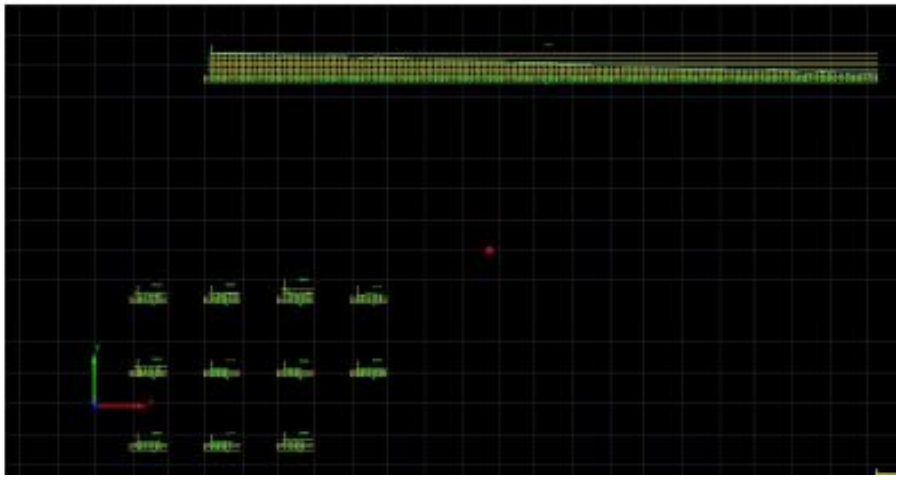

Figure 3-193 Generate Section View effects

3.10.9 Delete

Click the delete button, click on the screen, and then select the vector entity to delete. Left key to select the middle vector line, right-click to confirm the deletion.

Support the deletion of reference line, measured longitudinal section, reference cross section line and measured cross section line. If the reference line is deleted, the pop-up will indicate whether to delete the measured section line, measured section line and reference section line. If selected, delete; otherwise, do not delete.

3.10.10 Section Method Earthwork Calculation

According to the selected section, set the benchmark elevation generation section method earthwork calculation.

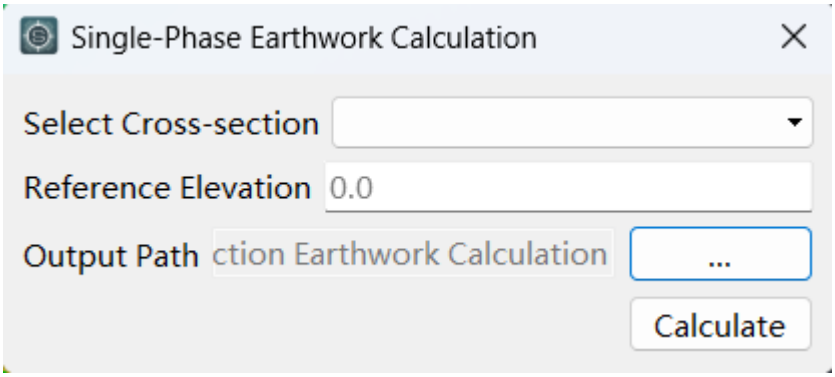

Figure 3-194 Section Method Earthwork Calculation interface

3.10.11 Multi-Period section Comparison

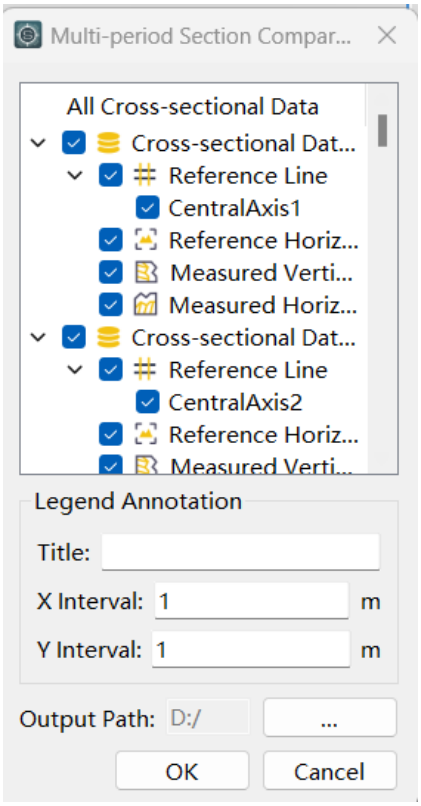

Figure 3-195 Multi-Period section Comparison interface

Support the comparison of section data and output the comparison report, and support the comparison report export of the following situations:

(1) Reference center line, comparison and export of measured longitudinal section lines of two sets of data.

(2) Reference cross section line, comparison and export of measured cross section lines of two sets of data.

(3) Uncheck reference center line, comparison and export of measured cross section lines of two sets of data.

(4) Uncheck reference cross section line, comparison and export of measured longitudinal section lines of two sets of data.

3.10.12 Lower left section tree control

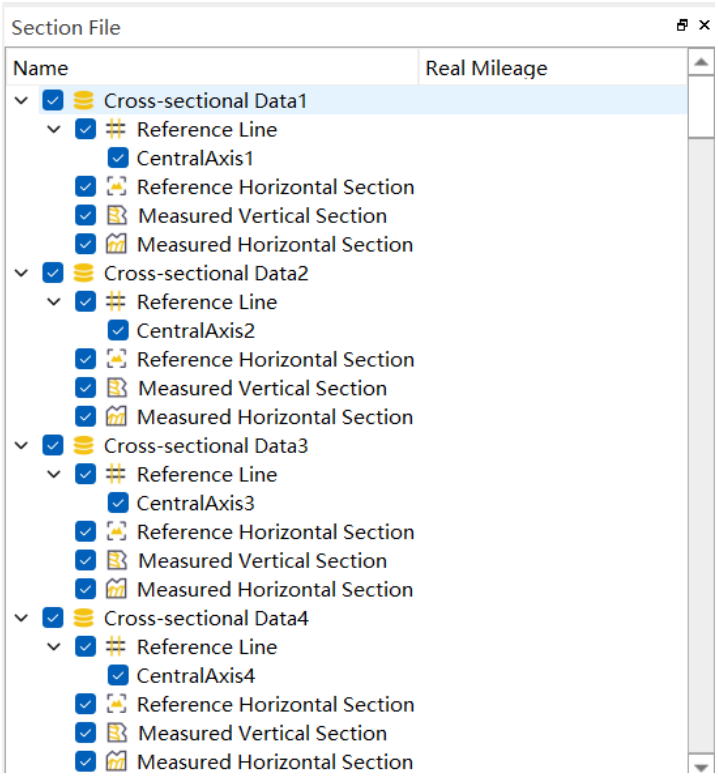

Figure 3-196 Lower left section tree control interface

In the engineering application, once the reference line, reference cross-section, measured longitudinal section, and measured cross-section are generated, they will be displayed in the section tree control in the lower left corner. The system supports operations such as clearing, deleting, and zooming on the nodes. It allows zooming to layers for both primary and secondary nodes, as well as clearing data. When "clear" is selected, the section data will be cleared. For tertiary nodes, it supports zooming to the layer and deletion. Once deleted, the corresponding entity will also be removed from the view. Additionally, it supports toggling the visibility of vector entities.

3.10.13 Calculation of tunnel overshoot and undershoot

<Calculation of tunnel overshoot and undershoot> mainly inputs and calculates the over-and-under-excavation calculation section template.

The user clicks [Engineering Applications] -> "Calculation of tunnel overshoot and undershoot", and the software pops up Calculation of tunnel overshoot and undershoot interface. The user clicks "Add" in the right-click menu of the section template list or "Add" at the bottom of the list, and then clicks "Add" in the right-click menu of the section template list or "Add" at the bottom of the list. The software pops up the section template input interface. The user enters the section template information and clicks "OK". The software exits the section template input interface, and a section template record is added to the section template list.

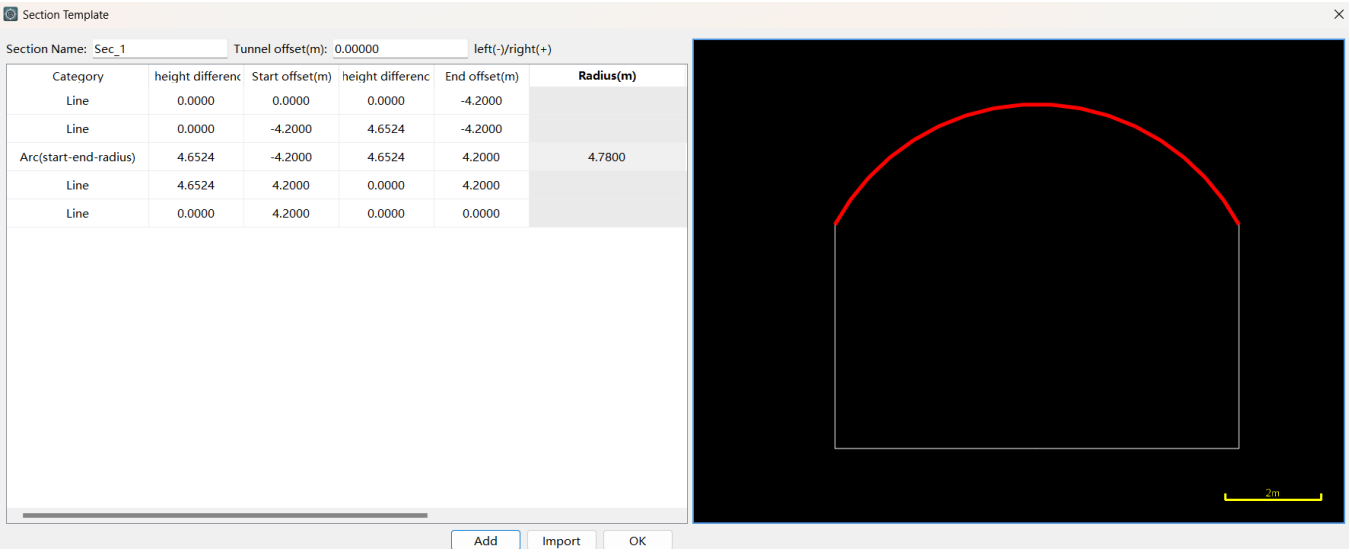

Figure 3-197 Calculation of tunnel overshoot and undershoot template editing interface

Perform cross-section matching, select the mileage to be calculated, click calculate, and a tunnel over-excavation and under-excavation report in DXF format will be automatically generated.

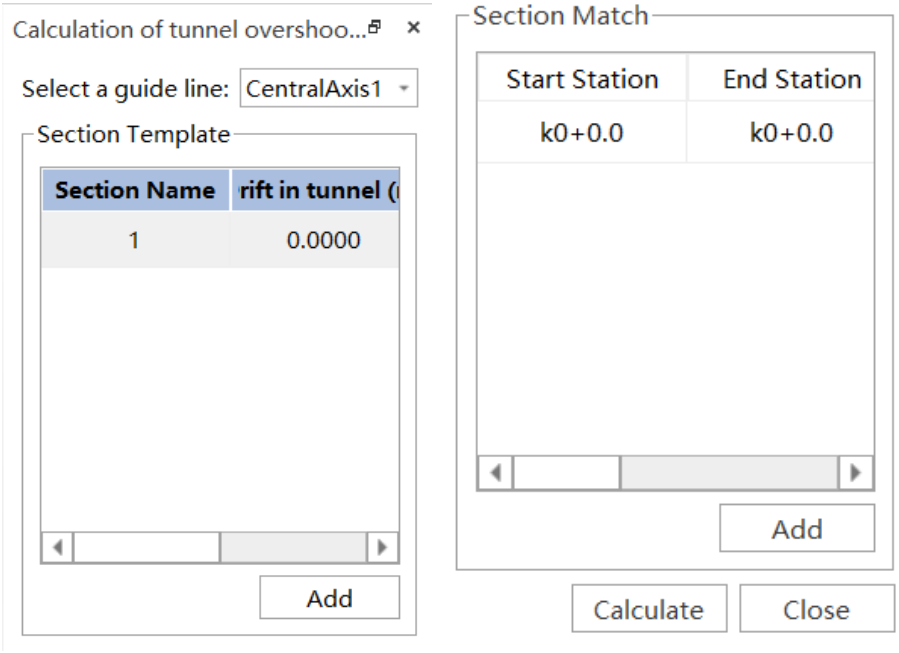

Figure 3-198 Tunnel over- and under-excavation interface

3.10.14 Calculation of cross-sectional flow area

<Calculation of cross-sectional flow area> is mainly based on a design height, which calculates the flow area of the generated measured longitudinal section.

The user clicks on [Section Area Calculation] under [Engineering Applications] to display a dialog box for section area calculation. Enter the design height, enter the title, click OK, and generate a report for section area calculation. The report presents a comparison chart based on design height and longitudinal section, as well as a flow area table and flow area chart based on design height data.

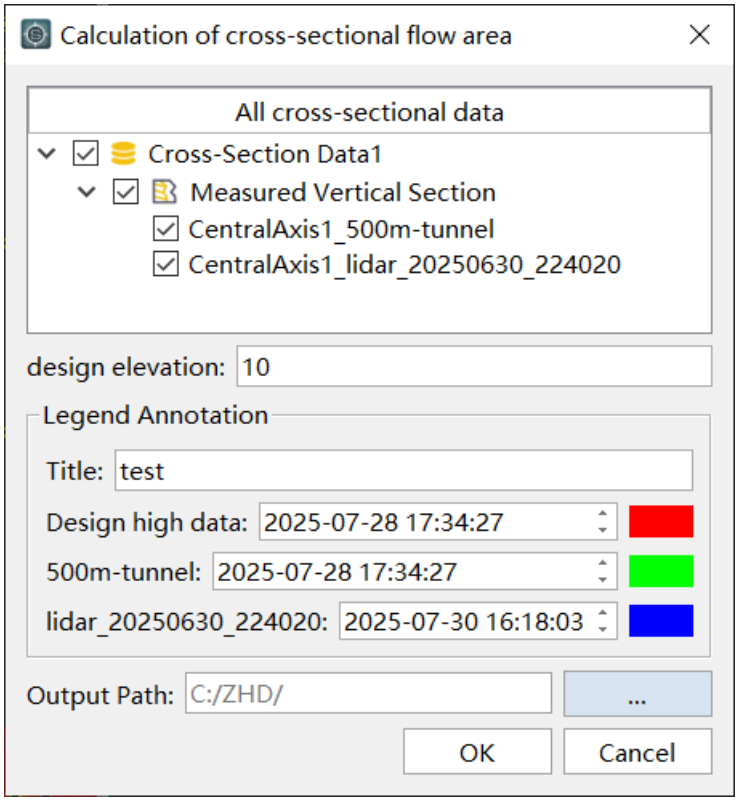

Figure 3-199 Calculation interface for cross-sectional flow area interface

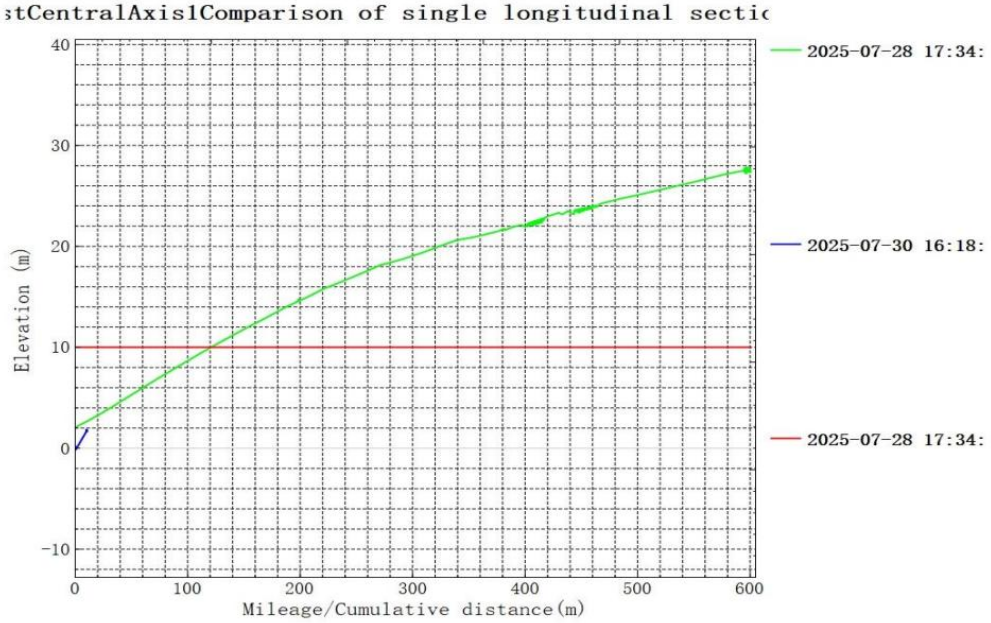

Figure 3-200 Longitudinal section comparison diagram

Point cloud phase number	Measure time	Processed cross-section file	Area
1	20250728173427	CentralAxis1_500m-tunnel	483.092
2	20250730161803	CentralAxis1_lidar_20250630_224020	94.657

Figure 3-201 Statistical Table of Overflow Area

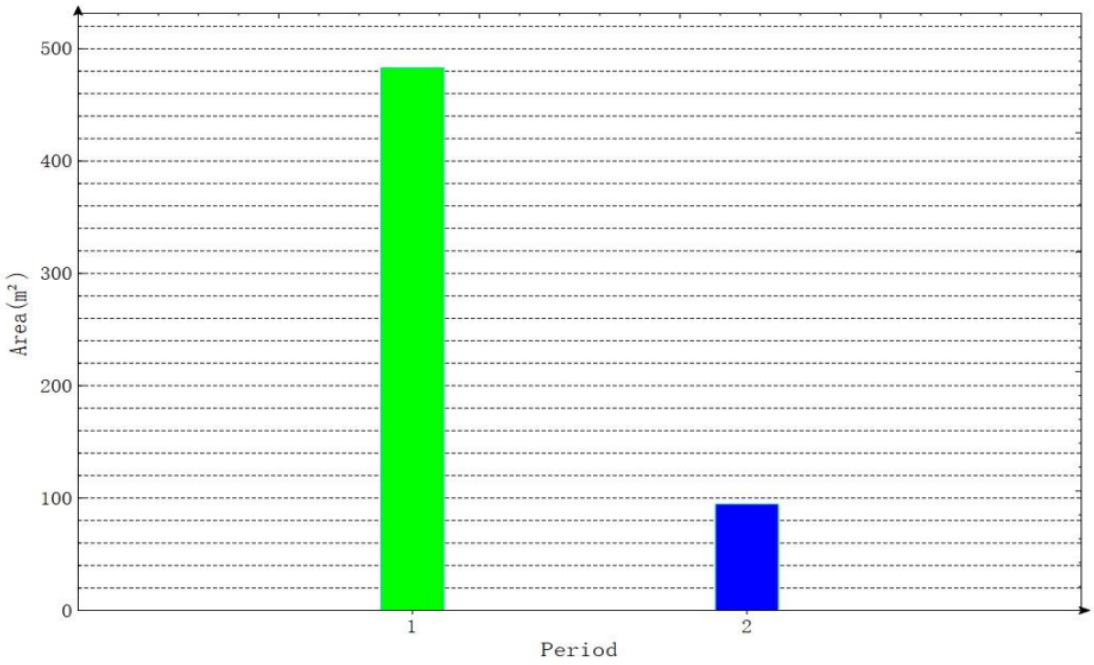

Figure 3-202 Statistical chart of overcurrent area

3.10.15 Cross-section sedimentation analysis

<Cross-section sedimentation analysis> is primarily based on a selected reference. It involves inputting two cross-section lines and two reference lines, where the intersection of these lines forms a calculation area. By setting the interval for analysis lines and sampling interval, a cross-section sedimentation analysis report can be generated.

Click [Cross-section sedimentation analysis] under [Engineering Applications] to open the "Cross-section sedimentation analysis" dialog box. Import the original point cloud data of two phases. Import two cross-section lines (in DXF format). These lines are opposite to each other and serve to enclose the sub-region for analysis, Import two reference lines (in DXF format). These reference lines intersect with each other and also intersect with the cross-section lines. In the displayed " Cross-section sedimentation analysis " dialog box:

(1) Select the reference point cloud.

(2) Import the two cross-section line files, which will be displayed in the 3D view.

(3) Import the two reference line files, which will also be displayed in the 3D view.

(4) Set the interval for analysis lines. The analysis lines start from Reference Line 1 and are spaced at the specified interval while extending toward Reference Line 2.

Click [OK] to start calculations and export the analysis report.

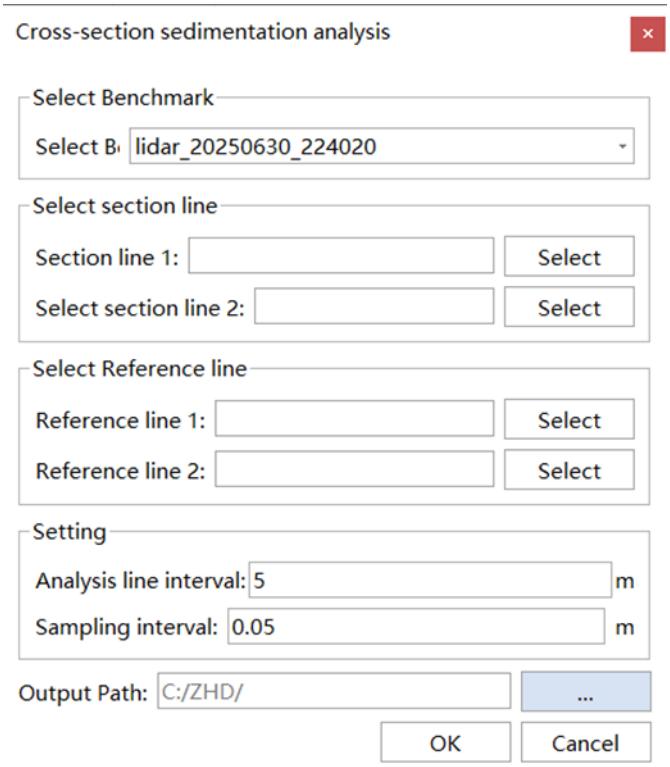

Figure 3-203 Cross-section sedimentation analysis interface

3.11 Draw

The <Draw> menu is used to add various point, line, and surface entities, and to perform operations such as deleting, line interruption, two-point interruption, connection, copying, extension, and smoothing on entities. When drawing entities, you can select the layer and coding attributes, and support viewing the attributes of various point, line, and surface entities.

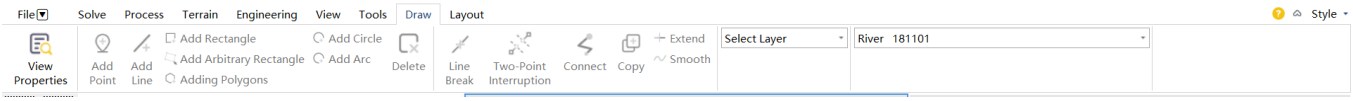

Figure 3-204 The vector-editing interface

3.11.1 View the properties

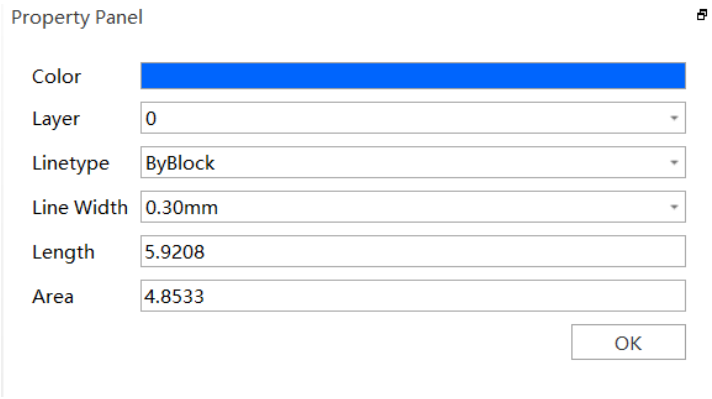

Figure 3-205 View the attribute interface

Click [View Properties] under [Draw], select a reference centerline, or measured longitudinal section line, or reference cross section line, or measured cross section line, and display the [Properties Panel] interface when [Properties Panel] is open in [Layout]. You can edit the color, layer, line type, and line width, but the length and area cannot be edited.

3.11.2 Add Point

Click [Draw] -> "Add Point" or the shortcut toolbar "Add Point", select the layer and code in the drop-down box, and left-click the position to add the point on the view.

3.11.3 Add Line

Click [Draw] -> "Add Line" or "Add Line" in the shortcut toolbar, select the layer and code in the drop-down box, click the nodes of the view to add lines in turn with the left mouse button, and right mouse button to end adding lines.

3.11.4 Add Rectangle

Click [Draw]-> "Add Rectangle" or "Add Rectangle" in the shortcut toolbar, select the layer and code in the drop-down box, left-click the view to add the starting point of the rectangle, move the mouse, then left-click the end point of the rectangle, and then then left-click the opposite corner of the rectangle to end adding the rectangle.

3.11.5 Add Arbitrary Rectangle

Click [Draw]-> "Add Arbitrary Rectangle " or "Add Arbitrary Rectangle " in the shortcut toolbar, select the layer and code in the drop-down box, left-click the view to add the starting point of the rectangle, move the mouse, then left-click the end point of the rectangle, and then left-click the opposite corner of the rectangle to end adding the rectangle.

3.11.6 Add Polygon

Click [Draw]-> "Add Polygon" or "Add Polygon" in the shortcut toolbar, select the layer and code in the drop-down box, click the left mouse button on the drawing to add a polygon node, right click to end the drawing, and the polygon will automatically close at the beginning and end without filling, completing the drawing of a single polygon entity.

3.11.7 Add Circle

Click [Draw]-> "Add Circle" or "Add Circle" in the shortcut toolbar, select the layer and code in the drop-down box, and then click the left mouse button on the drawing to add a circular node. Select the node as the position on the arc, add three nodes to judge and end the drawing command. The triangle formed by the three selected points must be an acute triangle. After judging that the inscribed triangle of the circle meets the requirements, submit the drawn circle and save it to the corresponding vector file.

3.11.8 Delete

Click [Draw]-> "delete", and then choose to delete the entity, When selecting, you can select the entity by the left mouse click or box selection,, if the choice is wrong, or more, or choose less, can hold the ctrl key to select point or box for increasing or reducing the selection. After the selection is completed, right-click to complete the delete operation.

3.11.9 Line Break

Click [Draw]-> "Line Break" or "Line Break" in the shortcut toolbar, to select the line by the left mouse click or box selection select single or multiple closed line (surface) entities, break the intersecting line entities, and remove the part within the closed surface to obtain the new line segment and the original line segment after truncation.

3.11.10 Two-Point interruption

Click [Draw]-> "Two-Point interruption" or "Two-Point interruption" in the shortcut toolbar, select the nearest line entity within the threshold range by clicking the left mouse button, and directly specify the first point on the line according to the selected position, click the left mouse button to select the second point on the line, break the line into three segments according to the two points, remove the middle part between the two points, and keep the remaining two parts to obtain the newly added line object and the original entity after truncation, and submit the modification to the vector data.

3.11.11 Connect

Click [Draw] -> "Connect" or shortcut bar "Connect", left click select one end of the line, left click select one end of the other line, to connect.

3.11.12 Copy

Click [Draw] -> "Copy" or shortcut bar "copy", left click to select the entity to copy, move the mouse, and then click the left button to copy.

3.11.13 Extend

Click [Draw] -> "Extend" or shortcut bar "Extend". On the drawing, left-click the mouse to select the line entity to be extended, and determine the distance from the first and last breakpoints of the line according to the selected position. Use the near end as the starting point for extension. On the drawing, left-click the mouse to select another line entity as the target entity to be extended to. Extend according to the selected endpoint and the direction of the previous point (the previous node and breakpoint are taken to form the tangent direction after the curve or arc is broken into a line) of the line entity to be extended, and obtain the intersection with the target line entity. Add the intersection position to the line entity to be extended as the new endpoint position, and submit the line entity after extension to the vector data.

3.11.14 Smooth

Click [Draw]-> "Smooth" or "Smooth" on the shortcut toolbar, you can select the line entities that need to be smoothed by the left mouse click or box selection. If you make a wrong selection, or select too many or too few, you can hold down the ctrl key to click or box select to increase or decrease the selection. After the selection is complete, right-click to complete the smoothing operation.

3.11.15 Add Arc

Click [Draw]-> "Add Arc" or "Add Arc" on the shortcut toolbar, select the layer and code in the drop-down box, and then click the left mouse button on the drawing to add an arc node. After adding the three nodes, judge and end the single arc drawing. Draw the arc composed of three points through the selected three points, and write the selected layer and Cass code into the new entity.

3.12 Architectural

The <Architectural > menu is used to slice point clouds, draw slices, and export drawing data.

3.12.1 Point Cloud Slice

Click [Architectural]->"Point Cloud Slice" button to automatically display the section view and pop up the point cloud slice property page. The properties display the following contents: show/hide slice arrow button, reference plane (including X-axis, Y-axis, Z-axis, any three points), point selection interface including XYZ value and point selection button, drawing button, close button. The coordinates of the point selection interface default to the average value of point cloud XYZ, supporting three points to determine the reference plane, and two methods of determining the plane by X-axis, Y-axis, and Z-axis.

After determining the plane, you can select points on the view by selecting points, or select points by manually entering point coordinates in the display box. After entering the value, press Enter to modify the parameters and take effect.

At the same time, you can customize the thickness value of the slice, which will take effect as soon as it is entered, and the input thickness value will be displayed in the section view and drawing window.

Click Draw to jump to the drawing interface and perform drawing operations on the current slice.

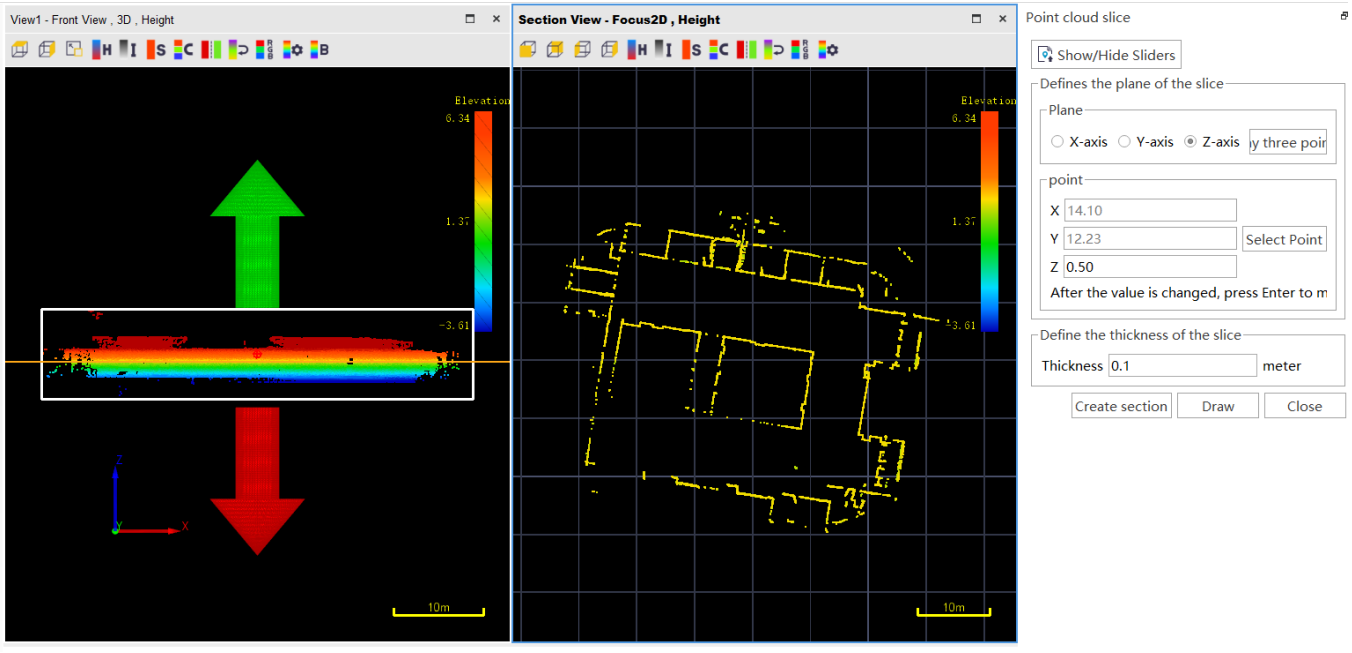

Figure 3-206 Point cloud slices interface

3.12.2 Export

Click [Architectural]->"Export" button to jump to the export interface, and export the currently drawn vector file. It supports merged export, and the output format runs DXF and DWG. The export options support 3D and 2D. It also supports exporting the drawn slices in the specified las format.

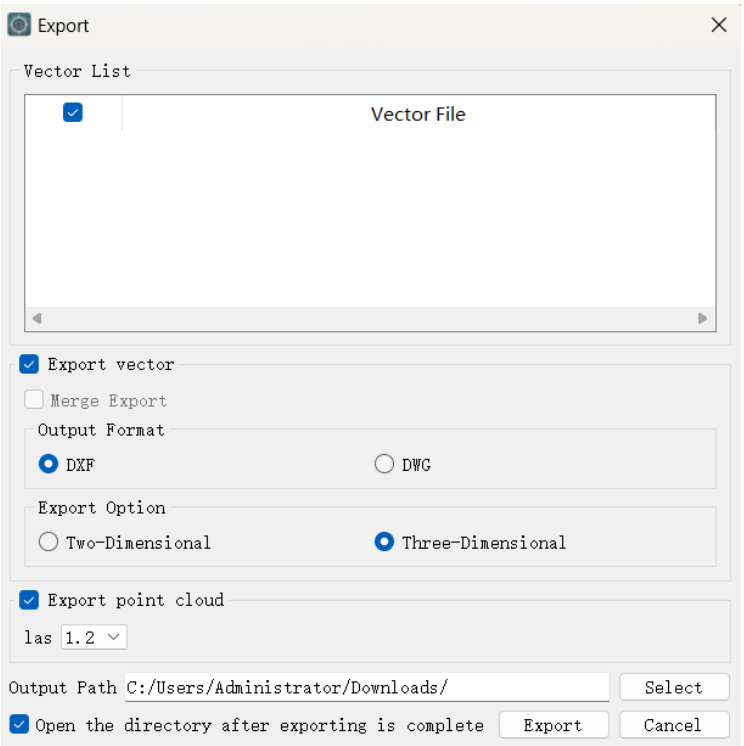

Figure 3-207 Export interface

3.13 SLAM

3.13.1 SLAM Processing

SLAM point cloud processing can calculate the POS trajectory and point cloud from the collected SLAM data. After creating a new project or opening a project, click <SLAM> → < SLAM Processing > on the main interface menu to pop up the point cloud solution dialog box, in which the project name and task name under the project folder are automatically read. If different tasks have different acquisition modes or need to be solved by different solution methods, users can select project/task settings respectively. In addition, you can check the functions of point cloud coloring, moving object removal, point cloud denoising and point cloud smoothing as required. After checking, you can synchronize processing during point cloud processing. When there is a separate acquisition control point in the acquisition process, you can check <Control Point> and click <Edit> → <Import Control Point> → check whether to enable, so as to improve the accuracy of point cloud processing by matching the coordinates of control points.

When selecting PPK solution, you can choose the source of the base station, which defaults to built-in RTCM. When selecting a GNSS file, you need to locate the file path and enter the base station coordinates and antenna parameters. Alternatively, you can select from favorites. See 3.3.1.1 for details. After setting is complete, click OK.

The selectable point cloud formats for outputting point clouds include HLZ and LAS. The format version can be selected from the dropdown menu.

The coordinate system settings, as described in 3.3.1.3, encompass projection, datum, ellipsoid transformation, plane transformation, elevation fitting, grid correction, translation parameters, options, parameter calculation, import, and coordinate system management. If you check apply to all projects, the coordinate transformation parameters of the current project will be applied to all other projects. If you do not need to apply them to other projects, do not check this option.

If you need to solve all projects, check solve all projects. If not checked, it will solve the current project. After setting, click start solving.

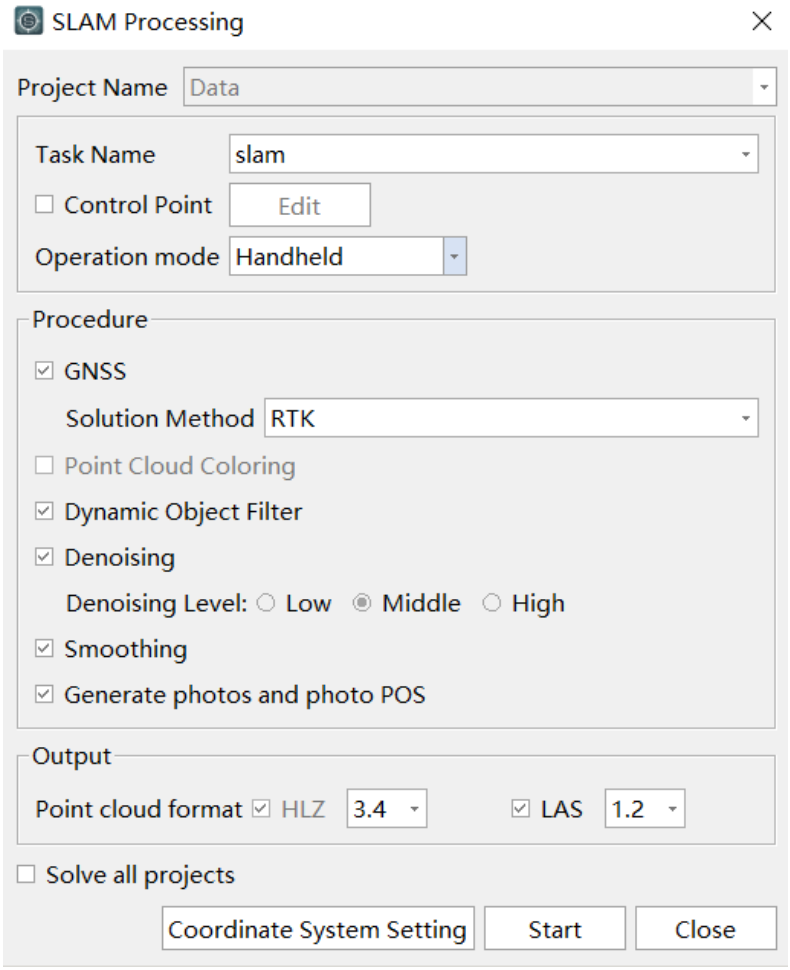

Figure 3-208 SLAM Processing interface

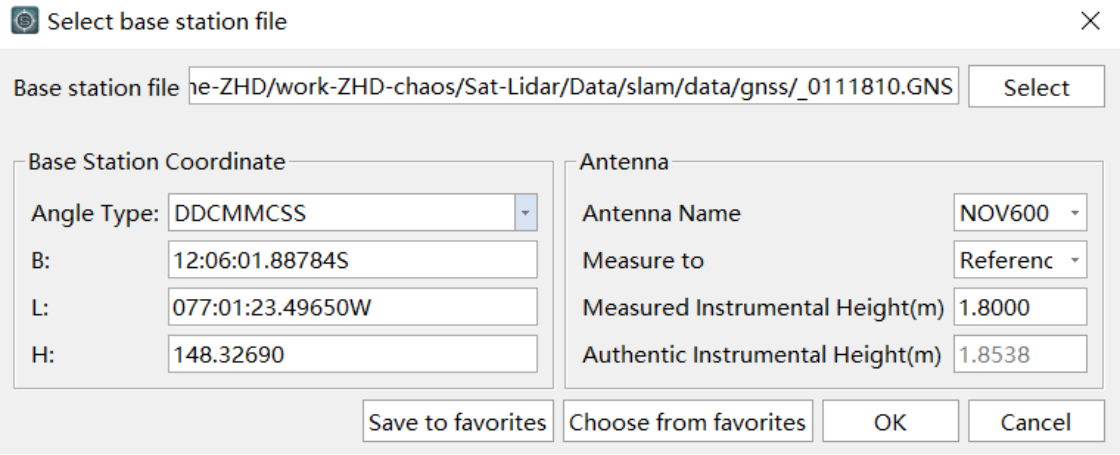

Figure 3-209 Select base station file interface

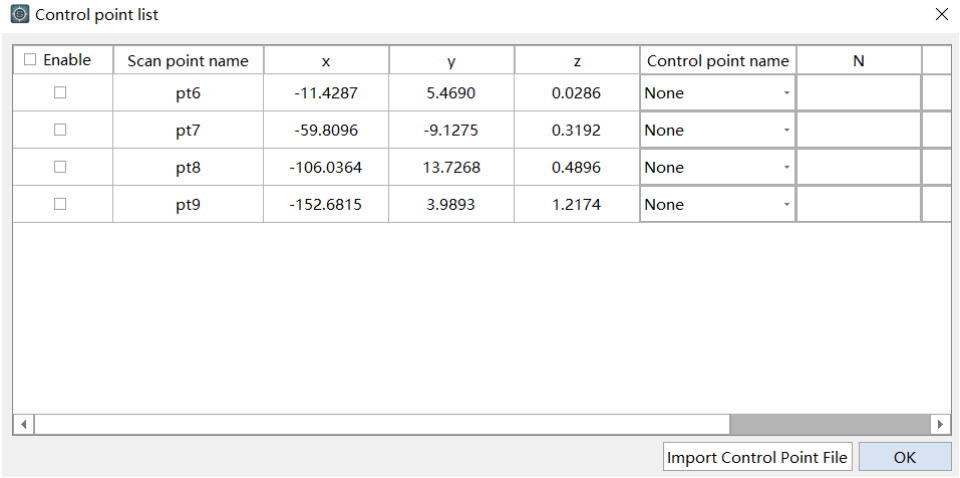

Figure 3-210 Import Control Point interface

3.13.2 Sampling Points Align

SLAM control point concatenation is mainly used to concatenate point cloud data from different collection tasks through the collection point pairs during the job. Click on the menu<SLAM>→ <Sampling Points Align>on the main interface to bring up the Sampling Points Align dialog box.

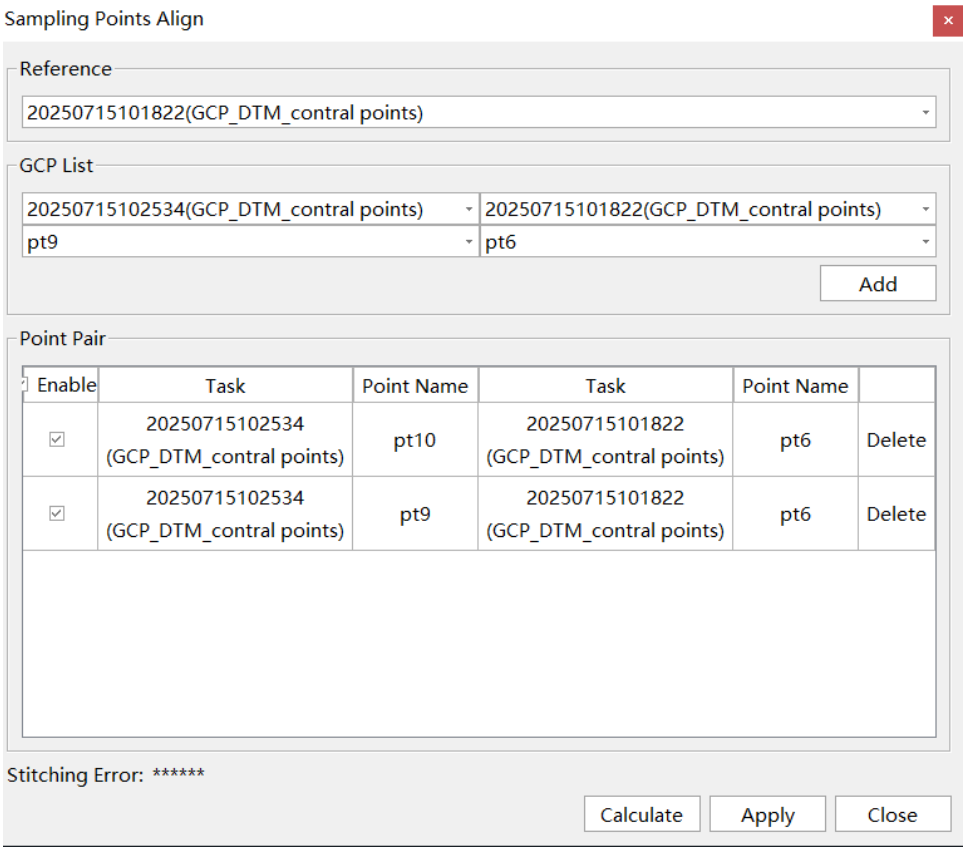

Figure 3-211 Sampling Points Align interface

3.13.3 Coloring

SLAM point cloud coloring can be performed after selecting the model file. Click on the menu<SLAM>→<Coloring>on the main interface, and a point cloud coloring dialog box will pop up. Select the point cloud path and model folder path, check the point cloud that needs to be colored, and click OK to color it.

Figure 3-212 Import Control Point interface

3.13.4 Point Cloud Denoising

SLAM point cloud denoising mainly involves denoising and smoothing point cloud data. Click on the<SLAM>->< Denoising>menu on the main interface to bring up the Point Cloud Denoising dialog box. By default, the dialog box will select Point Cloud Denoising and Point Cloud Smoothing, and list the currently loaded point cloud files in the software. You can also select the point cloud files that need to be processed. If point cloud smoothing is not required, you can uncheck them.

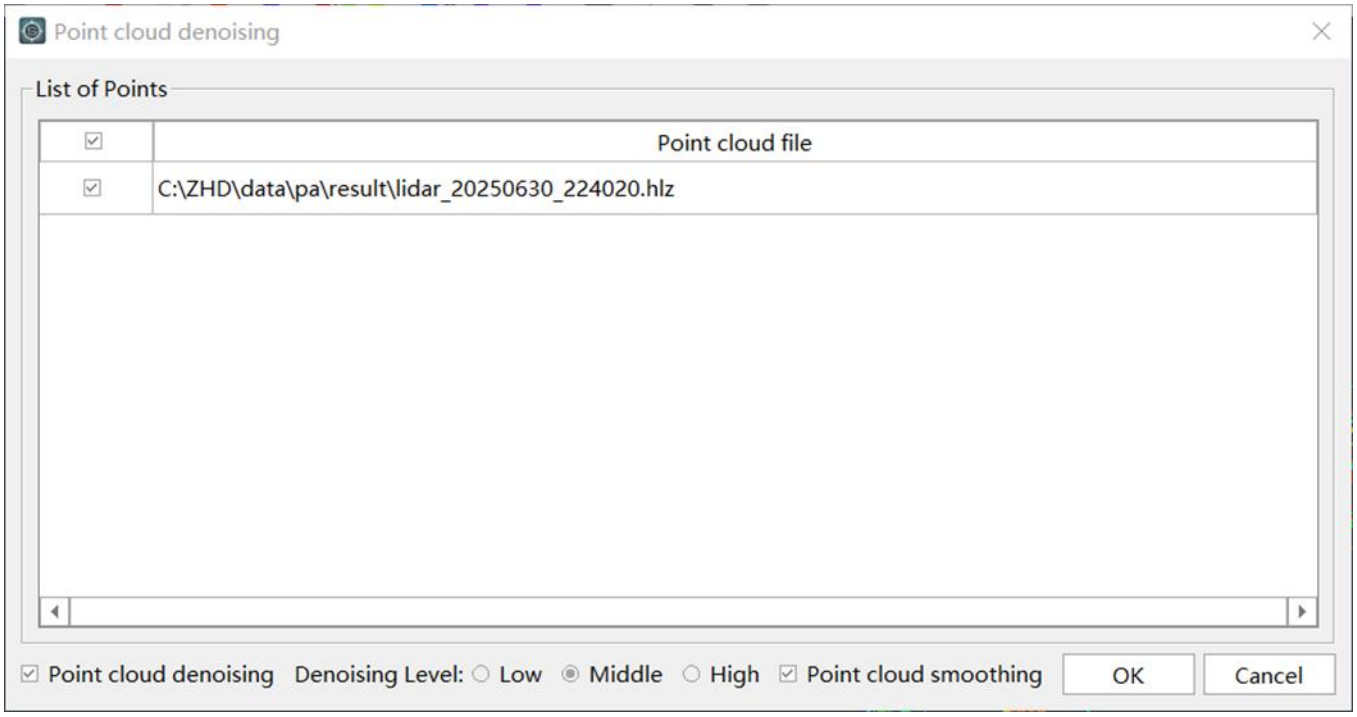

Figure 3-213 Point Cloud Denoising

3.13.5 Manual denoising

SLAM manual denoising mainly involves denoising and classifying post-processing point cloud data. Click on the menu<SLAM>→<Manual denoising>on the main interface, and a point cloud denoising dialog box will pop up. Select the initial category and target category in the pop-up window, and select the target point cloud in the box. If the box selection is incorrect, click undo. After confirming that the box selection is correct, save and exit.

3.13.6 Collection Point

SLAM collection mainly exports the point coordinates collected in SLAM mode during the job process. Click on the<SLAM>-><Collection Point>menu on the main interface, and a collection point dialog box will pop up. Select the project name and task name, and the collection point information will be automatically read. Confirm the export type (export the collection point of the current task or the collection points of all tasks under this project), and then click Export to bring up the collection point export dialog box.

File: You can specify the export file name and select the export file format, .txt, .csv.

Location: Specify the export file location.

Custom format settings: When there is no template, the exported content and field order can be specified. If selected on the left and added to the right, it will be considered as the specified field; You can choose to export in the order of collection or in the order of point numbers in the export order; Floating point precision can specify the number of decimal places in a numerical value; The delimiter

can specify the delimiter symbol between fields; If the format file header is checked, the exported file will contain data headers.

Template management: Templates can be fixed and exported through customization.

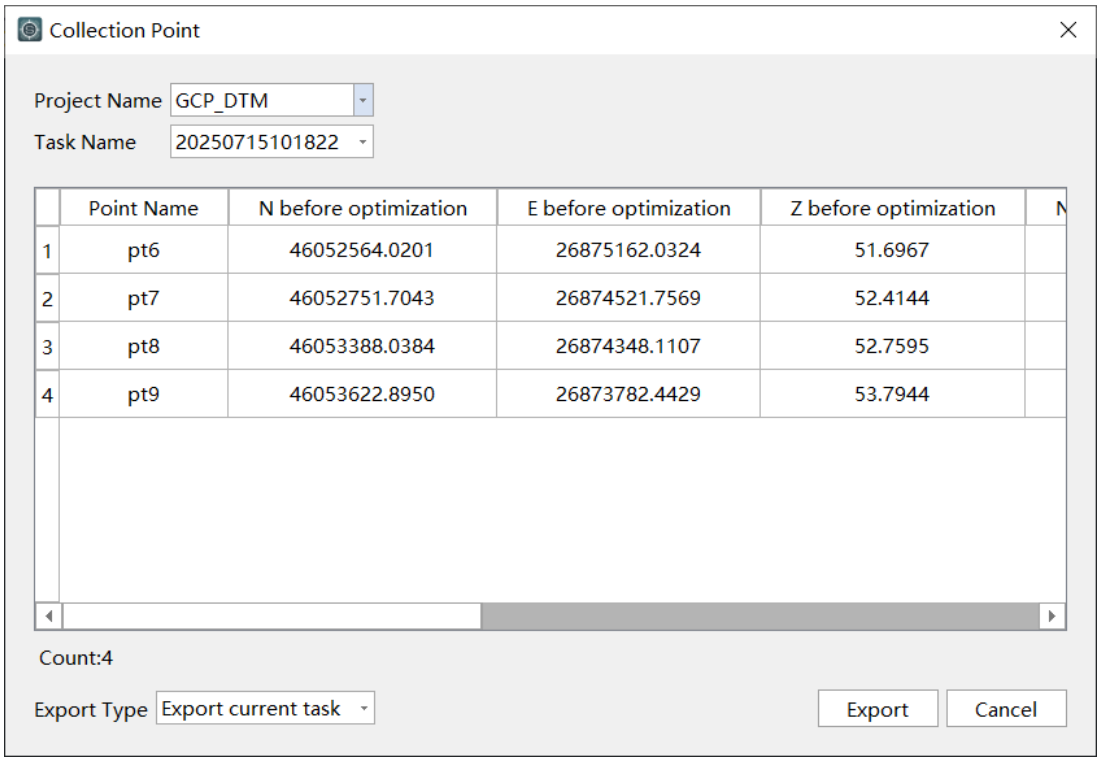

Figure 3-214 Export of collection points

3.13.7 Point Cloud Export

On the main interface, click on the menu <SLAM> ->< Export> to bring up a dialog box. Users can select the desired export options, surface category, output format, and point cloud attributes for customized export.

Export options: <Filter> supports setting parameters to dilute and export point clouds; <Class> supports selecting a specific category for export, with multiple options available. The default category is all categories; <Regular Export> means complete export; <Subsection> means that each original scanned file is exported to multiple target point cloud files in segments based on GNSS time; 'Split export' means that each original scanned file is exported to multiple point cloud files based on the polygons in the polygon file, and the polygons involved in the export can be set through field values; <Volume Export> means that point cloud data is divided into files according to the set size for output (this setting is only valid for las format point clouds).

The output formats include: hlz, las, laz, pts, xyz, pcd, and e57, with multiple export formats available. Click the export button to export the selected point cloud.

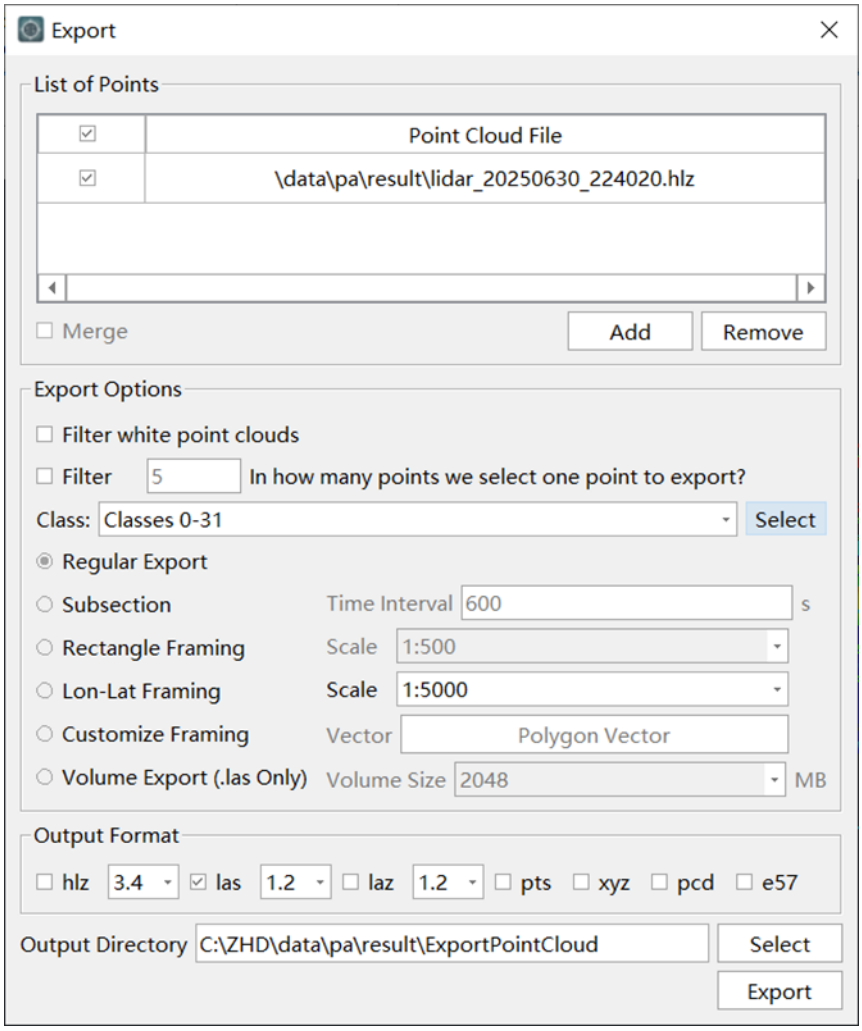

Figure 3-215 Point Cloud Export

3.13.8 Related Control Points

Click on the menu <SLAM> ->< Related Control Points > on the main interface to support associating control points in txt and csv formats. After selecting the control point file, click OK to open the ASCII file window.

When the control point file has a title or header, it is necessary to set the first few lines of the control point to be ignored; Then correspond the coordinate orientation of each column of data for the control point, set it up, click apply, and display the control point in the current point cloud.

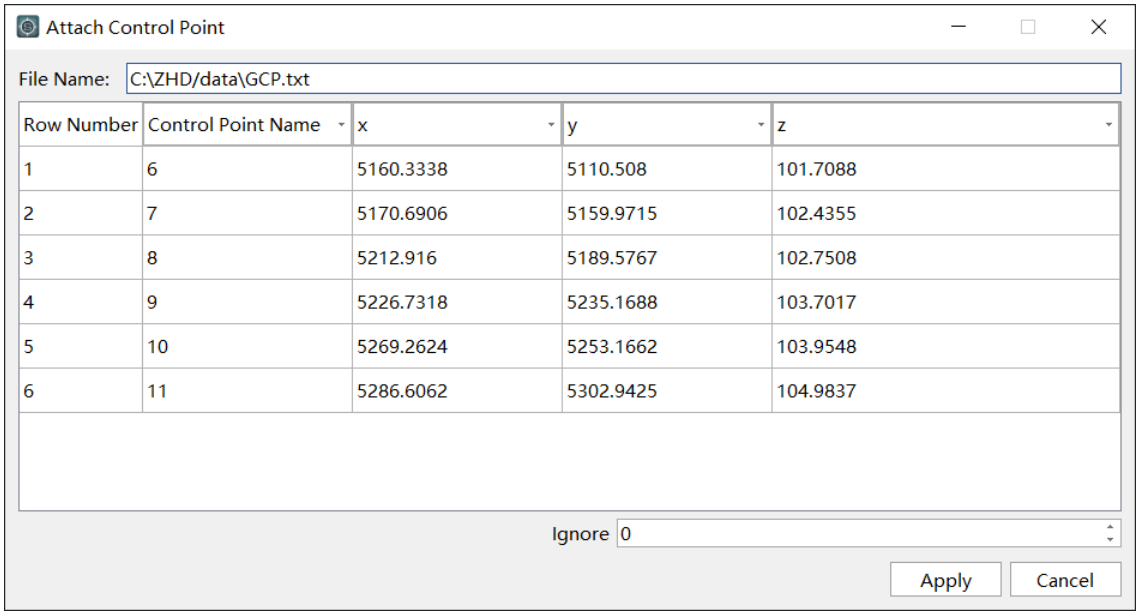

Figure 3-216 Related Control Points

Notes: 1. Ignoring the header of the control point may result in the failure of importing the control point;

2. Incorrect correspondence of control point coordinates to xyz can result in the control point not being together with the point cloud.

3.13.9 Accuracy Inspection

After loading the point cloud and associating it with control points, enter <SLAM> → < Accuracy Inspection > to enter the Accuracy Inspection dialog box. Accuracy Inspection provides two methods for point selection: manual point selection and automatic search. The accuracy verification function is not bound to the project and can be imported separately into las or hlz to support verification.

1.Manual point selection

By selecting scanning points to perform accuracy verification with control points, the results of accuracy verification can be obtained.

Click on precision check, and the page will bring up a magnifying glass view for point selection. Select the matching scanning point at the corresponding position of each control point, left click to confirm the point selection. After the scanning point selection is successful, the corresponding scanning point will be filled to the right of the control point list. If you are not satisfied with the selected point, you can reposition it to the corresponding scanning point position and select the scanning point again, or manually modify the scanning point coordinates and press Enter to confirm. At this point, complete the selection of all scanning points, click to display the results, and the system will automatically calculate the accuracy of the current data based on the correspondence between the control points and the scanning points. At this point, clicking on export report will redirect you to the report save interface.

2.Automatic search

When clicking on automatic search, the program first searches for target points near the control point. Currently, it only supports square checkerboard grids with a size of 50cm * 50cm targets. If it cannot be found, then a point cloud of 1m * 1m in size is found within a 1m range above and below the control point. After finding it, a weighted coordinate is calculated based on the zy distance from the point to the control point. If normal is selected, the scanning point xyz will not be grayed out. If it is an elevation, the scanning point xy will be grayed out. If it is a plane, the scanning point z will be grayed out.

The formula for calculating the mean square error σ_x in the X direction is:

$$\sigma_x = \sqrt{\frac{\sum_{i=1}^{n} X_i^2}{n}}$$

The formula for calculating the mean square error σ_y in the Y direction is:

$$\sigma_y = \sqrt{\frac{\sum_{i=1}^{n} Y_i^2}{n}}$$

The formula for calculating the mean square error σ_z in the Z direction is:

$$\sigma_z = \sqrt{\frac{\sum_{i=1}^{n} Z_i^2}{n}}$$

The calculation formula for the error σ_{plane} X in the plane is:

$$\sigma_{plane} = \sqrt{\sigma_x^2 + \sigma_y^2}$$

In the formula, X_i, Y_i and Z_i respectively represent the coordinate error values between the scanning point and the control point in the X, Y, and Z directions.

Surface category selection: After selecting a category, only the points corresponding to the category will be searched during automatic search. The category can be selected from a certain category or all categories, and the category defaults to all category points; Manual point selection in the view is not limited by the category set; The magnifying glass view displays the specified category point cloud.

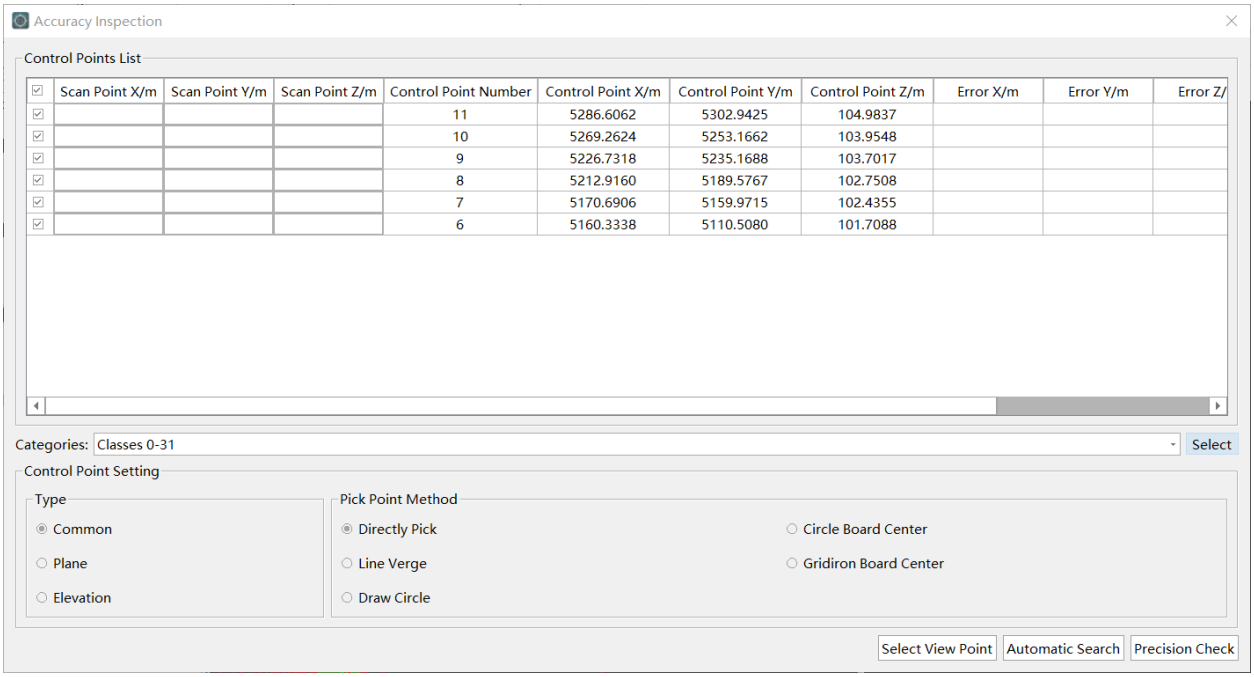

Figure 3-217 Accuracy Inspection main

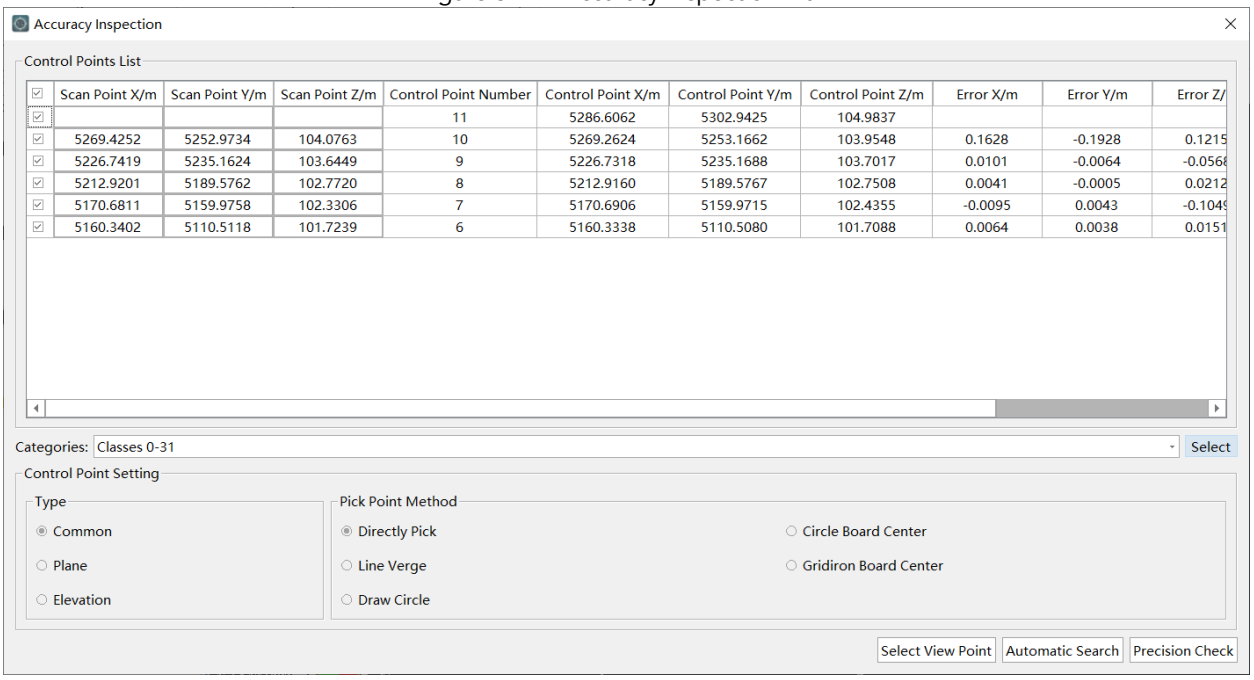

Figure 3-218 Automatic Search

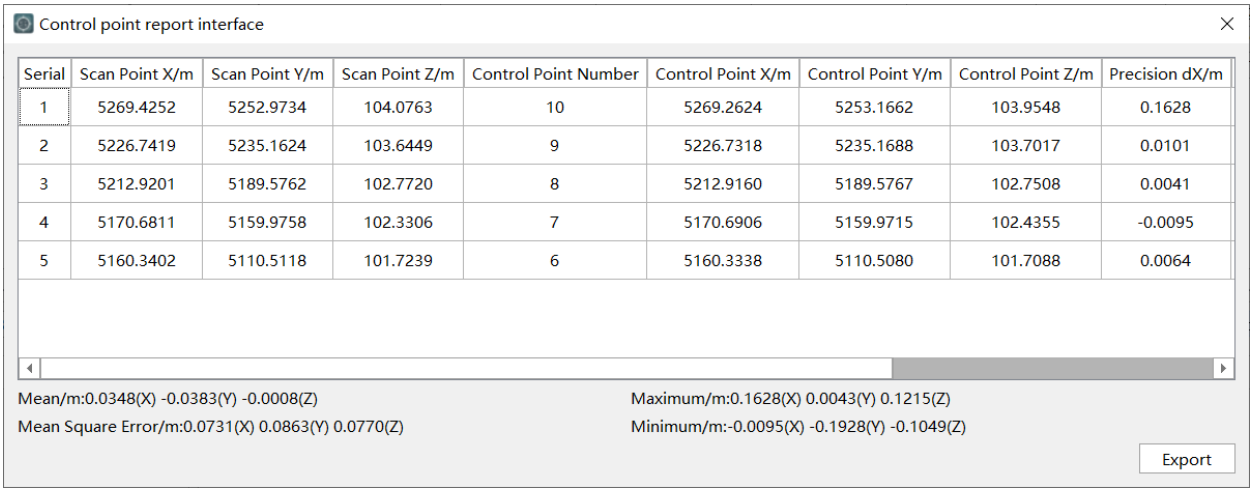

Figure 3-219 Precision Check

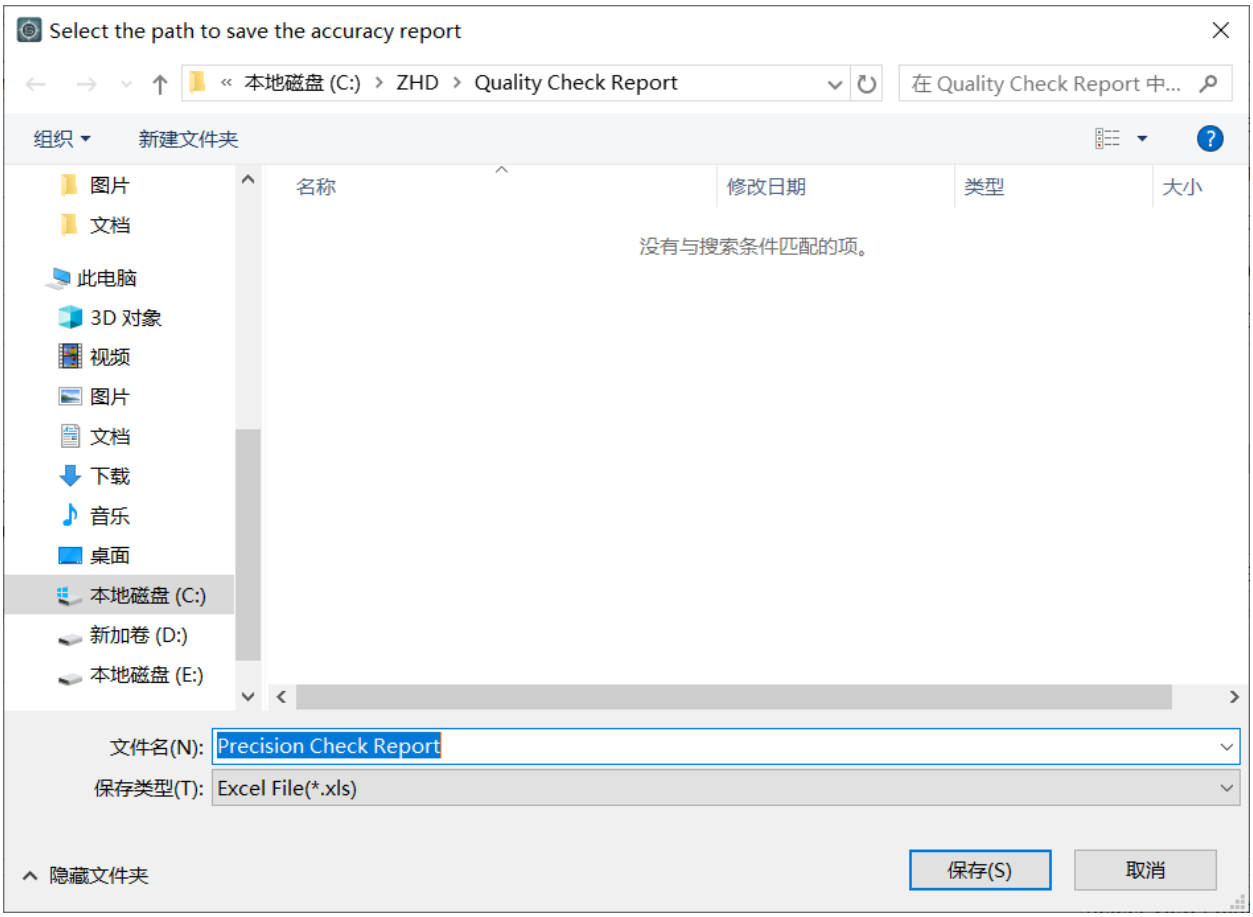

Figure 3-220 Export Report

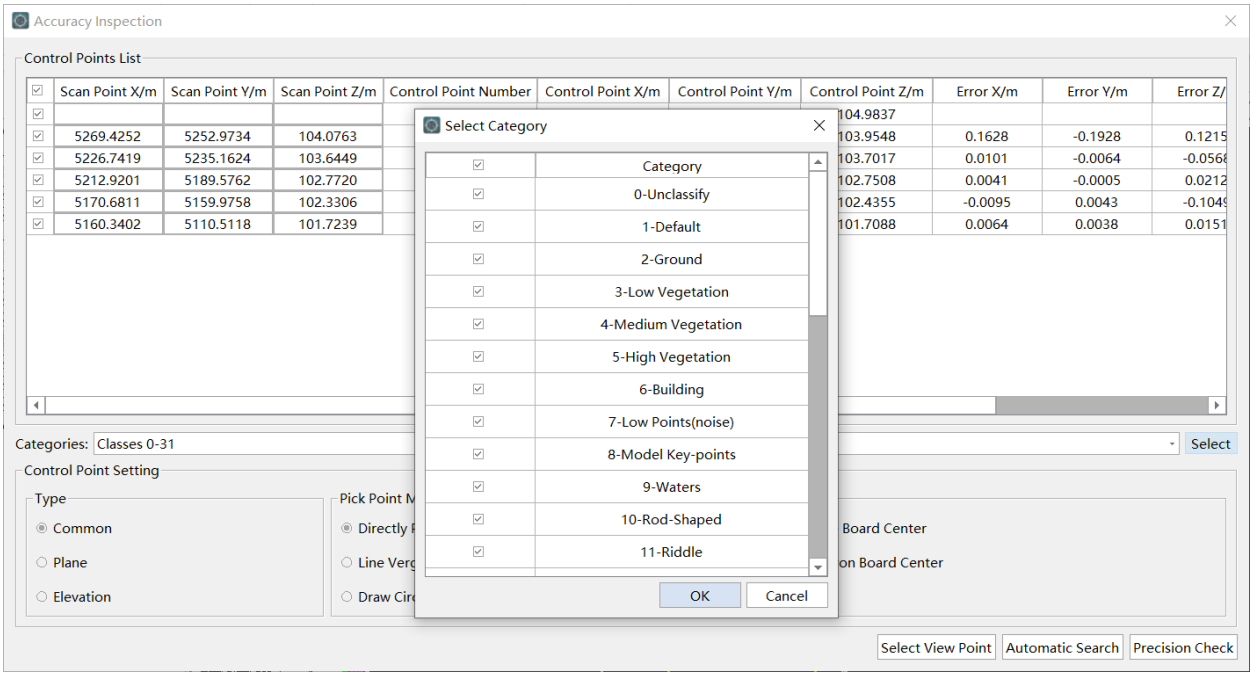

Figure 3-221 Category Selection

Control point types: including regular, planar, and elevation control points. Horizontal control points are only valid for X and Y coordinates, while elevation control points are only valid for Z coordinates.

There are five point selection methods: ordinary point selection, fitting intersection points, fitting center points, fitting circular targets, and fitting checkerboard targets. Ordinary point selection can be directly left clicked on the magnifying glass view for point selection; The fitting intersection point is determined by identifying the intersection point of two lines to select a point; Fitting the center of a circle refers to selecting two points on the magnifying glass view to determine the diameter of the circle, and moving the mouse to determine the shape of the circle. At this point, the center of the circle is the determined point. Fitting a circular target is automatically fitting the center of the target based on the selected point cloud. Fitting a checkerboard target is selecting the middle point position of the checkerboard points.

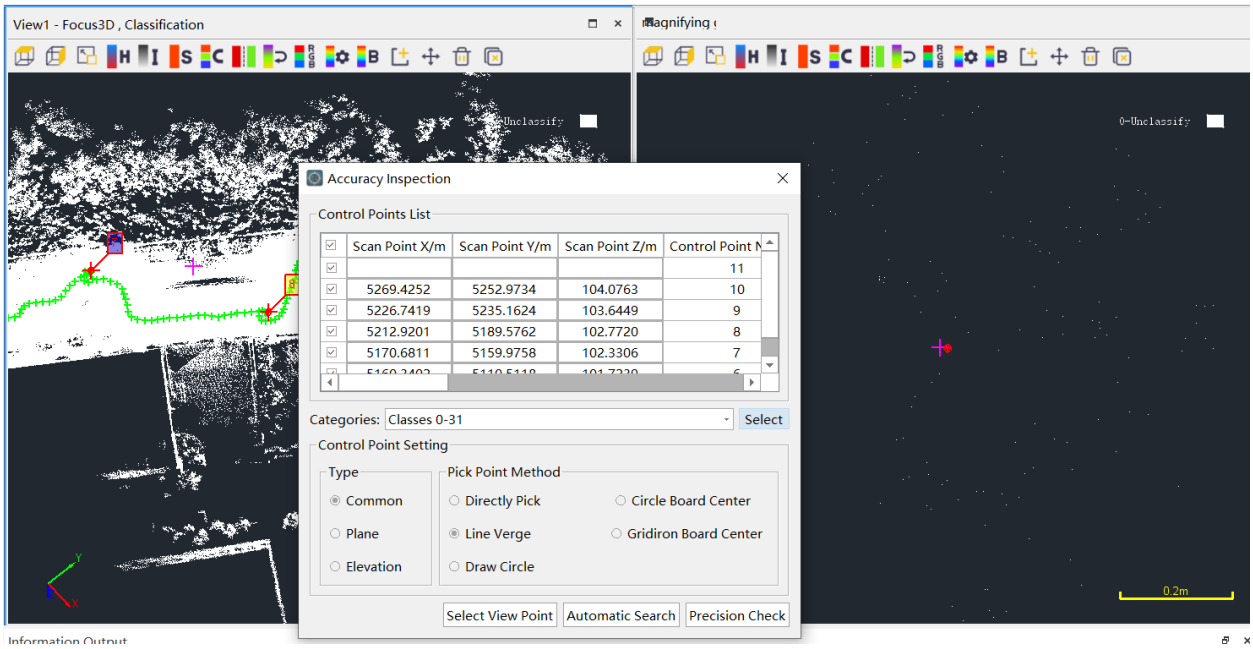

Figure 3-222 Line Verge

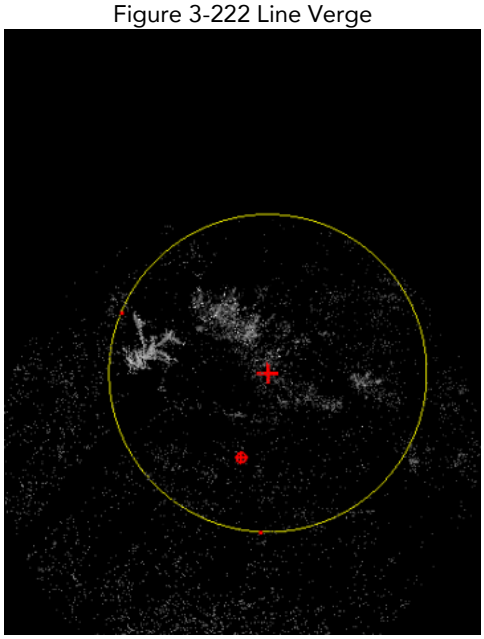

Figure 3-223 Draw Circle

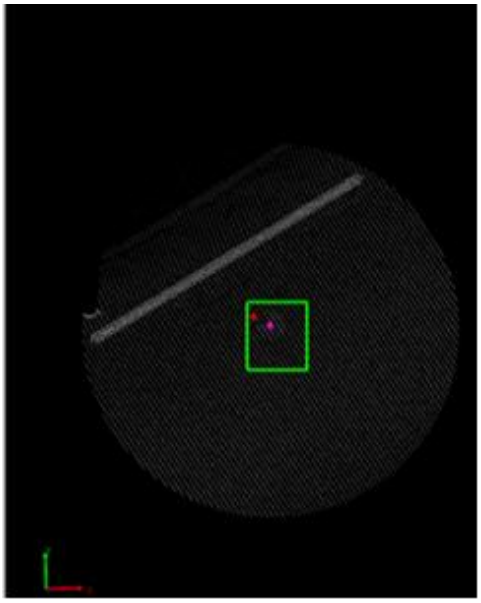

Figure 3-224 Circle Board Center

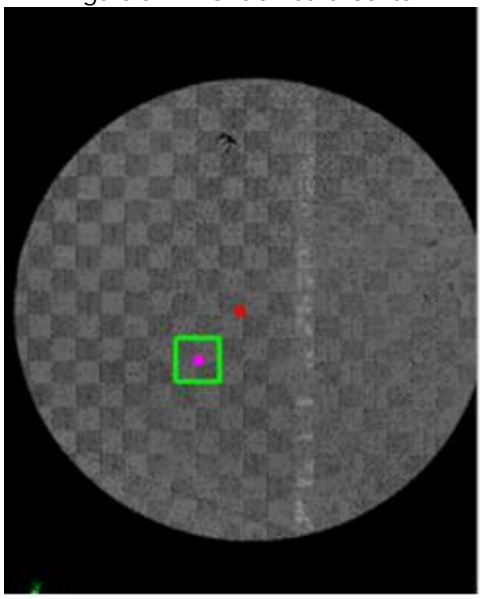

Figure 3-225 Chessboard Center

3.13.10 Earthwork Calculations

After loading the point cloud, enter <SLAM> → <Earthwork Calculations> to enter the Earthwork Calculations dialog box. Earthwork calculation mainly includes single phase earthwork calculation and multi phase earthwork comparison. The calculation format supports HLZ, LAS, Tif, and allows for the selection of basic elevation, setting range calculation, and more.

3.13.10.1 Single-Period Calculation

Single-Period Calculation requires selecting a point cloud file that supports (HLZ, LAS, TIF) formats. The dropdown menu allows you to select the file to import the view, while the 'Select' option on the right allows you to export a point cloud file from your local computer.

The benchmark elevation can be set through the right side bounding box, including minimum value, maximum value, average value, self selected point, and customization. Among them, the self selected point can be directly clicked from the view to obtain a benchmark elevation point; Customization can manually fill in an elevation value as the reference elevation.

The algorithm has options of Triangulation and Latticework, with Triangulation being the default. The grid size can be set by default (0.2m), which determines the density of thinning. The larger this value, the sparser the point cloud, the lower the calculation accuracy, and the faster the speed. When calculating large point cloud data, the parameter size can be appropriately increased to achieve faster calculation speed.

The point cloud category can be selected either from the dropdown menu or by specifying the category via the [Select] button on the right for calculation purposes.

The prescribed range can be left unspecified or specified. The specified method is to read range points from files in formats such as .shp, .dxf, .txt, .csv, or to select ranges from views. Clicking on 'Calculate' and waiting will bring up the calculation results. After the calculation is completed, the results can be exported in PDF, HTML, and DOC formats.

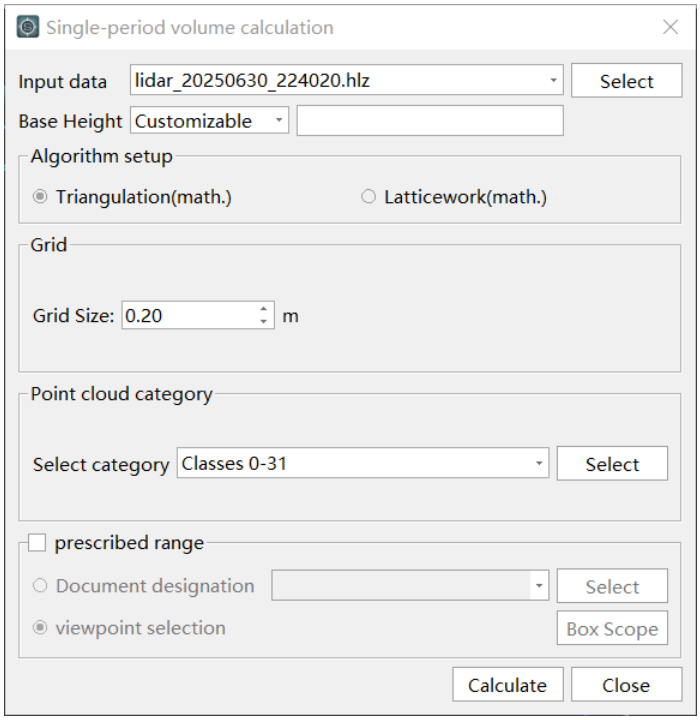

Figure 3-226 Single-period volume calculation

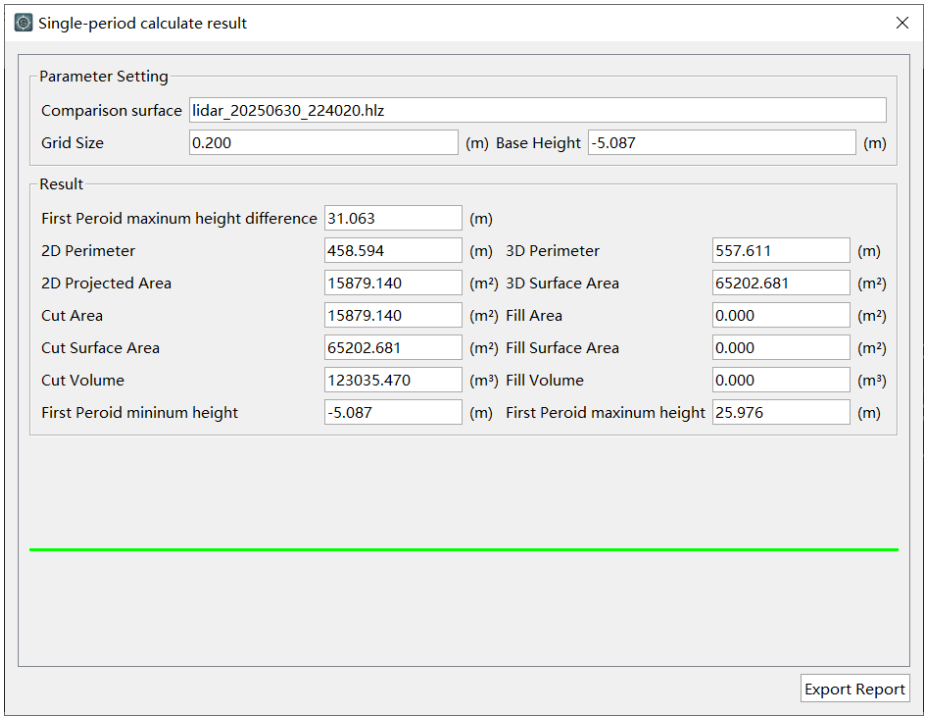

Figure 3-227 Earthwork calculation results

3.13.10.2 Multi-Phase Comparison

Multi-phase earthwork comparison is shown in the figure. Multi-phase earthwork comparison should include at least two phases of data for calculation. The "+" on the right side of the list can add point clouds to the list. By default, the data behind the list refers to the data in the previous list, and their reference logic can be adjusted by using the up and down arrows. The "-" can remove the selected data. The algorithm settings are the same as those for single period data, and the grid size is set to the size of thinning, with a default value of 0.2m. The specified range is also consistent with single period calculations. Clicking on 'Calculate' and waiting will bring up the calculation results. After calculation, the calculation results can be exported in formats such as PDF, HTML, CSV, XLS, and TXT.

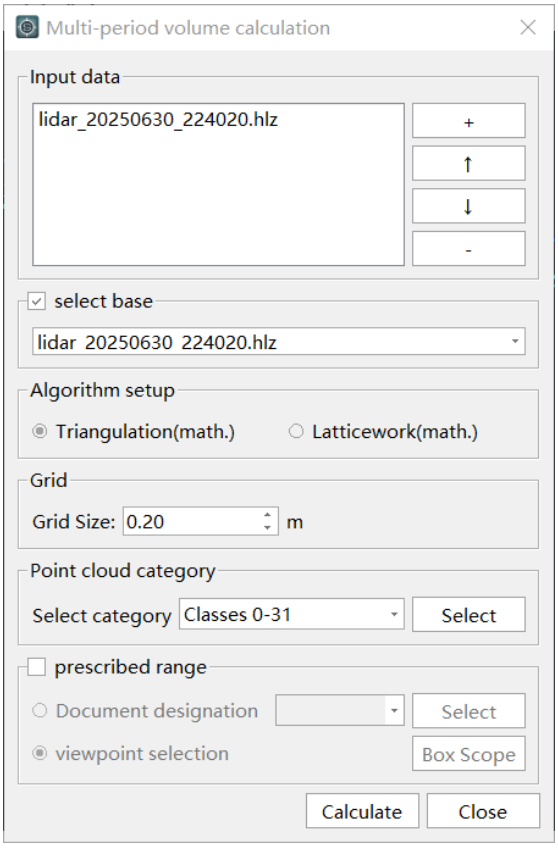

Figure 3-228 Multi-phase earthwork comparison

3.14 3DGS

3DGS modeling requires special attention to minimum computer configuration requirements: an NVIDIA graphics card (20, 30, or 40 series) with at least 8GB of VRAM and at least 32GB of RAM.Assuming the host hardware meets these minimum requirements, install the 3DGS module, open the SLAM project data, select the SLAM task, and on the main interface, click the menu <3D Gaussian modeling > → <Modeling>. In the pop-up dialog box, select the scene type "Large Scene" or "Small Scene," and click <Generate Model> to begin 3DGS modeling.Note: When using this module, after installing the Hi-Lidar version, the Gaussian modeling module must also be installed, and the person registering must authorize the Gaussian modeling module during registration.

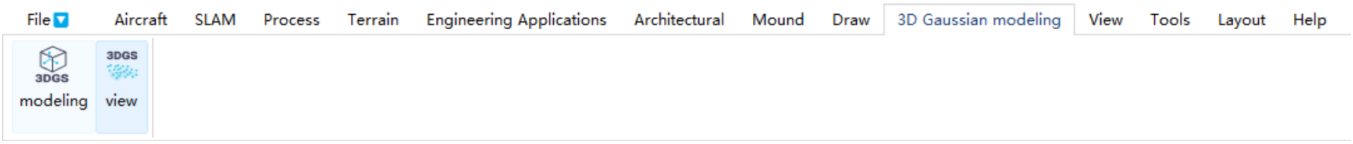

Figure 3-229 3D Gaussian Modeling Menu

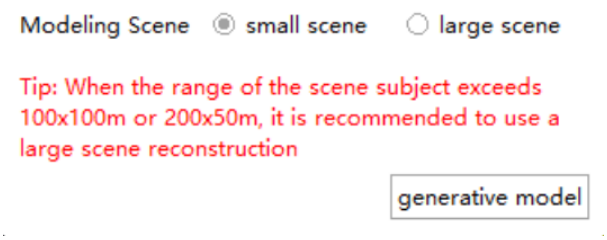

Figure 3-230 3D Gaussian Modeling - Scene Type Selection

After completing the 3D Gaussian model, click the menu <3D Gaussian Modeling> → <Browse> to enter the 3D Gaussian browsing interface, which supports browsing, rotating, scaling, and translating the 3D Gaussian model.

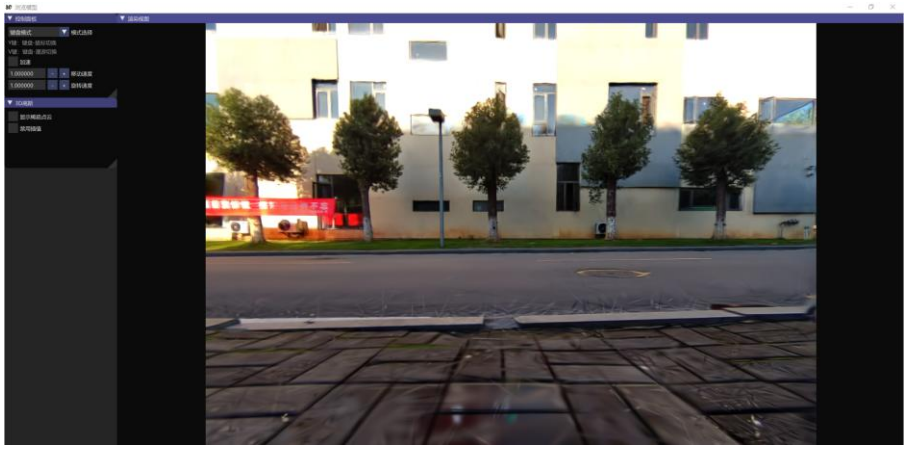

Figure 3-231 3D Gaussian Browser Interface

Headquarter:

Stora Åvägen 21,
436 34 ASKIM, Sweden
info@satlabgeo.com

Regional Offices:

Warsaw, Poland
Jičín, Czech Republic
Ankara, Turkey
Scottsdale, USA
Singapore
Hong Kong
Dubai, UAE

www.satlabgeo.com