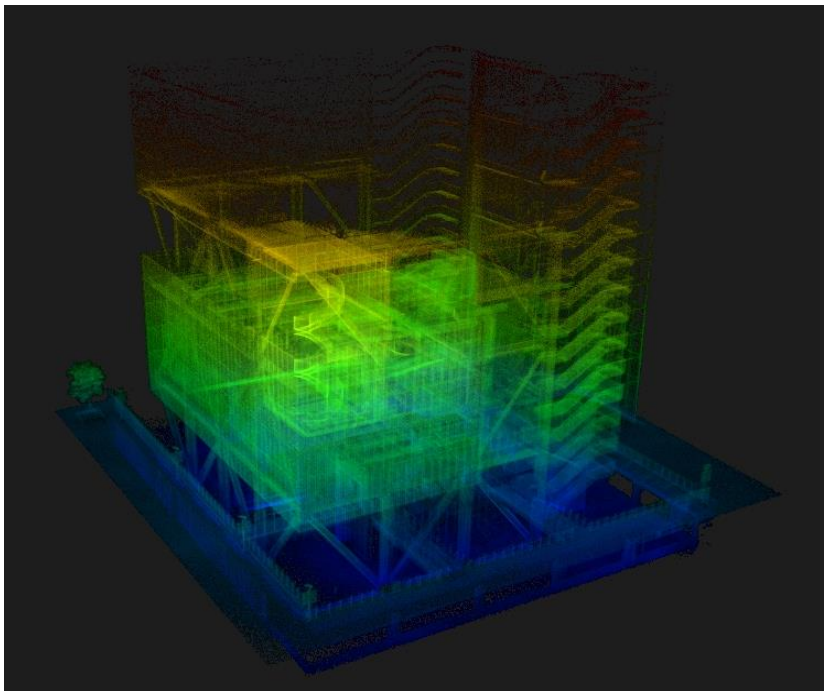
FJDTrion™



# FJD Trion Model

## Point Cloud Processing Software

# User Manual

July 2025 | Rev.205

**Copyright Notice:**



**Read Before Use:**

| | |
|---|---|
|  | **Operate in strict accordance with this software user manual.**<br>If you have any questions during use, contact the service personnel. |

**Disclaimer:**

- The purchased products, services, and features are stipulated by the contract. All or part of the products, services, and features described in this manual may not be within the scope of your purchase or usage. Unless otherwise specified in the contract, all the content in this manual is provided "AS IS" without warranties of any kind, express or implied.
- The content of this manual is subject to change due to product upgrades and other reasons. FJDynamics reserves the right to modify the content of this manual without notice.
- This manual only provides guidance for use of this product. Every effort has been made in the preparation of this manual to ensure accuracy of the content, but no information in this manual constitutes a warranty of any kind, express or implied.

目录

# 1 Overview

FJD Trion Model is point cloud processing software developed by FJ Dynamics Co., Ltd. (FJDynamics), and can be used to visualize and process point cloud data obtained by a laser scanner. It supports basic point cloud data processing, such as point cloud mapping, color assignment, denoising, point cloud visualization, etc., and adapts to the industry's common point cloud format; at the same time, it also supports point cloud-based industry applications, such as two-dimensional vector line drawing, volume measurement, and forestry data computation, etc., which provides a one-stop solution for the industries of construction, forestry, mining, and public safety.

To use FJD Trion Model correctly and efficiently, read this manual before use, so that you can learn its modules, functions, features, and basic operation procedures. All features in this manual are demonstrated in Windows 11. The operations in other operating systems are similar, and are not described again.

# 2 FJD Trion Model

## 2.1 Configuration Requirements

FJD Trion Model must be run in the following environment:

**1. Hardware:**

**Recommended configuration:**

- Processor: Intel® Core™ i7-10700 CPU @ 2.90 GHz (or an AMD CPU with equal performance)
- Memory: 32 GB
- Graphics card: NVIDIA GeForce RTX 4080
- Hard disk drive: 1 TB
- Screen resolution: 1920 × 1080 or above

**Minimum configuration:**

- Processor: Intel® Core ™ i5-2300 CPU @ 3.10 GHz (or an AMD CPU with equal performance) or above
- Memory: 16 GB
- Graphics card: NVIDIA GeForce RTX 3060
- Hard disk drive: 512 GB
- Screen resolution: 1920 × 1080 or above

**2. Reality Modeling Configuration:**

**Recommended configuration:**

- Processor: Intel® Core™ i7-11700 or AMD Ryzen 7 5700G or higher
- Memory: 64GB
- Graphics card: NVIDIA GeForce RTX4080
- Hard disk drive: 1TB (preferably SSD)
- Screen resolution: 1920x1080

**Minimum configuration:**

- Processor: Intel® Core™ i5-11500 or AMD Ryzen 5 3600 or higher
- Memory: 32GB
- Graphics card: NVIDIA GeForce RTX 2060 SUPER or higher (8GB or higher VRAM); AMD graphics cards are not supported.
- Hard disk drive: 512GB (preferably SSD)
- Screen resolution: 1920x1080

**3. Software:**

- Operating system: Windows 8 or above (only Windows X64 system, Windows 11 system is recommended)

## 2.2 Download

Download the FJD Trion Model package at:
https://www.fjdynamics.com/product/fjd-trion-model

## 2.3 Installation

1. Decompress the FJD Trion Model package, right-click  , and select **Run as administrator.**

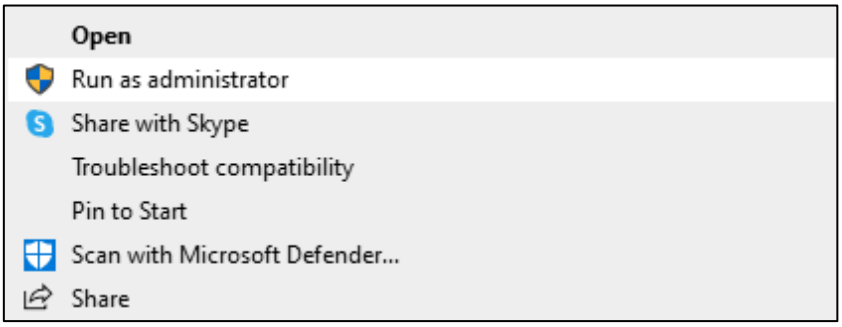



**Figure 1.** Running the program

2. Select the setup language from the ten options including"简体中文","English", "繁體中文", "Español", "CIS", "Italiano", "Français", "Português", "Deutsch", and "日本語", and click **OK**.

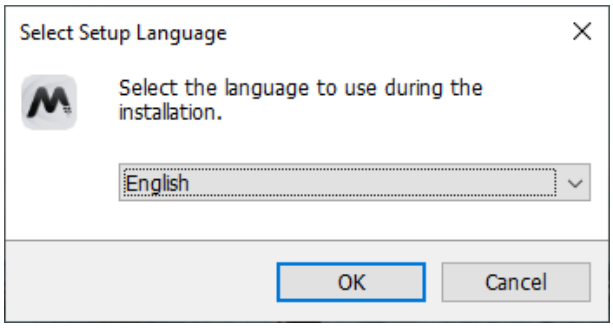



**Figure 2.** Selecting the setup language

3. Select **I accept the agreement** to accept the **License Agreement**, and then click **Next**; otherwise, the installation will be exited.

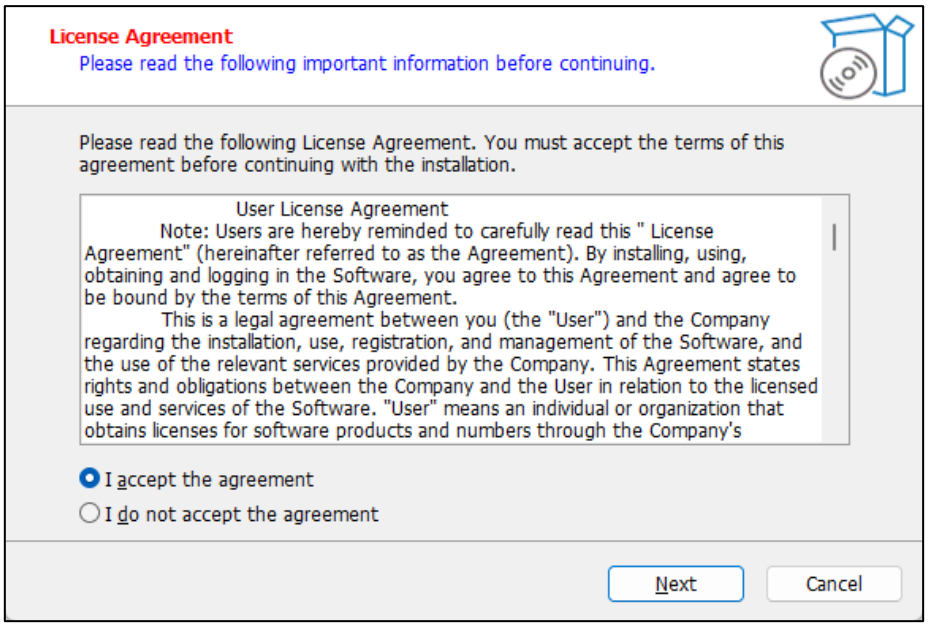



**Figure 3.** Accepting the license agreement

4. Select the target installation location, and then click **Next**. You are recommended to install FJD Trion Model in a drive (D drive recommended) other than C drive. Otherwise, a running exception may occur.

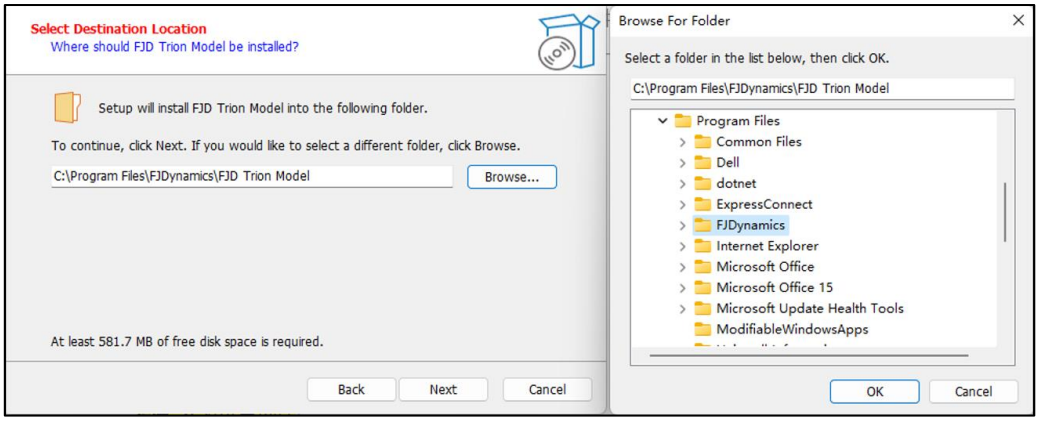



**Figure 4.** Selecting the installation location

5. Click **Install**. After the installation is completed, the window disappears, and a desktop shortcut  of FJD Trion Model is generated.

## 2.4 Update

FJD Trion Model only supports package updates currently. You can download the latest package from FJDynamics' official website (https://www.fjdynamics.com).

## 2.5 Uninstallation

For example, in Windows 11, choose **Start** > **All apps**, find FJD Trion Model in the list, right-click it, and select **Uninstall**.


**Figure 5.** Uninstalling FJD Trion Model

You can also go to the software installation directory and click  to uninstall it.

## 2.6 Activation

FJD Trion Model is a LiDAR point cloud processing software developed by FJDynamics, and can be used only with our authorization.

When purchasing the software, you can apply for an activation code based on your actual needs. The following figures show the license setting window and successful activation window.

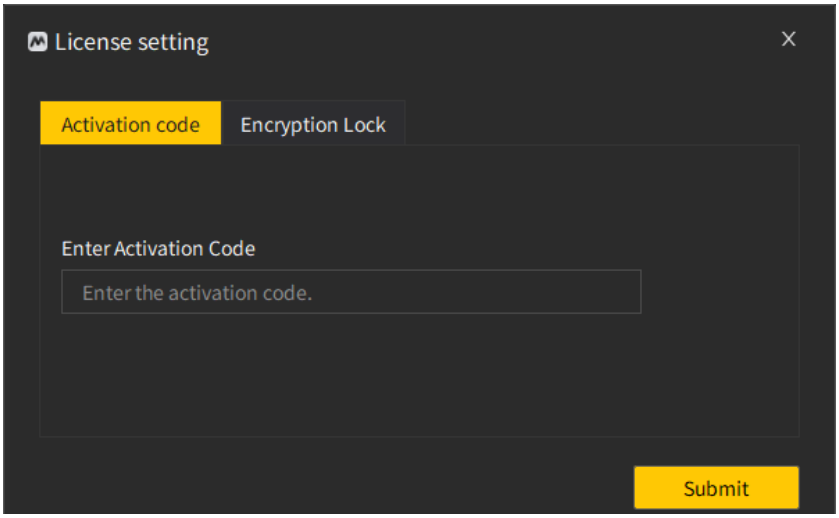



**Figure 6.** License setting

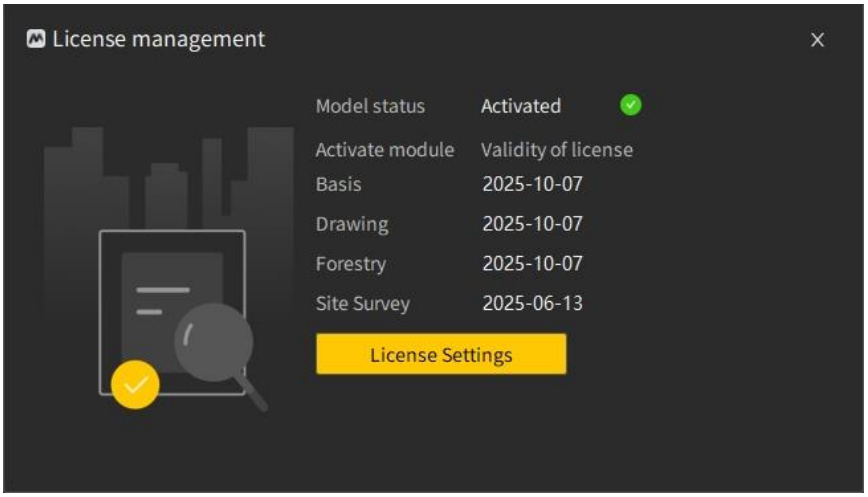



**Figure 7.** Successful activation by activation code

**Note:**

When the license expires, contact Dynamics' technical support.

## 2.7 Language

FJD Trion Model currently supports ten languages including Simplified Chinese, English, Traditional Chinese, Japanese, French, Italian, Português, German and Spanish, etc. Users can switch according to their needs. Click **Help**, and in the drop-down menu, click **Language** to select the appropriate language.

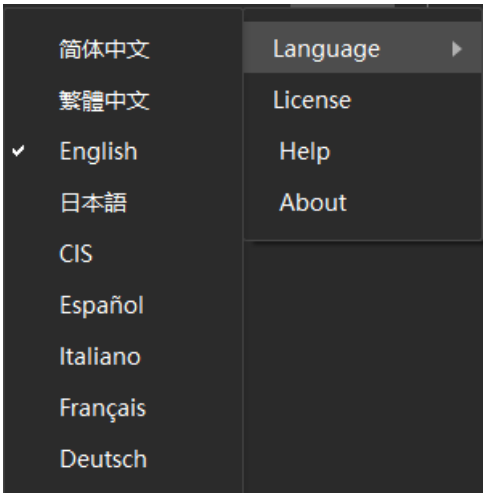



**Figure 8.** Selecting the language

# 2.8 Manual

The user manual is embedded in FJD Trion Model, and you can search this manual for specific operations.

Click **Help**, and select **Manual** in the drop-down list.

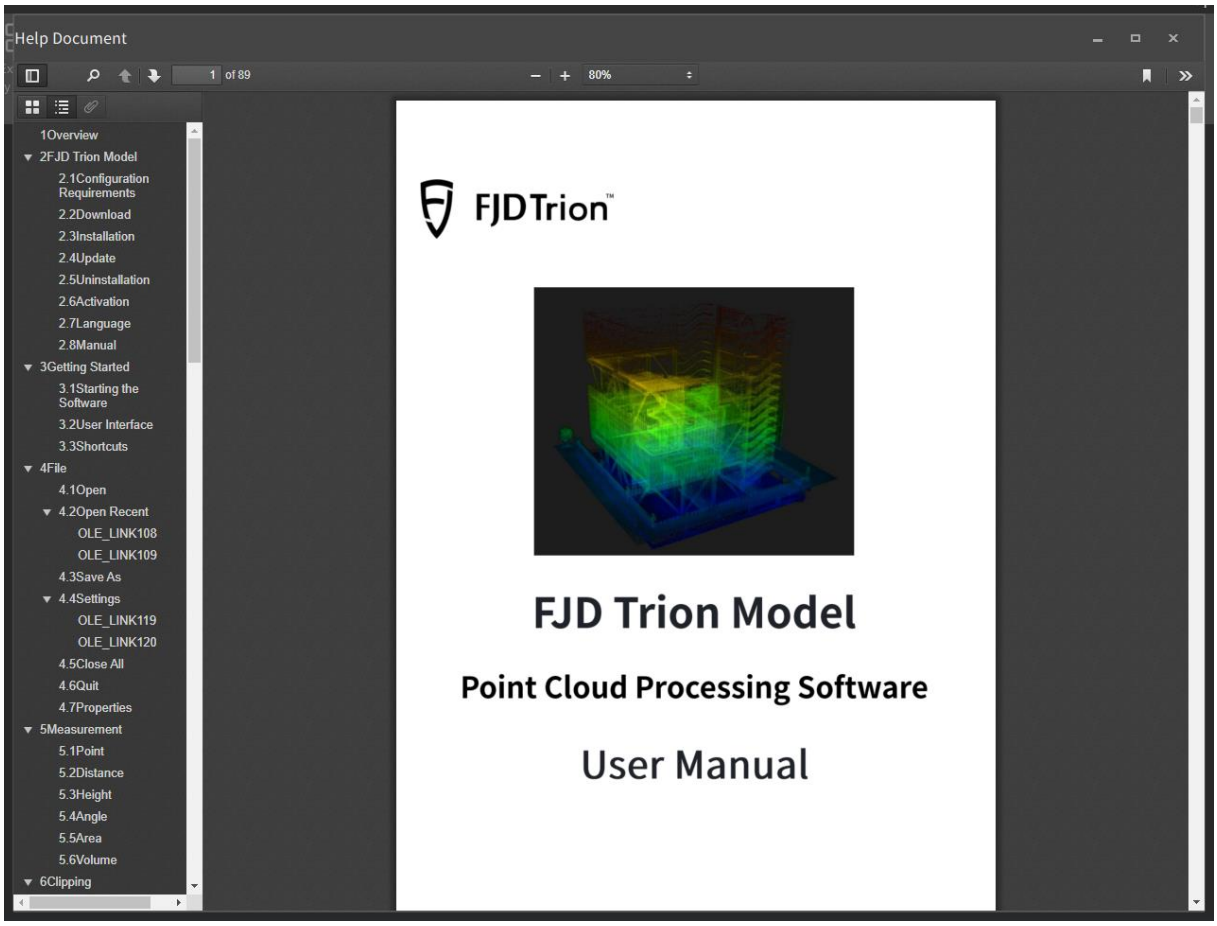



**Figure 9.** Manual

# 2.9 About

Click **About** to check the software version and perform online updates.

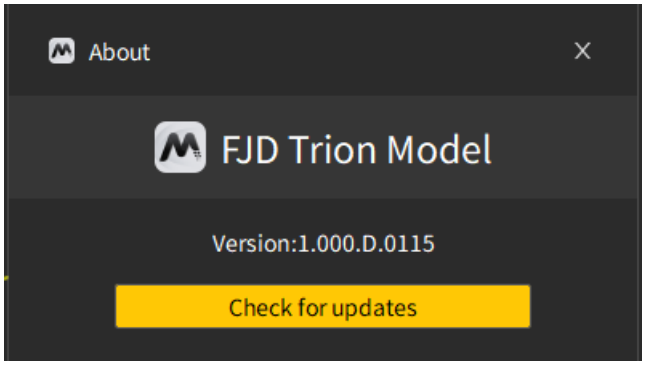



**Figure 10.** Software update

# 2.10 Share

**Description:**

Uploading point cloud data or roaming video to the cloud platform, you can view the point cloud or video remotely by clicking this link.

**Operation procedure:**

1. Select a point cloud file or video, click the **Share** button, and a pop-up window for uploading to the cloud will appear on the screen;

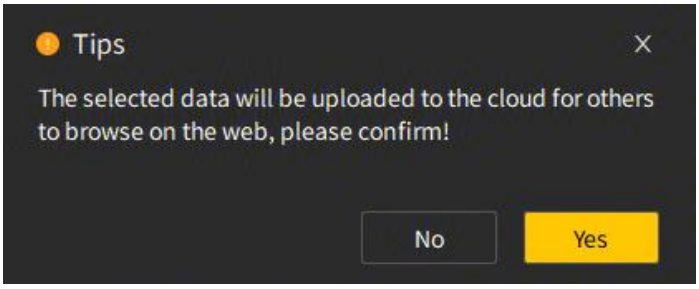



**Figure 11.** Share

2. Click **Yes** the data will be uploaded; During the data uploading process, click “Minimize” to upload data in the background without affecting the use of other functions; click **Cancel** to cancel the data sharing;

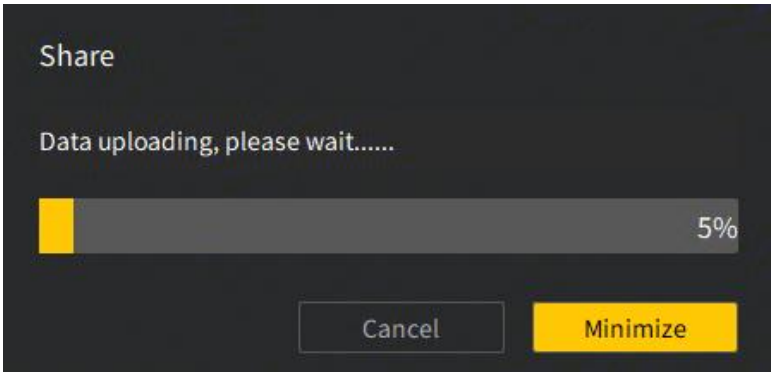



**Figure 12.** Upload

3. After the upload is successful, the user can copy the link to the clipboard and open it in a browser to view the data.

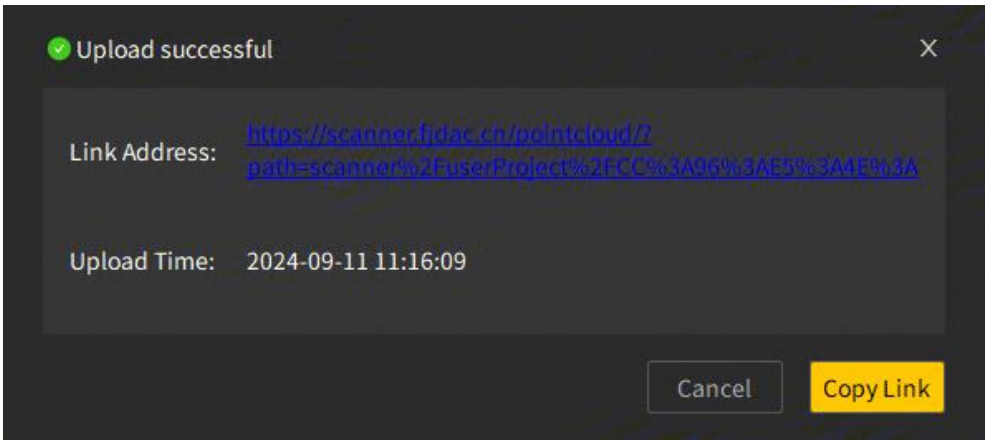



**Figure 13.** Share link

4. When you select the data that has been successfully shared and want to share it again, the sharing link will pop up directly. At this time, you can update or copy the link directly to share.

# 3 Getting Started

## 3.1 Starting the Software

Double-click the FJD Trion Model icon with the left mouse button to start the program. In the startup interface, choose "**Open Project**" or "**New Project**" to enter the software, or directly double-click to select a recently opened project file to enter.

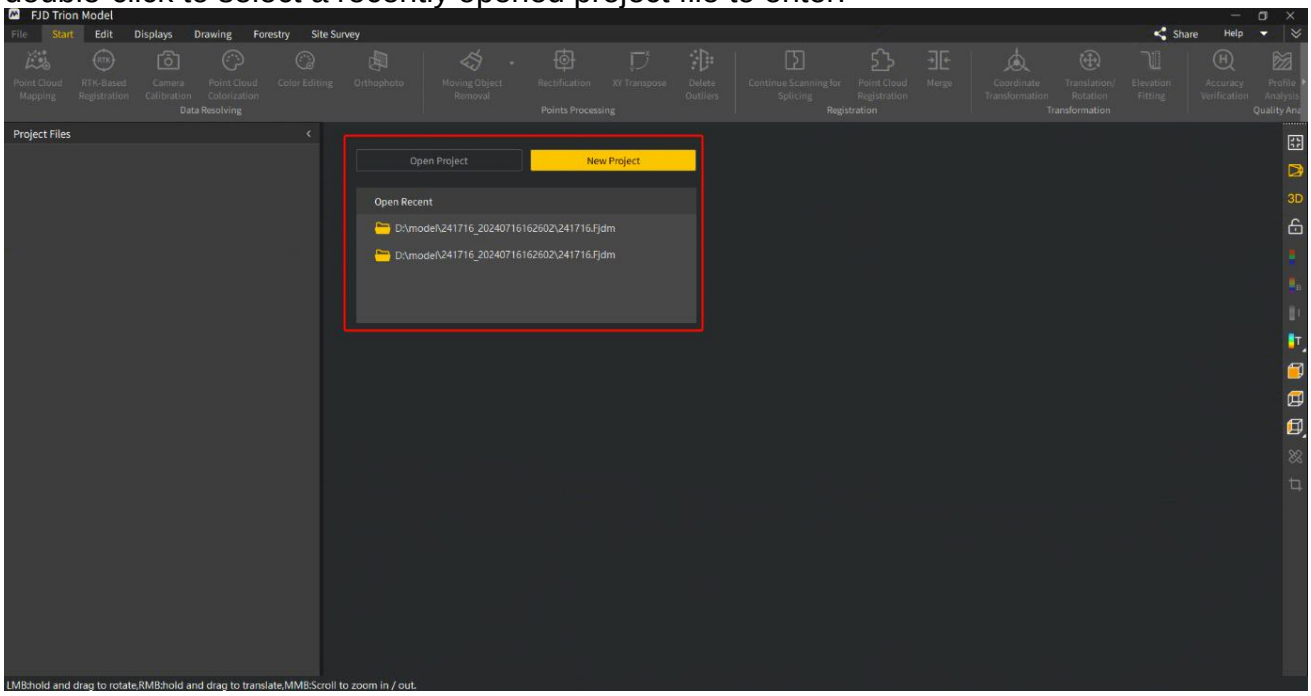



**Figure 14.** Launch Interface

### 3.1.1 New Project

Input the project name and select the save path to create a new project.

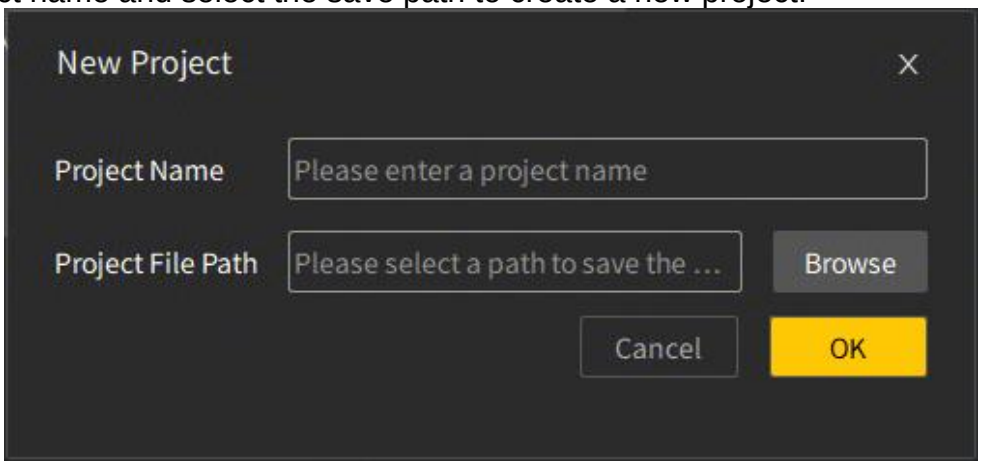



**Figure 15.** New Project

### 3.1.2 Open Project

Open the folder and select the created project file “**Fjdm**”, customers can continue to process the data in this project file.

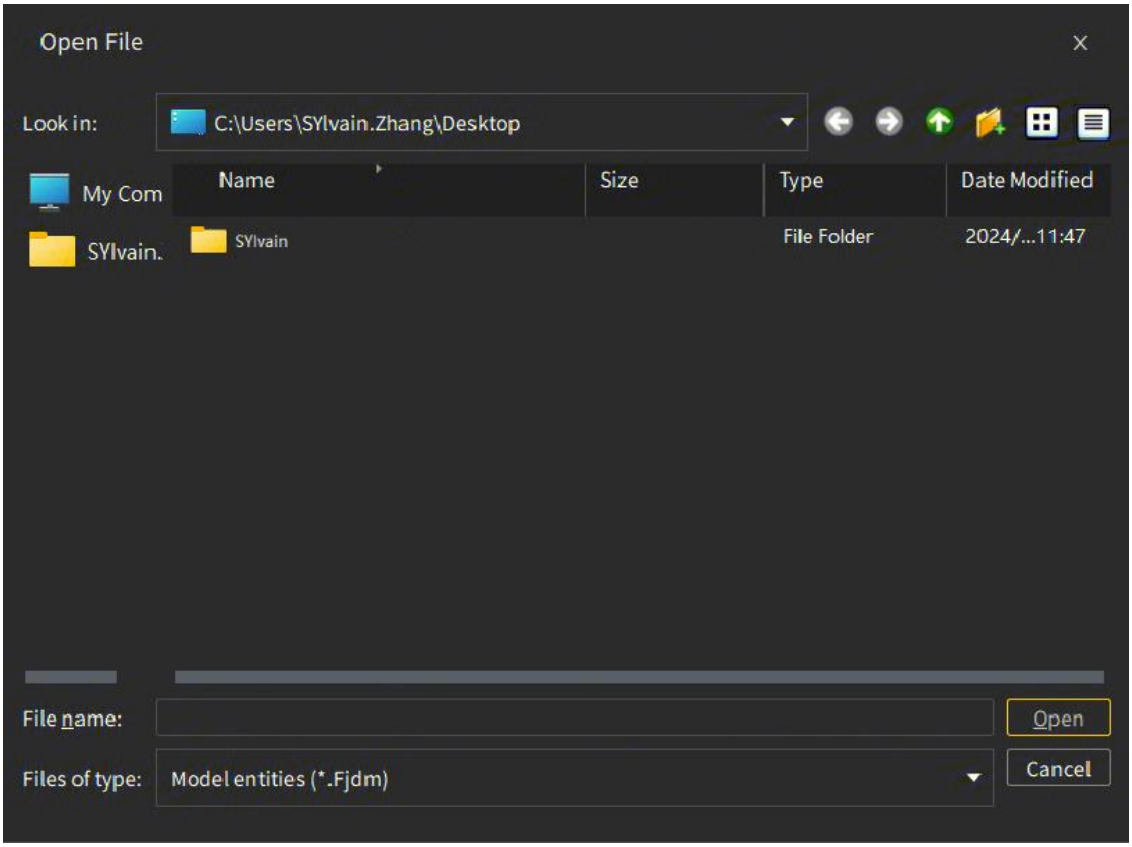



**Figure 16.** Open Project

Note: After opening or creating a new project, you can perform data processing in the project.

## 3.2 User Interface

The layout of the software interface is shown below:

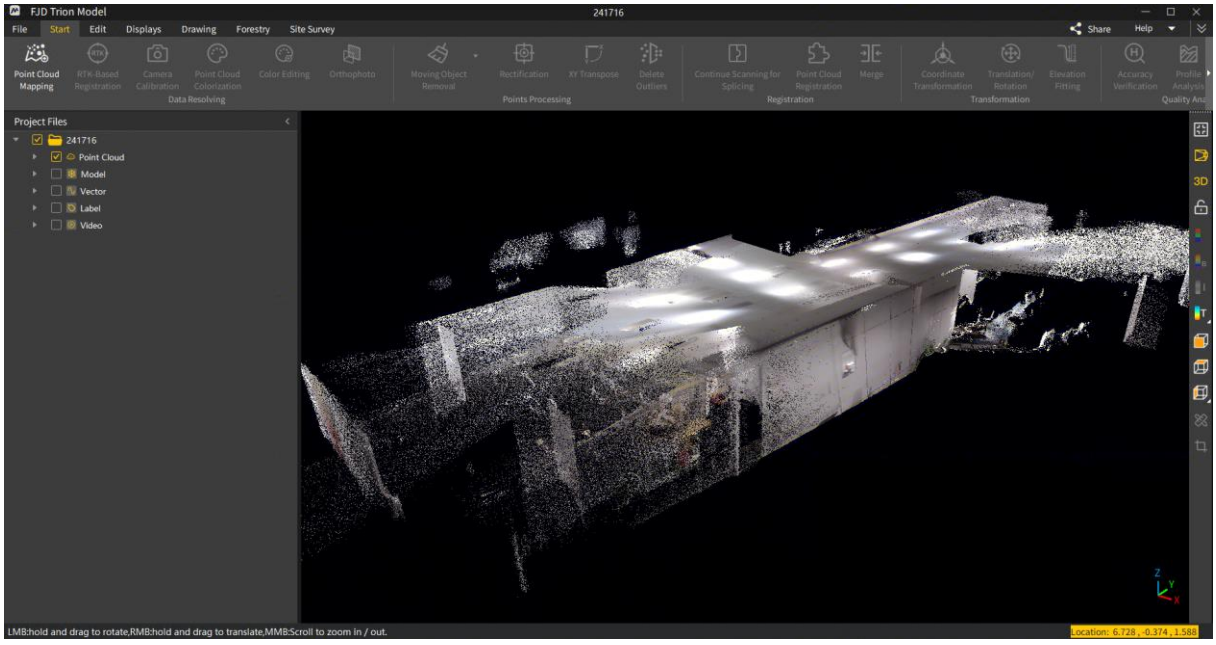



**Figure 17.** Software interface

The following components are included:

| | |
|---|---|
| Menu bar | Software Feature Selection |
| Share/Help | Share Data/User Help |
| Toolbar | Toggle View Display Style/Cutting/Measuring |
| Project file | Project Data Management |
| View Display | Display point cloud, model, video and other data |
| Point cloud information | Display point cloud position, point number and offset value |
| Operation Tips | Tips on how to operate the mouse in the 3D view area |

# 3.3 Help

Refer to 2.6 to 2.10.

# 3.4 Toolbar

In the point cloud general function area, users can drag and place the toolbar in a custom position to display it. On the toolbar, you can perform operations such as restoring views, rendering mode display, measurement, and clipping. The toolbar interface layout can be freely dragged to adapt to user operations.

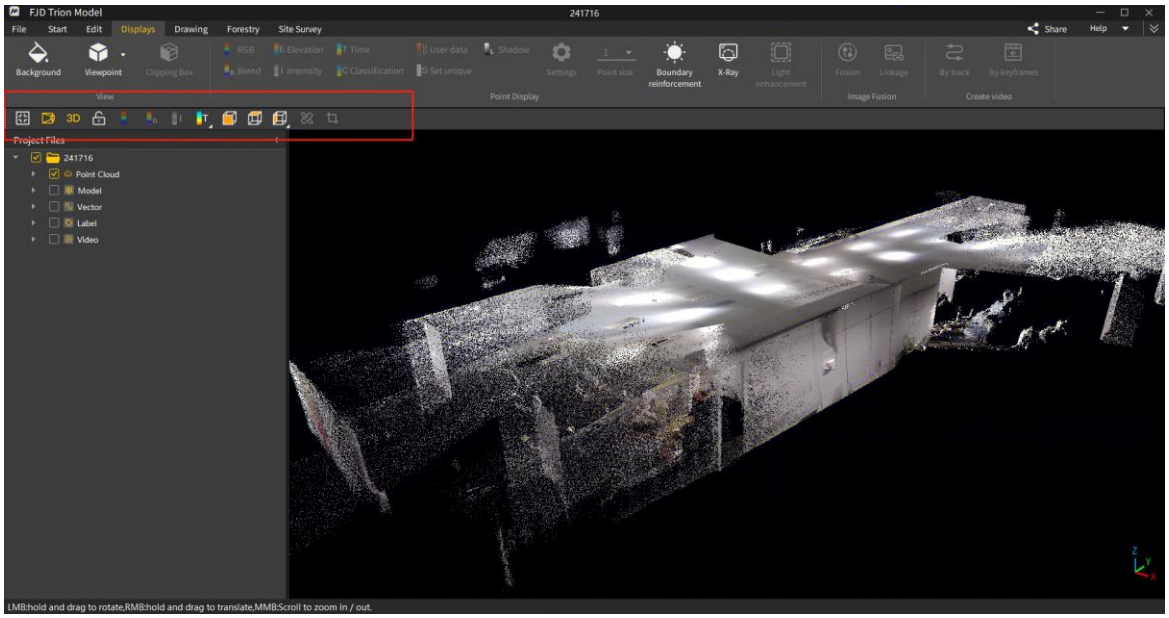



**Figure 18.** Reorganization of the layout

## 3.4.1 Restore the original scale and position

Click the toolbar  icon to restore the view to its initial scale and position for easy viewing.

## 3.4.2 Perspective Projection

By clicking  or  icon in the toolbar, the point cloud and model in the view will be displayed as orthographic projection or perspective projection; In perspective projection mode, objects appear larger when they are closer and smaller when they are farther away, which is more in line with human eye's observation of the real world.

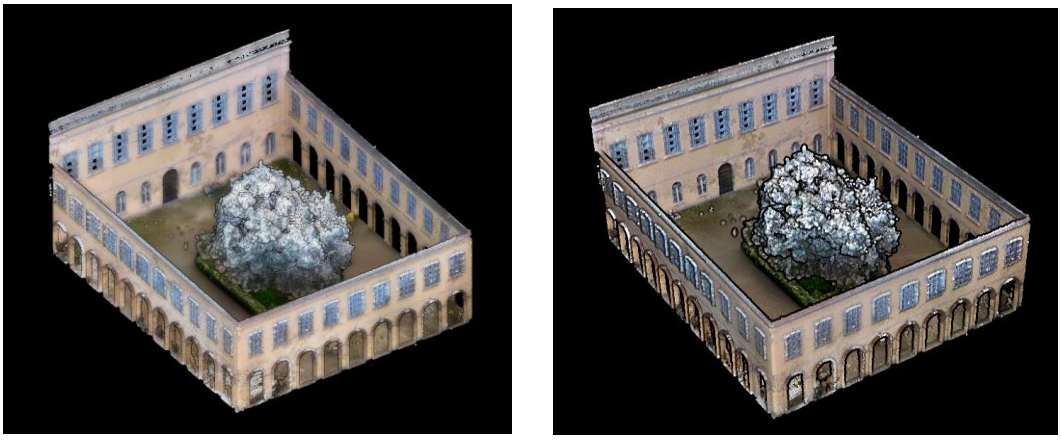


**Figure 19.** Left: front view, right: perspective

### 3.4.3 2D/3D View

Click 2D or 3D icon in the toolbar to switch the point cloud and model display in the view to 2D or 3D mode; for 2D view, the data is displayed in two-dimensional plane; for 3D view, the data is displayed in three-dimensional space. In general, you can choose to operate the data in 3D mode.

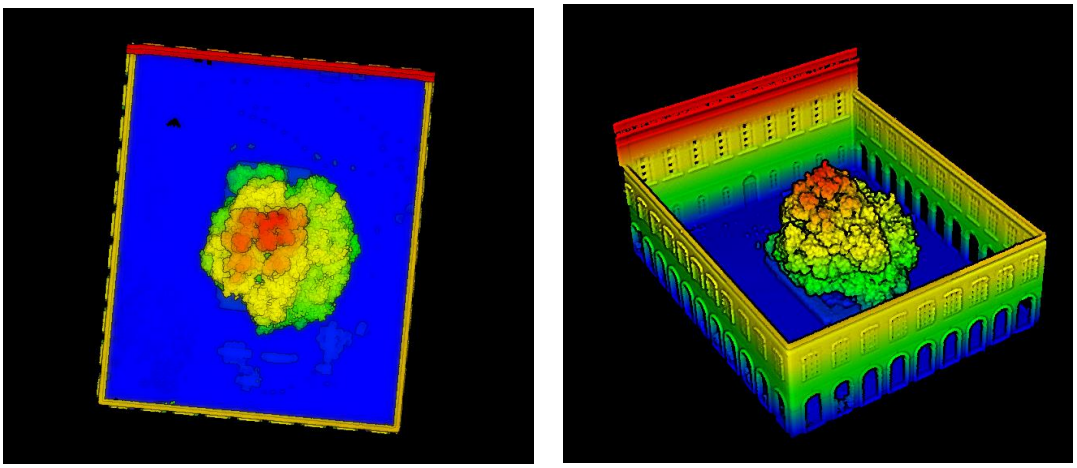


**Figure 20.** Left: 2D, right: 3D

### 3.4.4 Lock the View

After clicking **Lock the view**, the view is fixed and cannot be operated to rotate or pan; clicking **Unlock View** restores the user operation of the view.

### 3.4.5 Point Cloud Coloring

By clicking the point cloud coloring tool in the toolbar, you can quickly switch the way of point cloud display, refer to 7.2 for the specific way.

### 3.4.6 Viewpoint Switching

By clicking **Set front view**, **Set top view** and **Set right view** in the toolbar, you can quickly switch the 3D display view angle, refer to 7.1.2 for details.

### 3.4.7 Measurement Tools

Measurements are performed on point cloud or triangulation data to obtain the data required by the user. Measurement mainly includes: **Point Measurement**, **Distance measurement**, **Height measurement**, **Angle measurement** and **Area measurement**. When you select a point cloud or triangulation network in the file list, left click on the measurement icon in the toolbar to bring up the measurement toolbar. During the measurement process, you can click to undo the current operation, click to finish the measurement and save the result, synchronize the measurement result label in the project label file, and click to exit the measurement.

## I. Point measurement

**Description:**

The **Point Measurement** initial state is set to single-point measurement by default, which can be applied to point cloud, raster, and model data. By performing a single-point measurement on the data, you can interactively query the attribute information of a single point within the dataset.

**Operation procedure:**

1. Left mouse click in the project file to select the point cloud or raster data that has been opened;

2. Click the measurement function icon to activate the measurement toolbar;

3. Click **Point Measurement** icon, click the point on the point cloud model, pop-up as shown in the information tab, displaying the relative coordinates of a point as well as geographic coordinates of the information, according to the current point cloud rendering method to display the point of time, intensity, and other information.

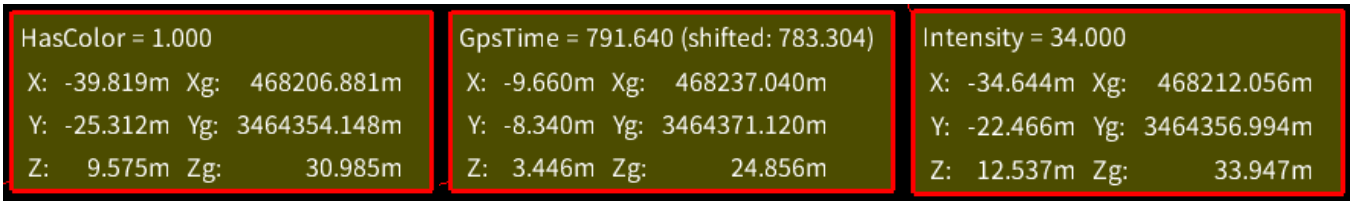



**Figure 21.** Single Point Measurement Results

## II. Distance measurement

**Description:**

The **Distance Measurement Tool** can be used with point cloud, raster, and model data. The **Distance Measurement Tool** uses left mouse clicks to interactively query distance information between multiple points. For 2D data, the measurement results represent flat distances, and for 3D data (e.g., point cloud and model data), the measurement results represent the distances between points in three-dimensional space.

**Operation procedure:**

1. Left mouse click in the project file to select the opened point cloud or raster data;

2. Click on the measurement function icon to activate the measurement toolbar;

3. Click on the **Distance Measurement** icon, left click on the scene of the two effective point, pop-up distance results as shown in the figure, showing the distance information between the two selected points. The following figure shows the results of distance measurement on point cloud data by taking two points and multiple points respectively.


**Figure 22.** Single-segment distance measurement

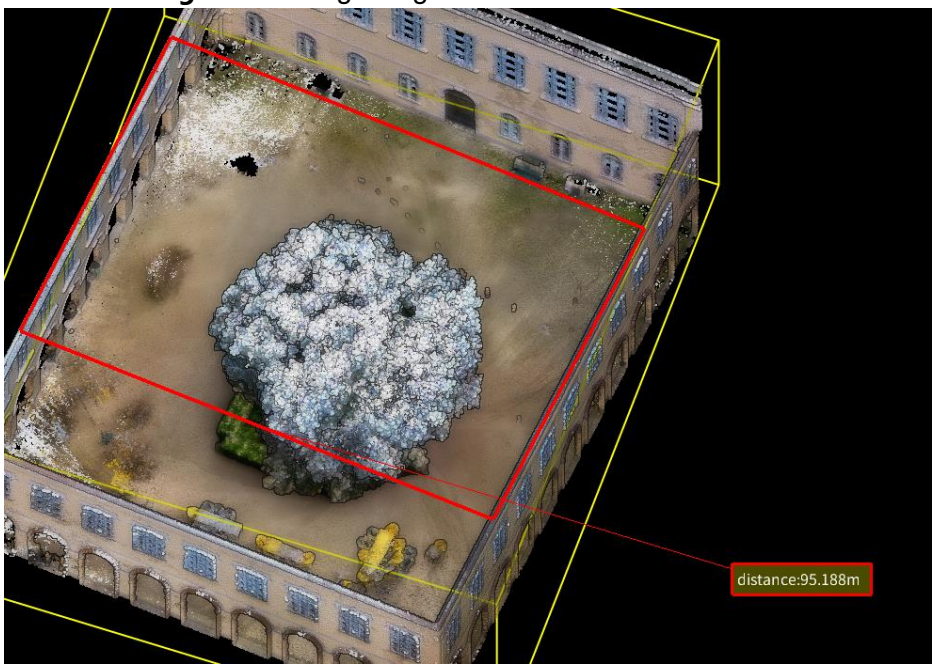



**Figure 23.** Multi-segment distance measurements

## III. Height measurement

**Description:**

The **Height Measurement Tool** works on both point cloud and model data. The height measurement tool interactively selects a measurement point by left mouse click to query the relative height between two points.

**Operation procedure:**

1. Left mouse click in the project file to select the point cloud or mesh data that has been opened;
2. Click on the measurement function icon to activate the measurement toolbar;
3. After clicking the **Height Measurement** icon, left mouse click to select two valid points in the scene and use them as two endpoints for height measurement;
4. After completing the selection of the two endpoints, the display area will draw the height display style between the two points in real time, and display the measurement results in real time in the form of labels.

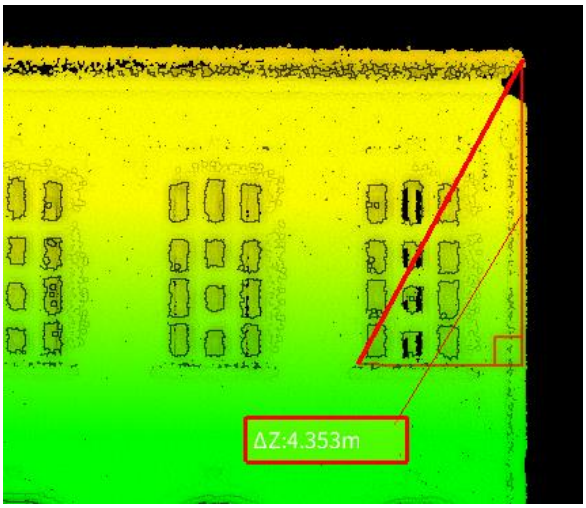



**Figure 24.** Height measurement result

## IV. Angle measurement

**Description:**

The **Angle Measurement Tool** works on point cloud, raster, and model data. Angle Measurement Tool uses the left mouse button to interactively select measurement points and query the angle between three points.

**Operation procedure:**

1. Left mouse click in the project file to select the opened point cloud or raster data;
2. Click on the measurement function icon to activate the measurement toolbar;
3. After clicking the Angle Measurement icon, when the Angle Measurement tool is active, left mouse click on the triangular mesh or point cloud data to select a point as the first point for angle measurement;
4. Left mouse click to select a point as the second point for angle measurement;
5. Double-click the left mouse button to select the angle measurement of the third point, the end of the measurement, the scene in real time to draw the measurement of the angle, and in the form of labels in real time to show the results of the measurement, as shown in the figure.

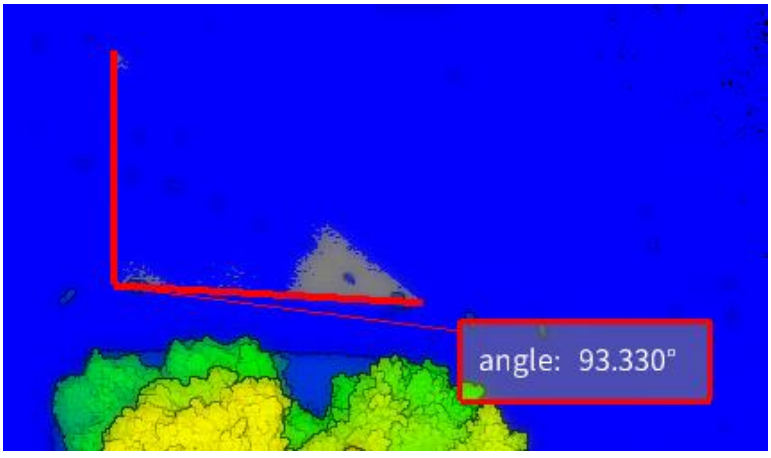



**Figure 25.** Angle measurement result

## V. Density measurement

**Description:**

The **Density Measurement Tool** is used to obtain the number of laser points per unit area by analyzing density, users can evaluate the spatial distribution and concentration of point cloud data, thereby assessing device performance and suitability for different application scenarios.

**Operation procedure:**

1. Left-click on the point cloud dataset in the project file to select the data to be measured.

2. Click on the measurement function icon to activate the measurement toolbar.

3. After clicking the density measurement icon, the function window will open. It supports two selection modes: square and polygon.

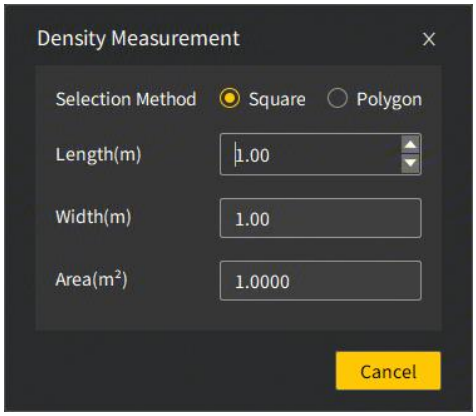



**Figure 26.** Density measurement function window

4. Once the tool is activated, the cursor will change from an arrow to a hand-shaped picker. Left-clicking on the point cloud will calculate and display the point density within the selected area.


**Figure 27.** Density measurement result

5. Click "Cancel" or "×" in the upper right corner of the window to exit the density measurement function. Clicking in non-point cloud areas of the view will not perform density measurement.

## VI. Area measurement

**Description:**

The **Area Measurement Tool** can be used for point cloud data and triangle mesh data. The area

measurement tool uses the left mouse button click to interactively draw triangles, and can query the area of the triangle region.

**Operation procedure:**

1. Left mouse click in the project file to select the opened point cloud or mesh data;

2. Click on the measurement function icon to activate the measurement toolbar;

3. After clicking the area measurement icon, left-click on the triangle mesh or point cloud data to select three points to complete the measurement, the following figure is the result of area measurement on the point cloud data.

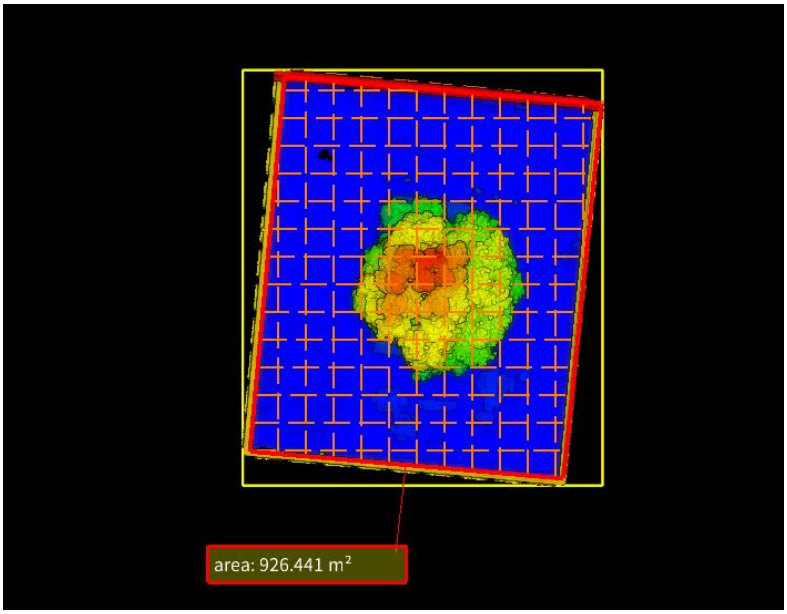



**Figure 28.** Area measurement result

### 3.4.8 Clipping

Regionally select point cloud data for retention or deletion.; left click on the project file to select the point cloud data, click **Clipping** in the toolbar to bring up the crop toolbar, the selection methods include circle, specified rectangle, custom rectangle and polygon; after completing the region selection, click to keep the point cloud data within the cropping range, and click to keep the point cloud data outside the cropping range. In case of cropping problem due to wrong data selection, you can click Undo icon to restore the initial state so as to crop again, click Clip icon to delete the cropped point cloud and keep only the selected point cloud, and click Split icon to split the cropped point cloud and save the cropped and retained point clouds into two point cloud data in the file tree, click Close icon to exit the clipping operation.

#### I. Specified circle

**Description:**

Quickly select the circular area for cutting operation.

**Operation procedure:**

1. Click the icon of the specified circle cutting function in the cutting toolbar;

2. Enter the cutting radius, left mouse click on the center of the circle, the scene will be drawn according to the radius value of the circular area;
3. After completing the selection of the cutting area, according to the project requirements, choose to cut the internal or external point cloud, to retain the target area;

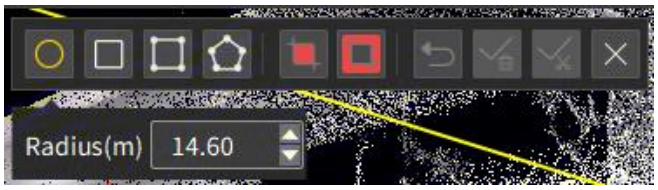



**Figure 29.** Specified circle radius

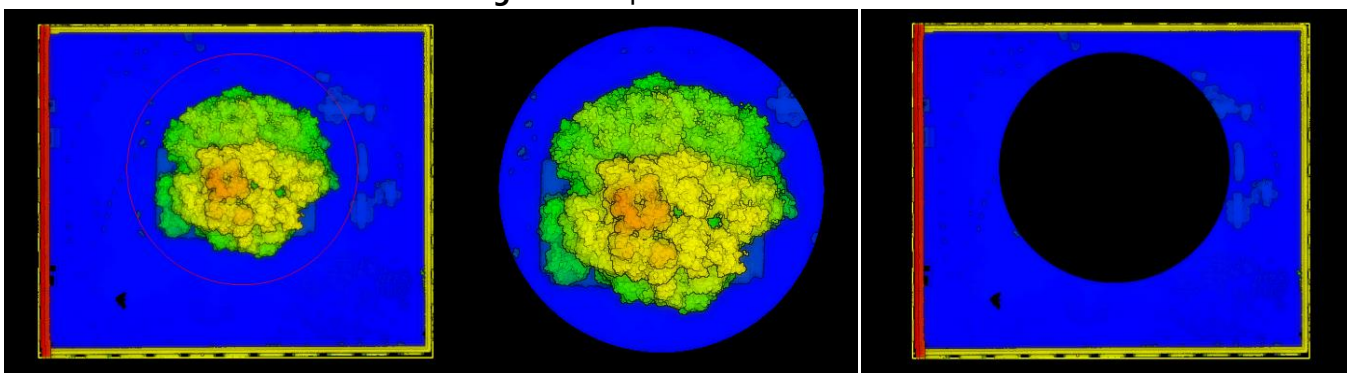


**Figure 30.** From left to right: area selection, point cloud inside the cut range, outside the cut range

4. Finish clipping the point cloud within the range, when cropping and saving, the point cloud outside the region will be deleted, and only one point cloud file will be retained in the file directory tree; when splitting and saving, the original point cloud will be split into two point cloud files; the user can choose the suitable saving method according to the project requirements.

## II. Specified rectangle

**Description:**
Accurately and quickly select the rectangular area for cutting operation according to the set length and width.
**Operation procedure:**

1. Click the specified rectangle cutting icon in the cutting toolbar;

2. Enter the length and width of the rectangle, the left mouse click on the center of the rectangle, the scene will be drawn according to the set length and width of the rectangular area, the user can continuously draw the specified rectangular area; cut and save the operation refer to the above specified circle.


**Figure 31.** Specified rectangle

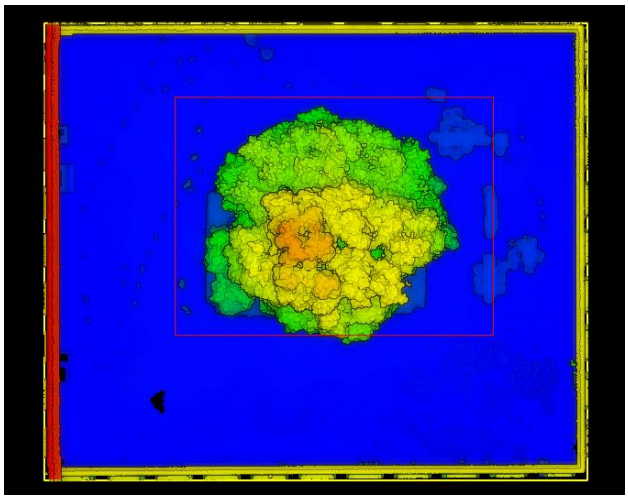


**Figure 32.** Specified rectangular area selection

### III. Rectangle

**Description:**
Customize the selected rectangular area for cropping operation.
**Operation procedure:**

1. Click the icon of rectangle cutting function in the cutting toolbar;

2. In the scene, the left mouse click to determine the first corner of the rectangle, drag the mouse and click again to complete the selection of the rectangular region; the user can continuously draw multiple rectangular regions, cut and save the operation refer to the above specified circle.

### IV. Polygon

**Description:**
Customize the drawing of polygonal areas for cropping operations.
**Operation procedure:**

1. Click the Polygon Crop icon in the Crop toolbar;

2. In the scene, the left mouse click to determine the first corner of the polygon, drag the mouse and click again to draw the next corner of the mouse, right mouse click to complete the polygon drawing; users can continuously draw multiple polygonal areas, cut and save the operation reference to the above specified circle.

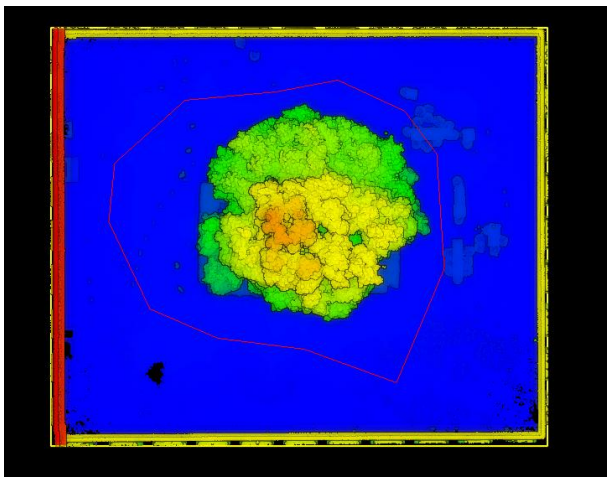


**Figure 33.** Polygon area selection

# 3.5 Project Files

Project data management, users can rename, view information, export and delete point cloud, model, vector, label and video in this interface; you can freely drag the interface layout of the project file or put it away to the left to increase the view display area.

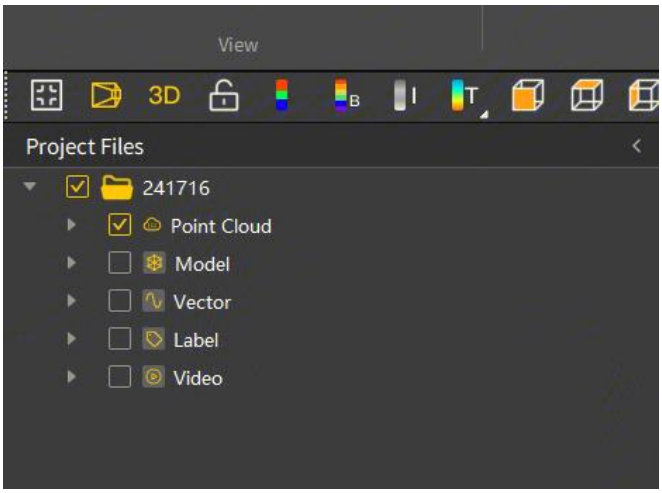



**Figure 34.** Project file interface

Right-clicking on a project file allows you to rename the project file, view the project save path and save the project.

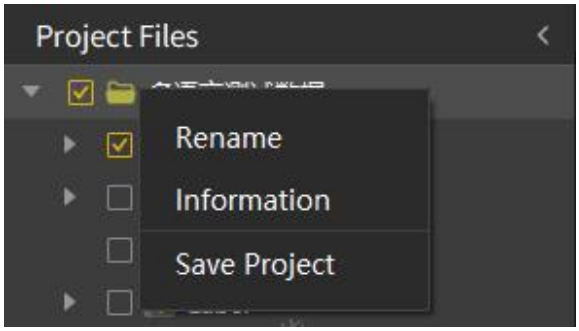



**Figure 35.** Engineering documents

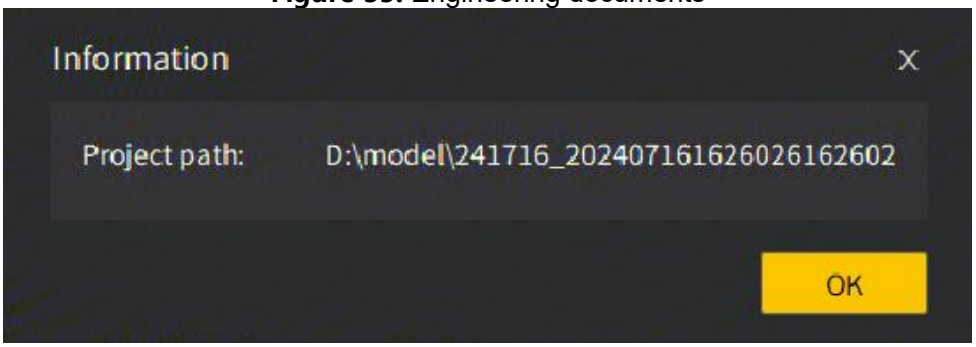



**Figure 36.** Engineering documents information

## 3.5.1 Point Cloud

**Description:**

Display and manage all point cloud files in the project. After the configuration file matched by point cloud is loaded successfully, it will be displayed in green on the right side. If it is loaded successfully but matched incorrectly, it will be displayed in orange. If the point cloud configuration file is not loaded, it will be displayed in red. Select a single point cloud data, right-click to rename the file and view the information. Select "Switch Display" to show or hide point cloud. Depending

on the type of point cloud, the right-click menu displays different content.

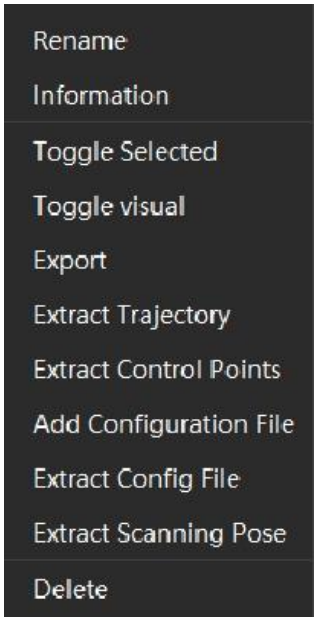



**Figure 37.** Right click on the cloud data toolbar

I Renamed

**Description:**

Modify the name of the point cloud data.

**Operation procedure:**

1. Right-click on the point cloud data and select Rename.
2. Enter a new name and click "enter" or the blank space to save.

II. Information

**Description:**

View the name of the point cloud data.

**Operation procedure:**

1. Right-click on point cloud data and select Info;
2. A pop-up information display window appears, allowing users to view the scanning time, number of points, and scalar field information of point cloud data.

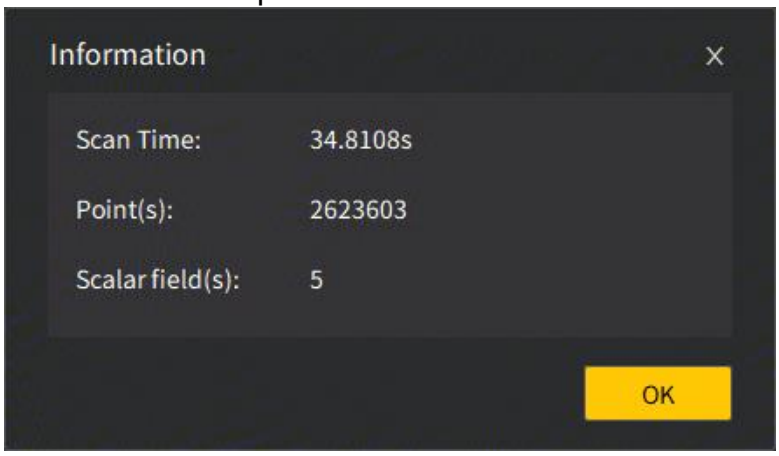



**Figure 38.** Information viewing

I Switch select/ Switch visual

1. Right-click on the point cloud file and click "Switch Selected". The point cloud will switch to the selected state, and the point cloud in the view will also switch.
2. Click "Switch Display" to show/hide point cloud data.

II Export

Select "Export" to save the current point cloud data to a local file, supporting common point cloud formats such as las, e57, ply, pts, etc.

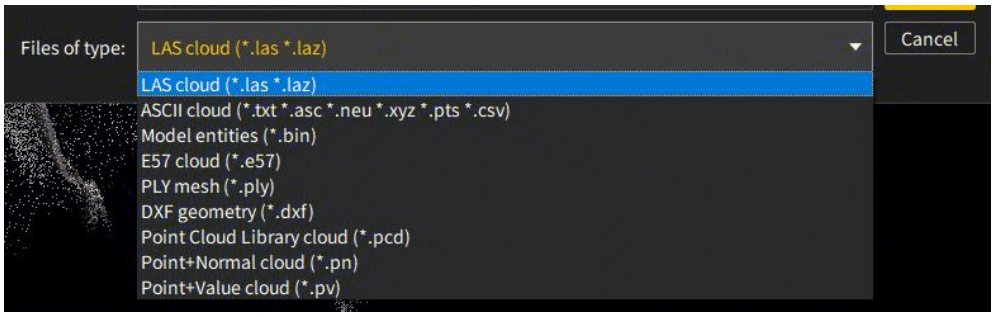



**Figure 39.** Point cloud data export

Ⅲ Extract trajectory

Select "Extract Trajectory" to export the trajectory point file (.trajectory.las) to the specified folder. Note that before extracting trajectory points and control points, a configuration file (.fjdata) needs to be added, otherwise it cannot be extracted and a prompt pop-up window will appear.


**Figure 40.** Trajectory file export

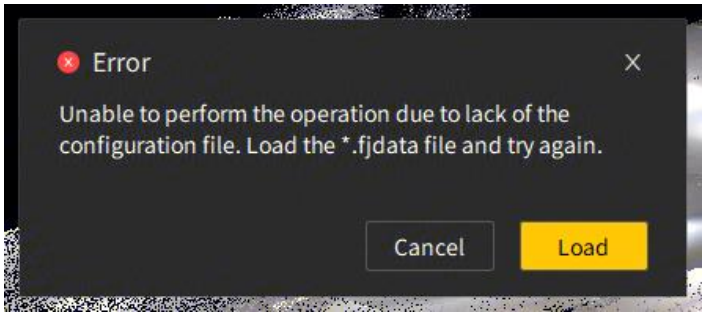



**Figure 41.** Export error message

Ⅳ Extract Control Points

By choosing the "Extract Control Points" option, you can export the control point file (.txt) to a specified folder. Please note that before extracting control points, you need to add the configuration file (.fjdata). If the point cloud file does not have control points, an error reminder popup window will appear to notify you of the missing control points.


**Figure 42.** Control points export

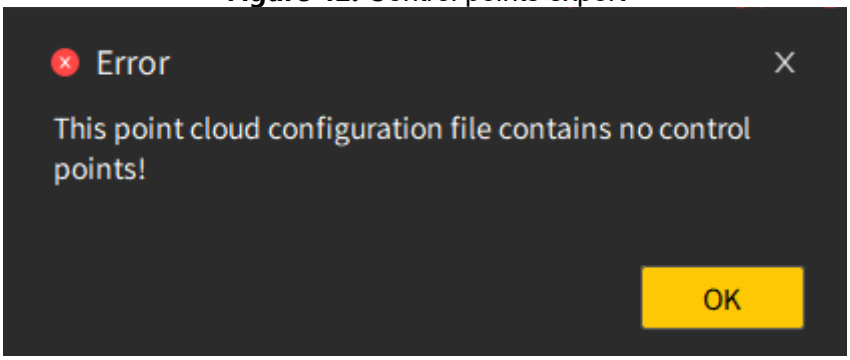



**Figure 43.** Export error message

V Add profile/extract profile/extract scan pose/delete

1. Select "Add Configuration File" or "Extract Configuration File" to import and export the configuration file (.fjdata);
2. Select "Extract Scan Pose" to export the pose file (.txt);
3. Select "Delete" to remove the point cloud file from the list.


**Figure 44.** Configuration files export

### 3.5.2 Model

**Description:**

Display as well as manage all model files in the project.

**Operation procedure:**

1. Select a model data, right mouse click to rename the file and view information;

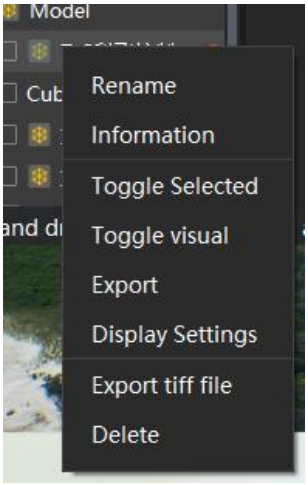



**Figure 45.** Right-click the model data toolbar

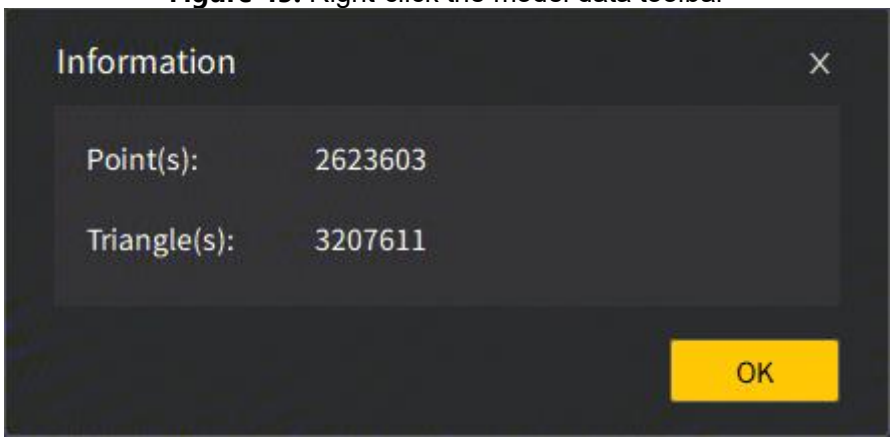



**Figure 46.** Model information

2. Select Export to save the current model data to a local file, support ply, obj, stl and other common model formats;

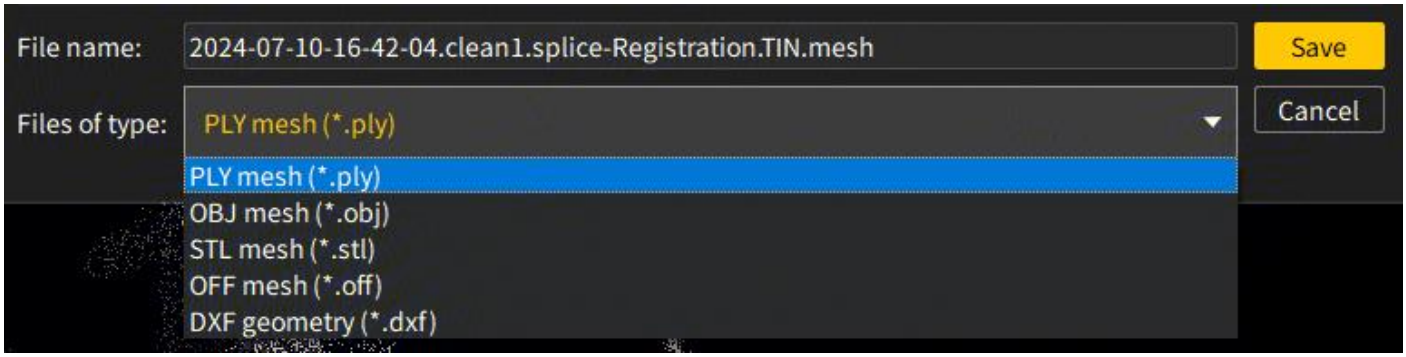



**Figure 47.** Model file export

3. Select Display Settings to set the display mode of the current model, which supports the display of vertex color, i.e. the original material of the model, or the display of custom color;

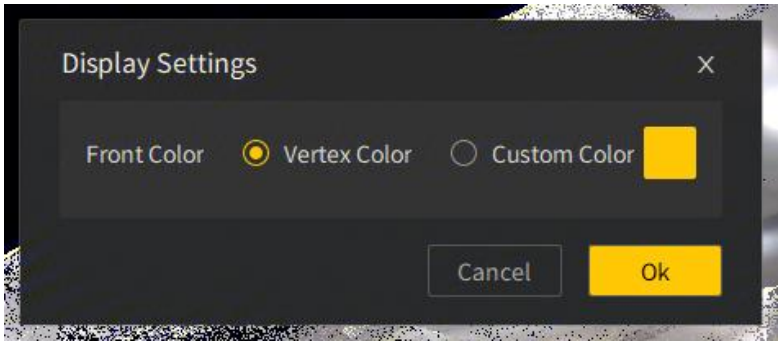



**Figure 48.** Vertex color display

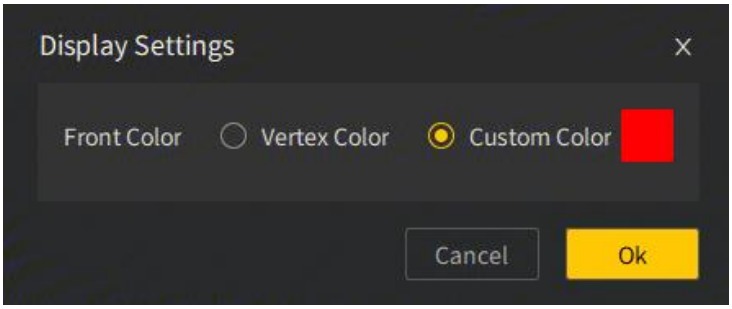



**Figure 49.** Custom color display

### 3.5.3 Vector

**Description:**

Display as well as manage all vector files in the project.

**Operation procedure:**

1. Select individual vector data and right mouse click to rename, toggle selection and display of files;

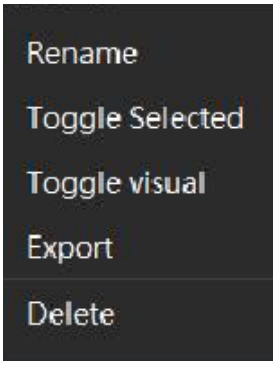



**Figure 50.** Right click the vector data toolbar

2. Select **Export** to save the current vector data to a local file, supporting bin, dxf and other common formats; select **Delete** to delete the corresponding file.

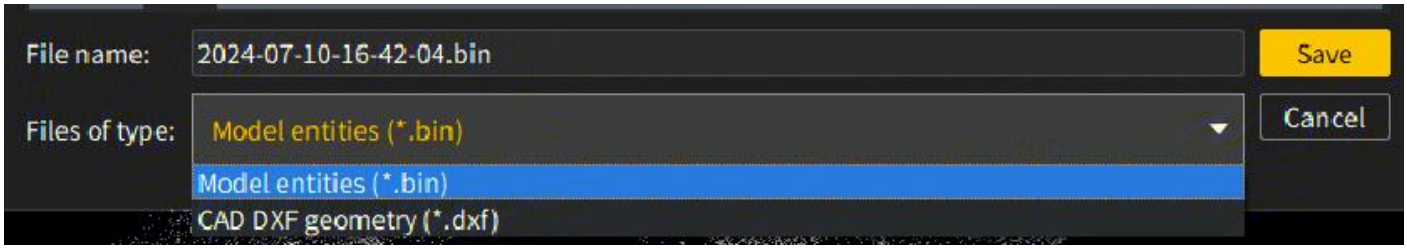



**Figure 51.** Vector file export

### 3.5.4 Label

**Description:**
Display as well as manage all labeled files in the project.
**Operation procedure:**
1. Select a label data, right mouse click to rename, switch select, display the file;

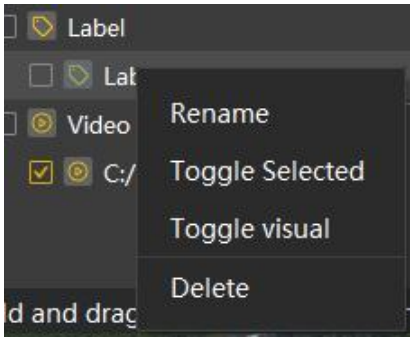



**Figure 52.** Right-click the label data toolbar
2. Select “Delete” to delete the corresponding file.

### 3.5.5 Video

**Description:**
Display as well as manage all video files in the project.
**Operation procedure:**
1. Select a video data, right mouse click to rename the file and view the information;

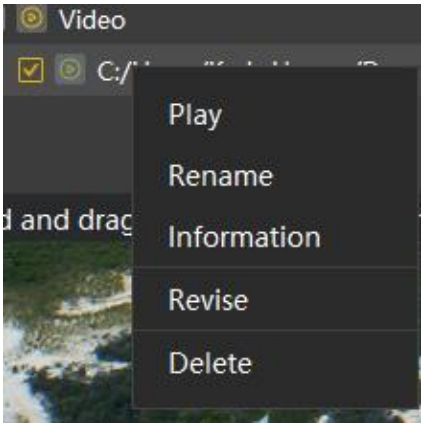



**Figure 53.** Right-click the video data toolbar


**Figure 54.** Video information

2. Select **Play**, the video playback interface will pop up in the view, and users can preview the video; select **Delete** to delete the corresponding file.

## 3.6 Shortcuts

| Operation | Function |
|---|---|
| Ctrl+O | Open project |
| Ctrl+S | Save project |
| Ctrl+Shift+S | save as |
| Alt+F4 | Quit |
| Ctrl+Z | Undo |
| Ctrl+L/R/F/K/T/B | Left/right/front/back/top/bottom views |
| Ctrl+left mouse button click on the project file | Select multiple files |
| Press down the left mouse button and drag in the 3D view area | Rotates the model |
| Press down the right mouse button and drag in the 3D view area | Translates the model |
| Scroll the scroll wheel of the mouse in the 3D view area | Zooms out and zooms in |

# 4 File

## 4.1 Open Project

Open a project file and display it in the 3D view, or use the shortcut **Ctrl+O** to open it (refer to 3.1.2 Open Project). If the project has been opened in the scene, the user will be prompted whether to save the current project, and the user can choose to save or not according to the actual situation.

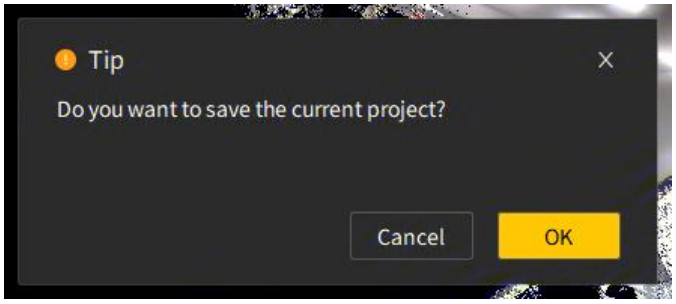



**Figure 55.** Tip

## 4.2 New Project

A new project file is created and displayed in the 3D view (refer to 3.1.1). If the project is already opened in the scene, the user will be prompted whether to save the current project, and the user can choose to save or not according to the actual situation.

## 4.3 Recent Project

Display the path information of the recently opened project, which is convenient for users to quickly open the recent project data; users hover over **Recently Opened** to display the history of the recently opened project, click on the corresponding project can be opened; users can click **Clear Menu** to delete the recently opened history.


**Figure 56.** Recently opened project

## 4.4 Add Data

Add external data to the project for operation, support point cloud, model, video and vector file data import, support ipadOS model exported scene file (.fjdscene) and project file (.fjdproj) data import, you can also drag the data to the software interface to add.

## 4.5 Export Data

Export single data or multiple data, supporting point cloud and model data; when selecting a single data, a data export window will pop up, and users can edit the name and select the file type; when selecting multiple data, the default save format will be set, point cloud defaults to las, and the model defaults to obj format.

## 4.6 Save Project

Users can quickly save the current project file by clicking **File-Save Project** or using the shortcut key **Ctrl+S**. The saved data is stored under the original file path.

## 4.7 Save Project As

Users can open the save file interface by clicking **File-Save Project As** or using the shortcut key **Ctrl+Shift+S**. Users can save the current project as a new project file by selecting the save path and entering the project name.

# 4.8 Settings

**Description:**

Settings include units, category attributes, and other settings. Unit settings include settings for units such as length, area, radius, volume, angle, etc. as well as the number of decimal places to display and whether to display a unit label after the value.

**Operation procedure:**

1. Users click **File** and then click **Settings** to pop up the following interface;

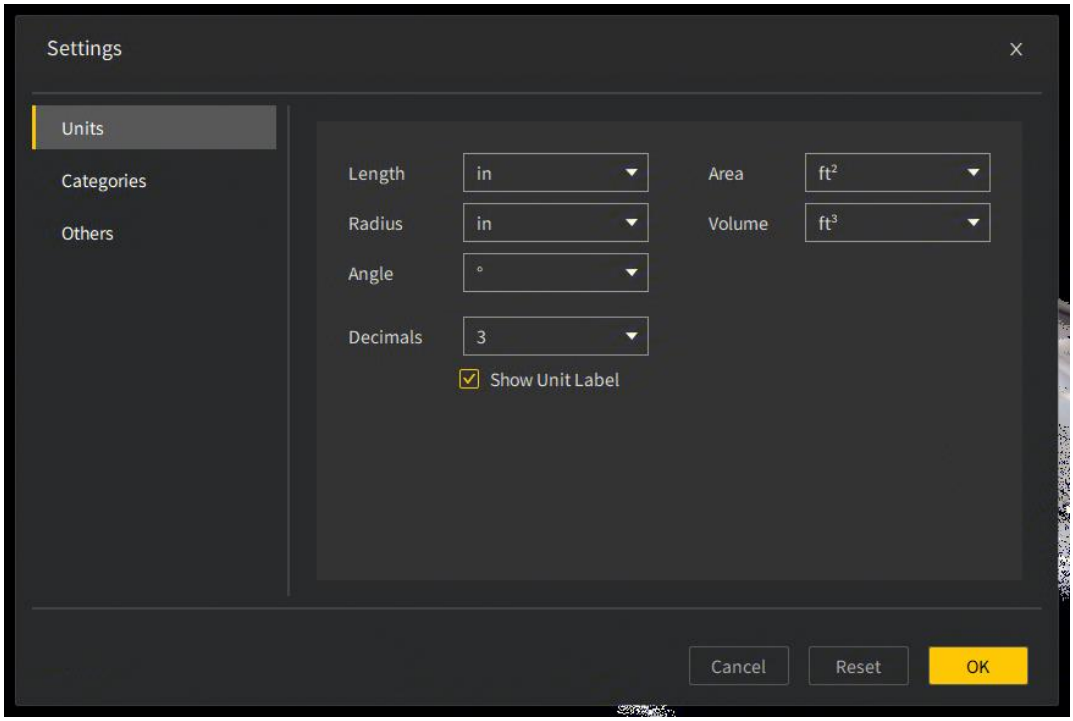



**Figure 57.** Settings interface

2. In the unit setting, you can set the unit and the number of decimal places of the data calculation results, and the following figure shows the comparison of the height measurement display results whether the unit is displayed or not;

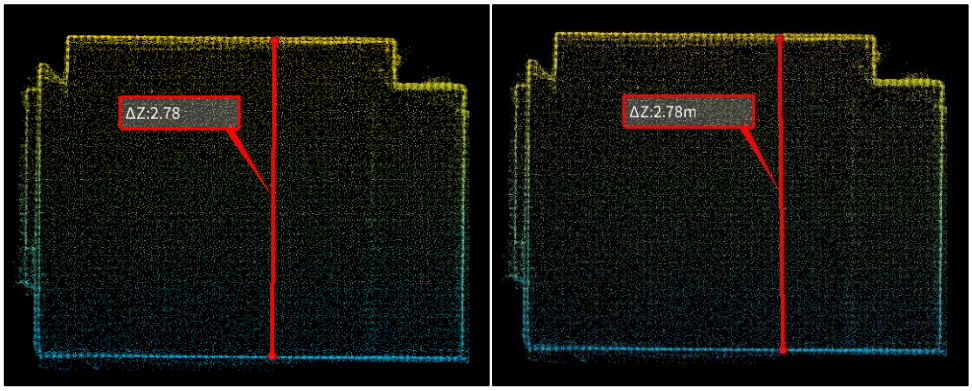



**Figure 58.** Unit display comparison

3. The category settings show the default point cloud category ID, category name and color, click “+Add” to add a new point cloud category, you can also modify the color of an existing category and the name of a category with a category ID greater than 15.

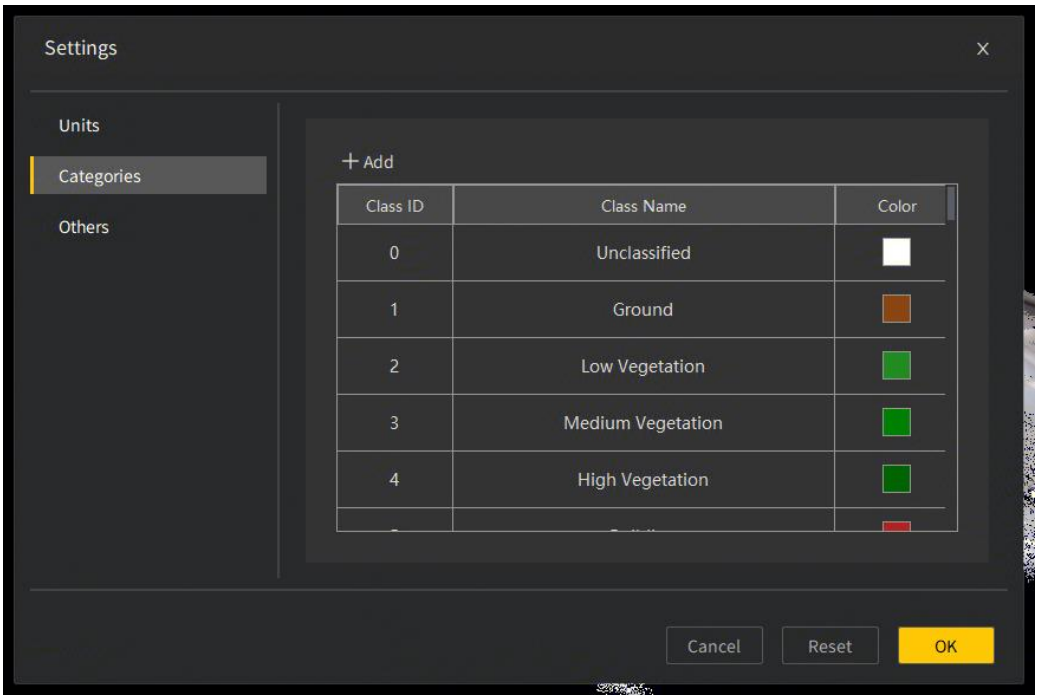



**Figure 59.** Categories setting interface

3. Other settings include default font size, point size, point style, whether to display the middle screen cross, scene brightness and contrast, as well as setting the software cache path, which can be set according to the specific needs of users.

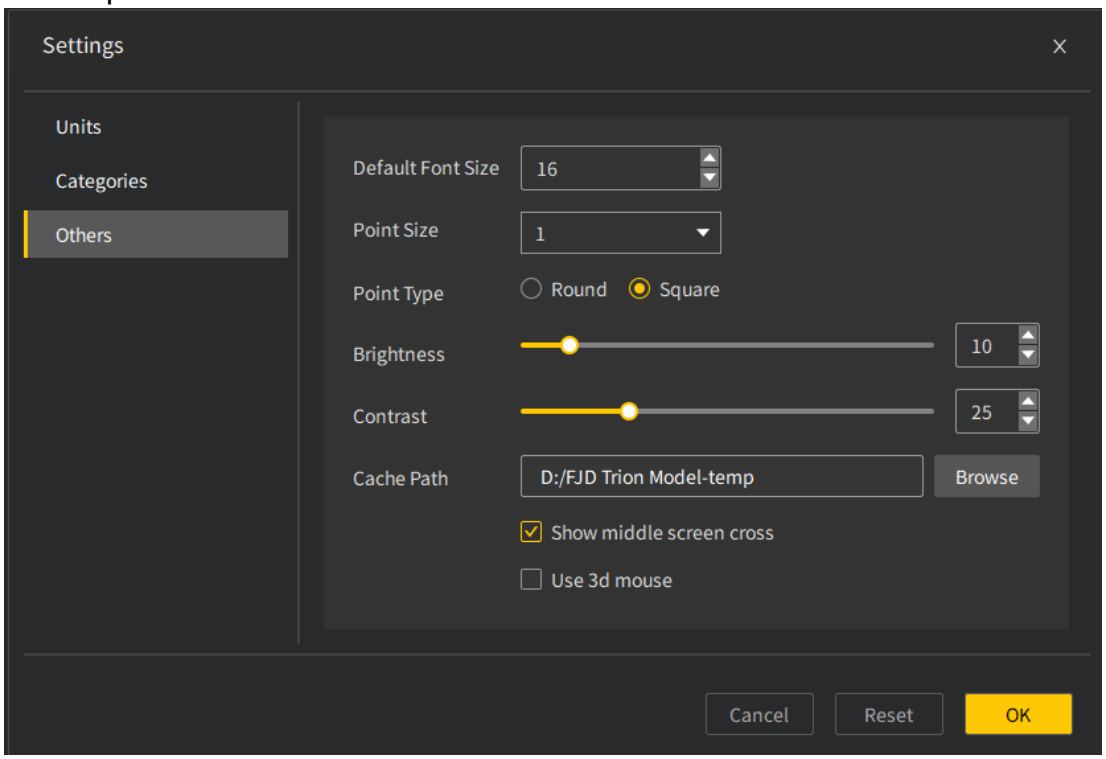



**Figure 60.** Others setting interface

4. Users according to the actual use of the scene and personal preferences to modify the relevant settings, click **OK** settings take effect, to restore the default settings when clicking **Reset**; The following is the display effect to modify the point style , the left figure for the round point style, the right figure for the square point style.

5. When the software needs to use a 3D mouse, you need to enable the 3D mouse settings.

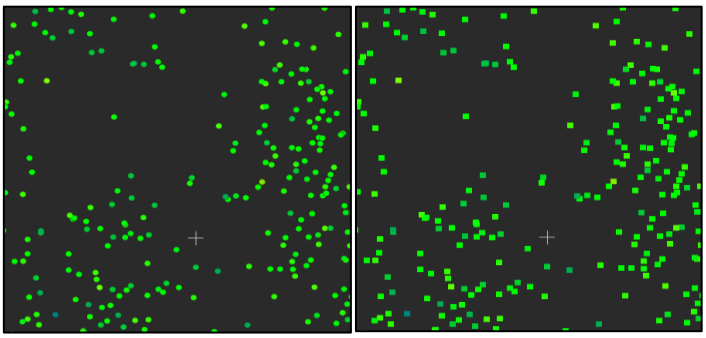


**Figure 61.** Comparison of the effect of point style modification

## 4.9 Quit

Users can close the software by clicking **File-Quit**, and the interface will pop up the following tips before exiting, so as to prevent users from exiting without saving data or misoperation.

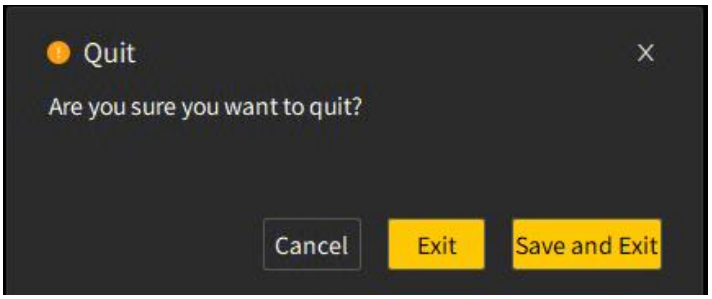



**Figure 62.** Quit the software

# 5 Start

## 5.1 Data Resolving

### 5.1.1 Point Cloud Mapping

**Description:**

Download raw data from FJDynamics 3D LiDAR scanners and generate point cloud maps via the FJD Trion Model software. The software currently supports post-processing and mapping for three in-house scanners: FJD Trion S1, FJD Trion S2, and FJD Trion P1.

**Operation procedure:**

1. Click  on the under the **Start** tab, or directly drag the .fjdslam file into the software to open the dialog box as shown below**.** Select the device model, scanning scenario, mapping range, and whether to generate a mapping report (to show mapping quality and related info), depending on the type of data to be processed.

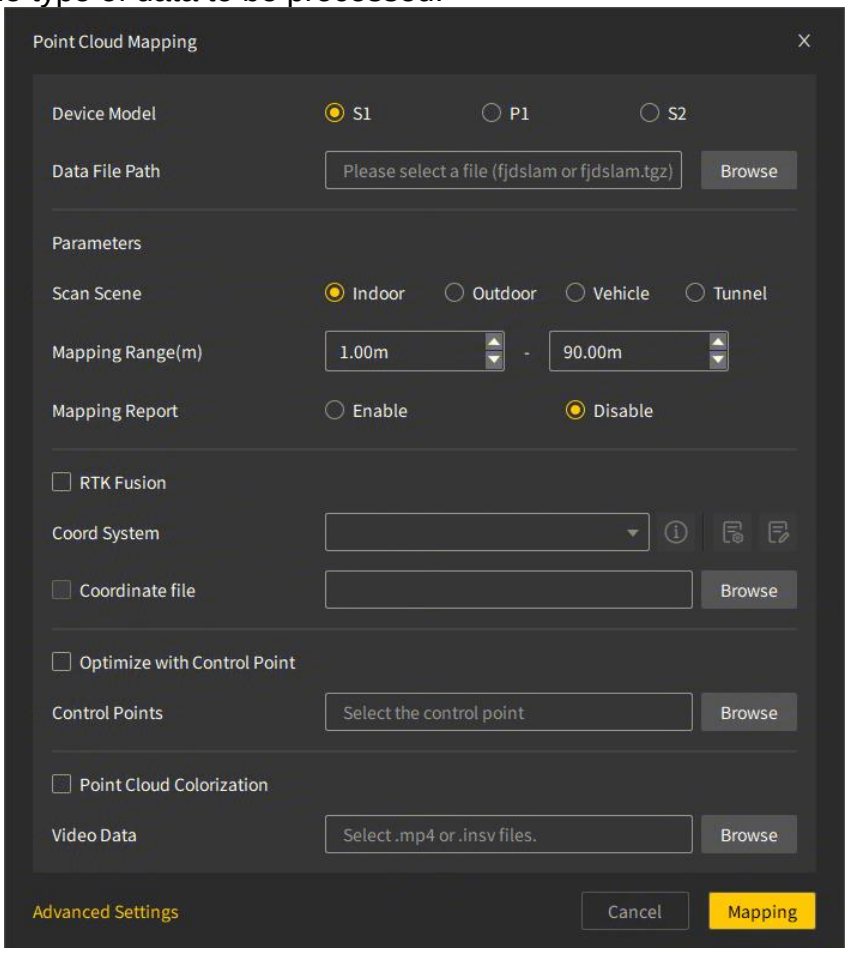



**Figure 63.** Point cloud mapping

2. In the **Advanced Settings**, you can configure mapping parameters (such as **loop closure enforcement**, **Coloring frequency** and **dynamic object removal**), RTK deviation values, and coloring parameters (including calibration file, colorization range, colorization frequency, and target removal). After completing the settings, click **Next**. To cancel mapping, click **Cancel**.

**Parameters:**

- Back to starting point: For data without a closed loop that returns to the starting point, the algorithm automatically enforces a closed loop solution to improve the mapping accuracy.
- Moving object removal: Automatically removes moving objects in the scene (e.g., people, vehicles).
- Mapping filtering: Addresses insufficient point cloud density in special scenarios.
- Densify point cloud: Enhances point cloud density.

- RTK receiver offsets: For RTK-equipped platforms (e.g., vehicle or backpack), you can adjust deviation settings if changes were made to the default structure in post-processing to obtain accurate mapping result.
- Calibration files: Allows importing of camera calibration data.
- Coloring range: Define the distance range used from panoramic video for point cloud colorization.
- Coloring frequency: Higher frequency improves result quality but will increase processing time
- Object removal: Remove unwanted objects from the panoramic video before colorization.

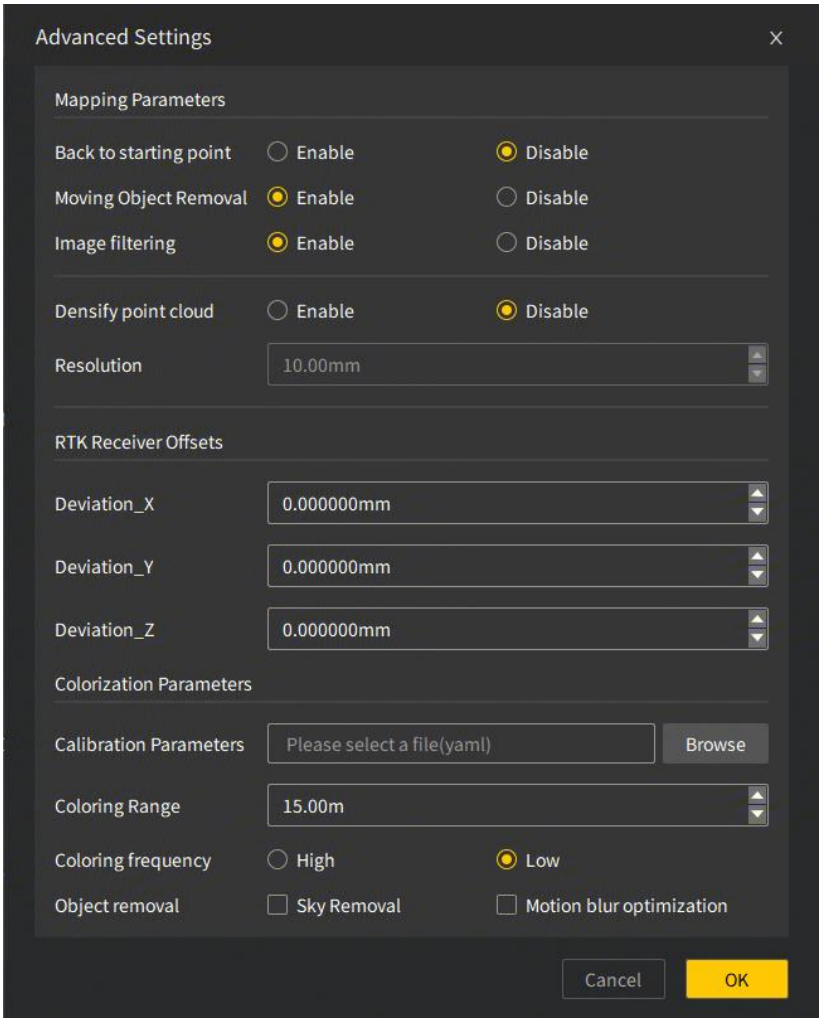



**Figure 64.** Advanced Settings

3. After selecting the mapping type, specify the data files and configure whether to enable RTK Integration, Control Point Optimization, or Point Cloud Colorization.

**RTK Integration** and **Control Point Optimization** cannot be enabled simultaneously. Enabling RTK Integration will automatically disable Control Point Optimization, while enabling Control Point Optimization will automatically disable RTK Integration and open the control point information viewer.

**Parameters:**

- RTK Fusion: If RTK mode is used in scanning, you only need to select the **Target coordinate system**, and after RTK fusion is enabled, RTK information will be used to assist in building maps and output the point cloud data of the target coordinate system. If PPK mode is used in scanning, you need to add the **Coordinate file**, which needs to be generated by **PPK Calculation** first, please refer to 5.1.3 PPK Calculation for details.
- Control Point Optimization: If control points were collected on-site, this function can be used to optimize the mapping result. A valid control point file must be selected for processing. The supported data format is: ID, X, Y, Z, and files can be in .txt or .xlsx format. TXT files may use

either comma or space as delimiters. Matching is prioritized by ID; if mismatched, the algorithm will automatically search for the corresponding points. At least 4 control points are required, and they should be spatially distributed and not located on the same plane.

- Point cloud colorization: If a camera mode was used during scanning, enabling this option will output colorized point cloud data. After selecting this option, choose the corresponding image file (MP4 or INSV format).

4. After completing the settings, click **OK** to start data analysis, the point cloud is displayed by default in the process of building, or you can uncheck the point cloud display. Display options include frame-by-frame display(showing the current frame only) or accumulative display (displaying all frames from the beginning to the current one).
5. To cancel the mapping process, click **Cancel**. To return to the previous step, click **Back**.
6. Once data parsing is complete, the result will be shown in the view window, and a new point cloud file will be added to the point cloud catalog tree.

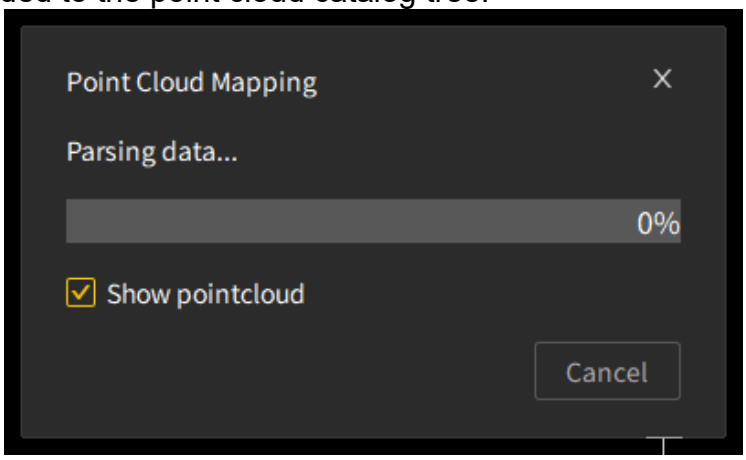



**Figure 65.** Data parsing progress

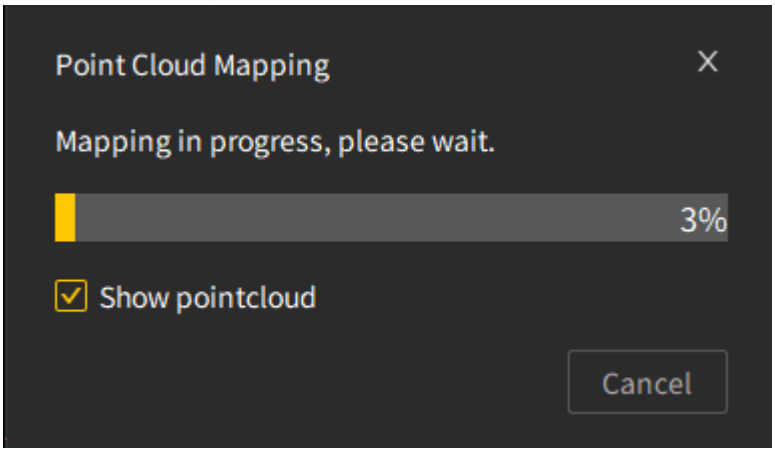



**Figure 66.** Mapping progress

**Note:**

- Mapping data is saved in the same path as the raw data file, so to simplify the mapping process.
- During mapping, the point cloud and path data is shown for your view.

### 5.1.2 RTK-based Registration

**Description:**

Aligns a point cloud with the point position data collected by an RTK receiver to represent the point cloud in a geographic coordinate system.

**Operation procedure:**

1. Click  on the **Start** tab, and the following window appears.

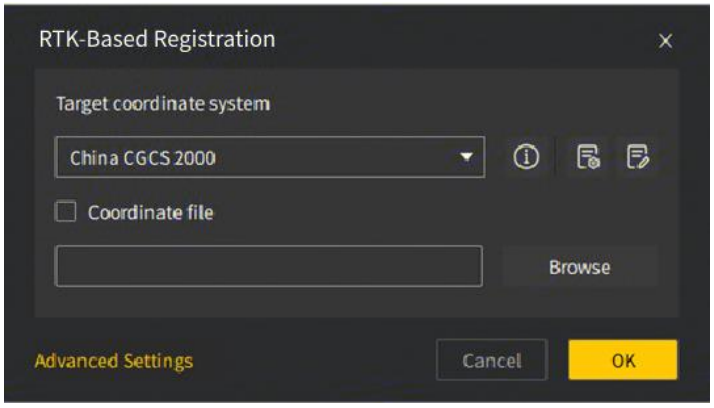



**Figure 67.** RTK-based registration

2. If RTK mode is used for scanning, you only need to select the target coordinate system. Click  to use predefined coordinates, it will bring up the following pop-up window to filter the coordinate system by continent and region.

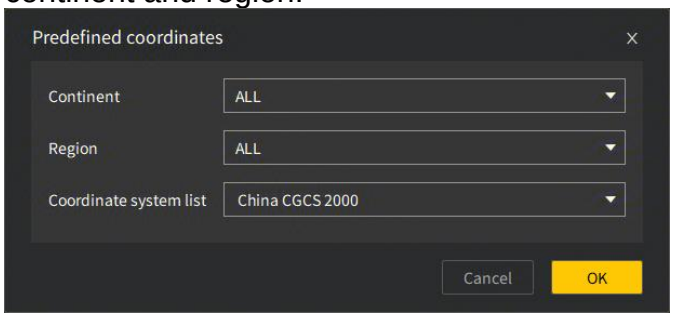



**Figure 68.** Predefined coordinates

3. After selecting the coordinate system and clicking **OK**, the detailed parameter information will appear, and you can modify it, or use the advanced parameters, and you need to re-enter the coordinate system name after modification.

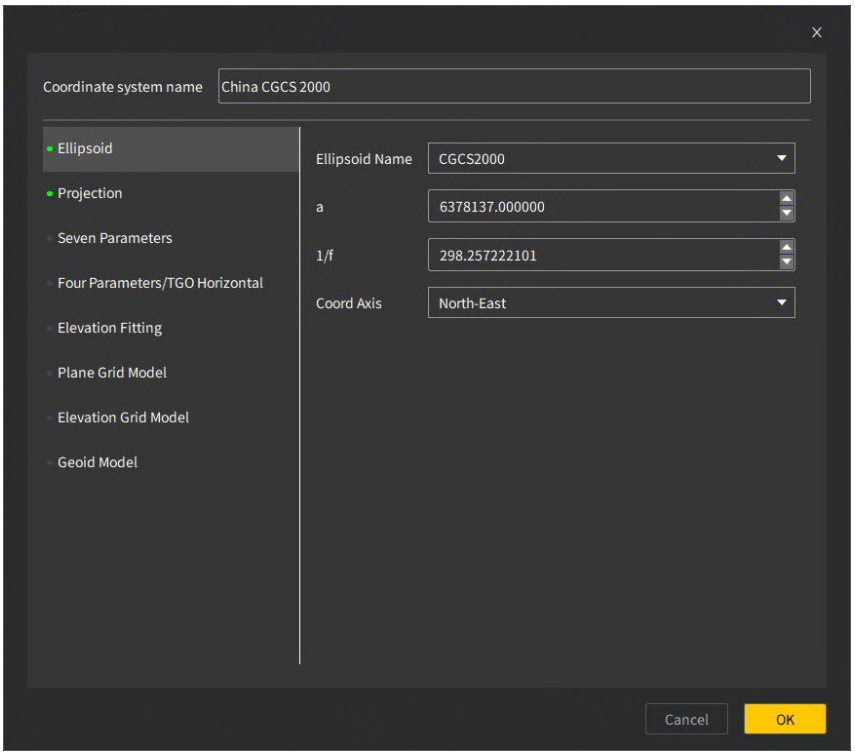



**Figure 69.** Detailed parameters of the coordinate system

4. When the user's target coordinate system does not exist in the drop-down list of geographic coordinate system, the user can choose the custom coordinate system, click  , the detailed parameters of the above coordinate system pop-up window will be displayed, set up and click **OK**, the calculation will be executed, the calculation will be successful, the target coordinate system will be outputted point cloud data, the coordinates of the measurement point will be displayed in the relative coordinates as well as the geographic coordinates.

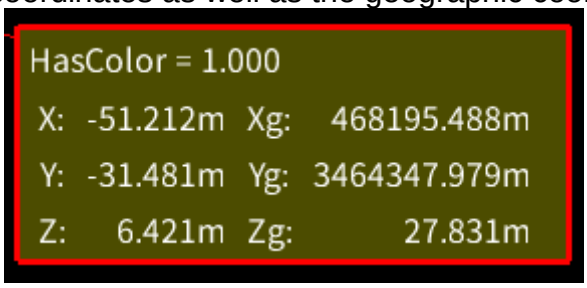



**Figure 70.** Coordinate display after RTK-based registration

5. If PPK mode is used for scanning, the coordinate file is also required, which supports the ".fjdrtk" format. The coordinate file needs to be generated using the **PPK Calculation**, details can be found in 5.1.3 **PPK Calculation**.

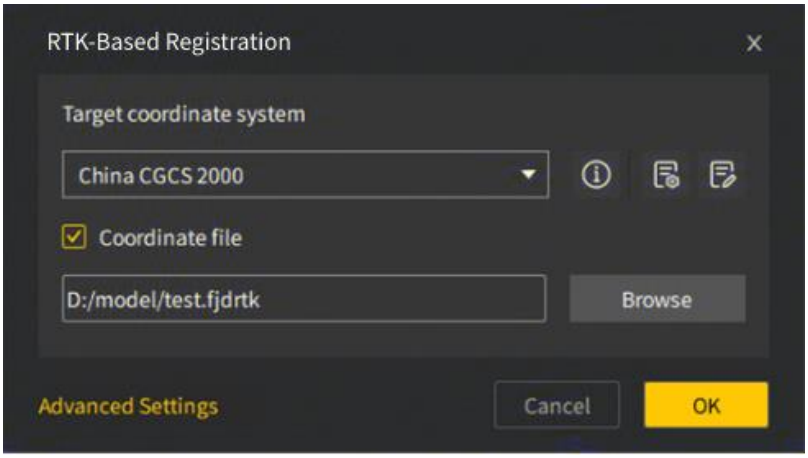



**Figure 71.** PPK calculation

6. If you are using an RTK receiver not from FJDynamics, click Advanced Settings on the RTK-based registration window, and enter the X/Y/Z axis offsets.

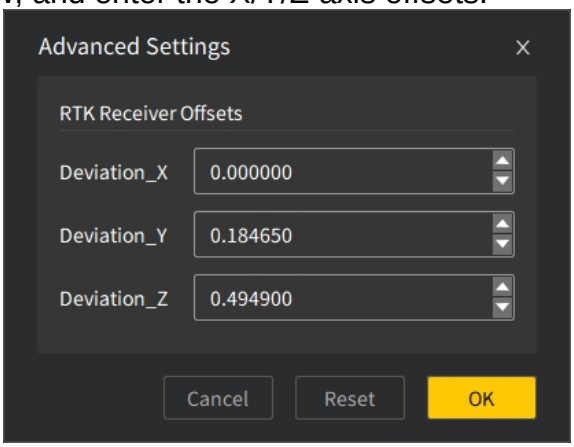



**Figure 72.** Advanced settings

## 5.1.3 PPK Calculation

**Description:**

The geographic coordinate information is solved from the source data collected by the base and mobile stations.

**Operation procedure:**

1. Click  on the Start tab and the window will pop up as shown below.

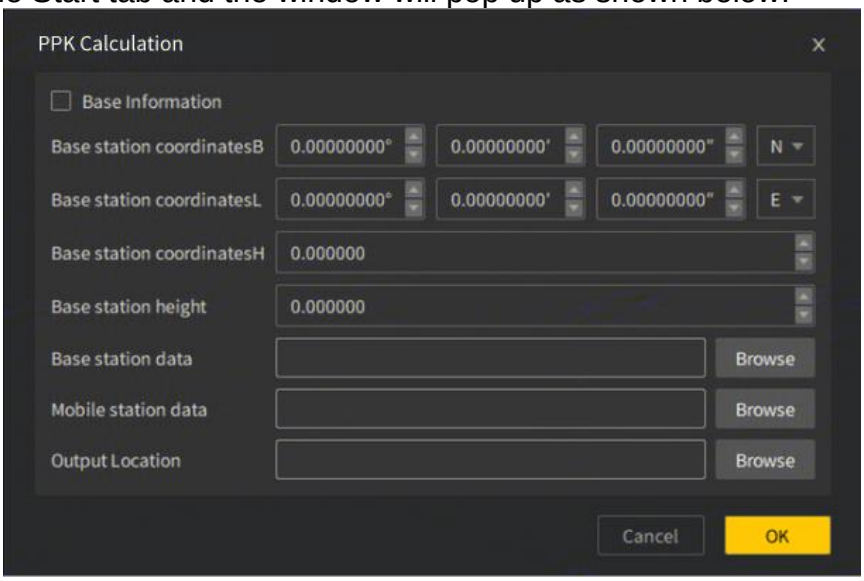



**Figure 73.** PPK calculation

2. If you need to input base station information manually, you need to check “Base Information” to activate the input box.

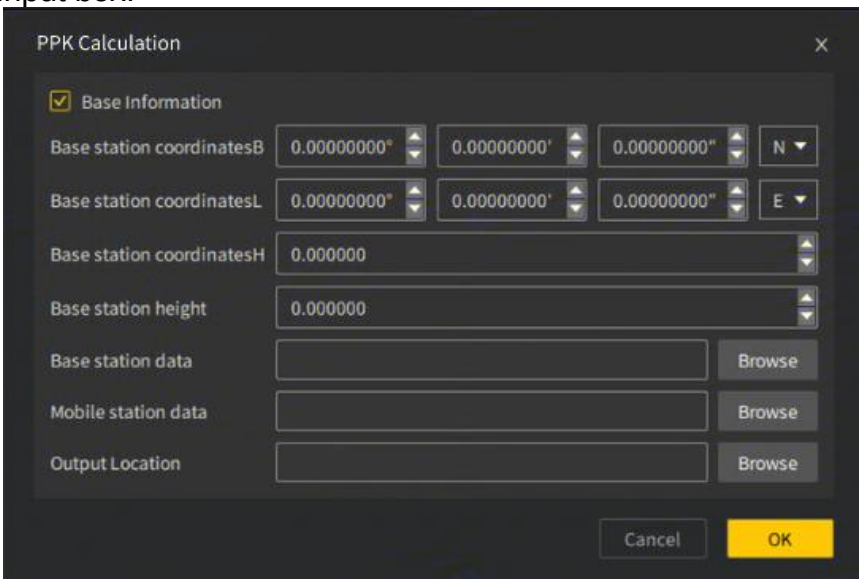



**Figure 74.** Input manually

3. If you don't need to input manually, you can import the base station data directly. Click Browse to select the base station data file, which supports .rtcm and .obs formats.

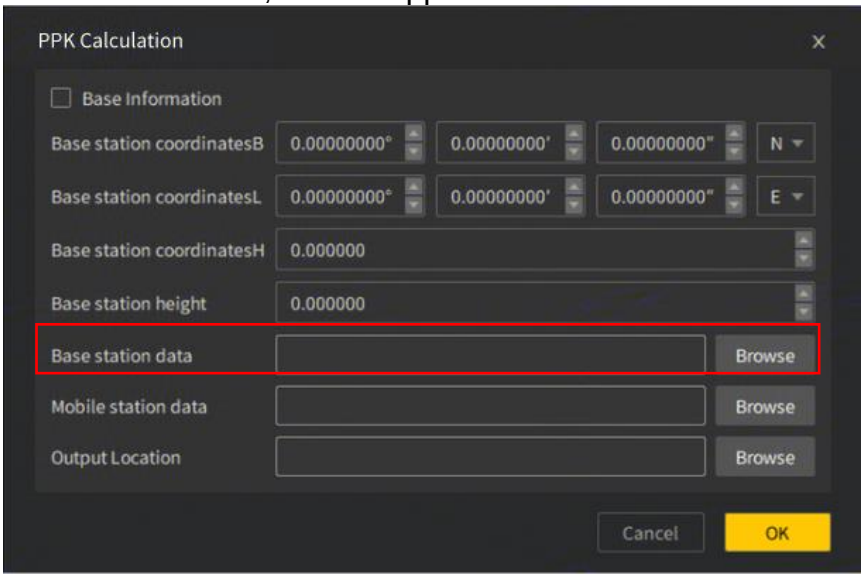


**Figure 75.** Import base station data

4. Then import the mobile station data and make the selection of the output location. After clicking **OK**, a .fjdrtk format file will be generated at the corresponding location, which can be used for RTK Fusion function during point cloud building or RTK-based Registration function.

### 5.1.4 Base Station Data

**Description:**

Acquisition of raw observation data from the base station.

**Operation procedure:**

1. Click the Base Station Data  and the following pop-up window will appear.

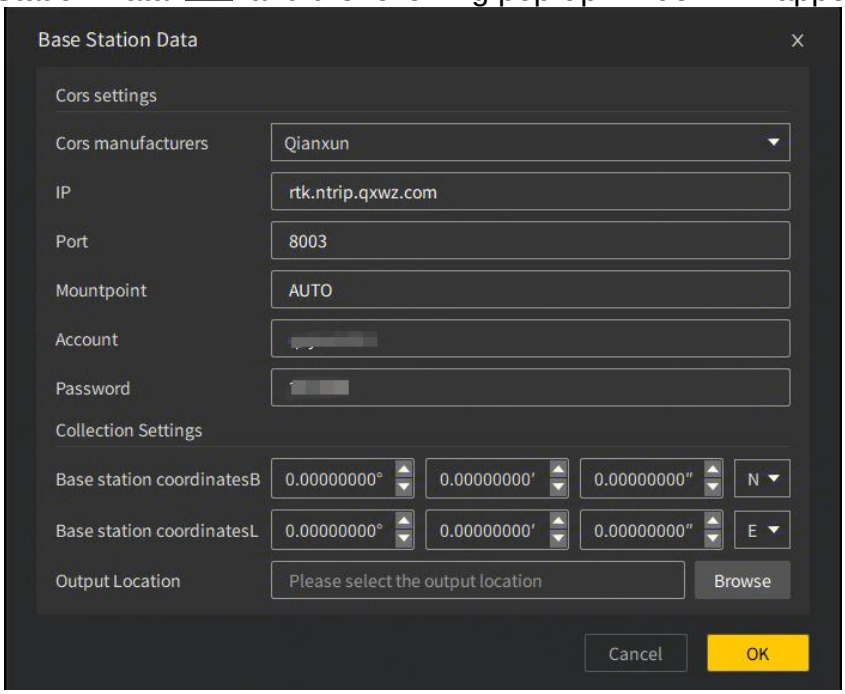



**Figure 76.** Base station data setting

2. Select Cors manufacturer in the drop-down box, then fill in the **IP address**, **Port**, **Mountpoint**, **Account** and **Password**, enter the **Latitude** and **Longitude** of the collection point, select the output location and click **OK** to start the base station data collection, the following window appears.


**Figure 77.** Base station data collection

3. When the acquisition is finished, click **Stop**, the ".rtcm" file will be generated under the output path just set.

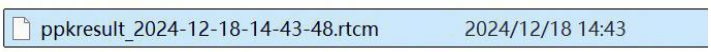



**Figure 78.** Base station data file

### 5.1.5 Camera Calibration

**Description:**

When scanning a scenario involving planar targets, the camera on the scanner captures a video of the scenario, and the system works out the camera and scanner parameter matrix to obtain the true color point cloud of better quality.

**Operation procedure:**

1. Select one or more (no more than three) point cloud files under **Objects** as required. To select more files at a time, click the files one by one while pressing the Ctrl key.
2. Click on the **Start** tab, and the following window appears. Select the point cloud name, config file, and image files, and click **Next**.

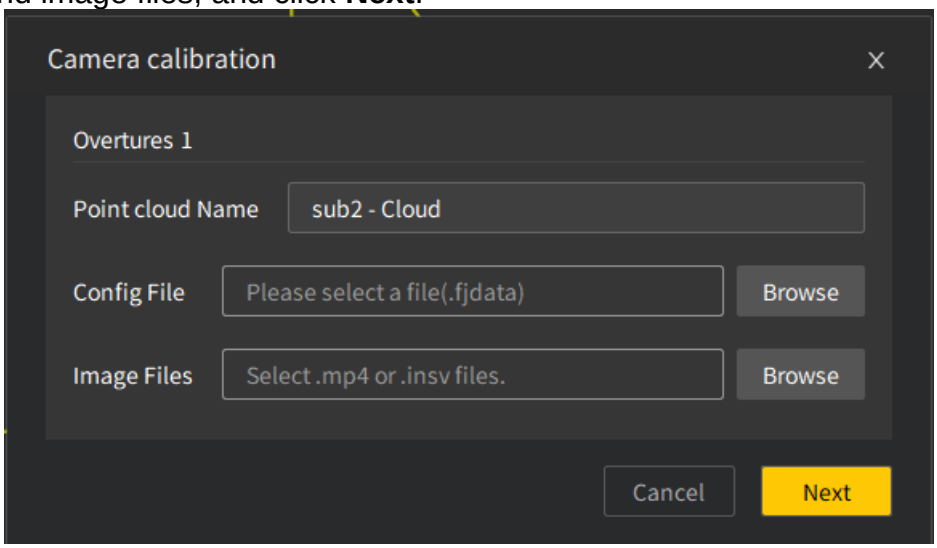



**Figure 79.** Camera calibration window

3. Pick the same number of target points on both the preview image and the point cloud for each state. If the calibration involves only one state, at least 12 target points must be picked. If the calibration involves more than one state, at least 4 target points must be picked for each state and the total number of target points picked must be no less than 12. Otherwise the following message appears after you click Next.

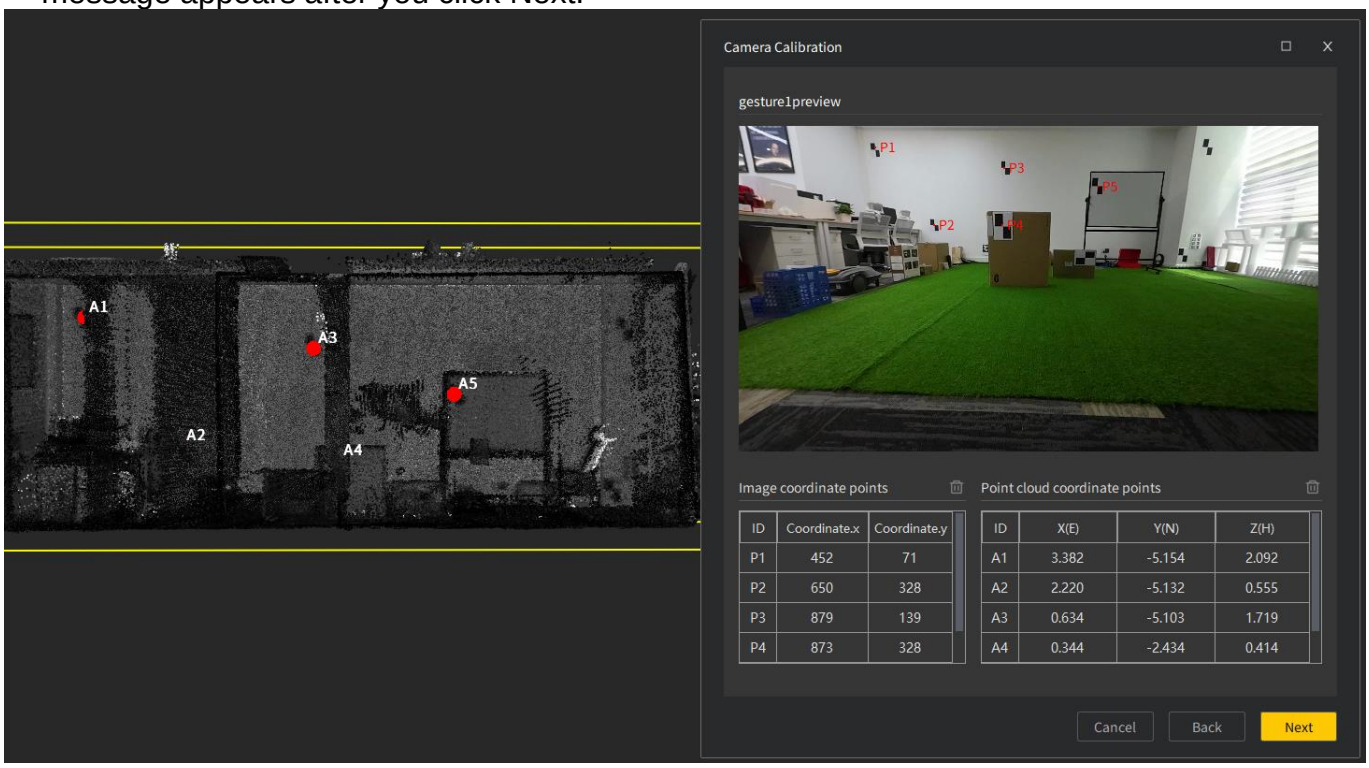



**Figure 80.** Picking target points

4. Click **Browse** to name the calibration file and select the storage path, click **Save**, and then click **OK**.

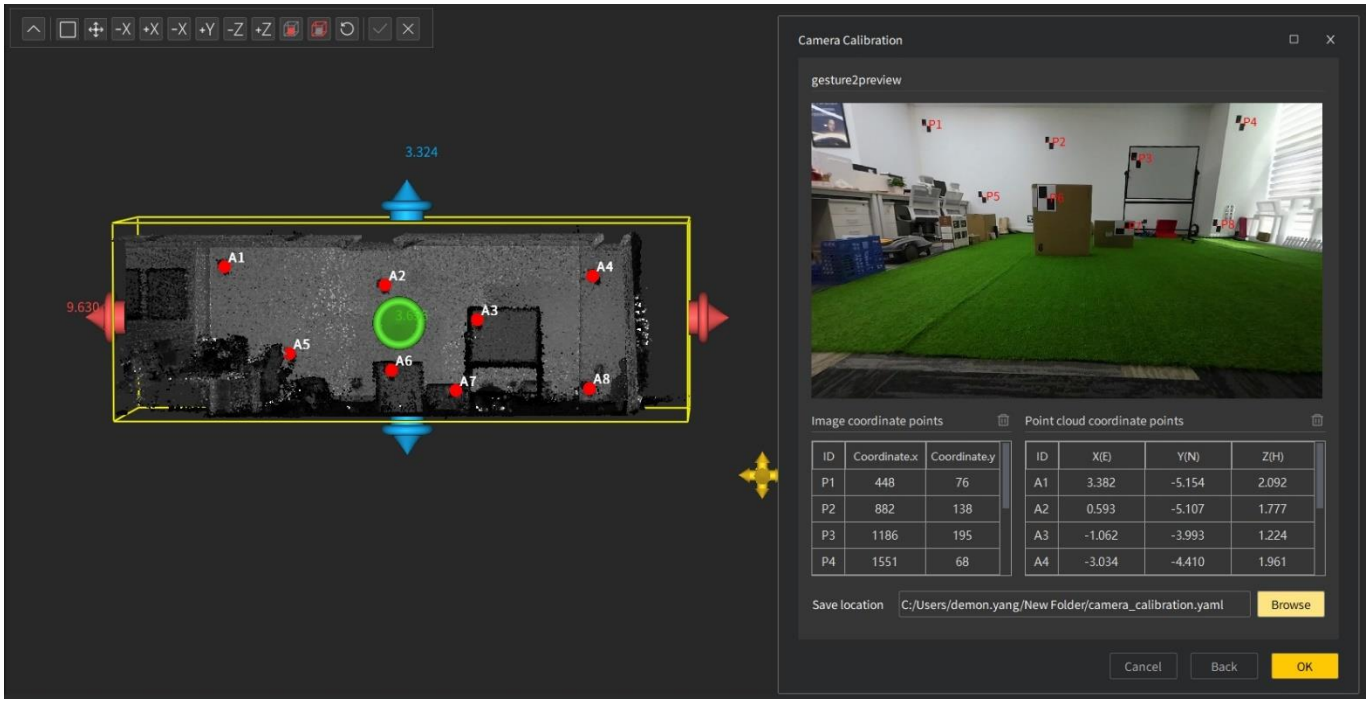



**Figure 81.** Selecting the storage path

5. A message will show up indicating whether the calculation is successful or failed. When the successful calculation message is displayed, click **OK** to colorize the point cloud.

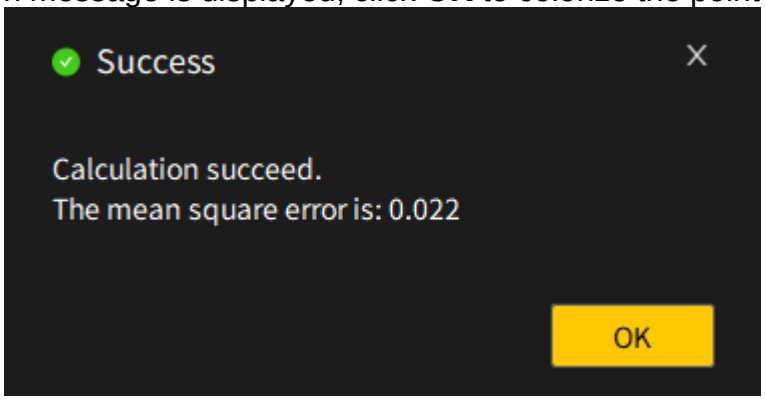



**Figure 82.** Successful calculation message

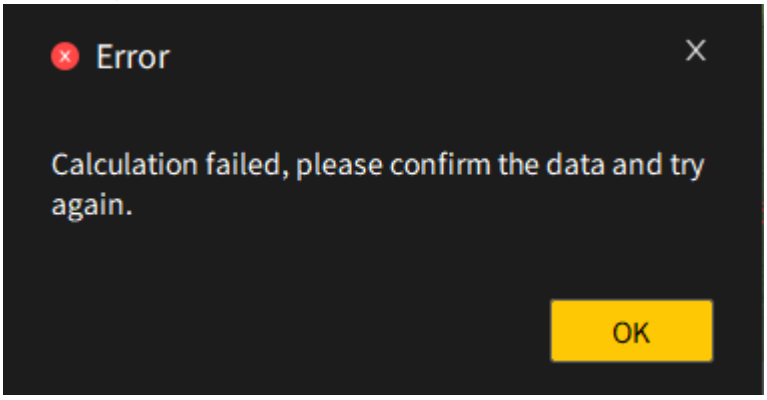



**Figure 83.** Unsuccessful calculation message

**Note:**

- Select the centers of planar targets as the target points.
- For the use of the camera, refer to the camera user manual. The camera calibration and point cloud colorization are unavailable if the camera is not purchased along with the scanner.

### 5.1.6 Point Cloud Colorization

**Description:**

Uses a series of algorithms to colorize the point cloud with true colors based on videos captured during scanning.

**Operation procedure:**

1. Click on the **Start** tab, and the following window appears, select the corresponding image data.

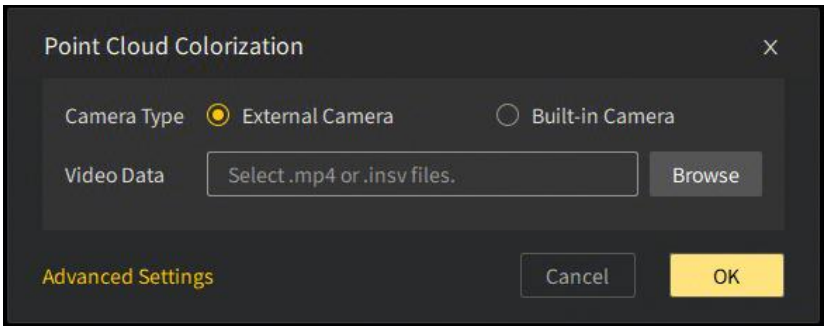



**Figure 84.** Select the image data

2. Click **Advanced Settings** to configure the detailed parameters of the colorization assignment.

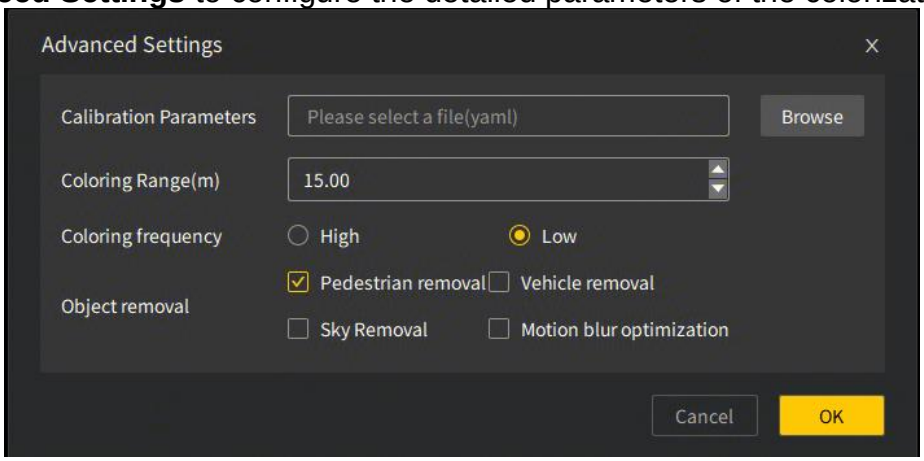



**Figure 85.** Advanced settings

3. When all required files are selected, click **OK** to start the point cloud colorization. The following window appears.

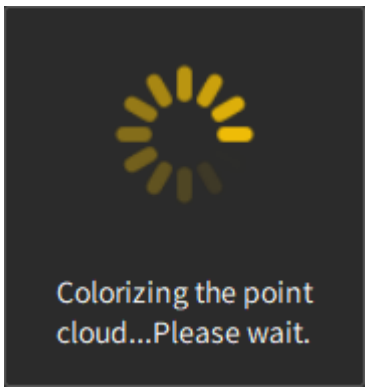



**Figure 86.** Colorizing the point cloud

4. The above window disappears when the colorization is completed. The colorized point cloud is displayed in the 3D view area.

**Figure 87.** A colorized building point cloud

5. The following window appears when the colorization fails. Ensure that the correct data is selected and try again.

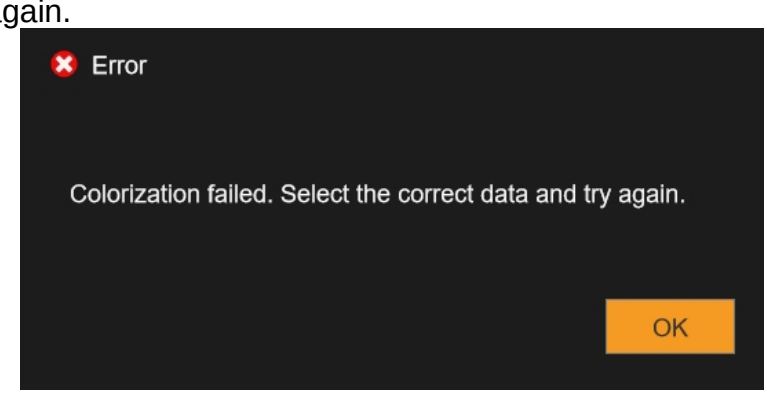



**Figure 88.** Unsuccessful point cloud colorization

### 5.1.7 Color Editing

**Description:**

During actual scanning, factors such as complex site conditions, drastic environmental changes, or lack of operator experience may lead to poor colorization results of the point cloud (e.g., incorrect colorization or missing color data). This can negatively impact the secondary use of the colored point cloud. Therefore, a manually controlled method is provided to allow users to modify or fill in point cloud colors, ensuring better colorization outcomes.

**Operation procedure:**

1. Click on the point cloud node in the project file to select the point cloud to be color edited, color editing function is activated and available, click the function button to open the function window.
2. By default, the "Specify Color" mode is enabled. In this mode, users can change the color of point cloud points that match a specific original color within the selected region to a target color. Selection tools including rectangle, polygon, and lasso are available for selecting regions. After selection, users can use the color picker to sample colors directly from the screen or manually set the original and target color values.

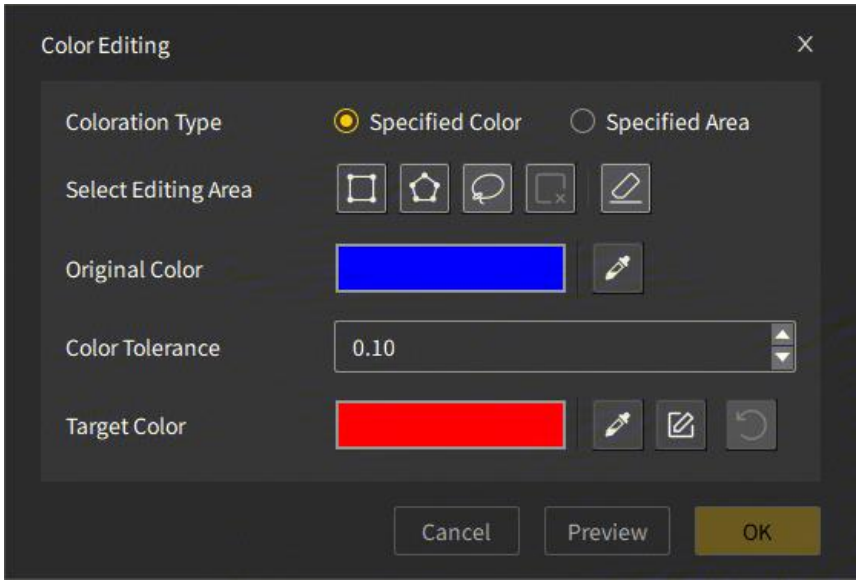



**Figure 89.** Specified Color

3. Once the color picker tool is activated, the mouse cursor enters the picking state. After picking, the color information is updated accordingly.
4. When the "Edit Color Values" button is clicked, a color setting window will appear. After setting the original and target colors, click Preview to view the effect of the changes in the point cloud. Click OK to confirm and update the display of the point cloud.
5. If the "Specify Area" mode is selected instead, all point cloud colors within the selected region will be changed to the specified target color.

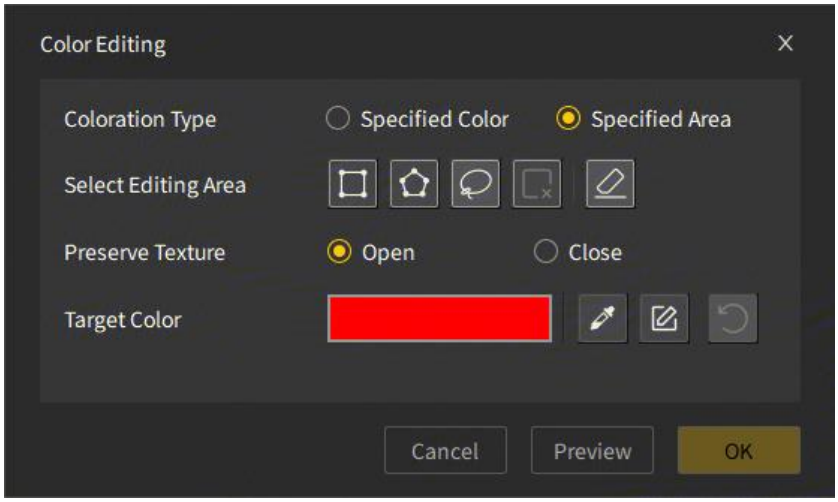



**Figure 90.** Specified Area

### 5.1.8 Orthophoto

**Description:**

Generates orthophotos with point clouds.

**Operation procedure:**

1. Select a point cloud under **Objects**.
2. The **Orthophoto** feature becomes available. Click  , and the following window appears.

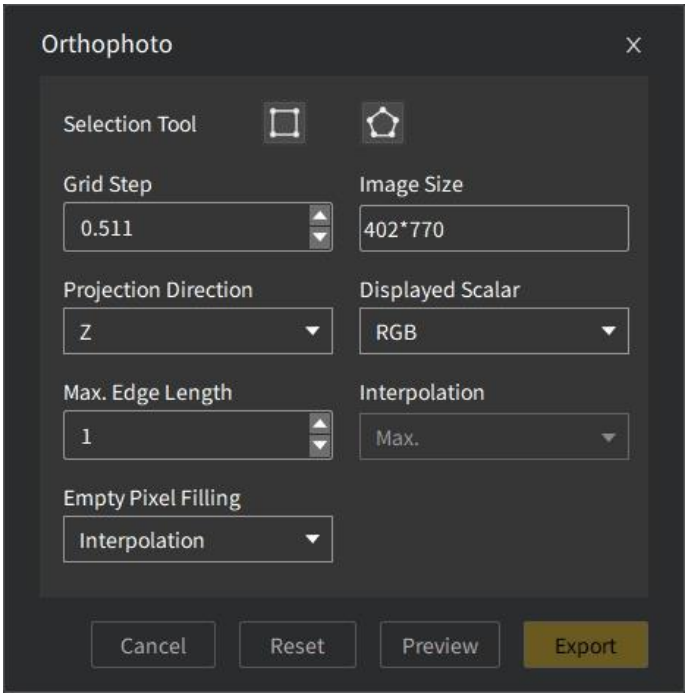



**Figure 91.** Orthophoto parameter settings

3. Select an area by rectangular or polygon selection in the same way as **Clipping**. Only one area can be selected at a time.
4. Setting **Grid Step** automatically updates **Image Size**. The grid step ranges from 0.001 to 99, and its default value is calculated with the formula: (max. edge length×2.5)/1920. The grid step increases or decreases by 0.001 each time.
5. Set **Displayed Scalar** based on the scalar properties of the point cloud. It is recommended to select RGB, or elevation if RGB is unavailable.
6. The interpolation can be **Max.**, **Mean**, or **Min.** and is set to **Mean** by default.
7. The empty pixel filling can be **Max.**, **Mean**, **Min.**, or **Interpolation** and is set to **Interpolation** by default.
8. The max. edge length ranges from 0 to 1000 and is set to 1 by default, meaning no limit on the max. edge length. The parameter only appears when the empty pixel filling is set to **Interpolation**.
9. Click **Preview** and wait. Then, the orthophoto appears. Click **Reset** to restore the original settings and close the preview. The orthophoto can be exported directly without a preview.

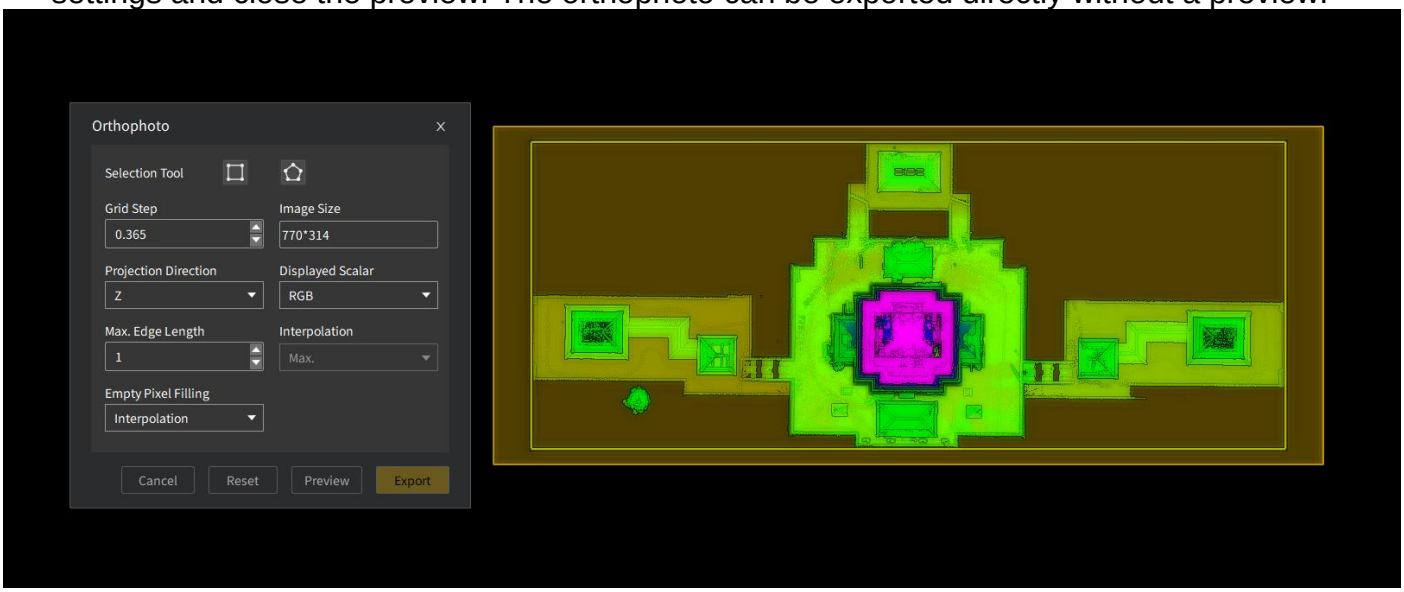



**Figure 92.** Orthophoto preview

10. Click **Export**, and the export setting window appears.

11. Set the parameters and select the storage path to export the orthophoto as a *.tif or *.tiff file, containing coordinate data. By default, the first file is named "orthophoto" and the subsequent files have a number added to its name starting from 1. The file name can be changed if required.

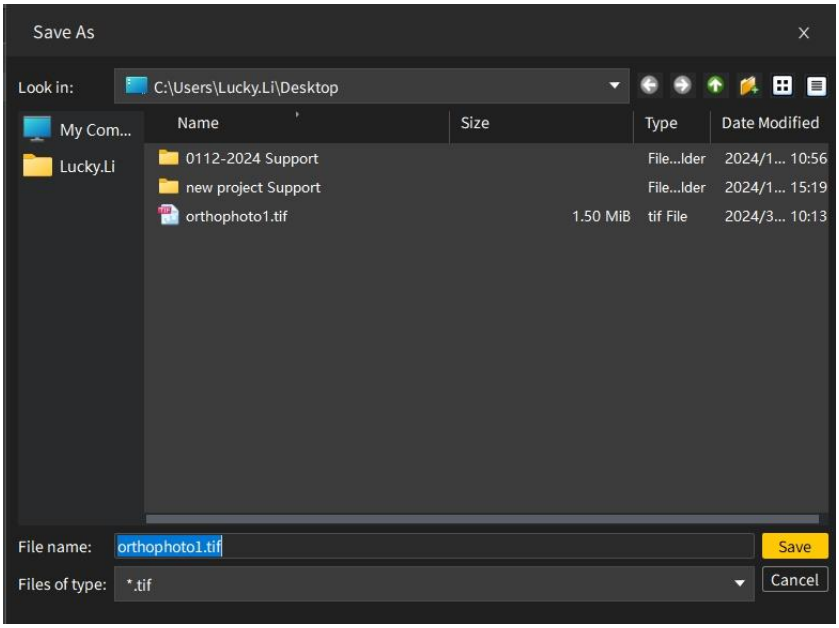



**Figure 93.** Orthophoto export

# 5.2 Point Processing

## 5.2.1 Moving Object Removal

Remove the abnormal point cloud caused by moving pedestrians or vehicles during the scanning process to ensure the display effect of the final point cloud, and solve the problem of misplaced scanning data or excessive noise in crowded areas; provide two ways of **Automatic Removal** and **Manual Removal**.

I. Automatic Removal

**Description:**

Moving objects in the scene are automatically removed by an algorithm.

**Operation procedure:**

1. In the project file, select the point cloud data that needs to be removed by moving objects, the **Moving Object Removal** will be activated and available, select **Automatic Removal** from the drop-down list, the window shown below will appear, enter the **Removal Grade** and click **OK** to perform automatic removal;
2. Automatic removal cannot proceed if the selected point cloud lacks an associated configuration file. Please ensure the configuration file is added beforehand;
3. The larger the value of **Removal Grade**, the stronger the removal effect and the more points will be removed;

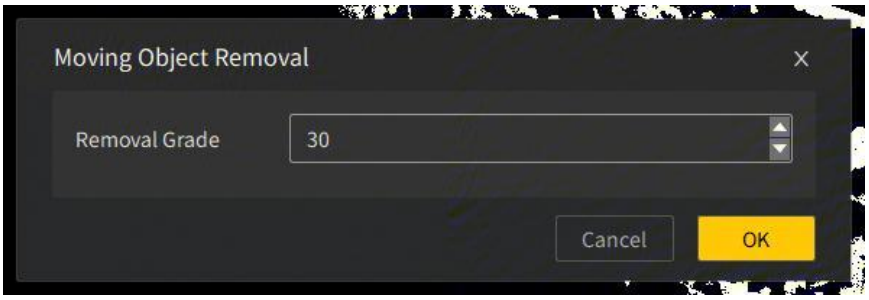



**Figure 94.** Moving object removal window

4. Click **OK** to enter the calculation process, displaying the progress of moving object removal. The greater the value of the **Removal Grade**, the better the moving object removal effect, and the longer the calculation time;

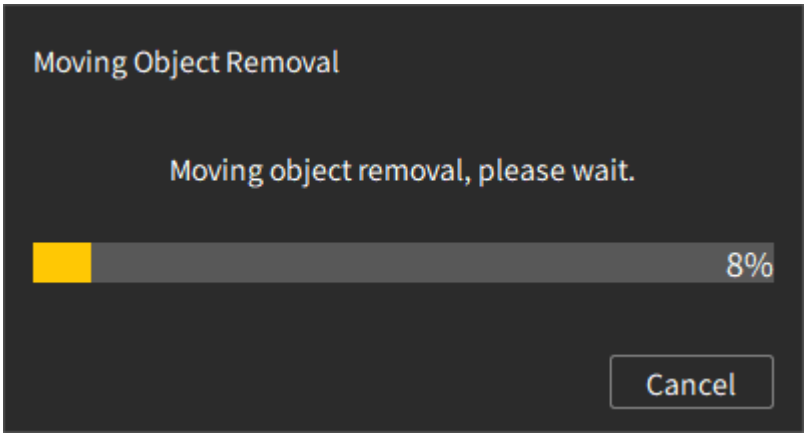



**Figure 95.** Moving object removal progress

5. After completing the calculation, the interface will be updated to show the point cloud data after the moving objects are removed. The following figure shows the comparison of the effect before and after removing the moving objects:

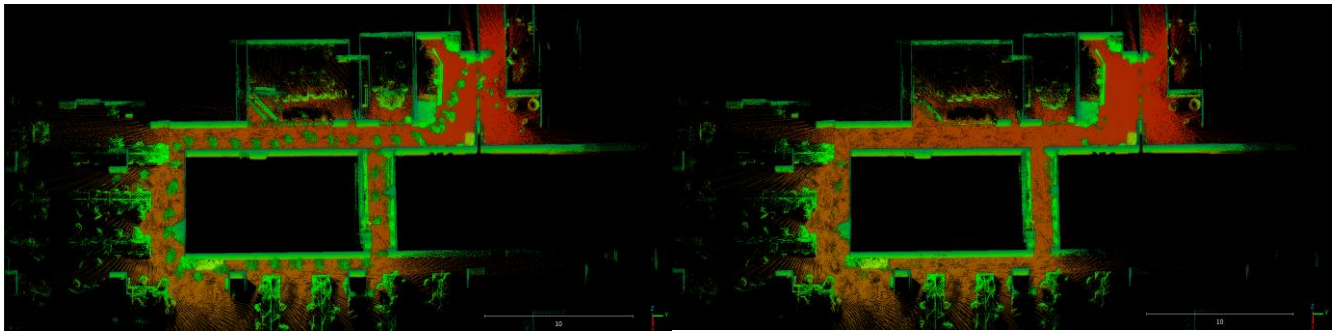


**Figure 96.** Before and after moving object removal

II. Manual Removal

**Description:**

Select the remove area by defining the eraser or specify the selection area by track;

**Operation procedure:**

1. Left mouse click in the project file list to select a point cloud data; select **Moving Object Removal**, select **Manual Removal,** display the **Manual Removal** window, only display the selected point cloud data after entering this function; for the convenience of users to choose, the point cloud is automatically displayed as per intensity;
2. Update the bottom bar prompt “Please pick up two points on the point cloud to define the length of the eraser”, in the 3D view area mouse switch to pick up the point state, pick up two points on the point cloud data as the rectangle length L, the mouse switch to the general state after completing the selection, “redefine "button available, previously disabled;
3. If the definition of the eraser is not satisfied, you can click **Redefine** to clear the interface has been the effect, the mouse switch back to the pickup state, can select again;
4. Complete the selection to update the length of the eraser parameters, while automatically updating the width and height according to the proportionality relationship, also supports the user to enter the definition, and at the same time, draw a green rectangle in the 3D view; set the removing mode to define the shape of the eraser; inside the zone: the eraser is a green rectangle, remove the moving objects in the rectangle; above the zone: draw a red cube of the same length and width but with a height of 2m above it, with support for modifying the height and removing moving objects in the red area above the cube.; below the zone: the opposite of above the zone;

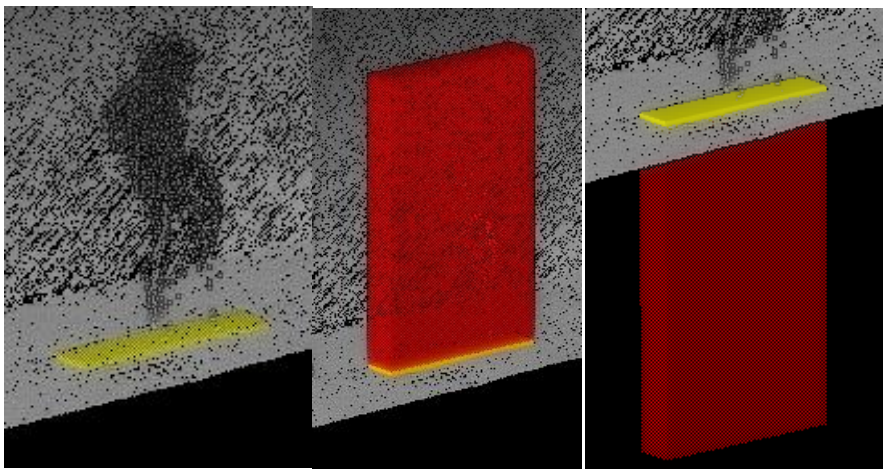


**Figure 97.** Inside、Above、Below the zone

5. **Preview**, **Apply** are available. After setting the relevant parameters, the user can drag the drawing cube through the point cloud area of the moving object to be deleted. After completing

the drawing, it will show the effect of selected moving objects, and then click the **Preview** to show the effect after removal;

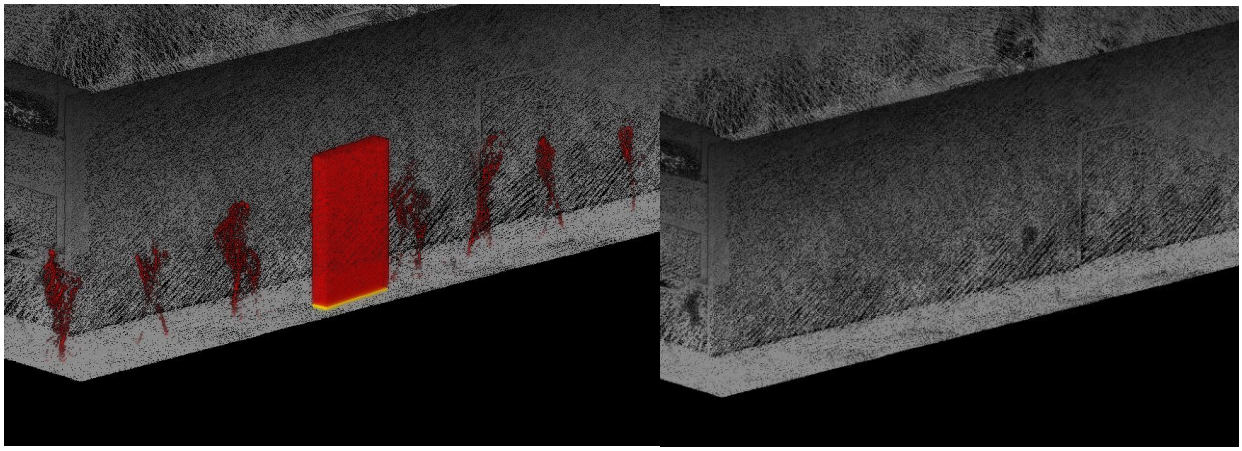


**Figure 98.** Before and after moving object removal

6. The interface pops up a preview window, and the interface view is refreshed after the preview is completed. If you are not satisfied with the preview result, you can **Ctrl+Z** to undo the removing effect;
7. After confirming that you are satisfied with the current effect, you can continue with the removing operation. Click **Apply** to save all removing effects and close the function window. Generate a point cloud file of the same level as the source file, with the same file name as the **Automatic Removal** rules;

### 5.2.2 Rectification

I. **Automatic correction**

**Description:**

Adjusts the normal of point cloud data across different planes based on axis-aligned constraints, aligning Z-values to a common plane to enhance the overall visualization of the point cloud.

**Operation procedure:**

1. Select a point cloud file under **Objects**, click the **Start** tab, and then select **Rectification**.
2. The rectification of the selected point cloud starts. A window appears indicating that the rectification is in progress.

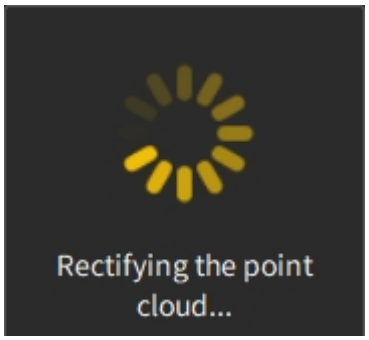



**Figure 99.** Rectification in progress

3. The window disappears when the rectification is complete. A new point cloud file is added to the list at the same level as the original point cloud file, and the file name is *original point cloud name*.adjust*X*, where *X* indicates the number of rectifications that have been done to the point cloud.
4. After rectification, only the new point cloud is shown in the 3D view area.
5. The following window appears if the point cloud cannot be rectified.

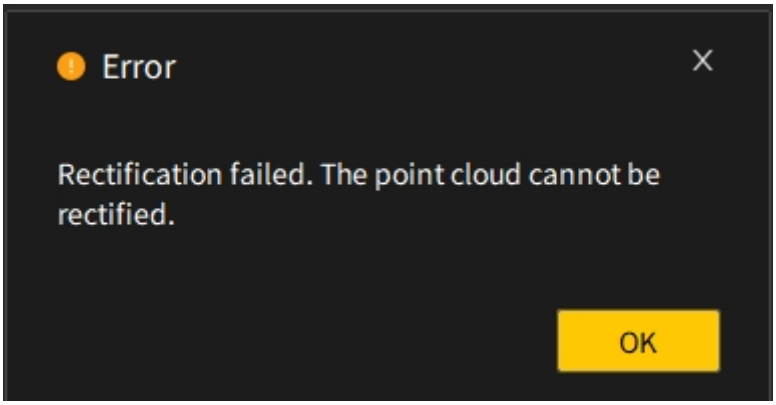



**Figure 100.** Rectification error prompt

Note: Only one point cloud can be rectified at a time. Hidden point clouds must be shown before they can be rectified.

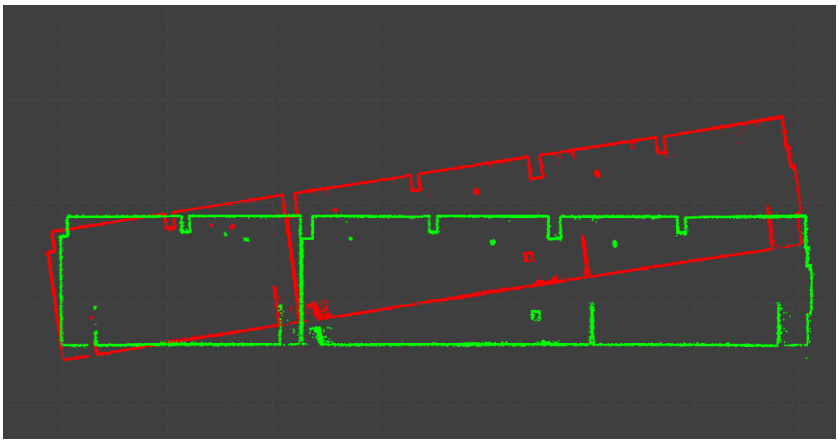


**Figure 101.** Effects before and after point cloud rectification

## II. **Manual correction**

**Description:**

Allows manual correction of the point cloud orientation by constructing a reference plane and assigning a specified direction to adjust the normal.

**Operation procedure:**

1. Select the point cloud data in the project file Under the **Start** tab, click **Point Cloud Correction**, then choose **Manual Correction** to open the corresponding dialog.

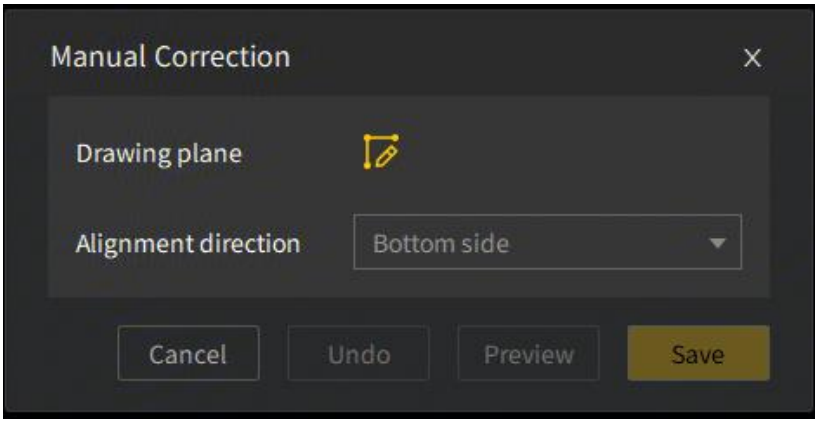



**Figure 102.** Manual correction pop-up window

2. Left-click three points on the point cloud to construct a plane. The arrow on the plane indicates the positive direction (normal vector) of the surface.

- Note: The direction of the plane's normal depends on the point selection order. Clockwise and counterclockwise selections will generate opposite directions.

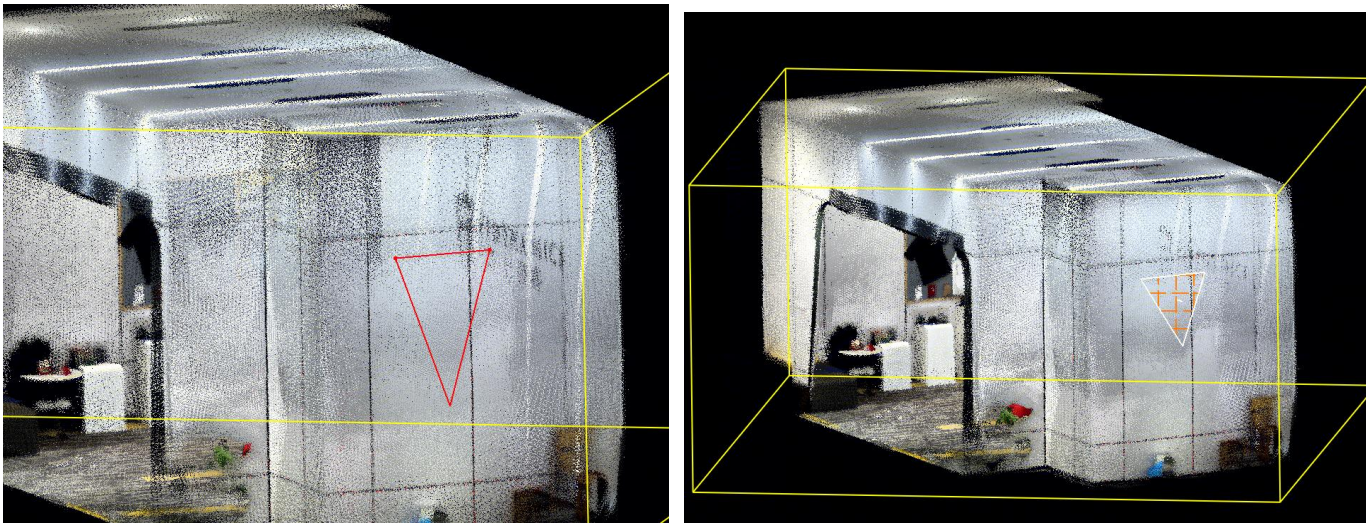


**Figure 103.** Draw a plane

3. Select the direction in which the plane is to be aligned, and use the front, back, left, right, top, and bottom sides of the bounding box as reference planes for alignment. Click Preview to correct the point cloud as a whole, and click Save to directly correct and save the original point cloud in this way.

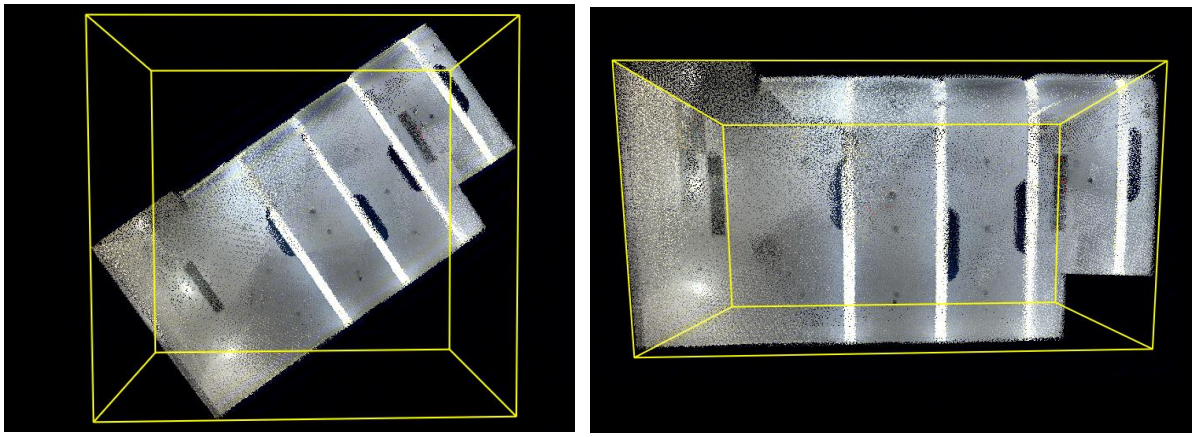


**Figure 104.** Before and After Manual Correction

### 5.2.3 Densify Point Cloud

**Description:**

Increase the point density of point cloud data.

**Operation procedure:**

1. Select the point cloud data in the project file, click the "Start" tab, and click the **Densify Point Cloud** function icon. A pop-up window will appear with a default resolution of 10 mm, which can be set manually.

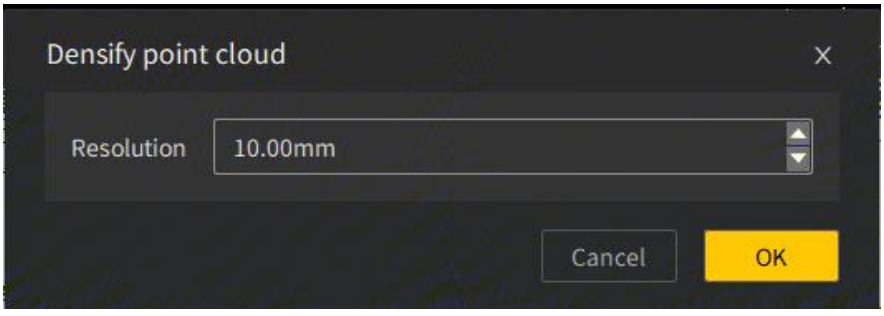



**Figure 105.** Densify Point Cloud pop-up window

2. . After completing the settings, click OK to start the calculation. Once finished, a new, denser point cloud dataset will be generated.

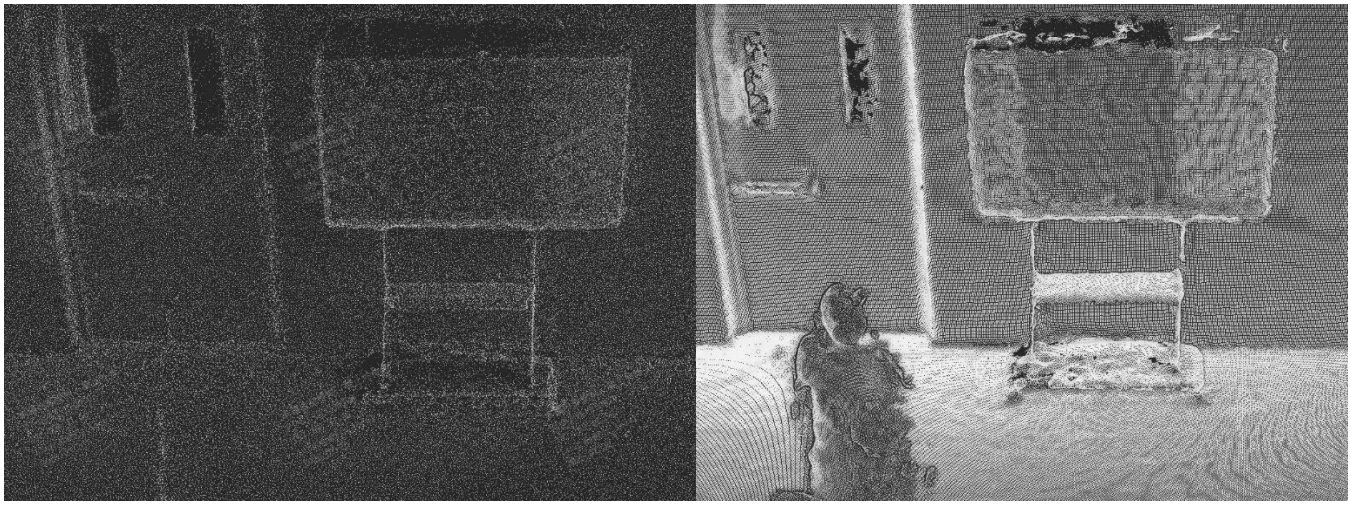


**Figure 106.** Effect comparison

### 5.2.4 Point Cloud Filling

**Description:**

Supports filling point cloud data to solve data holes caused by occlusion during scanning.

**Operation procedure:**

1. In the project file, select the point cloud dataset. When the  button becomes active, click it to open the Hole Filling window.
2. Select the area on the point cloud that needs to be filled. Click the **Fill** button to automatically compute and generate corresponding point cloud data in the selected region.
   - Note: If the selected area contains multiple separate holes, filling cannot be performed.

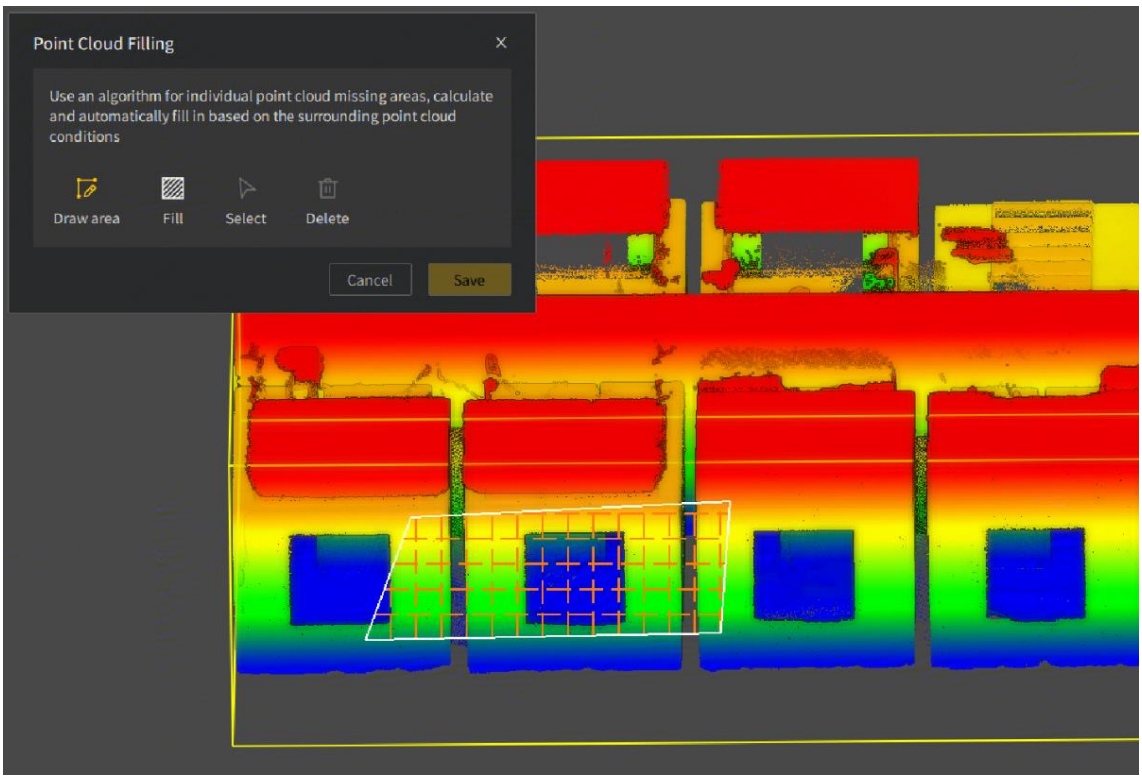



**Figure 107.** Point cloud filling

**3.** After filling is completed, click Save to update and save directly on the origin cloud.

### 5.2.5 XY Transpose

**Description:**

One-click transpose point cloud xy coordinates, output new point cloud data to meet the needs of users in different areas with different coordinate systems.

**Operation procedure:**

1. Under the point cloud node of the project file, left-click the mouse to select a point cloud data or **Ctrl+ Left-click** to select multiple point cloud data;
2. The XY transpose function is activated, click to perform the transpose operation to display the calculation waiting prompt box;
3. After completing the transpose, the waiting prompt box will be closed automatically, and the project file will be added with the converted result in the point cloud node. The naming format of the newly generated point cloud is: original point cloud name_Transpose number;
4. The number of consecutive operations on the same data is incremented from 1. Deleting the existing transpose results will not affect the numbering sequence;

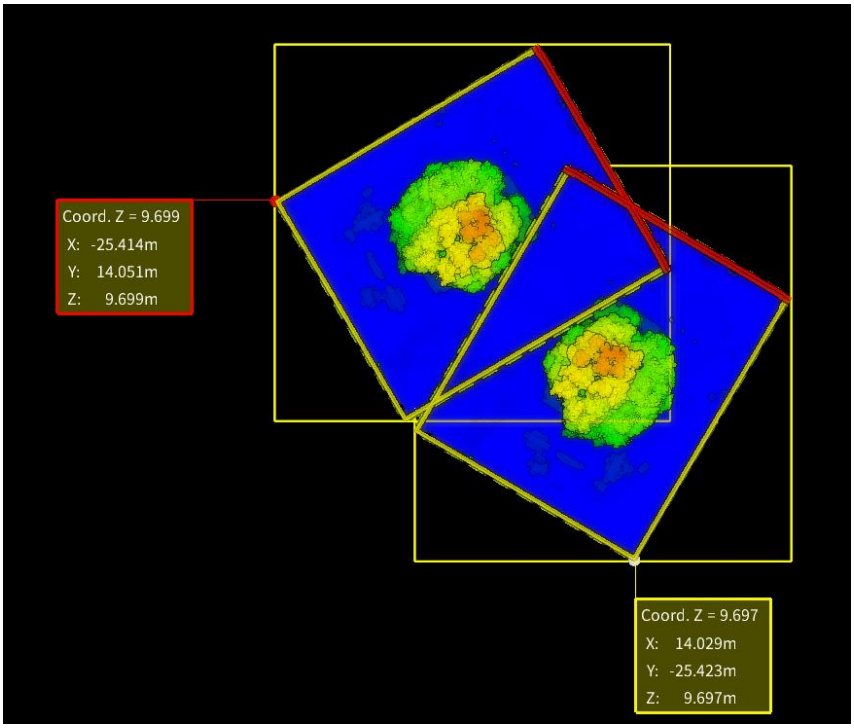



**Figure 108.** Changes before and after transpose

## 5.2.6 Delete Outliers

**Description:**

As the scanner generally generates point clouds of different densities, and there are unexpected points in the point clouds due to various external factors, to ensure accuracy, the outliers need to be filtered out. This function removes the outliers based on the statistical results.

**Operation procedure:**

1. Select a file under **Objects**.
2. Click , and the following window appears. Set the parameters, and click **OK**.

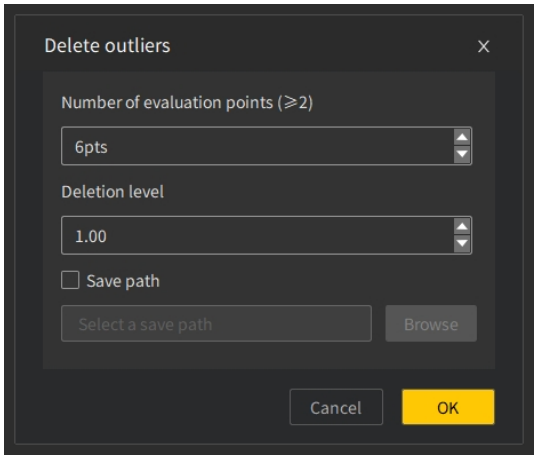



**Figure 109.** Outlier deletion screen

3. The following figures show the effects before and after outliers are deleted.

Note: The number of outliers that are deleted is not directly related to the number of evaluation points. The higher the deletion level, the more the outliers deleted.

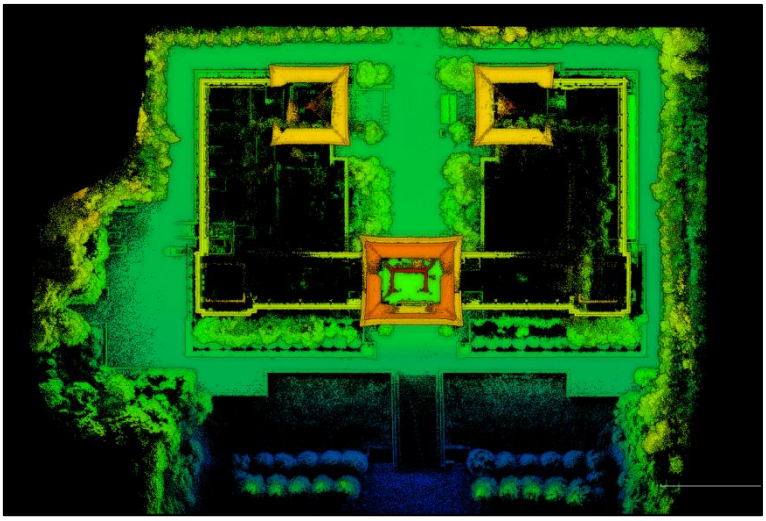


**Figure 110.** Before outliers are deleted

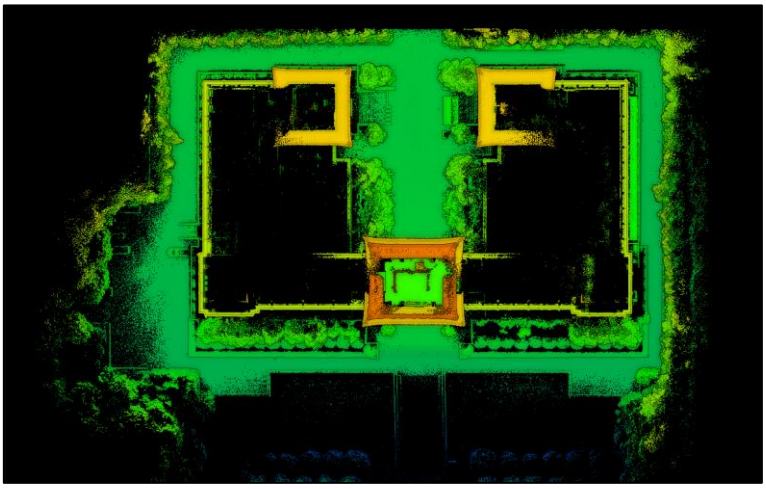


**Figure 111.** After outliers are deleted

### 5.2.7 Normal Calculation

**Description:**

Computes and writes normal vectors for each point in the point cloud, facilitating downstream modeling tasks in other software platforms.

**Operation procedure:**

1. In the project file list, select a single point cloud dataset. The  button will then become active. Click it to open the parameter settings window.
2. Upon opening the settings window, recommended values will be automatically generated based on the point cloud data. You may also manually adjust the parameters. The valid range is **[0.01, 10]**.

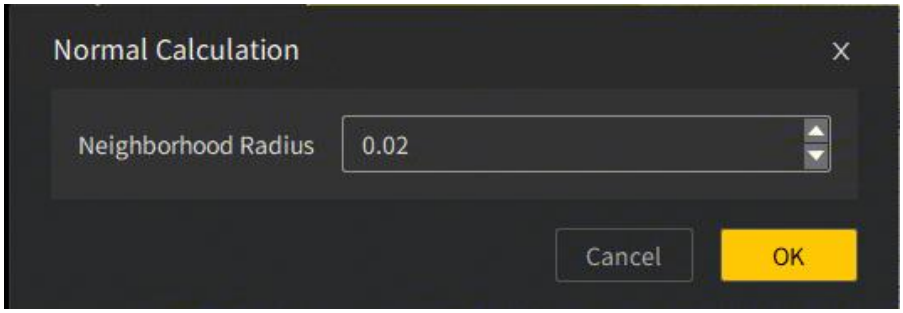



**Figure 112.** Setting window

3. After completing the parameter input, click OK to enter the calculation process. Once complete, the results will be directly updated in the current point cloud dataset.

### 5.2.8 Density Calculation

**Description:**

Calculates the spatial density distribution of the point cloud.

**Operation procedure:**

1. After selecting a single point cloud data in the project file list, the density calculation function button  becomes available. Click it to open the density calculation window.

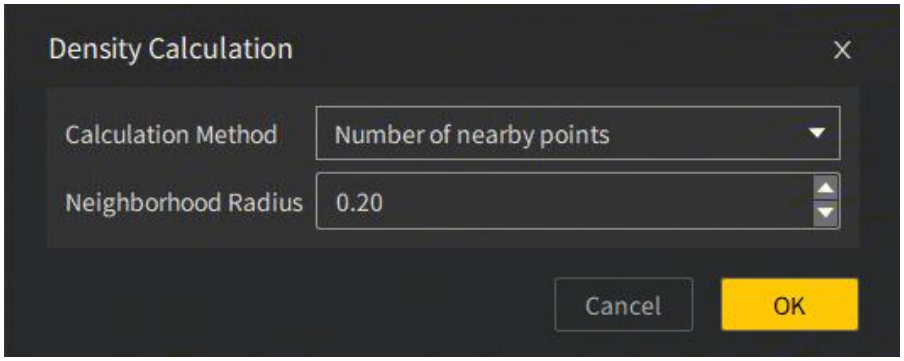



**Figure 113.** Density calculation window

2. The default calculation method is **Number of Neighboring Points within a Radius**, but you can also choose **Surface Density** or **Volume Density**.
   The default neighborhood radius is set to **0.2**, and can be manually adjusted within the range **[0.01, 10]**.
3. After setting the parameters, click **OK** to start the calculation. Once completed, the updated point cloud with density information will be displayed directly.

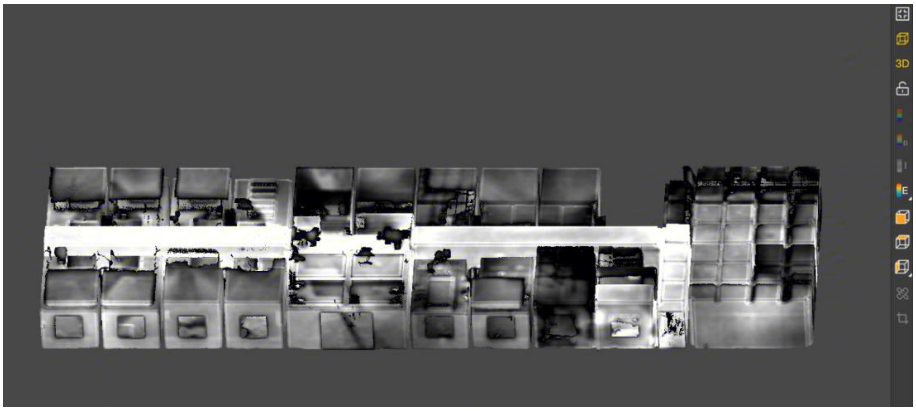


**Figure 114.** Density calculation result

# 5.3 Registration

## 5.3.1 Continue scanning for splicing

**Description:**

Registration for different projects of point cloud data.

**Operation procedure:**

1. In the project file list, click Ctrl + left mouse button to select multiple point cloud data to be spliced, and activate the continue scanning for splicing function icon.
2. Open the function window and select the configuration file accordingly.

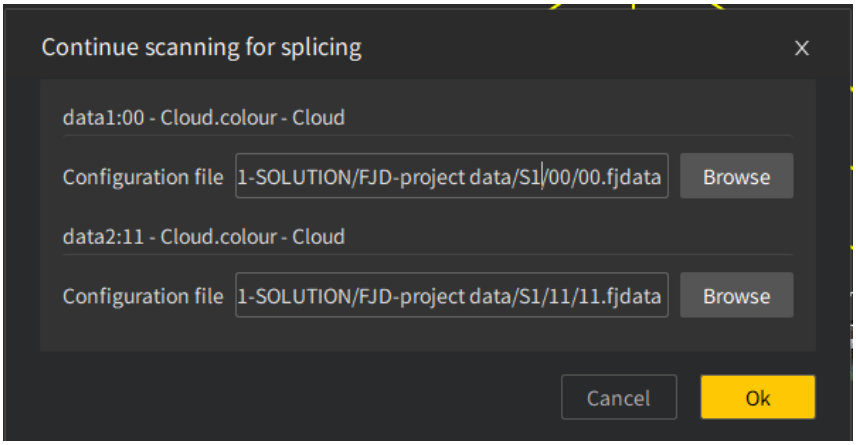



**Figure 115.** Continue scanning for splicing

3. After selecting the data, click "OK" to enter the point cloud splicing process.
4. When the processing finished, all point cloud data participating in the calculation is displayed by default.
5. When the total number of participating splicing point clouds exceeds 200 million point clouds, a warning pop-up window will be displayed with "Too many participating splicing points may cause splicing failure. Please confirm whether to continue!" The default option is "No".
6. Select "Yes" to perform the splicing calculation according to the normal process, and choose "No" to cancel the calculation.

## 5.3.2 Point Cloud Registration

**Description:**

The scanning of a large-scale scenario is often divided into multiple projects, generating separate point clouds. To obtain the complete point cloud data, you need to register the point clouds using the feature points in the overlapping area, the spherical targets, and the planar targets. Registration by coordinates and selected area are supported.

**Operation procedure:**

1. Hold down the Ctrl key and select the two point clouds to be registered under **Objects**. Click to open the **Point Cloud Registration** window.
2. View 1 loads the first selected point cloud data, view 2 displays another point cloud data, and the result view updates the registration results in real time.

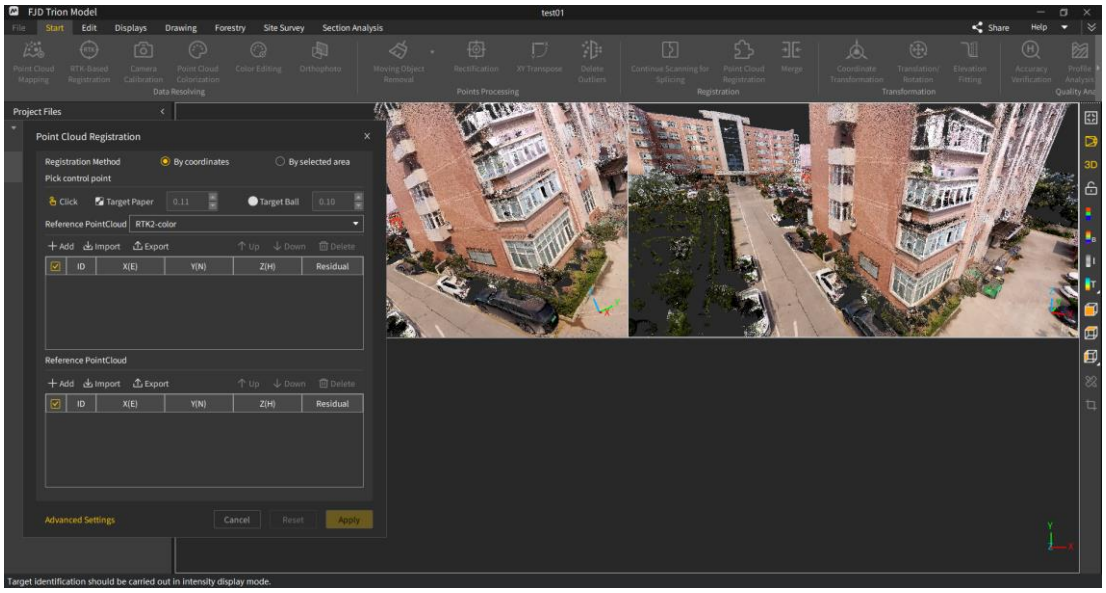



**Figure 116.** Point cloud registration window

3. Select By coordinates. Select the corresponding point pair in point cloud 1 and point cloud 2 respectively. When the number of selected point pairs is less than 3, only the selection point tag style will be displayed on the point cloud view.
4. Select a way to pick control points. Control points, picked by clicking by default, can also be picked by identifying the center of the spherical target or identifying the center of the planar target. Control points can also be imported in a .txt, .csv, .xls, or .xlsx file. Operation procedures for the three picking methods are as follows:

(a) By clicking

- Move the cursor to the view area, and the cursor changes into a hand cursor.
- Pick control points in the source point cloud, and they are shown in the list with their coordinates and numbered A0, A1, A2, and so on. Corresponding control points picked in the target point cloud are numbered R0, R1, R2, and so on.

(b) By identifying the center of the planar target

- Move the cursor to the view area, and the cursor changes into a hand cursor. Click the center of the planar target.
- If the center of the planar target is identified, the control point list is updated accordingly.
- If the center of the planar target is not identified, the cursor changes into a hand cursor again.

(c) By identifying the center of the spherical target

- The procedure is the same as that for identifying the center of the planar target.

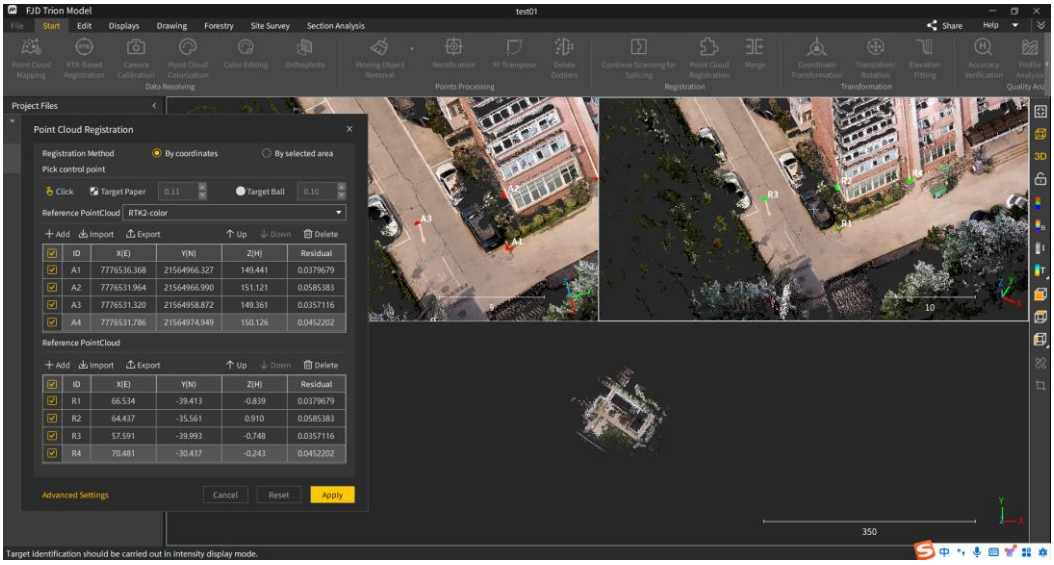



**Figure 117.** Picking points in the source point cloud

5. Click **Apply** to complete the point cloud registration.

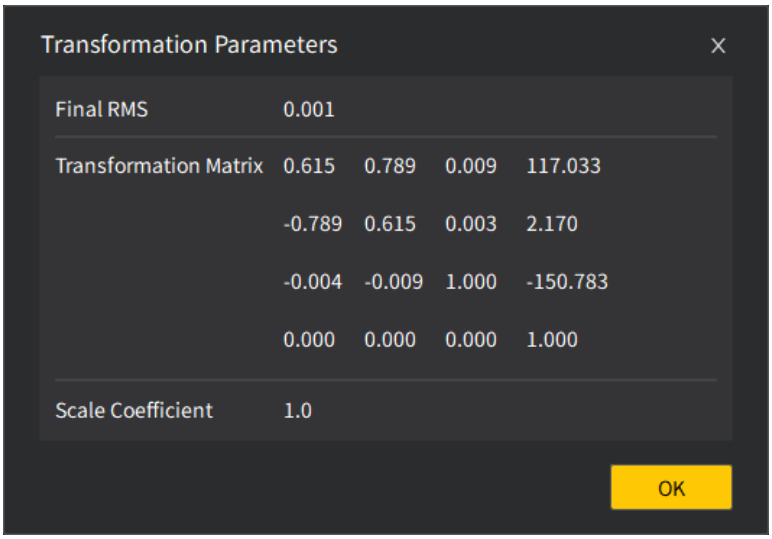



**Figure 118.** Transformation parameters

6. If the preview effect is not satisfactory, click **Reset** to adjust the control point coordinates or delete control points, and register again. When you delete a control point in the source point cloud, it prompts "Do you want to delete the equivalent reference point as well?". When you delete a control point in the target point cloud, it prompts "Do you want to delete the equivalent aligned point as well?". You can decide according to your need.

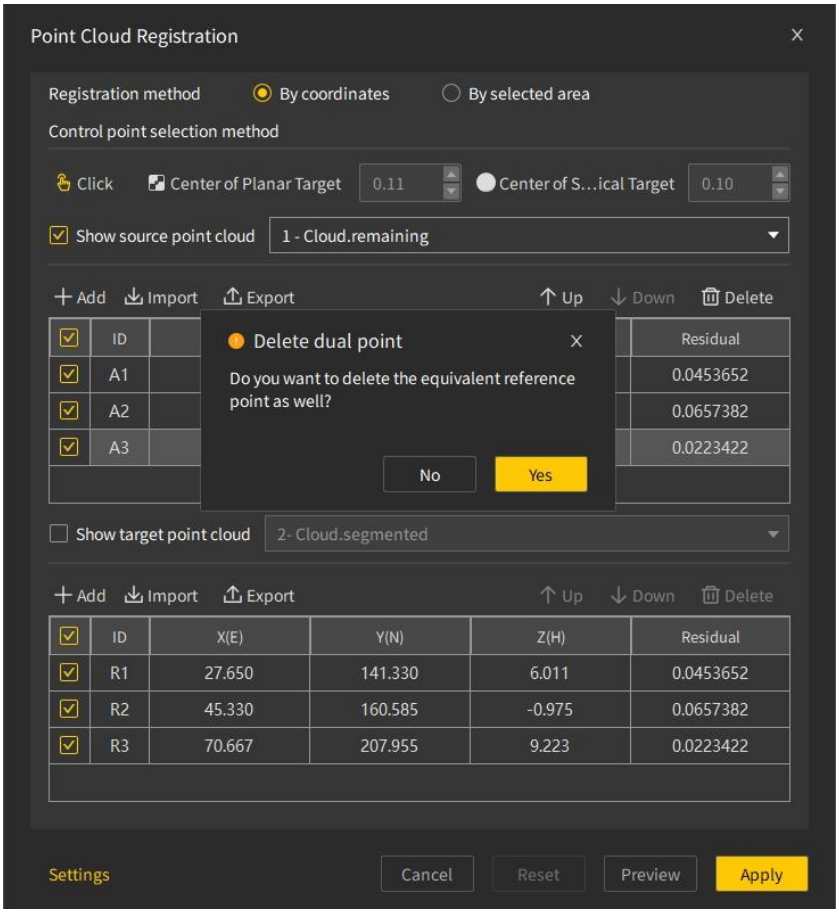



**Figure 119.** Prompt for deleting the control point in the source point cloud

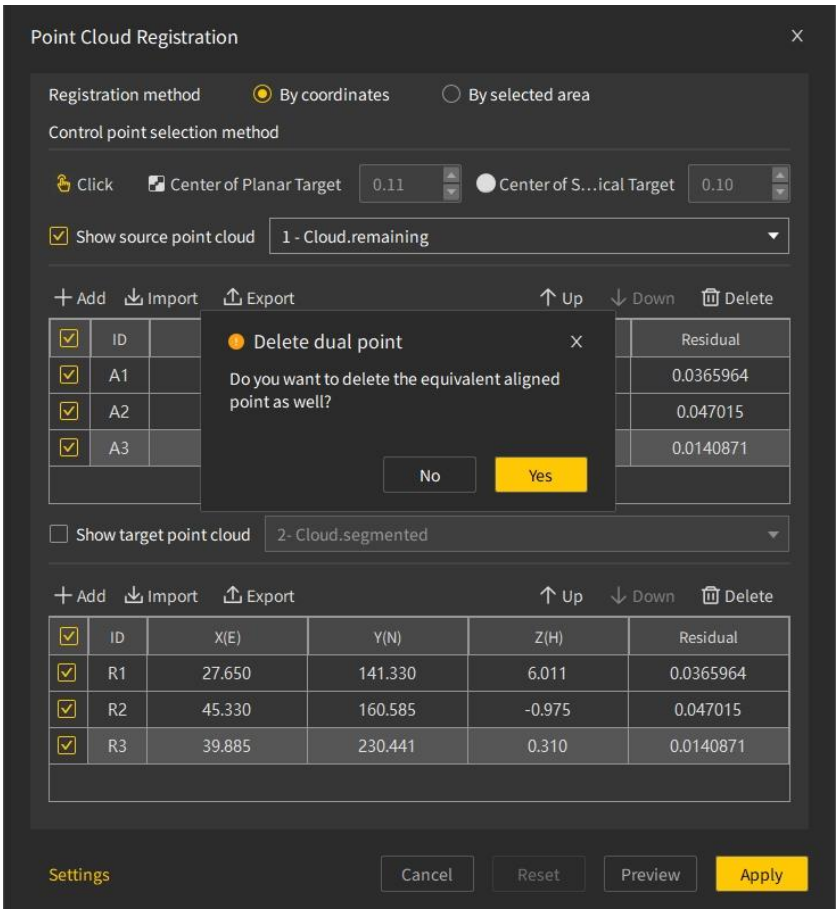



**Figure 120.** Prompt for deleting the control point in the target point cloud

7. Select **By selected area**. Select the point cloud to show, and select the public point cloud part with the default rectangular selection tool. After completing the area selection in point cloud 1

and point cloud 2, the preview calculation is automatically performed, and the registration preview effect is updated in the result view.

8. After the selected area is updated, the point cloud registration preview result will be updated in the result view;
9. Click "Reset" to cancel the effects of all selected points, clear the label, and cancel the registration preview effect;
10. When the preview effect meets the requirements, the user can click the "Apply" button to apply the registration result, and close the view main interface to update the registration effect;
11. Set the registration parameters.

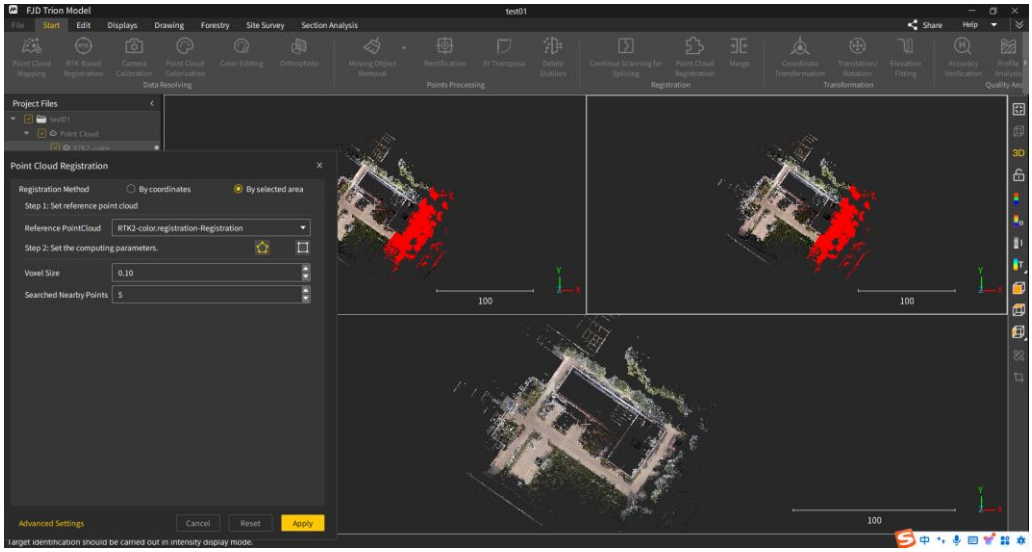



**Figure 121.** Select the public point cloud part

12. Click **Preview** to check the registration result. If it is unsatisfactory, click **Reset** to select another public point cloud part.
13. If the result is satisfactory, click **Apply** to apply the result.

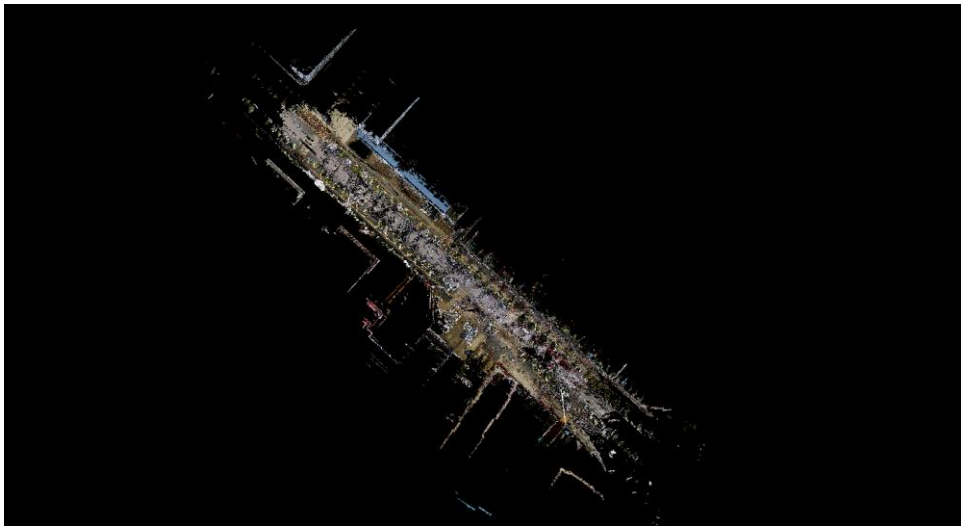


**Figure 122.** Registration result

14. When calculation is completed, the transformation parameters are shown. Click **OK** to close the registration and parameter windows.
15. The lower left corner of the registration interface supports fine registration parameter adjustment. Users can choose to enable or disable ICP registration according to the situation of the point cloud. The smaller the iteration threshold, the higher the registration accuracy. After starting the size scaling, to ensure the registration accuracy, the origin cloud will be scaled to a certain extent.

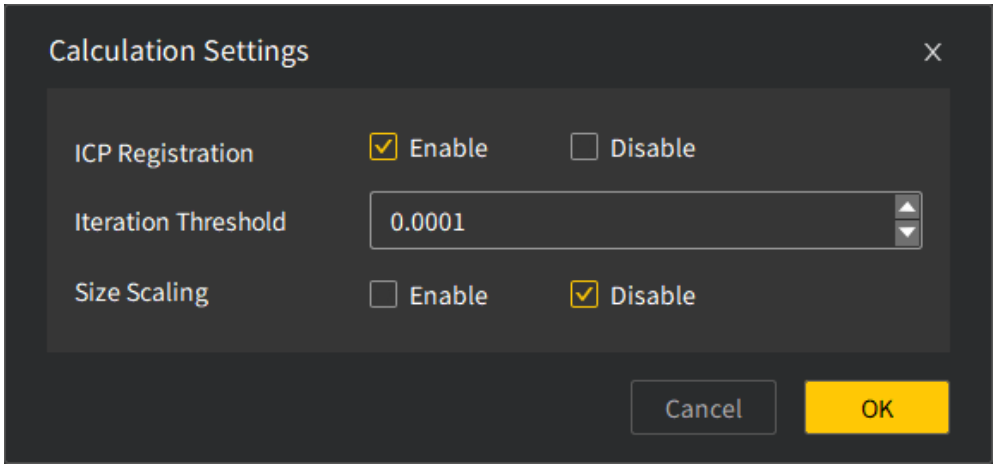



**Figure 123.** Calculation Settings

### 5.3.3 Splicing Optimization

**Description:**

Optimize the stitching of the point cloud after normal alignment to improve the stitching accuracy of the point cloud.

**Operation procedure:**

1. Open the point cloud data to be stitched, and in the project file list, use Ctrl+ Left mouse button to select at least 2 opened point cloud data.
2. Click, a pop-up window will appear as shown in the figure below, and you need to import configuration file if there was a point cloud missing it.

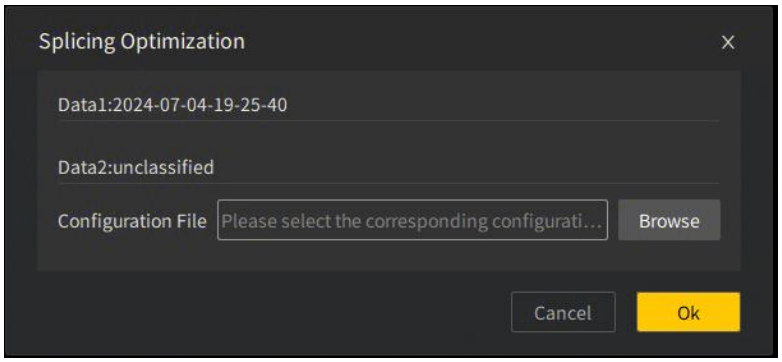



**Figure 124.** Splicing Optimization Popup 1

3. Or click without selecting any point cloud, and the following pop-up window will appear to add the point cloud and the configuration file to be stitched locally.

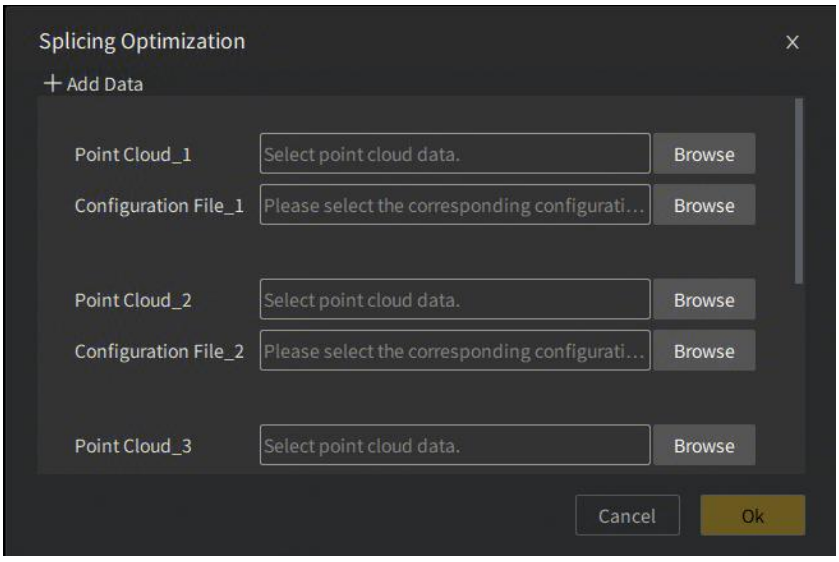



**Figure 125.** Splicing Optimization Popup 2

4. After selecting the point cloud data and clicking **OK**, the stitching calculation will start.

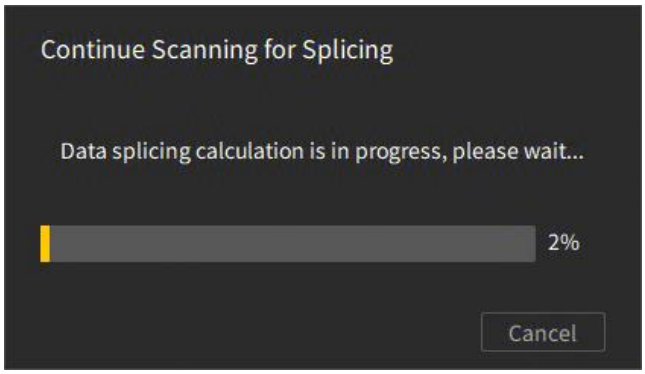



**Figure 126.** Splicing Optimization Calculating

### 5.3.4 Merge

**Description:**

Merges two or more point clouds into one.

**Operation procedure:**

1. Open the two point clouds to be merged under **Objects**, as shown below.

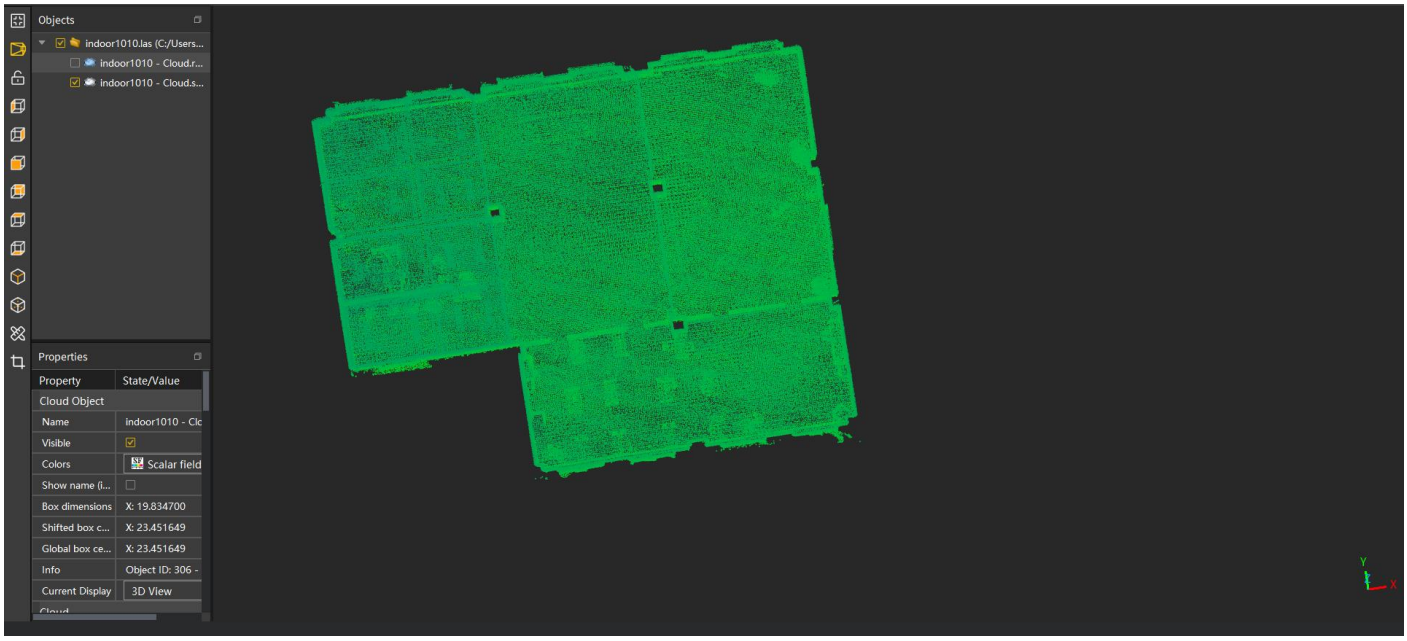



**Figure 127.** Point cloud 1

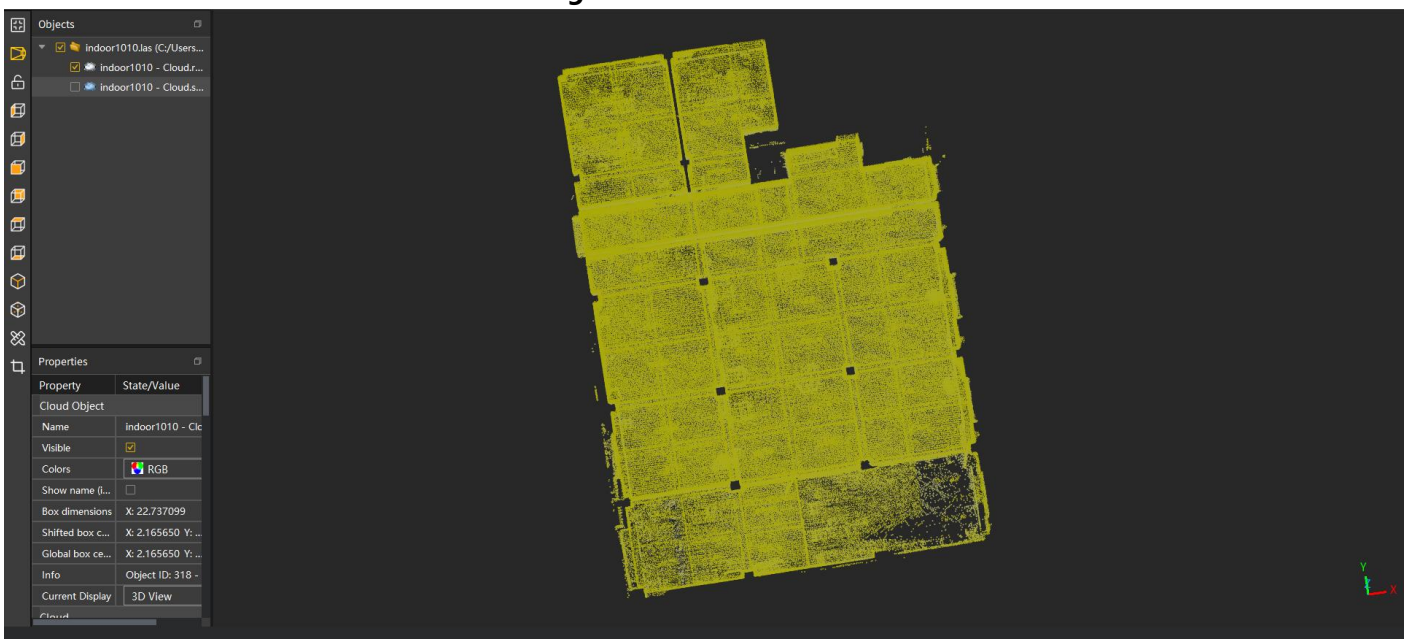


**Figure 128.** Point cloud 2

2. Hold down the Ctrl key and select the two point clouds.
3. Click , and the following window appears. Click **OK** to merge the clouds. The merging effect is as shown below.

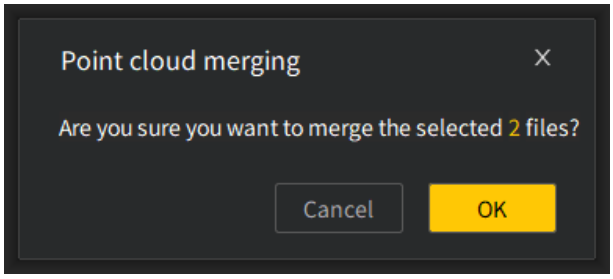



**Figure 129.** Merging confirmation window

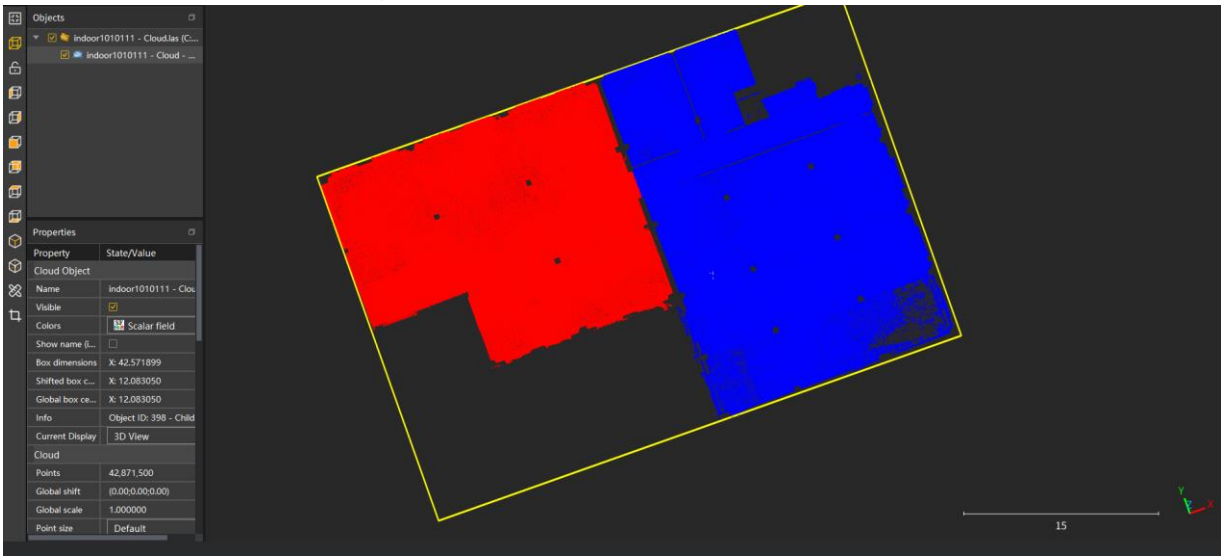



**Figure 130.** Point clouds merged

# 5.4 Transformation

## 5.4.1 Coordinate Transformation

Coordinate transformation refers to the process of transforming the coordinate system of a point cloud to another coordinate system. To process and analyze point clouds, you need to transform the original coordinate system of a point cloud, which may be the scanner coordinate system, to a world coordinate system, for example, the GPS coordinate system.

**Description:**

Transforms from a source coordinate system to a target coordinate system.

**Operation procedure:**

1. Select a file under **Objects**, and click to open the **Coordinate transformation** window. It supports both unknown and known matrix transformation and the order of points can be automatically corrected.
2. Select **Unknown matrix**, and calculate the matrix through selected points.

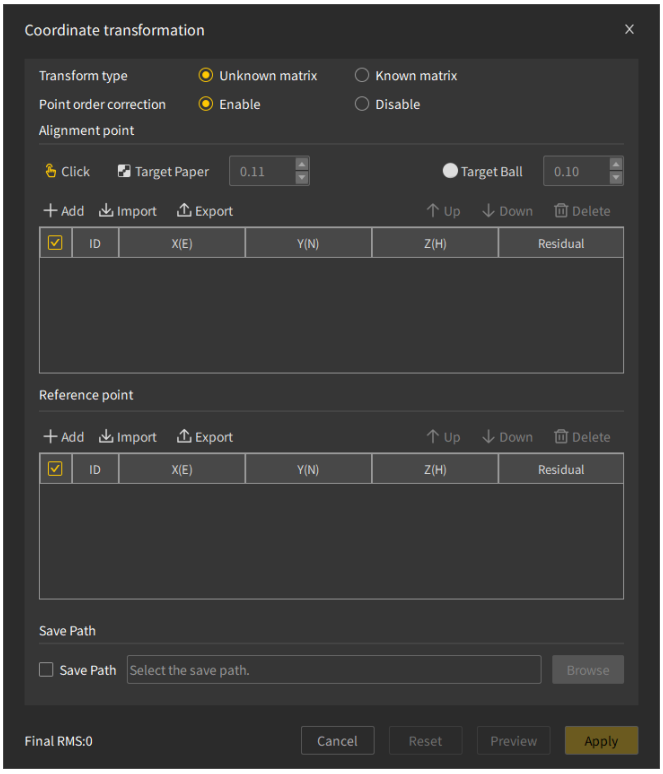



**Figure 131.** Coordinate transformation window

Pick the alignment points by clicking, identifying the center of the paper target, or identifying the center of the ball target. Refer to **7.3.1 Point Cloud Registration** for details of the three picking methods.

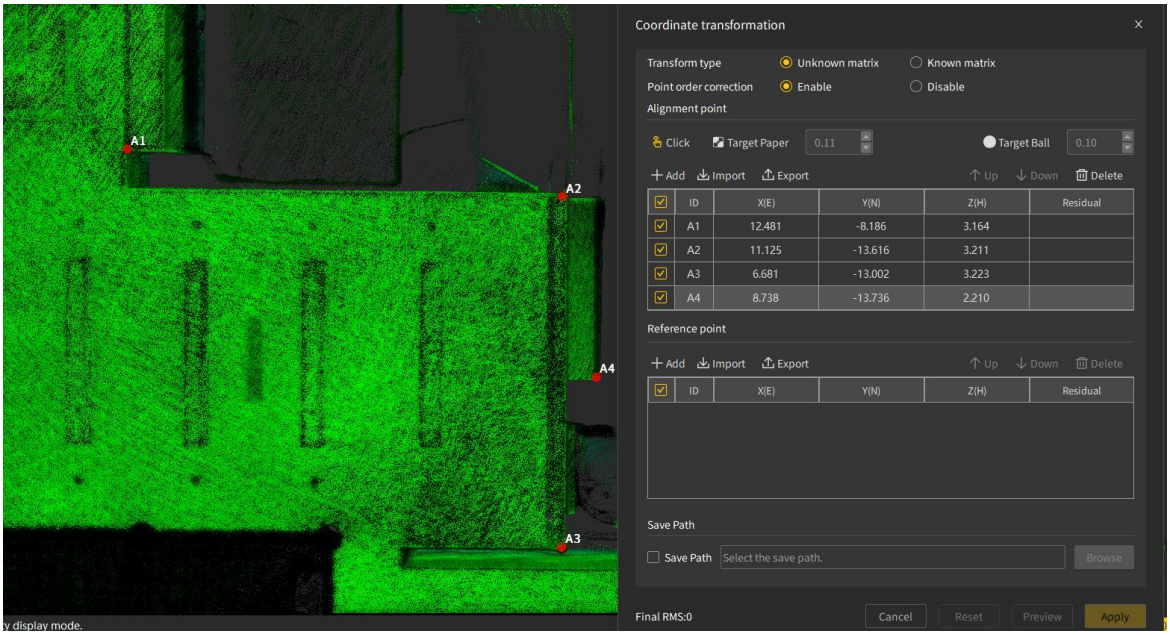



**Figure 132.** Control points picked by clicking

3. Enter the actual coordinates of the picked points by importing a file or inputting manually. You can import the actual coordinates in a .txt, .csv, .xls, or .xlsx file. You can also click **+Add**, double-click the table, and input the coordinates.

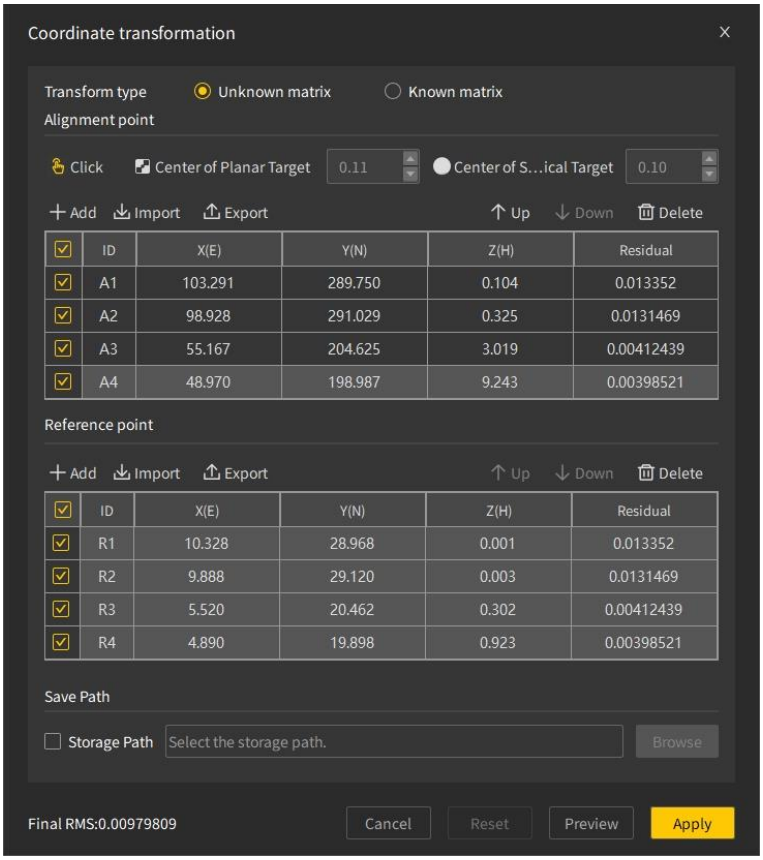



**Figure 133.** Entering the actual coordinates

4. When the number of the sets of actual coordinates is the same as the number of the alignment points and the number is greater than three, the residuals and RMS are calculated automatically. You can modify the alignment points and the actual coordinates and click **Preview** to refresh the display.
5. Click **Apply**. Transformation matrix parameters are displayed after the coordinate transformation is completed. Click **Save** to export the parameters.

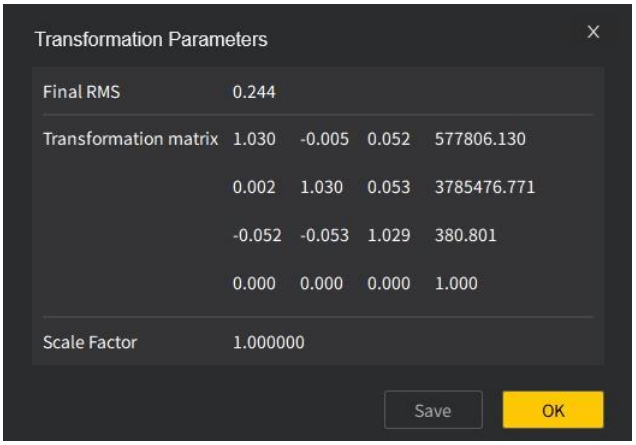



**Figure 134.** Transformation parameters

6. When the target coordinate system is the same as that in the last operation, select **Known matrix**, and click **Import** to import matrix parameters. Click **Preview** to view the transformation result, and click **Apply** to confirm the transformation.

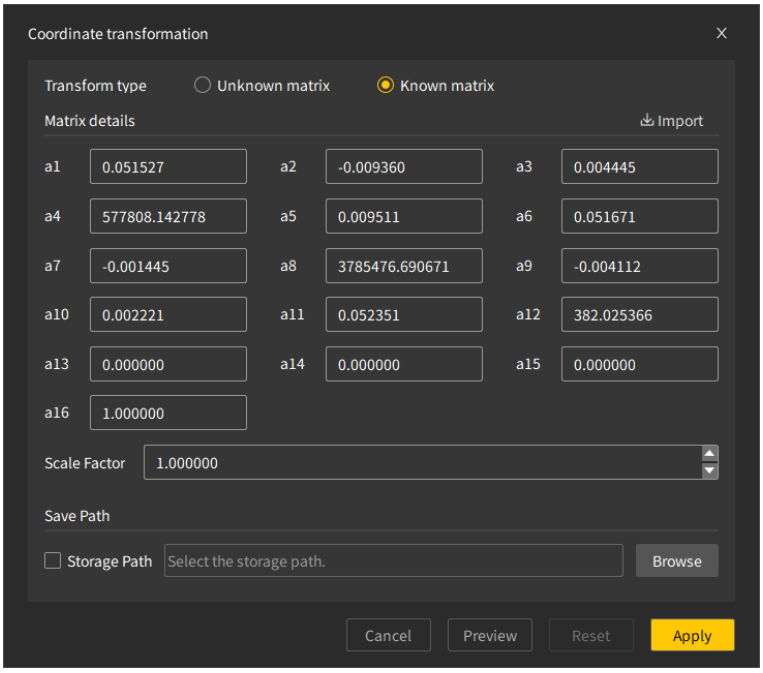



**Figure 135.** Known matrix transformation

7. If the transformation is not satisfactory, click **Reset** to adjust or delete the alignment points and the actual coordinates, and transform again. When you delete an alignment point, it prompts "Do you want to delete the equivalent reference point as well?". When you delete the actual coordinates, it prompts "Do you want to delete the equivalent aligned point as well?". You can decide according to your need.

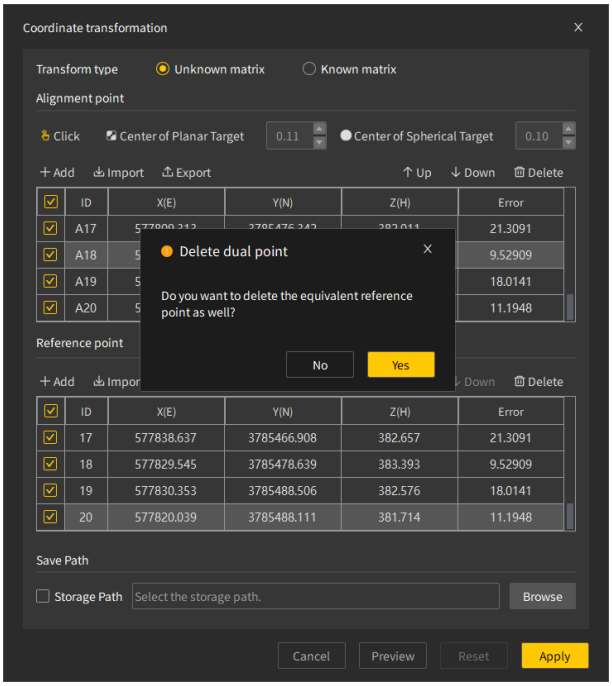



**Figure 136.** Prompt for deleting an alignment point

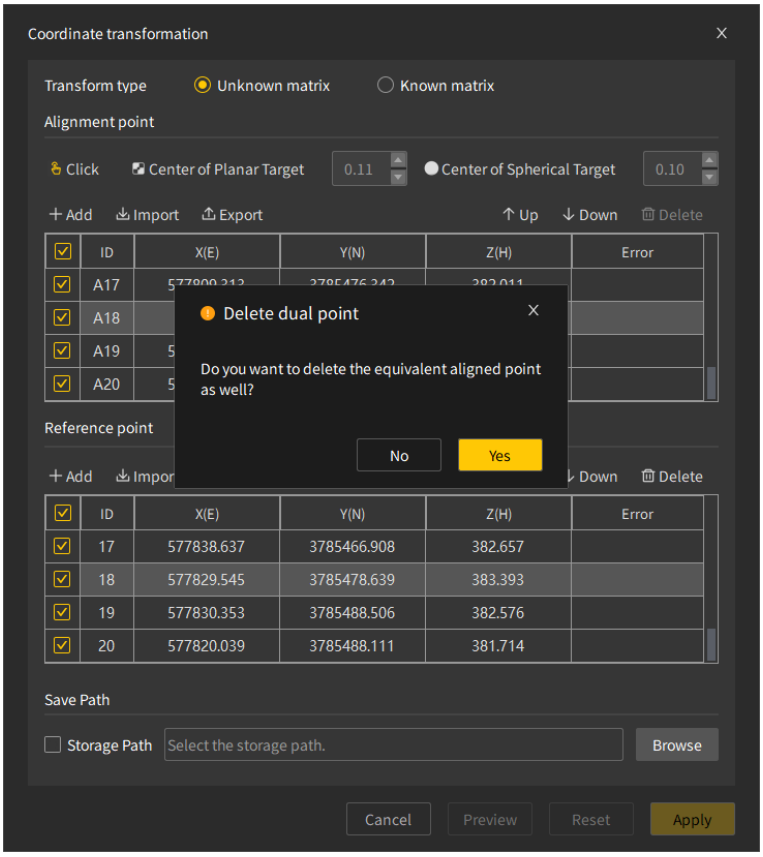



**Figure 137.** Prompt for deleting the actual coordinates

**Note:**

Select coordinate point pairs with a smaller residual to get better transformation results.

Points picked from a point cloud are hidden when the point cloud is hidden.

When point sequence correction is enabled, the system will automatically adjust the corresponding order of points based on the calculated point error. If you need to manually adjust the point sequence, you need to turn off point sequence correction.

### 5.4.2 Translation/Rotation

**Description:**

Rotates or translates a point cloud in any way, or rotates or translates the point cloud along an axis by a set value. After performing the operation, the coordinates of the point cloud will change accordingly.

**Operation procedure for rotation:**

1. Select a file under **Objects** to activate the **Translation/Rotation** function .
2. Press the left mouse button and drag to rotate the point cloud.

Note: Five rotation modes are available. If you select X, Y, or Z, you can press the left mouse button and drag to rotate the point cloud around the X axis, Y axis, or Z axis respectively. If you select XYZ, you can rotate the point cloud without restrictions. If you select None, the point cloud cannot be rotated in any direction.

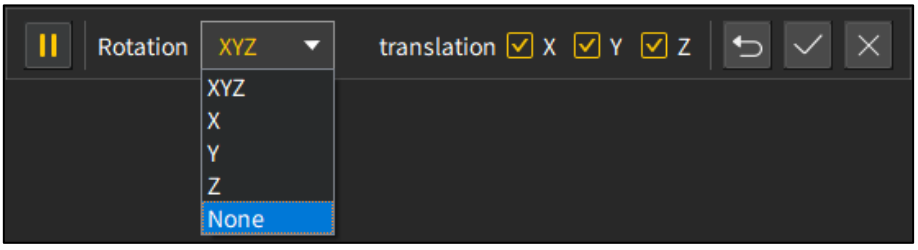



**Figure 138.** Selecting a rotation mode

3. Click ⏸ to pause the translation/rotation and enter the preview mode, and the icon changes into ▶. Click ▶ to resume the translation/rotation. Note: In the preview mode, you can translate and rotate the point cloud without restrictions.

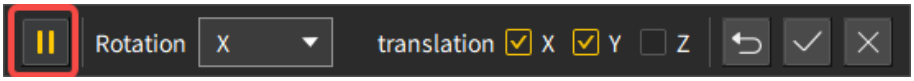



**Figure 139.** Pausing the translation/rotation

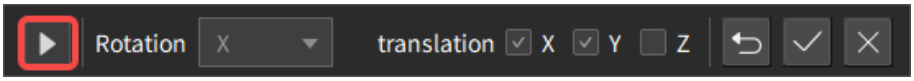



**Figure 140.** Resuming the translation/rotation

4. Click ✓ to apply the rotation, or click ✕ to exit the translation/rotation mode and restore the initial state. If the rotation is not satisfactory, click ↩ to restore the initial state and rotate again.

**Operation procedure for translation:**

1. Select a file under **Objects** to activate the **Translation/Rotation** function ✥. Press the right mouse button and drag to translate the point cloud.

Note: Translate the point cloud along the selected one or more axes. For example, if you select **X**, you can press the right mouse button and drag to translate the point cloud along the **X** axis.


**Figure 141.** Selecting a translation mode

2. Click ⏸ to pause the translation/rotation and enter the preview mode, and the icon changes into ▶. Click ▶ to resume the translation/rotation. Note: In the preview mode, you can translate and rotate the point cloud without restrictions.

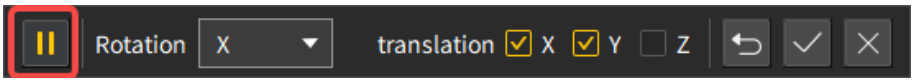



**Figure 142.** Pausing the translation/rotation

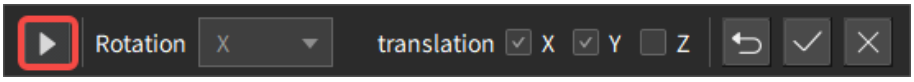



**Figure 143.** Resuming the translation/rotation

3. To translate or rotate more accurately: select the coordinate axis along which the rotation or translation takes place, enter the rotation angle or translation distance, and click ↻ (clockwise rotation), ↺ (anticlockwise rotation), + (positive translation), or − (negative translation).

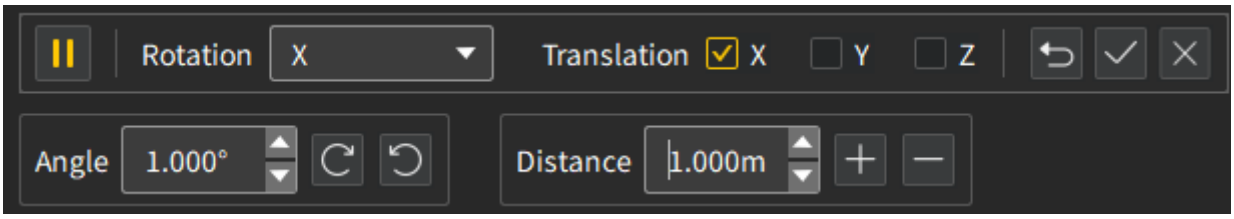



**Figure 144.** Accurate translation/rotation

4. Click ✓ to apply the translation, or click ✕ to exit the translation/rotation mode and restore the initial state. If the translation is not satisfactory, click ↩ to restore the initial state and translate again.

### 5.4.3 Elevation Fitting

**Description:**

Transforms the geodetic datum height to the normal height in a local elevation coordinate system for easy understanding.

**Operation procedure**

1. Select a file under **Objects**, and click . Fitting with unknown and known parameters are supported.
2. When parameters are unknown, select **Unknown parameters**, set **Fit Method** to **Fixed difference**, **Plane fitting**, **Quadric surface fitting**, or **Cubic surface fitting**, import control point coordinates in a .txt, .xlxs, or .csv file or manually add them, and then set the data storage location. Coordinates are listed in the X(E), Y(N), Z(H), and Z(h) columns.

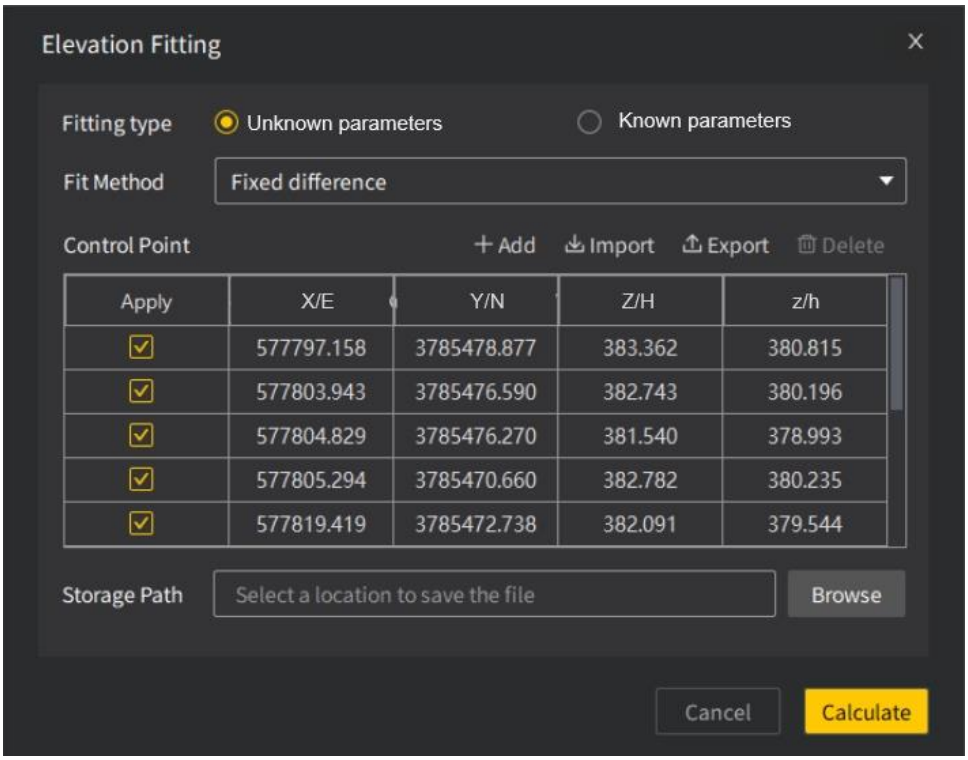



**Figure 145.** Fitting with unknown parameters

3. When parameters are known, select **Known parameters**, import parameters, and then set the data storage location.

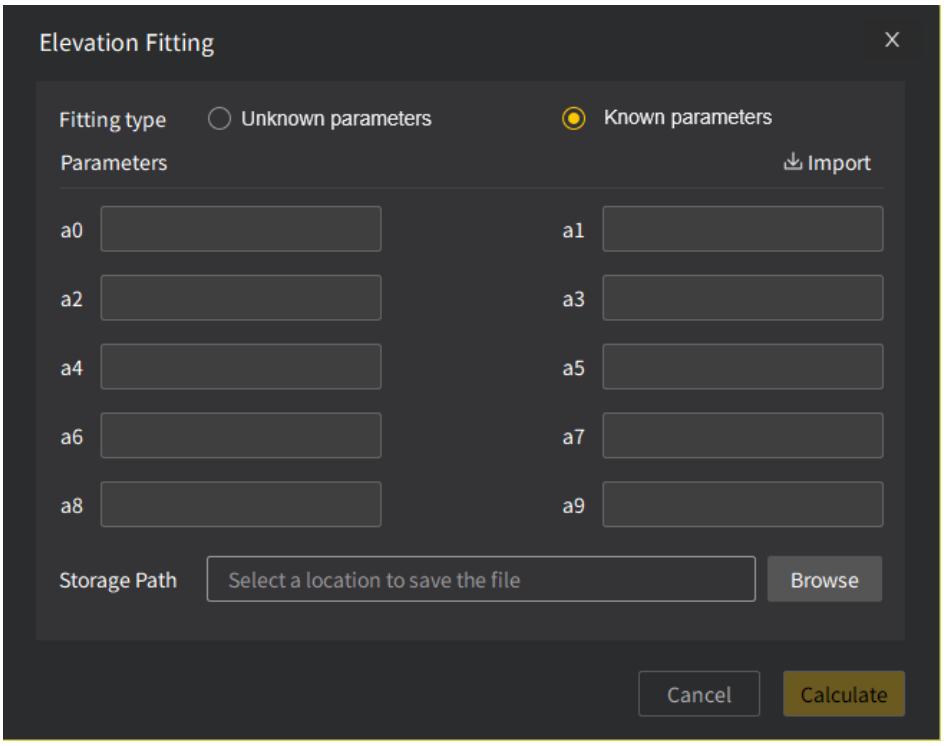



**Figure 146.** Fitting with known parameters

4. Click **Calculate**, and the calculation progress is displayed. When the calculation is completed, click **Save** to save parameters into a file named *original point cloud file name*_fitted_*system time*, and click **OK** to close the window.

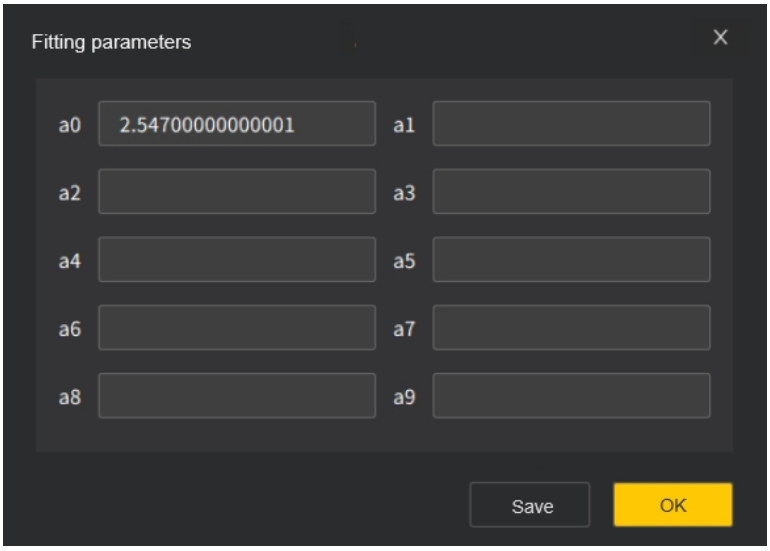



**Figure 147.** Fitting parameters

5. Fitting parameters are applied automatically after the calculation. A point cloud file named *original point cloud file name*.fitted appears under **Objects**. Only the point cloud formed after elevation fitting is automatically loaded and displayed in the 3D view area.

# 5.5 Quality Analysis

## 5.5.1 Accuracy Verification

**Description:**

Compares known coordinates of control points with scanned coordinates of such points to obtain the elevation and plane accuracy of the scanning.

**Elevation accuracy verification procedure:**

1. Click the **Start** tab and select **Accuracy Verification**.
2. Select **Elevation** as the comparison type and import control points in a .txt, .csv, .xls, or .xlsx file, which lists the data in the ID, X(E), Y(N), and Z(H) columns.

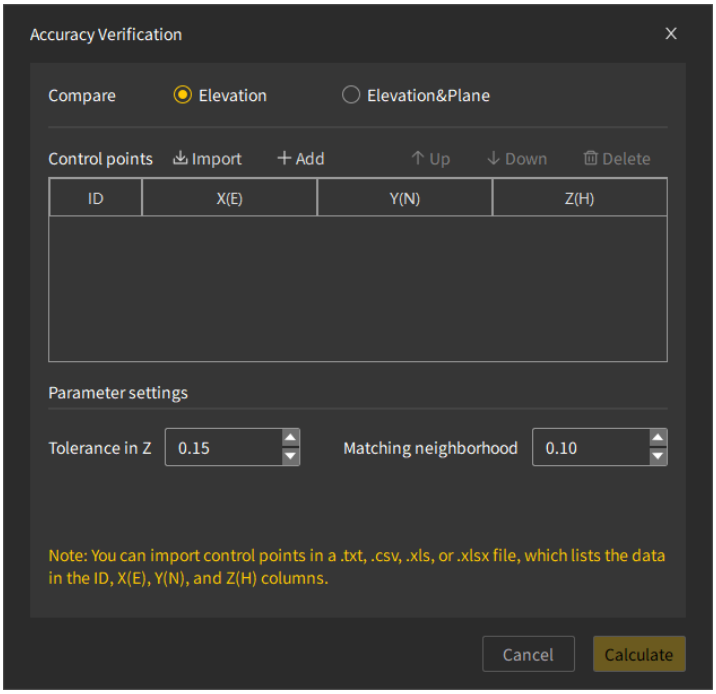



**Figure 148.** Elevation accuracy verification window

3. Select a control point row and click **Up**, **Down**, or **Delete** to move it up or down or delete it.
4. Set the parameters, and click **Calculate**. When the calculation is completed, the accuracy verification window disappears, and the **Reports** window appears.

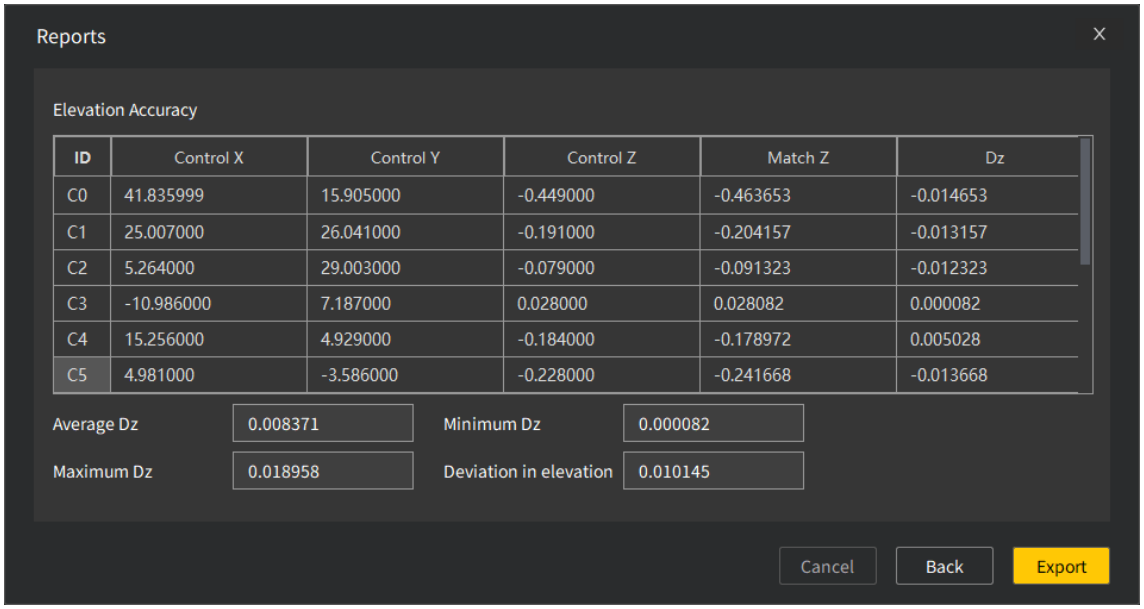



**Figure 149.** Elevation accuracy verification result window

5. Click **Export** to export the verification result into a .txt or .xlsx file.

**Elevation and plane accuracy verification procedure:**

1. Click the **Start** tab and select **Accuracy Verification**.
2. Select **Elevation&Plane** as the comparison type and import a control point file. Select the points corresponding to the control points from the point cloud. The scanned coordinates of the selected points are shown in the **Pick point information** table. Select a row and click **Up**, **Down**, or **Delete** to move it up or down or delete it.

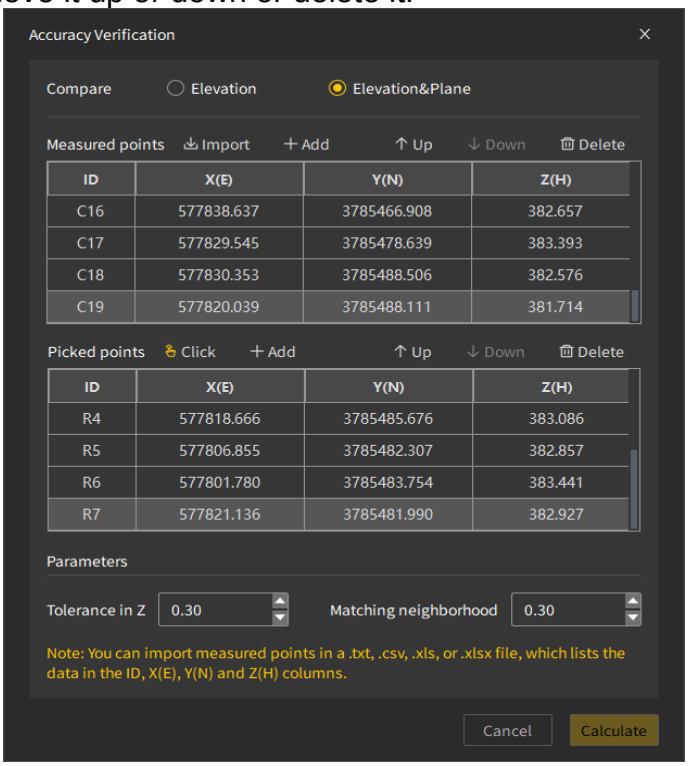



**Figure 150.** Elevation & plane accuracy verification window

3. Click any control point row and click **Delete** to delete it.
4. Set the parameters, and click **Calculate**. When the calculation is completed, the accuracy verification window disappears, and the **Reports** window appears. Click **Elevation Accuracy**

or **Plane Accuracy** to show different verification results. Select **Ds**, **X**, or **Y** for **Display Deviation** to show errors in different directions.

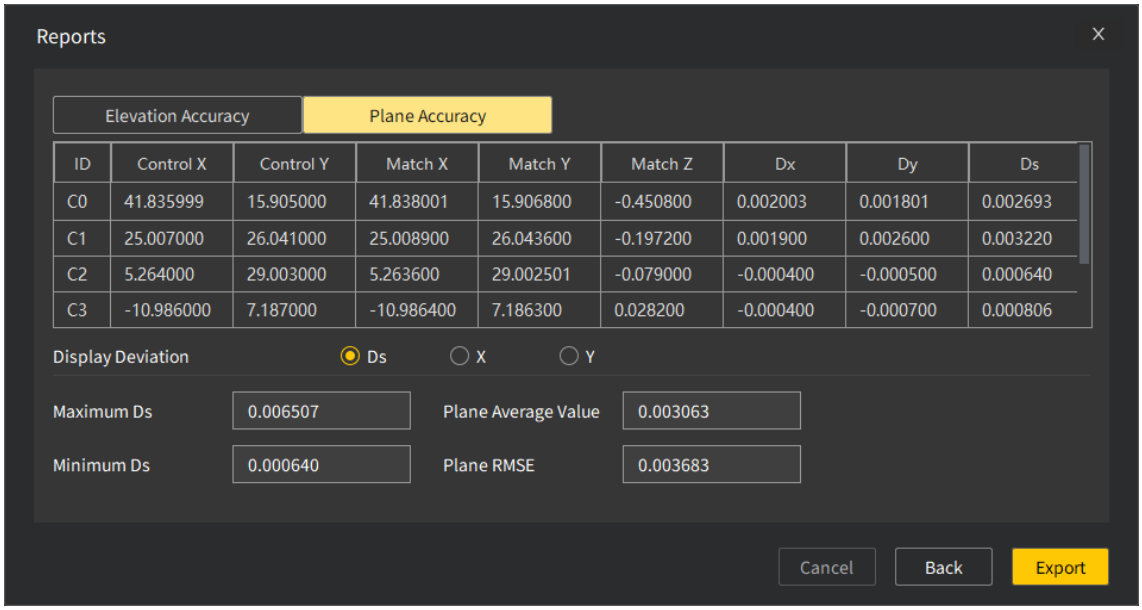



**Figure 151.** Elevation and plane accuracy verification result window

**Note:**

- The elevation accuracy verification result includes the average Dz, max. Dz, min. Dz, and elevation RMSE.
- The plane accuracy verification result includes the average Dx, max. Dx, min. Dx, average Dy, max. Dy, min. Dy, plane average value, and plane RMSE.

5. Click **Export** to export the elevation or plane accuracy verification result.

   **Note:**

   - You can drag the accuracy verification window outside the main interface, to provide more space for point cloud operations.
   - Double-click any point ID on the picked point or actual coordinate list to show the point in the center of the 3D view area.

### 5.5.2 Profile Analysis

**Description:**

Creates vertical or horizontal profiles of point clouds obtained from scanning in different views and measures the profile length, height, and slope, to evaluate the quality of single point clouds and the registration precision of multiple point clouds.

**Operation procedure:**

1. Select a point cloud file under **Objects**.
2. Click the **Start** tab, and select **Profile Analysis**.
3. Select  (arbitrary-view vertical slice),  (top-view vertical slice), or  (front-view horizontal slice).

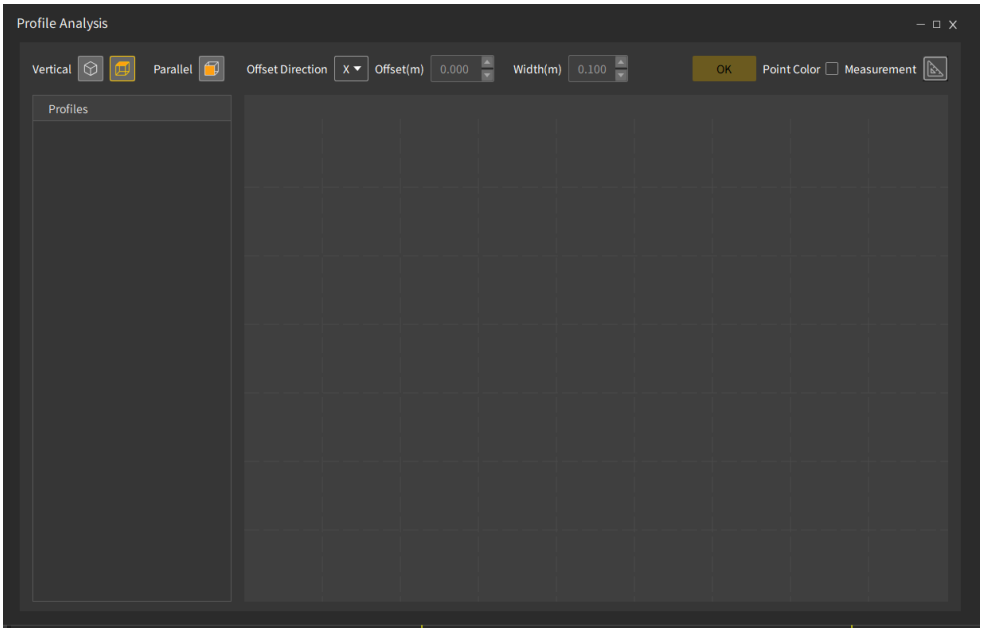



**Figure 152.** Profile analysis window

4. Click anywhere on the 3D view area to hide the profile analysis window. Then, click to select as many points as you need in the 3D view area to create a profile. Right-click to end the selection, or press the Esc key to undo all the point selection for the current profile. The following figures show the profile creating process and the profile analysis window after profile creation.

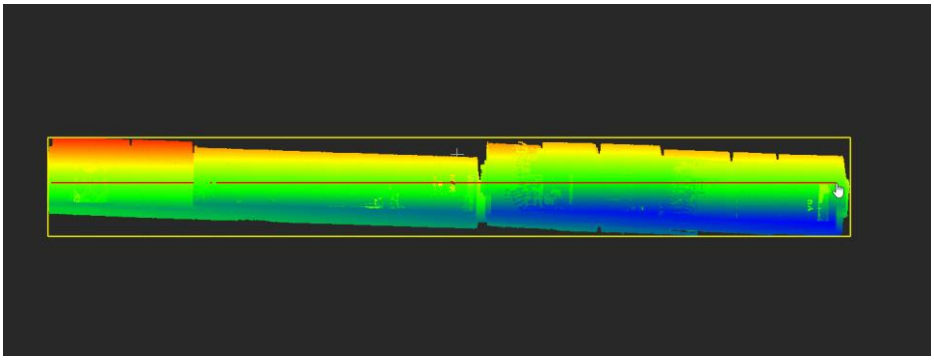


**Figure 153.** Creating a horizontal slice profile

5. Profile measurement is enabled by default when a profile is created. Click any two points in the profile view area to measure the slope, length, and height. The current measured values are shown in the upper right corner of the profile view area until the next measurement is completed.

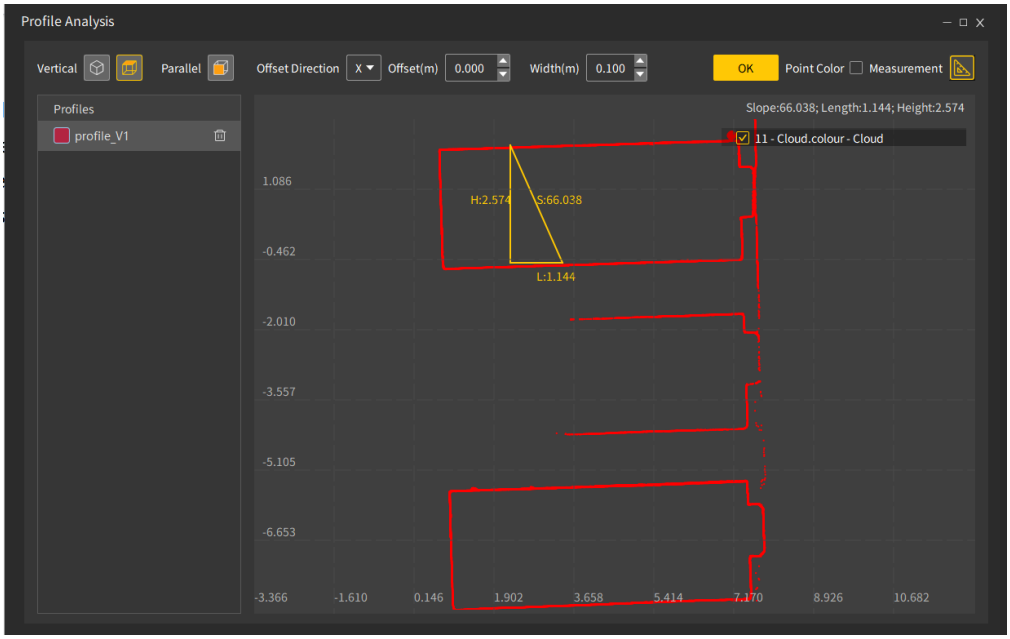



**Figure 154.** Profile measurement window

6. Adjust the **Offset(m)** value to offset a profile. When the value is 0, the profile does not offset. A positive value indicates a positive offset, while a negative value means a negative offset. The offset direction sets the moving direction of a slice. A vertical slice can offset in x or y direction, while a horizontal slice can offset in x or z direction. To modify the profile width, adjust the **Width(m)** value, or click the profile and drag its borders. The following figure shows that a profile is selected in the intensity rendering mode.

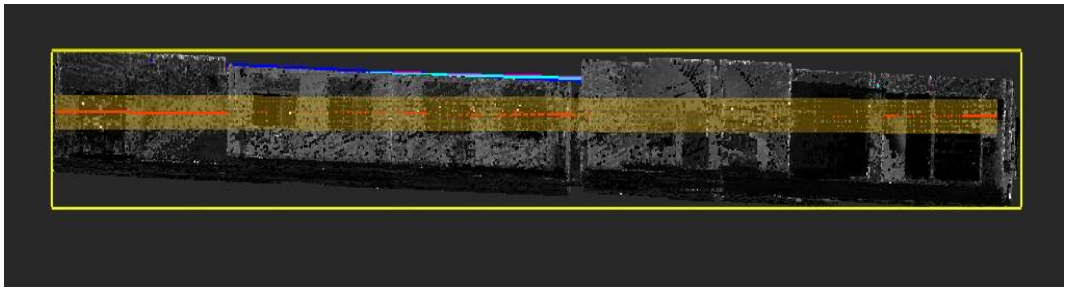


**Figure 155.** Adjusting the profile width

7. Multiple profiles can be created and color-coded. Select the required profile from the profile list.

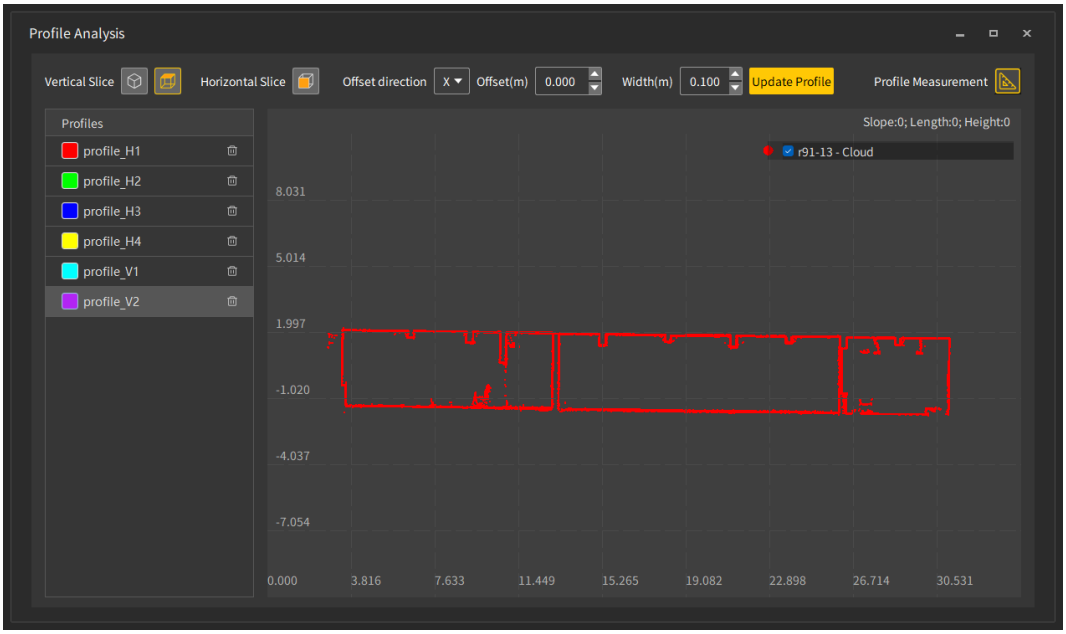


**Figure 156.** Multiple profiles created from the same point cloud

8. Profiles can be created from multiple color-coded point clouds. For example, 

indicates that the profile is created from two point clouds in red and green respectively.

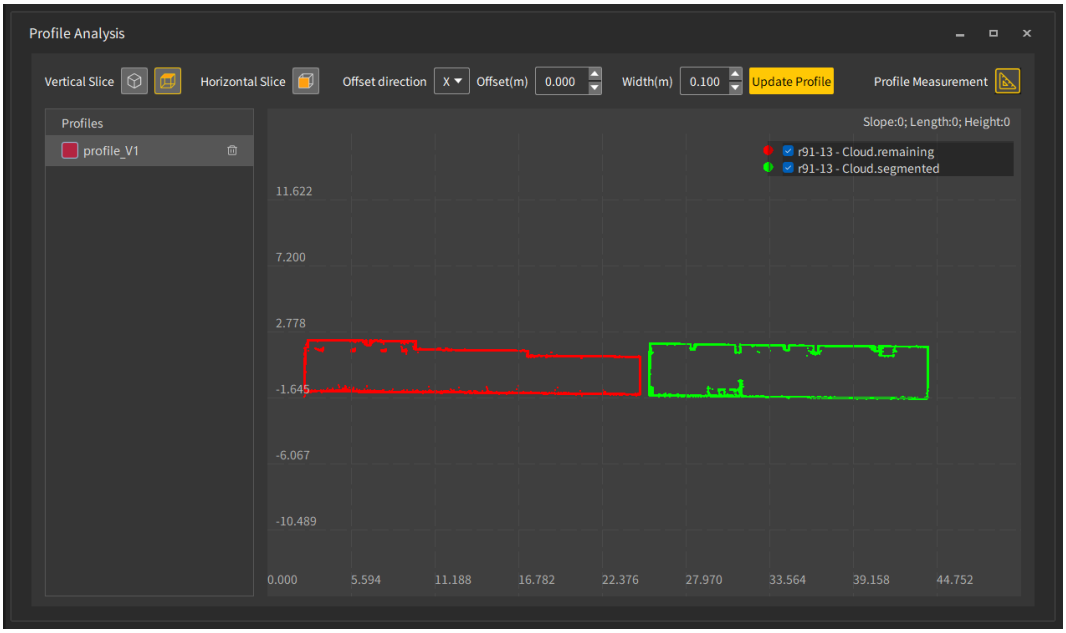



**Figure 157.** Profile created from two point clouds

**Note:**

- Profile analysis is not available when the point clouds are selected from different coordinate systems.

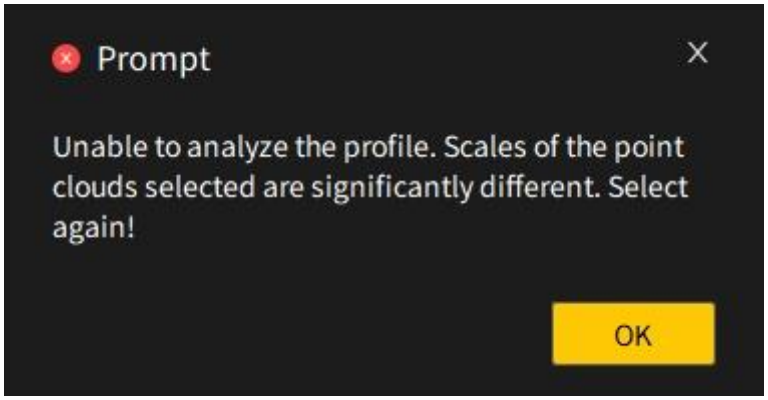



**Figure 158.** Prompt for unavailable profile analysis

- The profile can be rendered in the rendering mode applied.

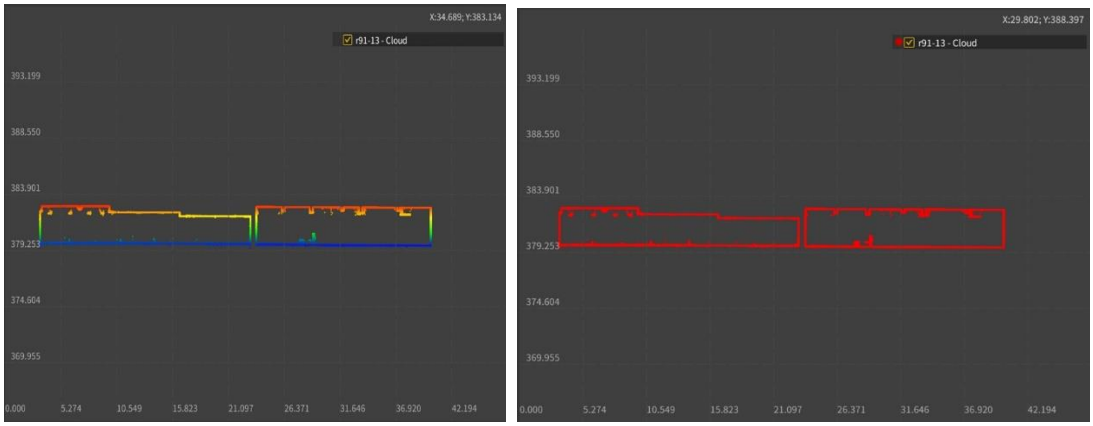



**Figure 159.** Profile after and before rendering

## 5.5.3 Density Evaluation

**Description:**

Performs statistical analysis of the spatial distribution and point density characteristics within a selected region of the point cloud.

**Operation procedure:**

1. Left-click a single point cloud dataset in the project file list;
2. The density evaluation function is activated. Click the density evaluation function icon to open the function window. The entire point cloud is analyzed by default. You can also use the selection tool to perform density analysis on the point cloud data of a certain area. Two selection methods are supported: square and polygon. When the **Density Evaluation** function is activated, click the density evaluation function icon to open the function window. By default, the entire point cloud will be analyzed. You can also select a specific region using either a rectangular or polygonal selection tool;

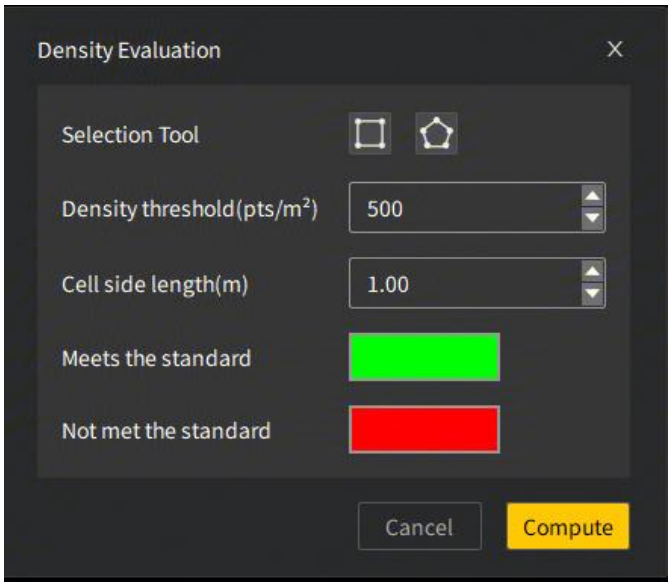



**Figure 160.** Density evaluation window

3. After selecting a tool, the cursor will change from an arrow to a hand icon. Left-click on the point cloud to define the analysis area. You can manually set the **density threshold**, **cell size**, and customize the **color scheme** for qualified and unqualified cells;

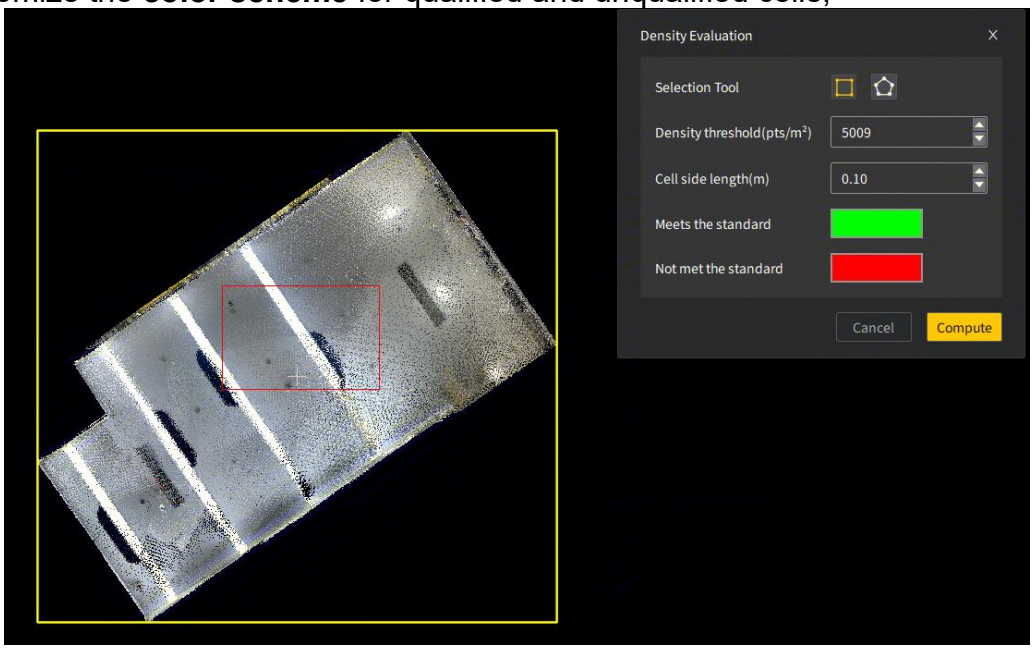



**Figure 161.** Select the point cloud for density evaluation

4. After setting the parameters and point cloud area, click Calculate to display the point cloud density in the selected area. A report can be generated directly from the results;

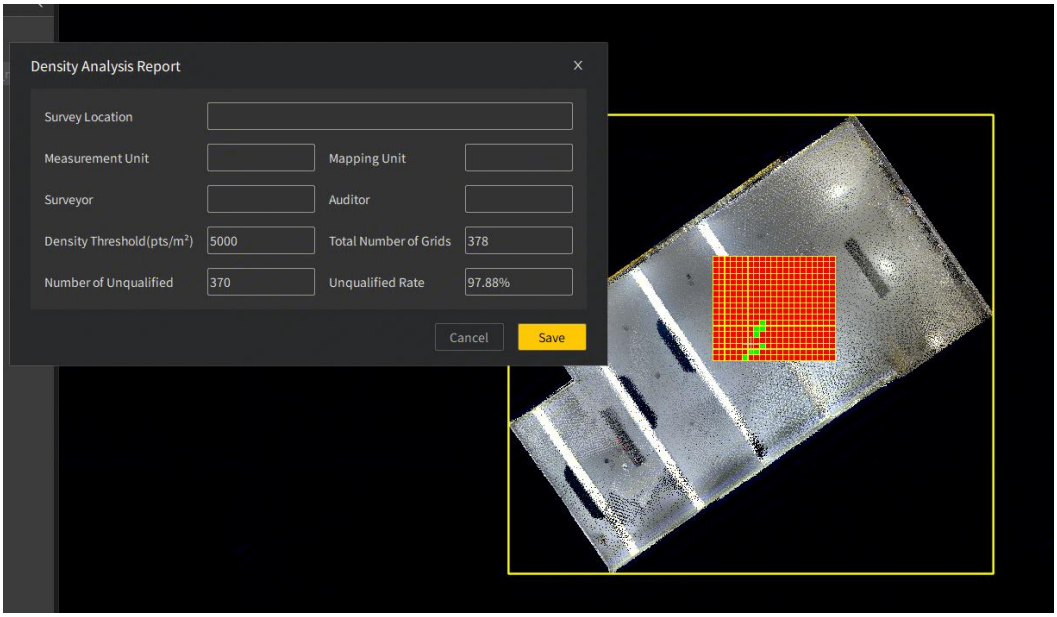



**Figure 162.** Density evaluation result

5. Click "Cancel" or click "×" in the upper right corner of the window to exit the density evaluation function. Left-clicking in non-point-cloud areas of the view during measurement will not trigger any density calculation.

# 5.6 Project Templates

## 5.6.1 Queue Processing

### I. Point Cloud Mapping

**Description:**

Supports performing the same operation on multiple datasets at once, currently available for point cloud coloring and point cloud mapping.

**Operation procedure:**

1. Click on the **Start** tab, and the queue processing window appears. Set the function option, device model, scanning scene, mapping mode, RTK fusion, and reconstruction range, and click **Next**.

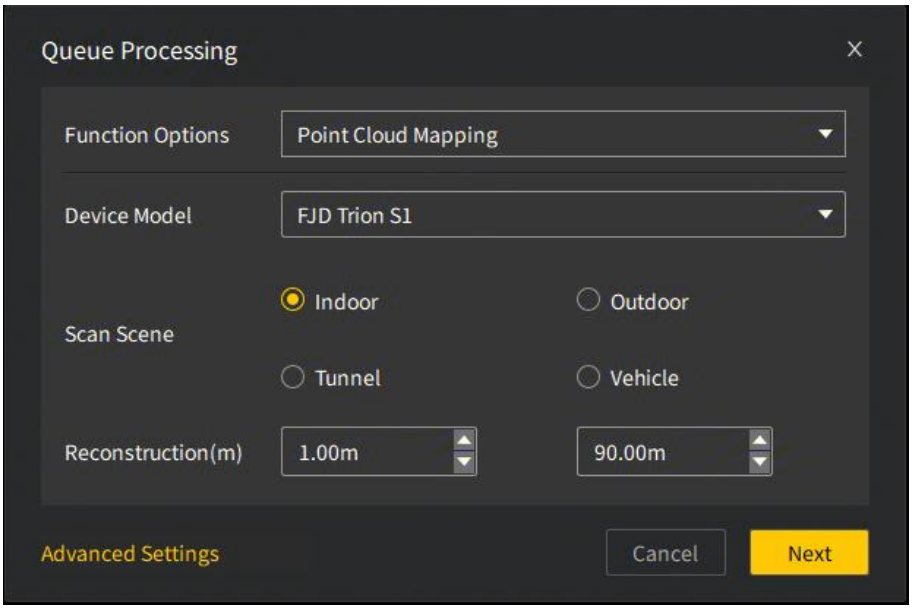



**Figure 163.** Queue processing

2. Advanced settings include **Back to starting point**, **Moving object removal**, and **Image filtering**.

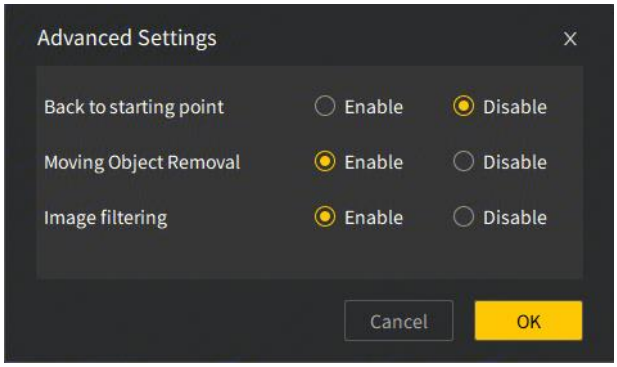



**Figure 164.** Advanced Settings

3. Select the compressed point cloud files for mapping. A maximum of 10 files can be processed simultaneously.

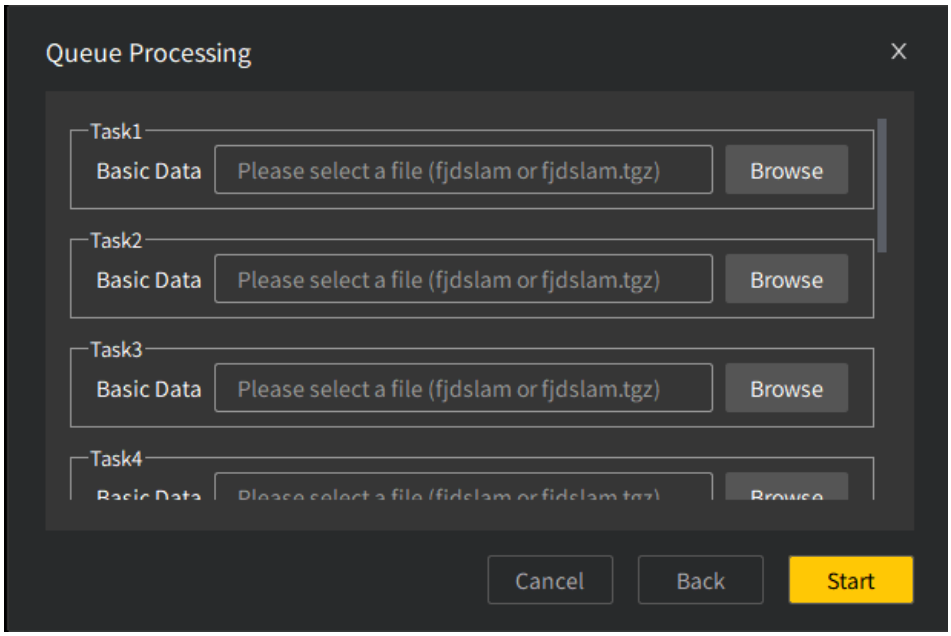



**Figure 165.** Queue processing for mapping

4. Click **Start** to start the mapping. The parameter setting window disappears and the progress window appears. The progress bar will indicate three possible states: Failed, Completed, or In Progress. Please monitor memory usage during processing to avoid failures caused by insufficient memory.

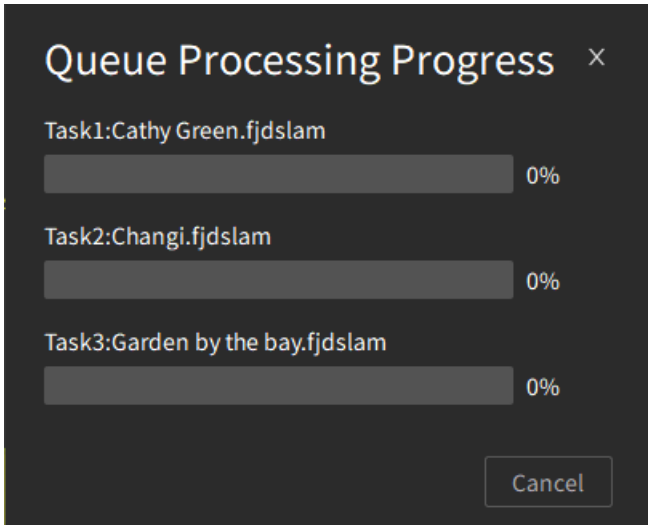



**Figure 166.** Queue processing progress

5. Once all tasks in the queue are completed, the progress window will close, and a load selection popup will appear. You can select which mapped data to load. Click "**OK**" after making the selection to proceed.

## II. Point Cloud Colorization

**Description:**

Batch execution of point cloud coloring function to improve coloring efficiency.

**Operation procedure:**

1. On the Start tab, select the Queue Processing function. Click the Queue Processing icon to open the function window. Choose the Point Cloud Colorization function, then select the base point cloud data and corresponding image data for coloring. A maximum of 5 files can be processed simultaneously. After completing the setup, click Start to begin coloring.
2. Coloring results are saved in the same directory as the origin cloud by default, and the naming rules remain unchanged.

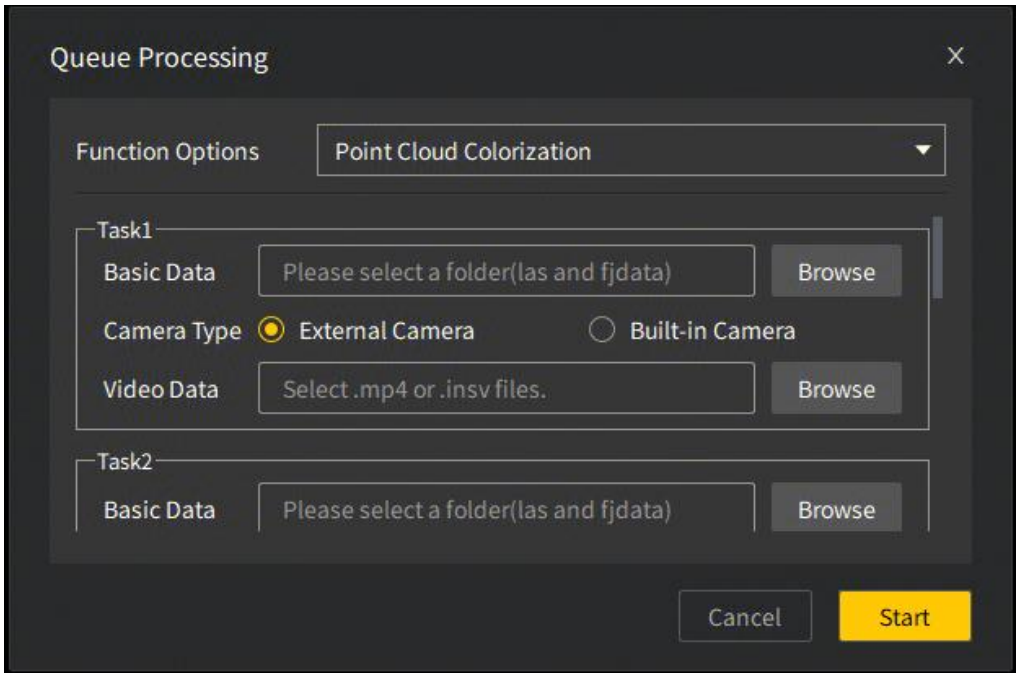



**Figure 167.** Queue processing progress

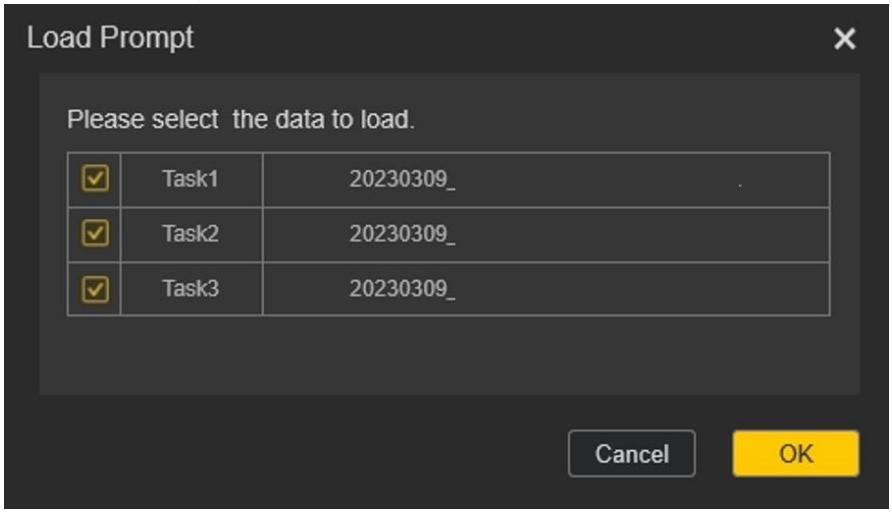



**Figure 168.** Selecting data to load

**Note:**

- A maximum of 10 tasks can be processed at a time.

- Ensure that there is sufficient memory available before queue processing.

# 6 Edit

## 6.1 Sampling

### 6.1.1 Subsample

**Description:**

Presents the features of the ground and objects with fewer points and achieves a balance between the point cloud density and the accuracy to greatly reduce the redundant data, improve the computing efficiency, and save the storage space. Subsampling can be based on the distance, density, or spatial structure.

**Operation procedure:**

1. Select a file under **Objects**.
2. Click .
3. Select a mode, set the parameter, and click **OK**.

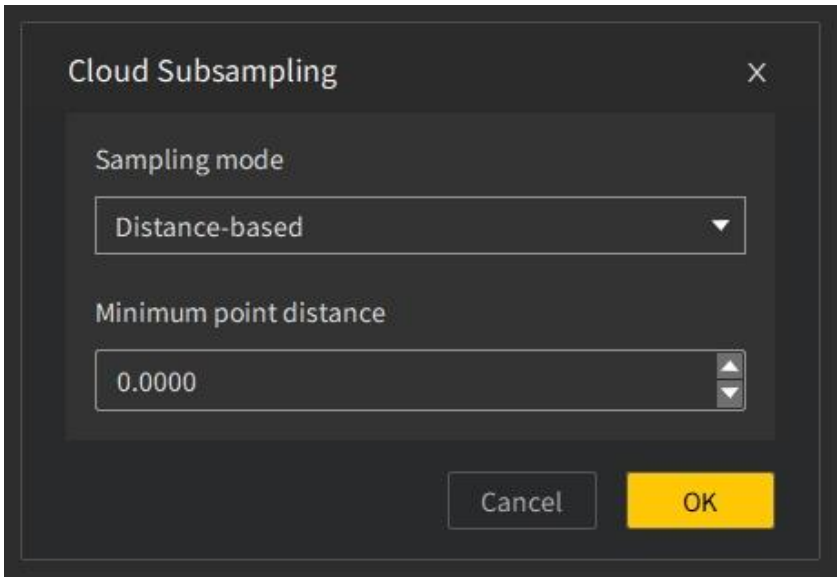



**Figure 169.** Cloud subsampling

The following figures show the effects before and after subsampling in different modes. When the **Density-based** mode is selected, the smaller the parameter, the fewer the points remained. When the **Distance-based** mode is selected, the larger the parameter, the fewer the points remained. When the **Spatial structure** mode is selected, the smaller the parameter, the fewer the points remained.

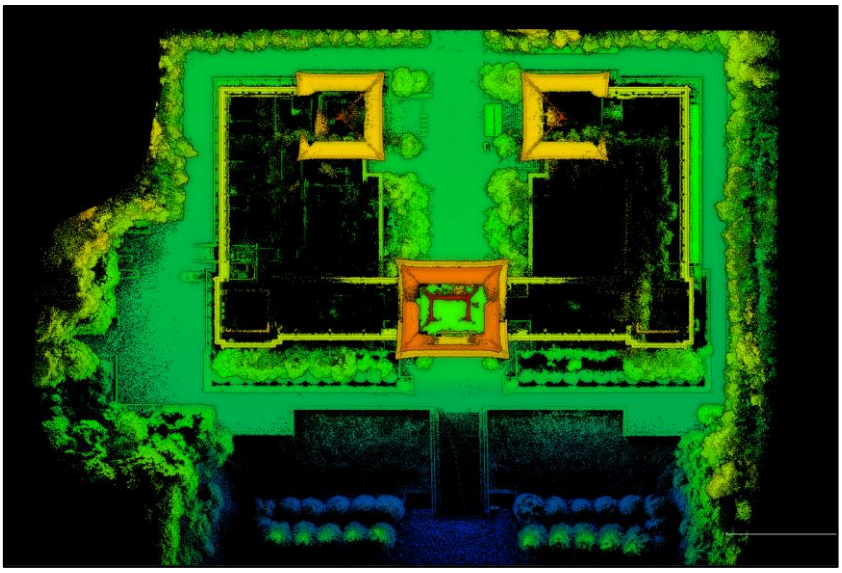


**Figure 170.** Raw data

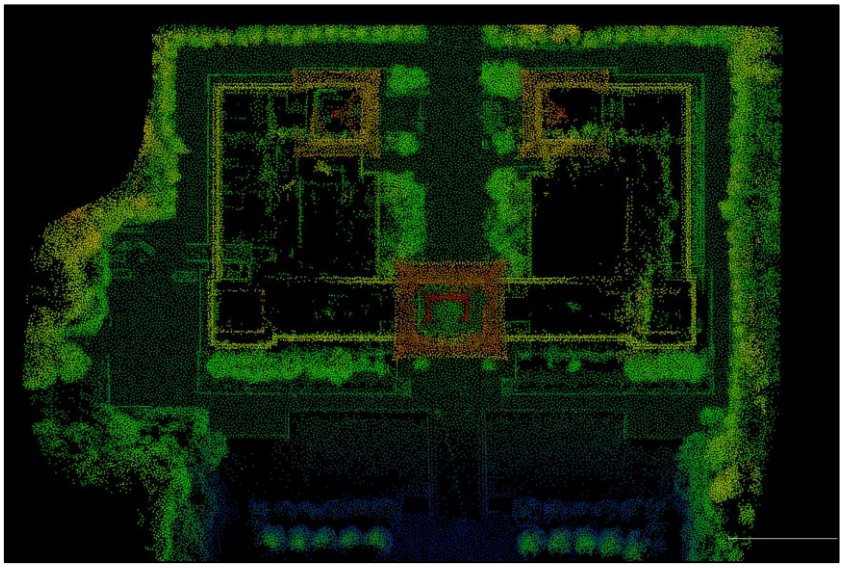


**Figure 171.** Distance-based subsampling

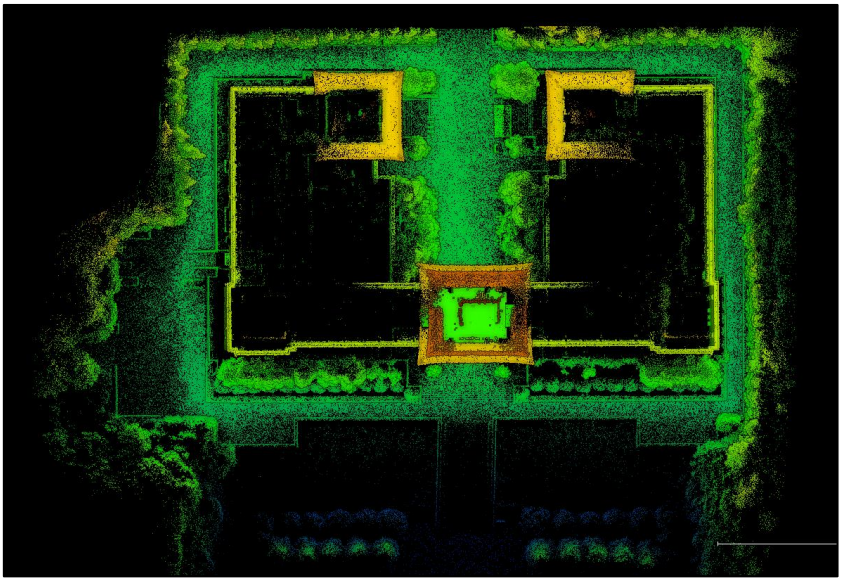


**Figure 172.** Density-based subsampling

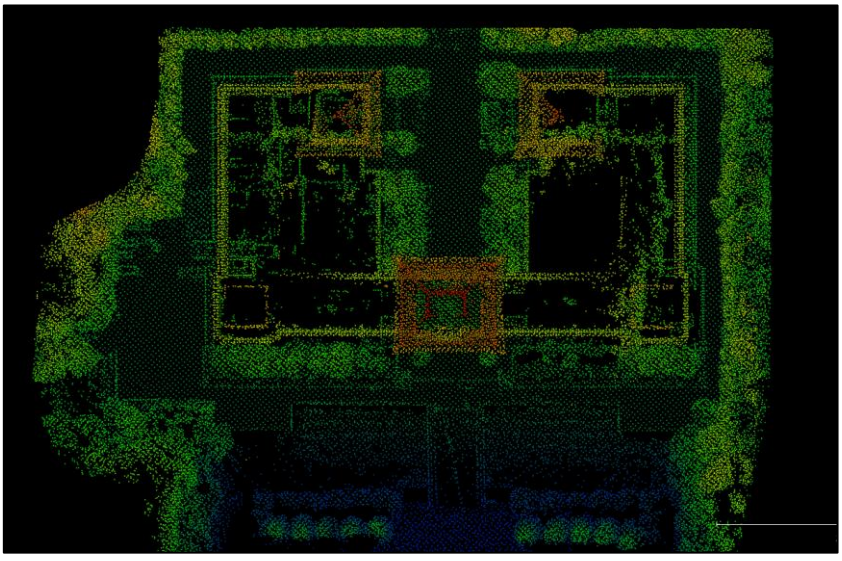


**Figure 173.** Spatial structure-based subsampling

# 6.2 Segment

## 6.2.1 Clip by Path

**Description:**

Clips the selected point clouds based on the scanning path. It is mainly used to remove abnormal point clouds that are generated because the scanner stays at a position for too long or follows an unreasonable path.

**Operation procedure:**

1. While pressing the Ctrl key, select a point cloud file and its corresponding scanning path file under **Objects**.
2. Click on the **Edit** tab to open the toolbar. Both the point cloud and the path are displayed in the 3D view area by default. Click to hide the point cloud. Pick any two points on the path, and the chronologically ascending section of the path is selected. If the wrong path section is selected, click to unselect it.

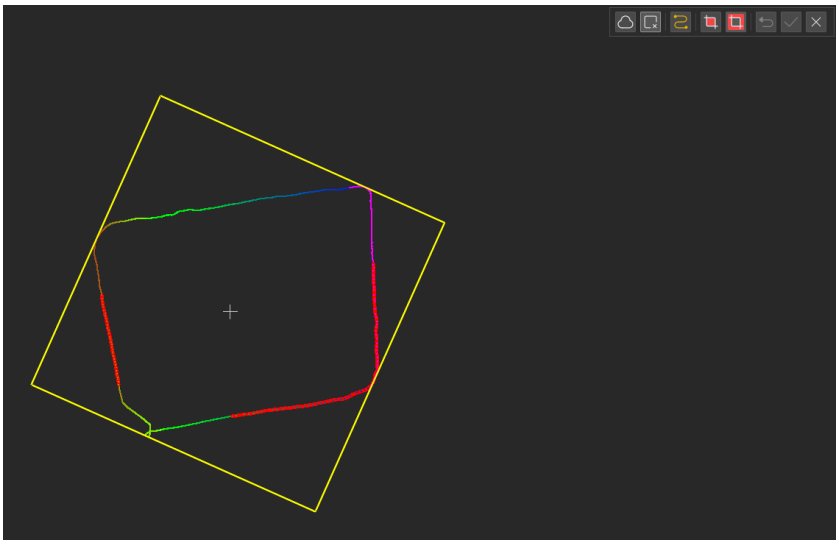


**Figure 174.** Path selected

3. Click or to clip the point cloud corresponding to the path section. The new point cloud generated after clipping is displayed automatically in the 3D view area.
4. Click to restore the original point cloud, or click to save the clipping result.
5. Click to exit the clipping operation.

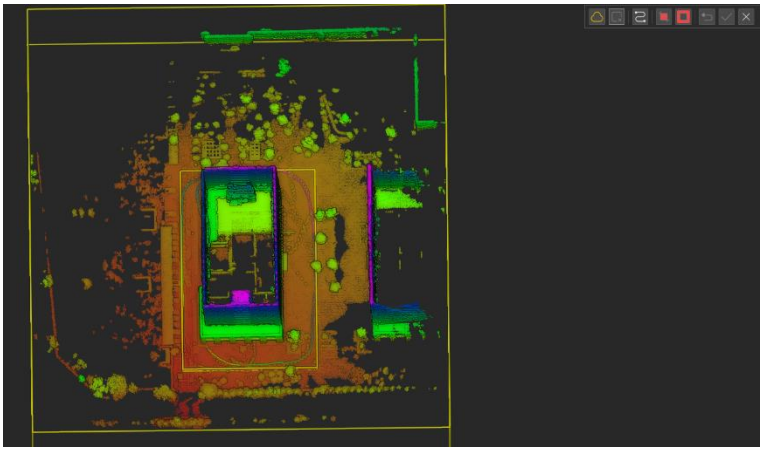


**Figure 175.** Point cloud before clipping

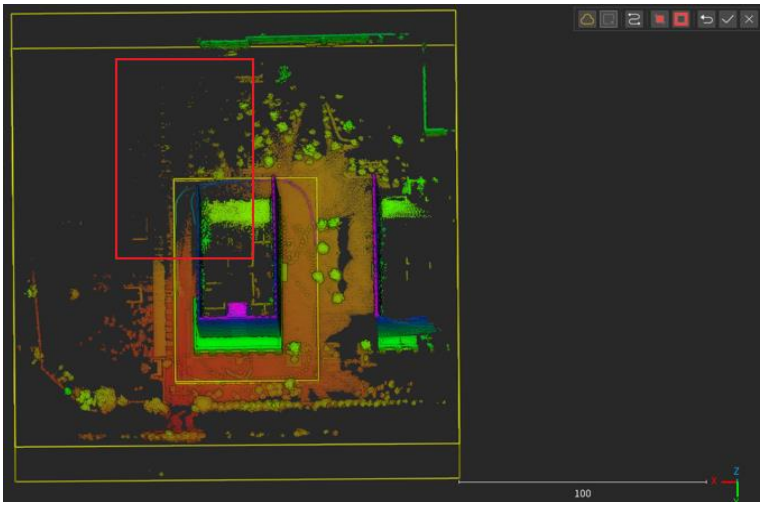


**Figure 176.** Point cloud after clipping

**Note:**

- Both a point cloud and its corresponding path data must be selected to enable the feature of clipping by path.
- The mismatch between the point cloud and the path data will result in clipping failure or incorrect clipping.

### 6.2.2 Data Extraction

**Description:**

Used to extract point cloud data based on different scalar attribute information, reducing point cloud size, avoiding unnecessary data interference, and enhancing point cloud visualization. Supports extraction by elevation, intensity, Pos distance, floor, and category.

**Operation procedure:**

1. Select the corresponding point cloud file to activate the function, click the  to open the settings window.

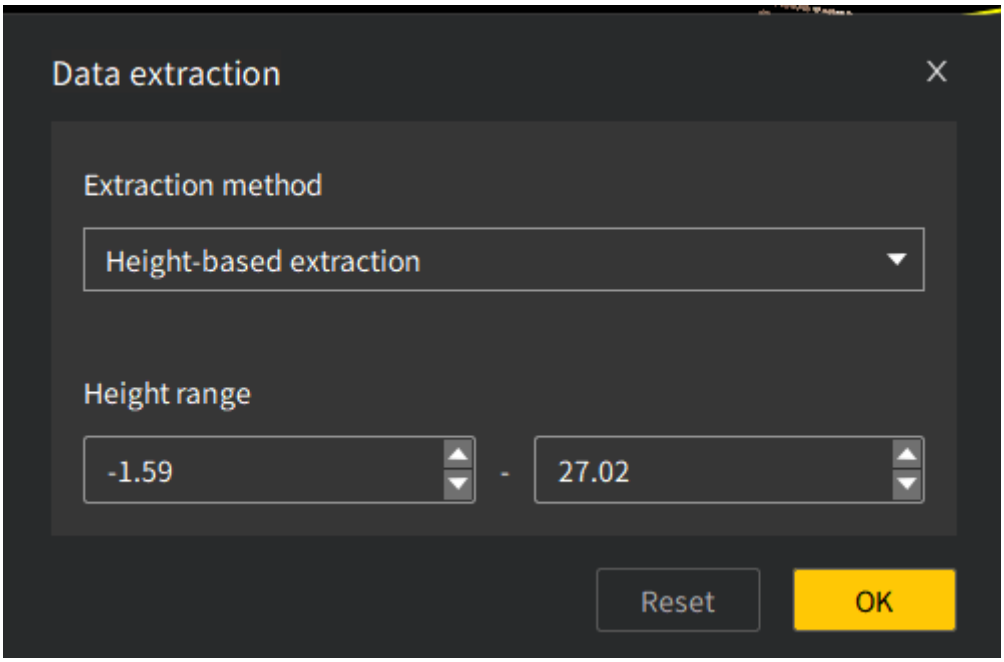



**Figure 177.** Data extraction

2. Select " **Elevation-Based Extraction**", the default elevation range displays the minimum and maximum elevation values of the current point cloud. Users can adjust the range as needed. The scene will retain only the point cloud data within the selected elevation range.

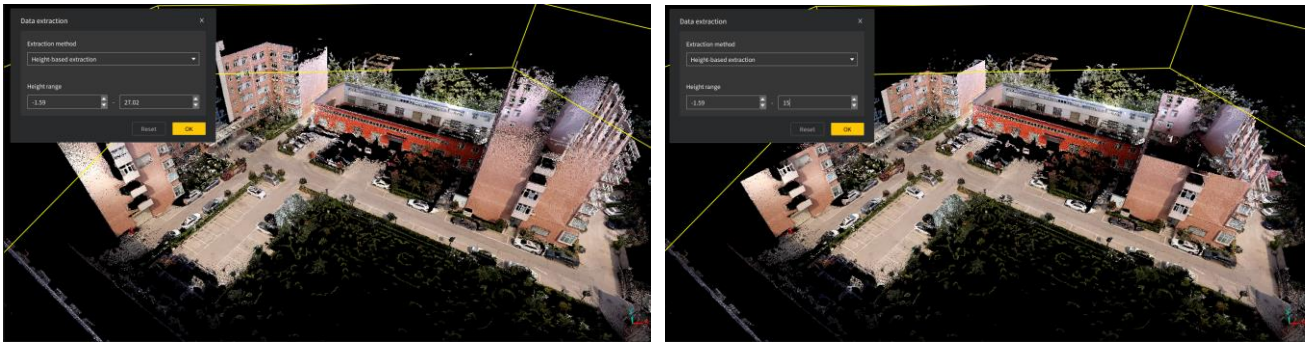


**Figure 178.** Elevation-based extraction

3. Select " Intensity-based extraction" to display the minimum and maximum intensity of the current point cloud by default. It can be used to remove noise such as water surface and glass. Before selecting the range, the intensity value at the noise point can be measured in advance for targeted extraction.

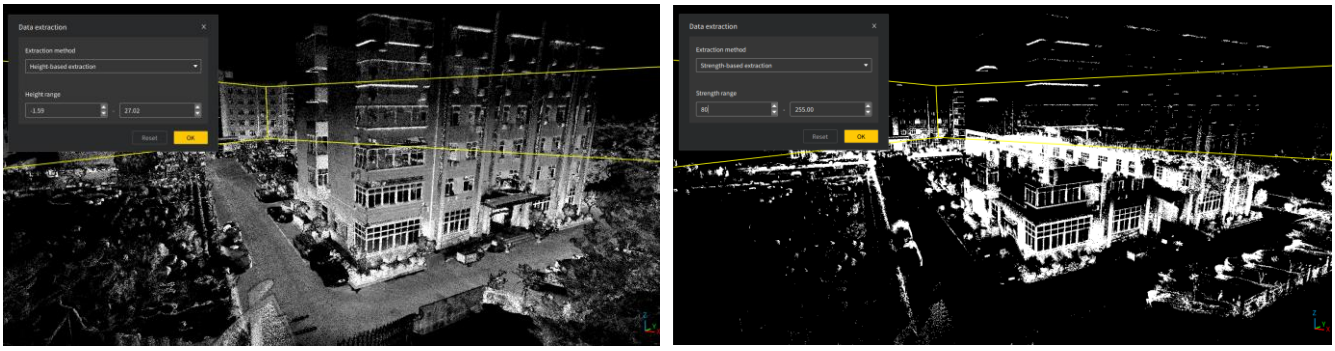


**Figure 179.** Intensity-based extraction

4. For point clouds with a loaded configuration file, you can select " POS distance-based extraction". The default mapping range is [0, 90]. Two extraction modes are available: 3D and 2D. 2D mode uses trajectory points as the center and extracts by clipping on the projection plane; 3D mode uses trajectory points as the center and extracts using a cylindrical range. The point cloud in the scene will be extracted based on the configured Pos distance range.

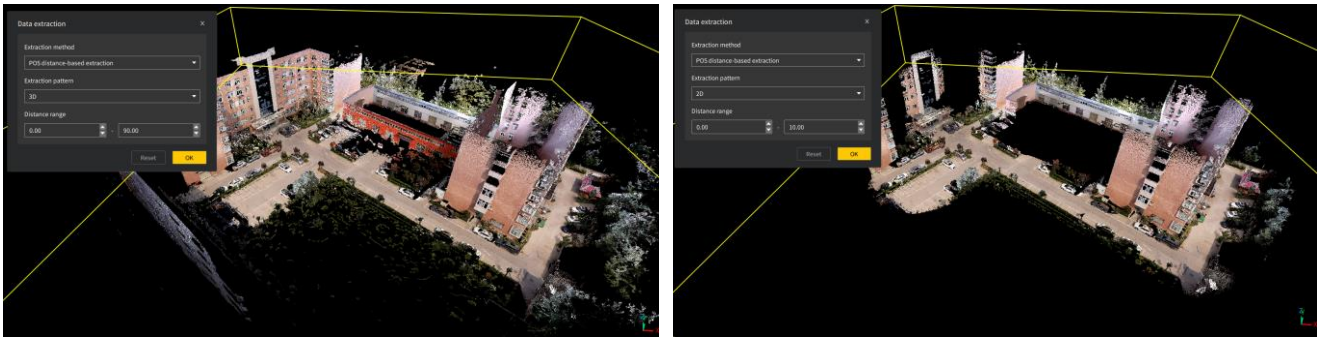


**Figure 180.** POS distance-based extraction

5. Select "Floor Extract", set the planar point recursion ratio (higher values reduce the detection ability for irregular floor slabs), and click OK to complete floor-level data extraction.

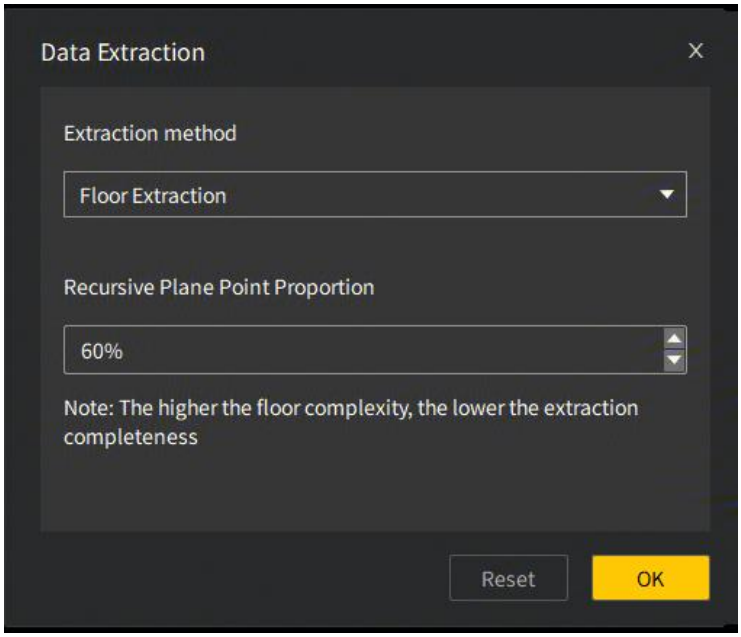



**Figure 181.** Floor Extract pop-up window

6. Select "Extract by Class" to extract according to the identified categories.

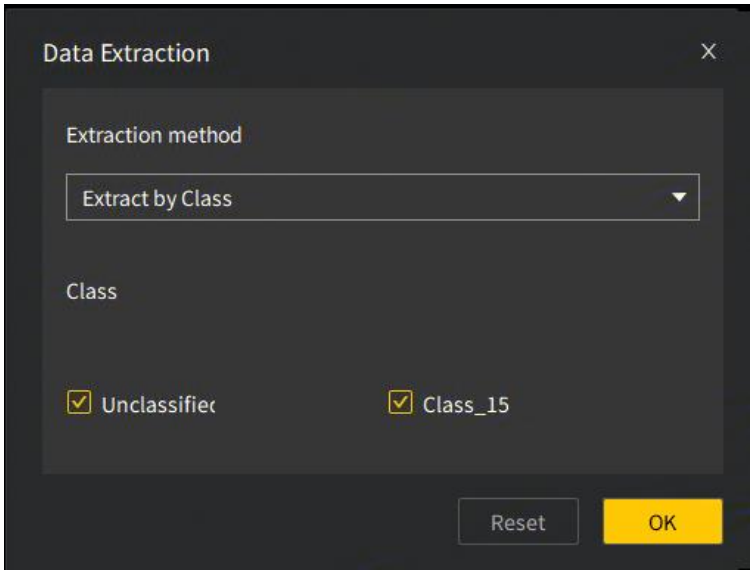



**Figure 182.** Extract by Class pop-up window

### 6.2.3 Slice Plane

**Description:**

Cut the plane from the point cloud image to obtain "quasi-plane" point cloud data with a certain thickness. Two cutting methods are provided: axial cutting and vertical screen cutting.

**Operation procedure:**

1. In the project file, click the left button of the mouse to select the point cloud data, and the slice plane function will be activated;
2. Click the slice plane function icon  to pop up the cutting operation interface as shown below;

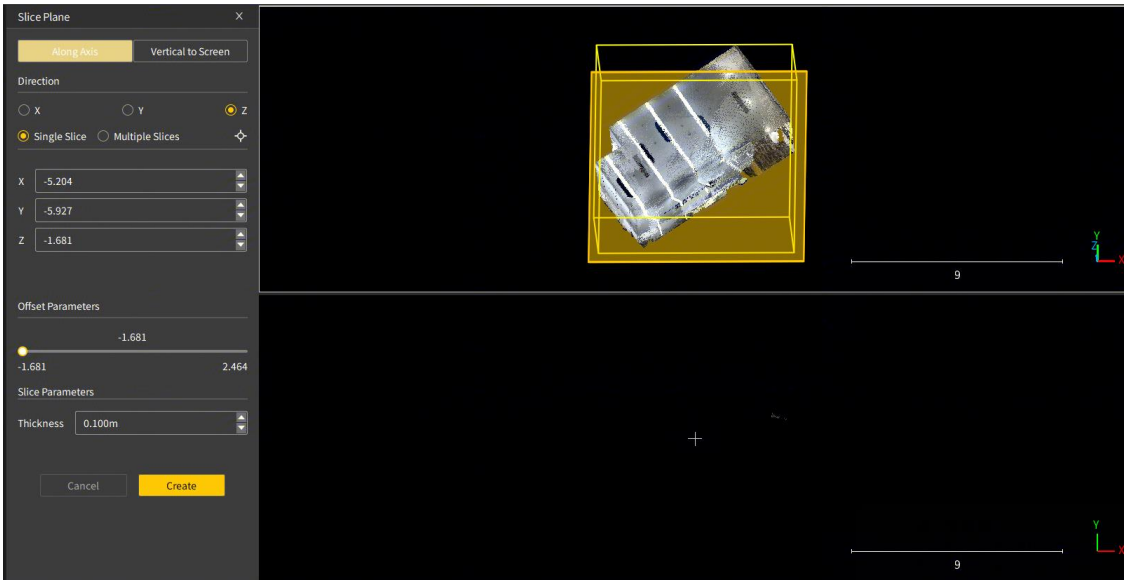



**Figure 183.** Slice plane operation interface

3. Axis cutting is adopted. Users can select the cutting direction. There are three directions to choose from: X, Y, and Z. Both **Single Slice** and **Multi-Slice** modes are supported.
4. Select single slicing, fine-tune the cutting position by inputting parameters or dragging the slider, and finally set the section thickness. Click the "Create" button to complete the plane cutting. After the creation is completed, the corresponding section file will be added to the project file;

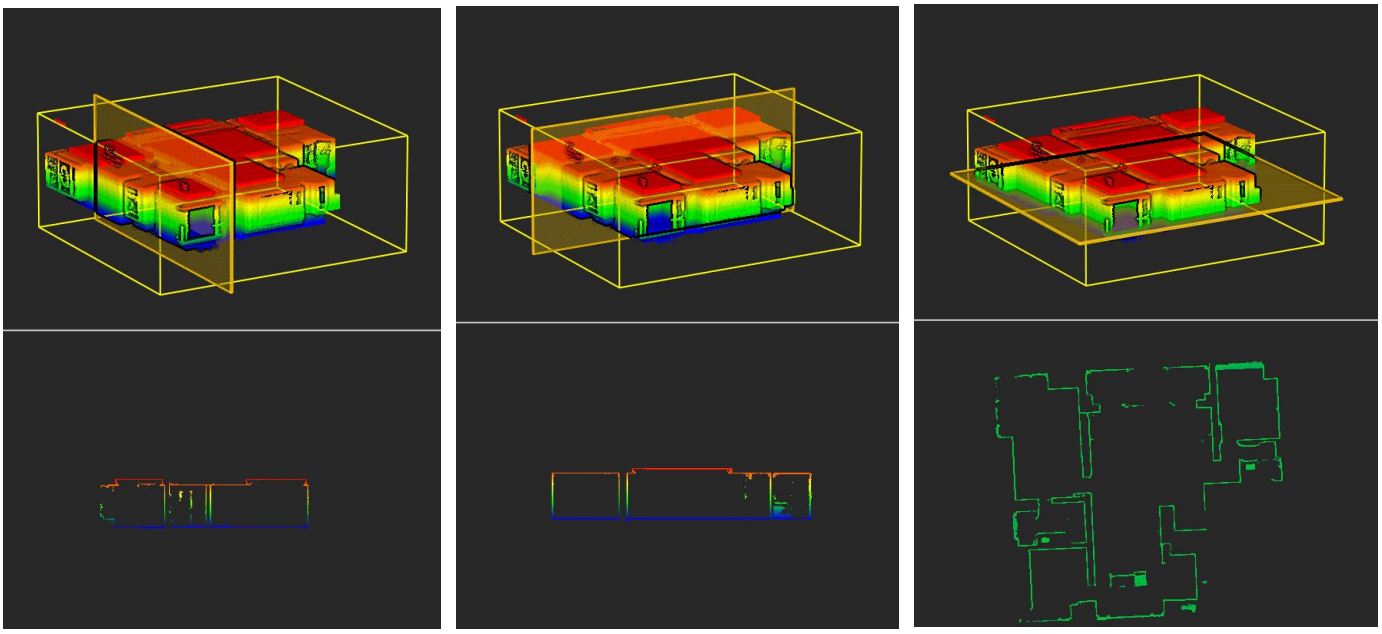


**Figure 184.** Single slice effect in X/Y/Z direction

5. In multi-slice mode, the starting position is automatically set to the minimum value along the selected axis. Users can define the slice interval and number of slices. By default, the system displays the first slice (i.e., the lowest one along the axis). The 2D view will automatically update to show the current slice. When the slice thickness is modified, the view will refresh accordingly. Adjusting the offset value will shift all slice planes together. Users can preview different slices by changing the slice index. The resulting slice plane files will be added as sibling files under the selected point cloud in the project file tree.
6. When using screen-perpendicular slicing, users can define a slice plane by selecting two points on the point cloud. The 2D view will update to display the slicing plane. The slice plane can be offset using the slider, and its thickness can be adjusted as needed. Axis parameters

may also be edited manually by double-clicking in the parameter table, as shown below. Click the “Create” button to generate the slice plane and update the project file list accordingly.

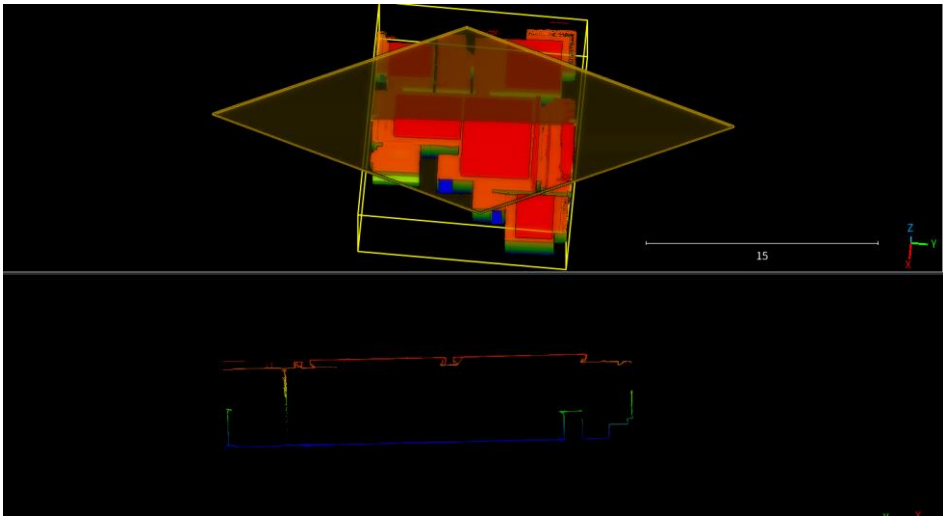



**Figure 185.** Vertical screen cutting results

# 6.3 Classification

**Description:**

Classifies the point clouds by scenario. This part covers five features, which are **Outdoor**, **Indoor**, **Manual**, **Extract by Class**, and **Floor**. The **Outdoor** scenario includes **Ground** and **Vegetation**. The **Indoor** scenario includes **Wall**, **Floor**, and **Ceiling**.

## 6.3.1 Automatic Classification

### I. Model File Classification

**Operation procedure:**

1. In the project file, left-click to select point cloud data;
2. Click the automatic classification function icon and select the model file classification;

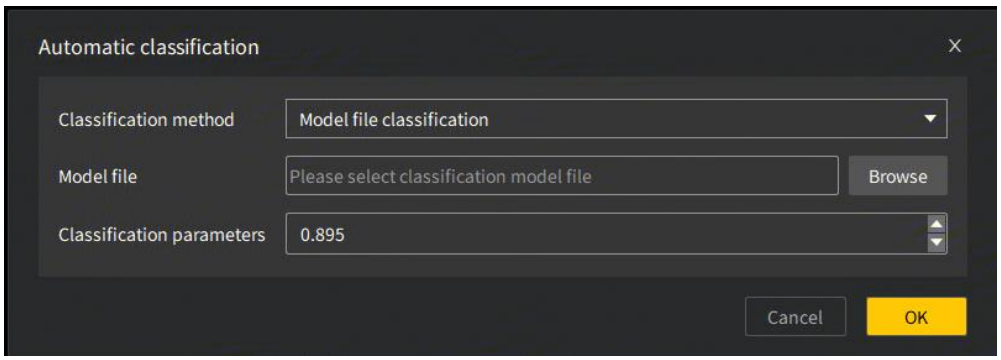



**Figure 186.** Model file classification

3. Select the model file(s) in .prm format (multiple files supported). Set the classification threshold (default: 0.895). A higher threshold yields more accurate classification results but may cause more omissions;
4. After configuration, click OK to start classification. Once completed, the classified point cloud data will be automatically displayed.

### II. Indoor Classification

**Operation procedure:**

1. Select a point cloud file under **Objects** to activate the indoor classification feature.

2. Click , and the following window appears.

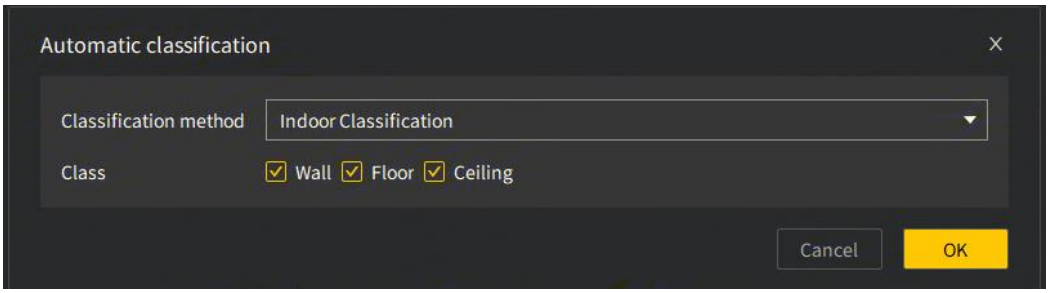
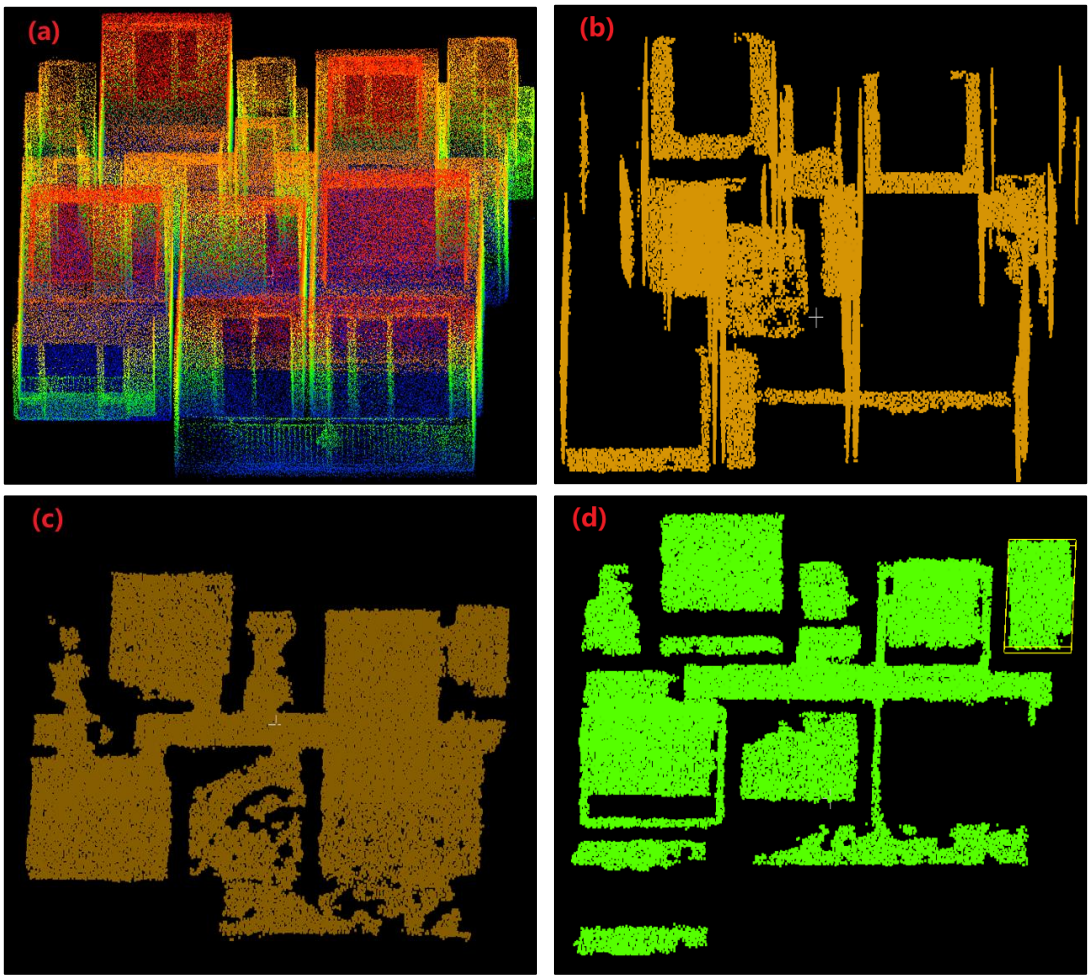



**Figure 187.** Indoor classification

3. Select one or more of Wall, Floor, and Ceiling, and then click **OK** to extract the clouds of the corresponding types. Multiple categories can be classified simultaneously. The images below show the classification results for wall, floor, and ceiling extracted from indoor scanning data.




(a) Indoor point cloud data; (b) Extracted wall point cloud; (c) Extracted floor point cloud; (d) Extracted ceiling point cloud

**Figure 188.** Indoor classification and extraction

## III. Outdoor Classification

**Operation procedure:**

1. Select a point cloud file under **Objects** to activate the outdoor classification feature.
2. Click , and the following window appears. Choose at least one target category to classify, such as **Ground**, **Vegetation**, or **Building**. Ground classification is subdivided based on slope into Flat Ground, Gentle Slope, and Steep Slope. Classification accuracy is inversely proportional to slope steepness: the steeper the slope, the lower the accuracy. For buildings,

smaller resolution values lead to higher extraction accuracy. In most cases, the default parameters provide good results.

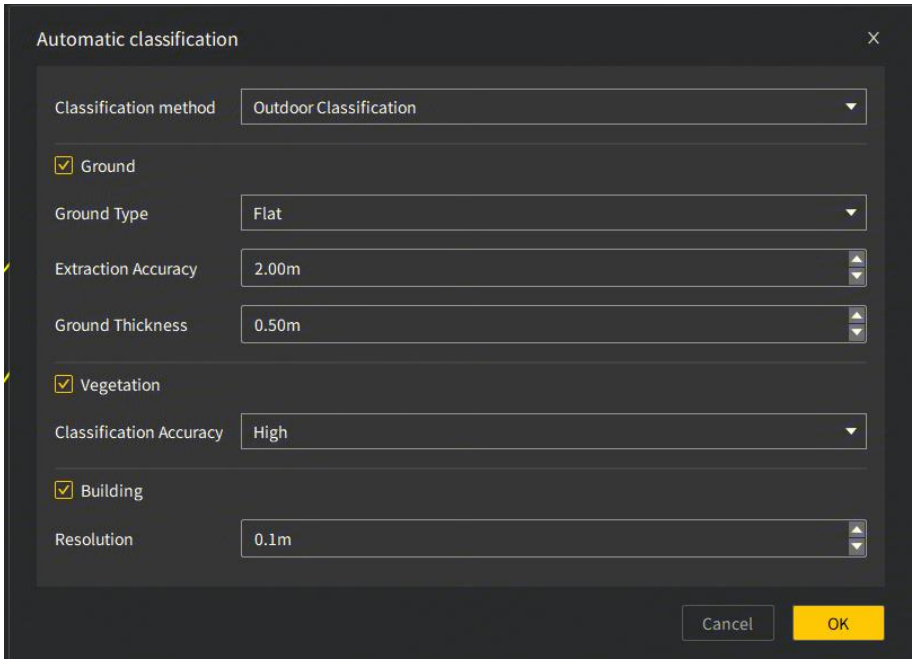



**Figure 189.** Outdoor classification

3. After selecting **Flat**, click **OK** to perform ground point cloud classification. The figures below show the classification results for ground, vegetation, and buildings within the scene.

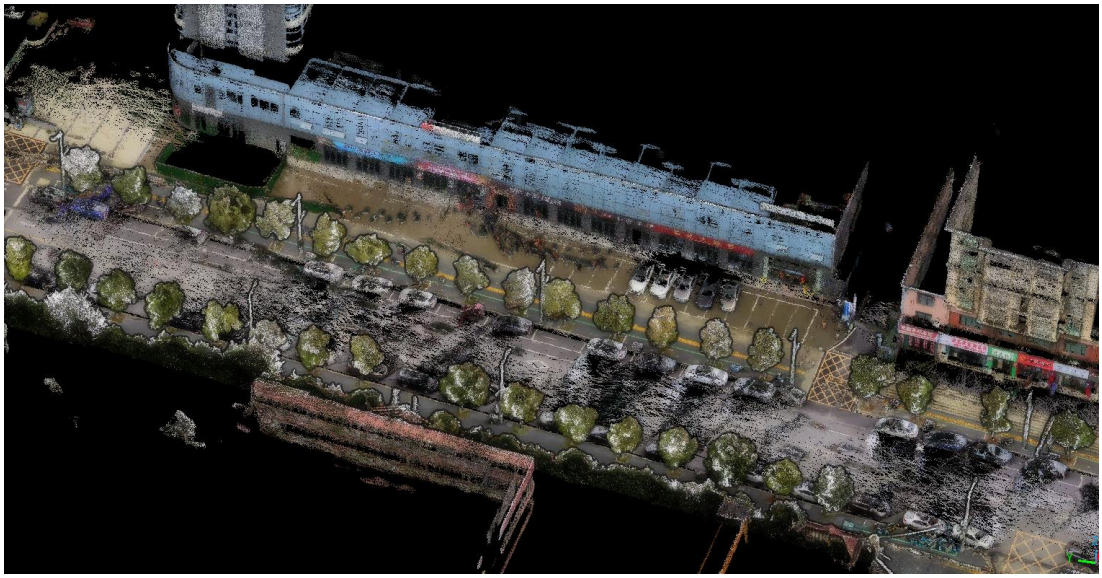


**Figure 190.** Raw point cloud

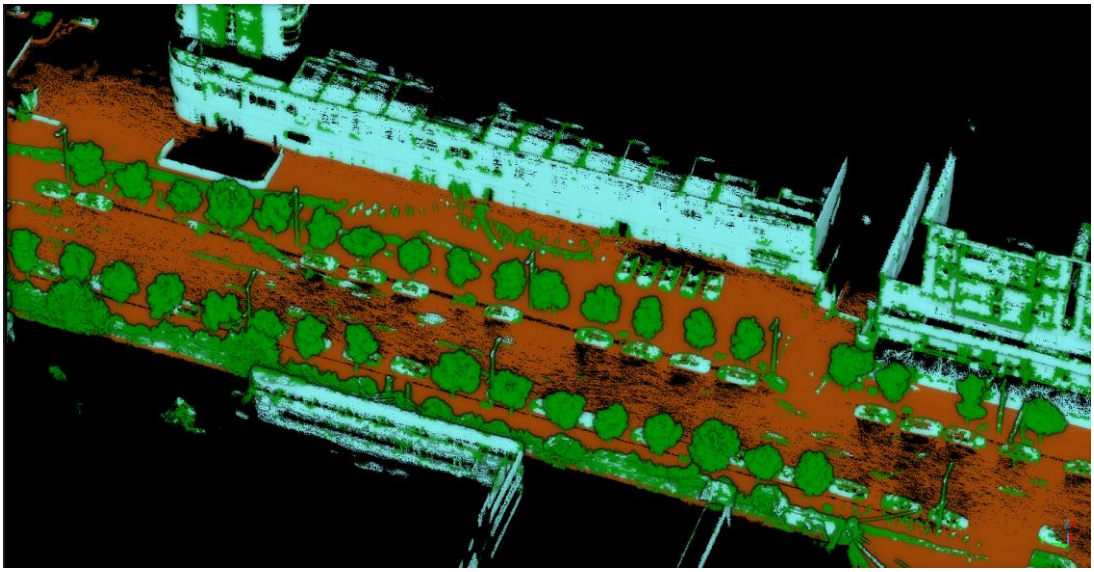


**Figure 191.** Classification result display

### 6.3.2 Manual

**Description:**

Manual classification allows you to assign categories to point cloud data by selecting regions using different selection tools, achieving the goal of point cloud classification.

**Operation procedure:**

1. Select a point cloud file under **Objects**.
2. Click on the **Edit** tab, and the following window appears. The point cloud will switch to category rendering mode.

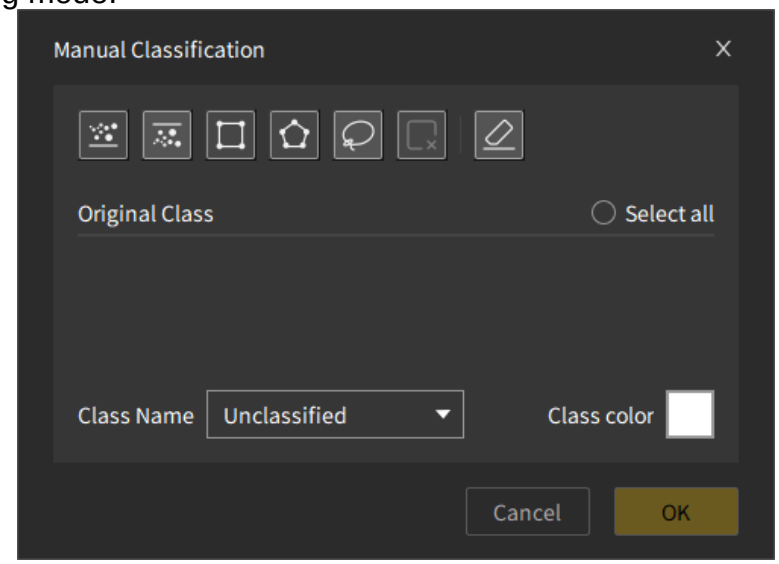



**Figure 192.** Manual classification (no data selected)

3. Click any selection tool to select a portion of the point cloud, and all the classes of the selected portion are automatically displayed under **Original Class**. The , , and **OK** buttons, and the **Class Name** and **Class color** fields become activated.
4. The selected portion is highlighted in red, and multiple portions can be selected. Click to undo the previous selection, or click to undo all the selections.
5. After selecting the point cloud, the software will by default apply category settings to unclassified points. You can use the checkboxes in front of category types to specify which points should be included in the classification.

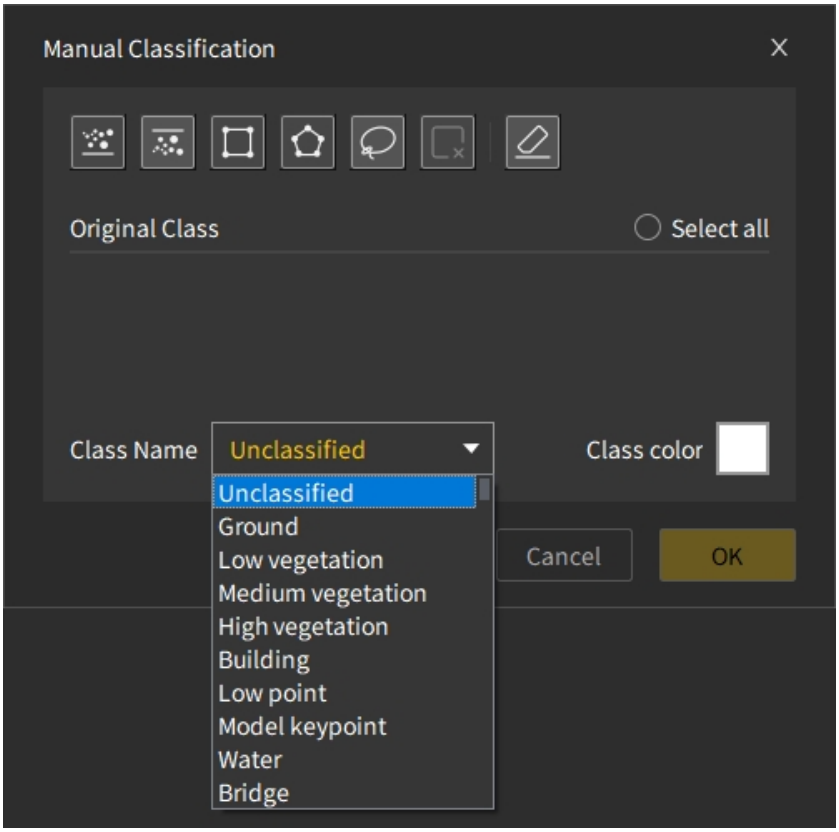



**Figure 193.** Manual classification (data selected)

6. After choosing a category name, the corresponding category color will be updated automatically. You can also click the Category Color button to customize the color.
7. Click **OK** to complete the settings. The selected state in the 3D view will be cleared, and the points within the selected range will be updated with the chosen color.
8. Click the **close** icon in the upper right corner of the **Manual Classification** window to exit the manual classification. The point cloud is rendered by classification by default.

**Figure 194.** Manual classification result

**Note:**

- Before any point cloud is selected, the class options are not displayed. The default category is Unclassified, with a default color of white, and the Confirm button is disabled.
- When selected points belong to multiple classes, operations will default to unclassified points. To reassign other classes, you must manually select the desired category.

### 6.3.3 Model Training

**Description:**

Use point cloud data to create classification model files.

**Operation procedure:**

1. Click the function button to display the model training pop-up window. You can select training data from the point cloud list, import it from an external source, or manually mark it in the point cloud (you need to select the point cloud first and then click the training model);
2. Manual labeling provides various point cloud selection tools to flexibly select data to be used for training;

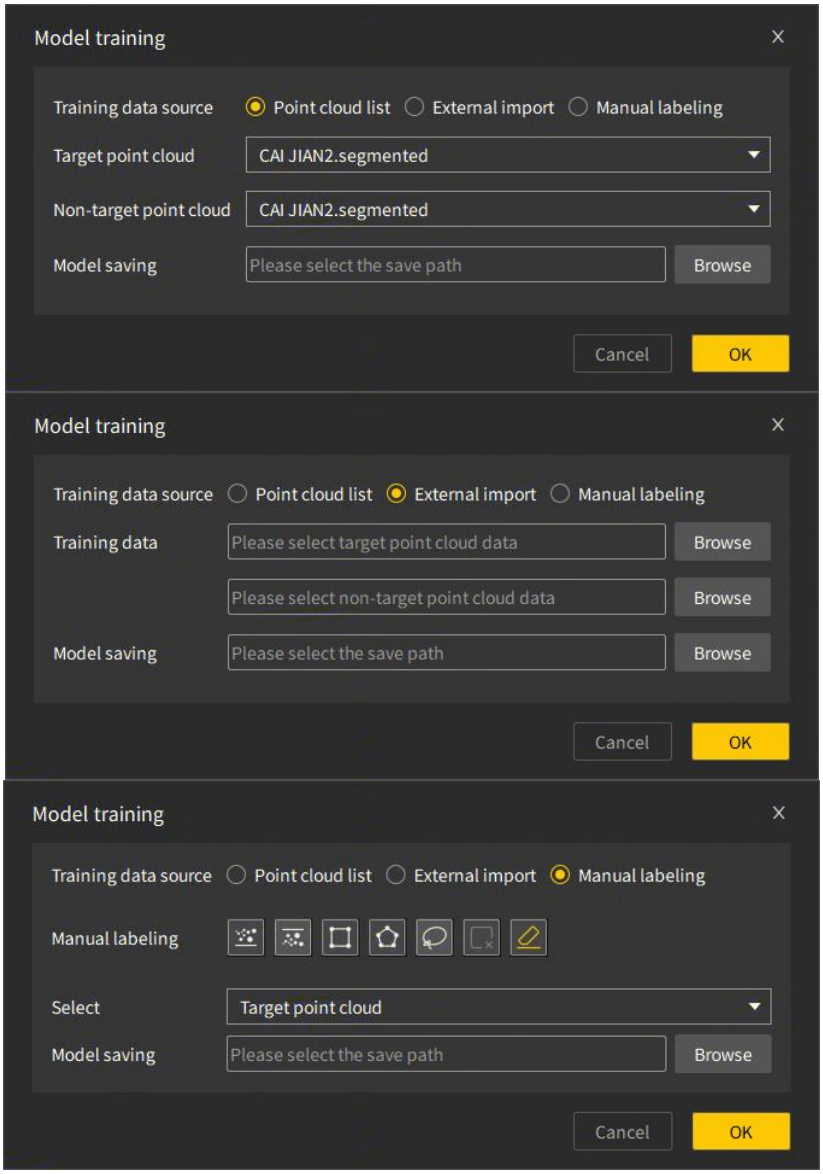



**Figure 195.** model training pop-up window

3. After specifying the Target and Non-target point clouds and setting the Save Path for the model, click **OK** to start training. Upon completion, a model file in .prm format will be generated;
4. For example, a point cloud file containing only trees can serve as the target point cloud, while another point cloud file containing only ground, buildings, streetlights, etc. can serve as the

non-target point cloud. The model trained in this way can be used to identify trees in the point cloud.

# 6.4 Note

**Description:**

Adds text or image notes to the selected point.

**Operation procedure:**

1. Select a point cloud file under **Objects** to activate the notes feature. Click , and the cursor pattern in the 3D view area changes from to .
2. Click a point to add the title, details, and images as shown below.

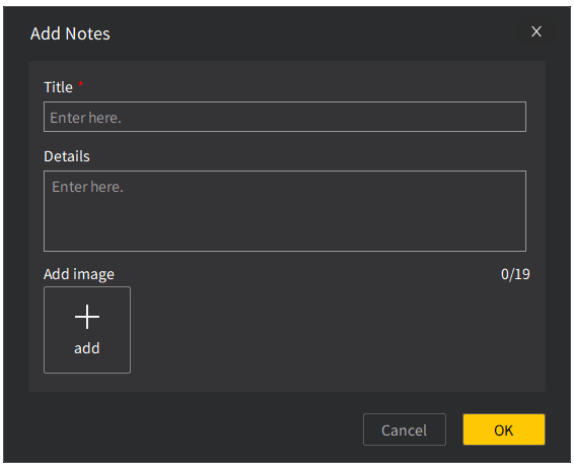



**Figure 196.** Adding notes

3. Click **OK**, and the note appears under **Objects** and also next to the point as a label, as shown below. Double-click a label to see the details or edit it.

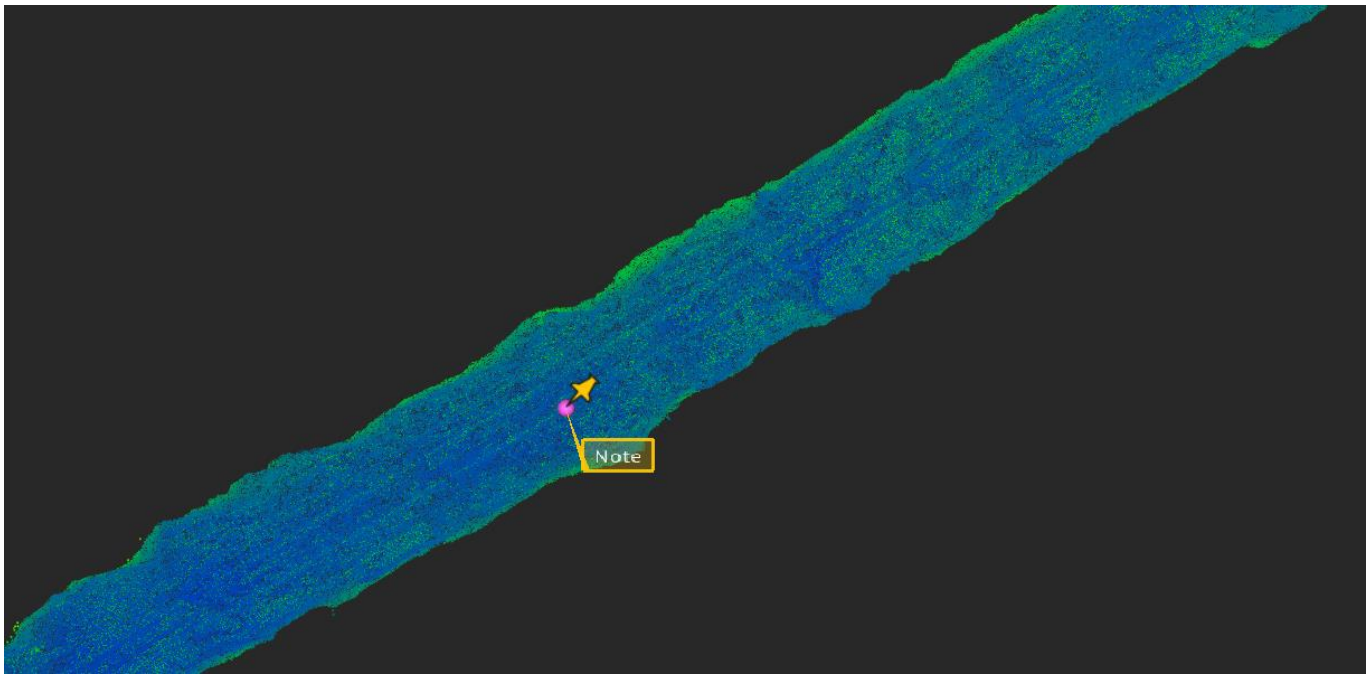



**Figure 197.** Note label

# 6.5 Triangular Mesh

## 6.5.1 Triangulation

**Description:**

Connects the scanned points into triangles according to certain rules, forming a mesh without overlapping triangles. The mesh density varies with the curvature of the object surface. Two triangulation types are supported: **TIN** and **Surface Triangular Mesh**.

**Operation procedure:**

1. Select a point cloud file under **Objects** to activate the mesh creation feature.
2. Click  .
3. Select the triangulation type, set the parameter, and click **OK**. The TIN construction method is often used to build models of approximate planes. After setting the hole filling threshold, holes are filled for points whose distance is less than the hole filling threshold; surface triangulation is often used in enclosed spaces such as mines. The larger the sampling interval, the longer the side length of the triangular surface constructed.
4. After the triangulation model is created, it is supported to save the file in OBJ, PLY and other formats.

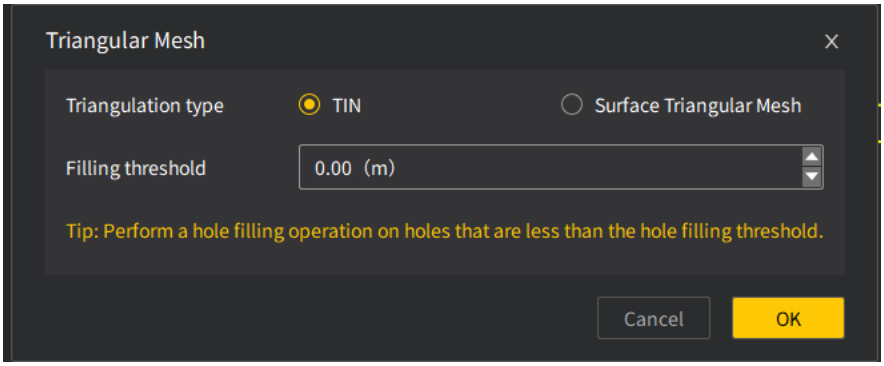



**Figure 198.** Creating a TIN mesh

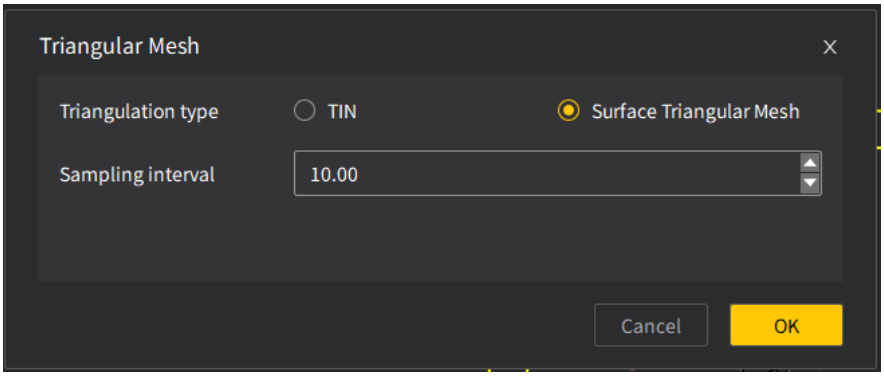



**Figure 199.** Creating a surface triangular mesh

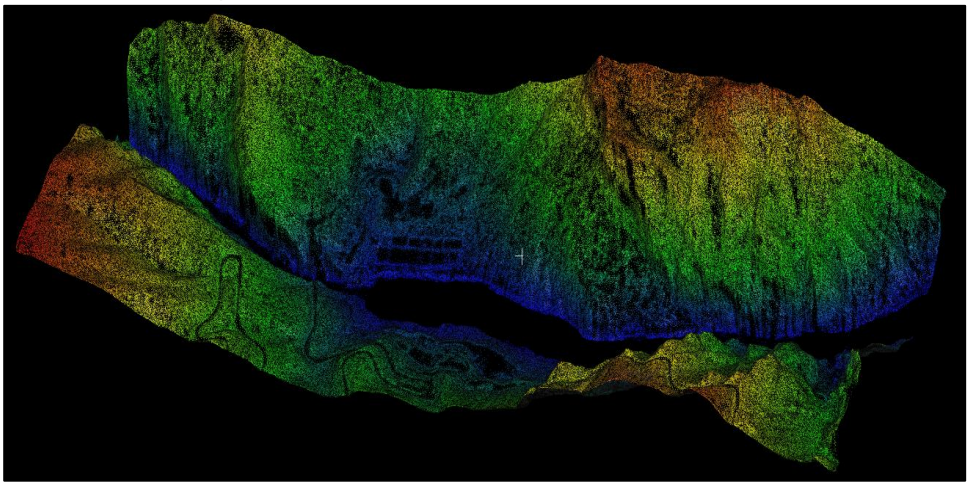


**Figure 200.** Raw point cloud

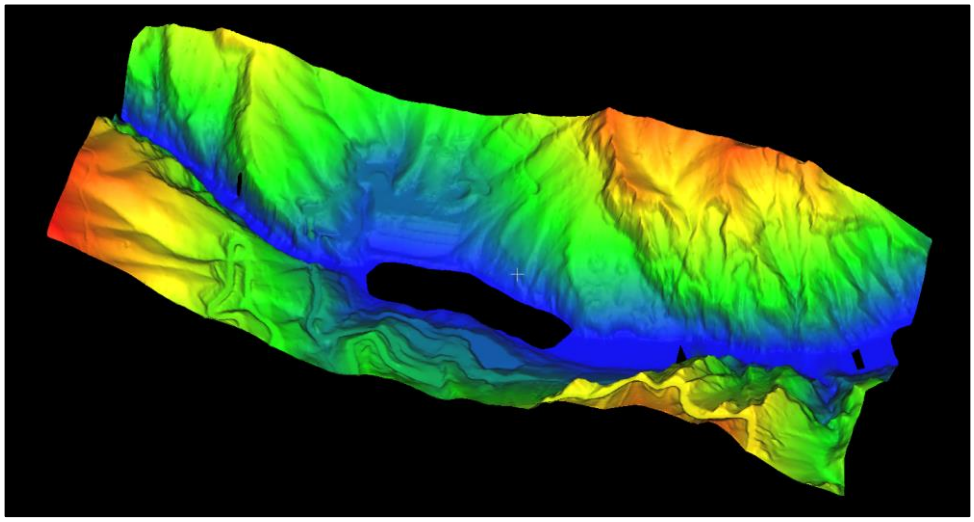


**Figure 201.** Mesh created

### 6.5.2 Contours

**Description:**

Extracts contour lines from a point cloud to reflect the topographic changes of a surveyed area.

**Operation procedure:**

1. Select a point cloud file under **Objects**.
2. Click on the **Edit** tab, and the following window appears. The display settings are collapsed by default.

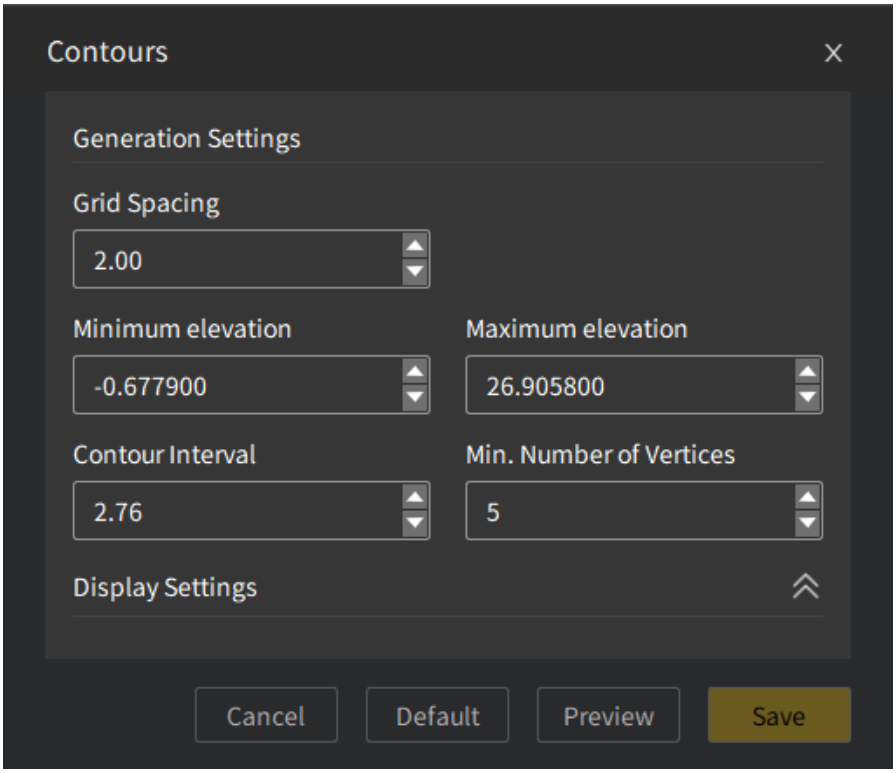



**Figure 202.** Contour generation settings

3. Complete the generation settings and display settings including the contour color, and the line width and color of the primary contour and index contour.

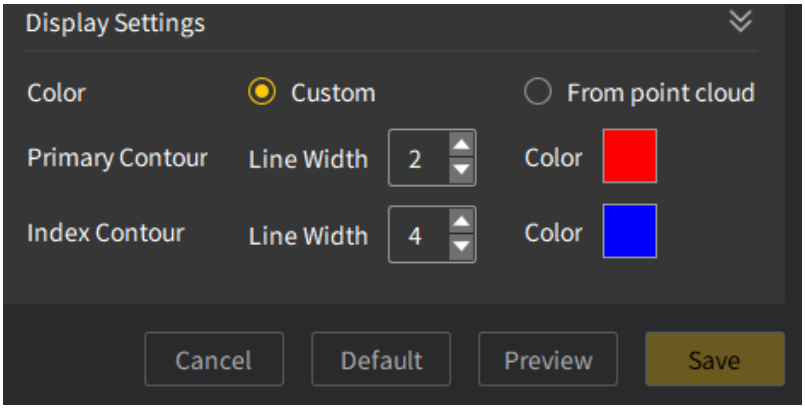



**Figure 203.** Contour display settings

4. Click **Preview** to display the generated contour lines in the 3D view area.

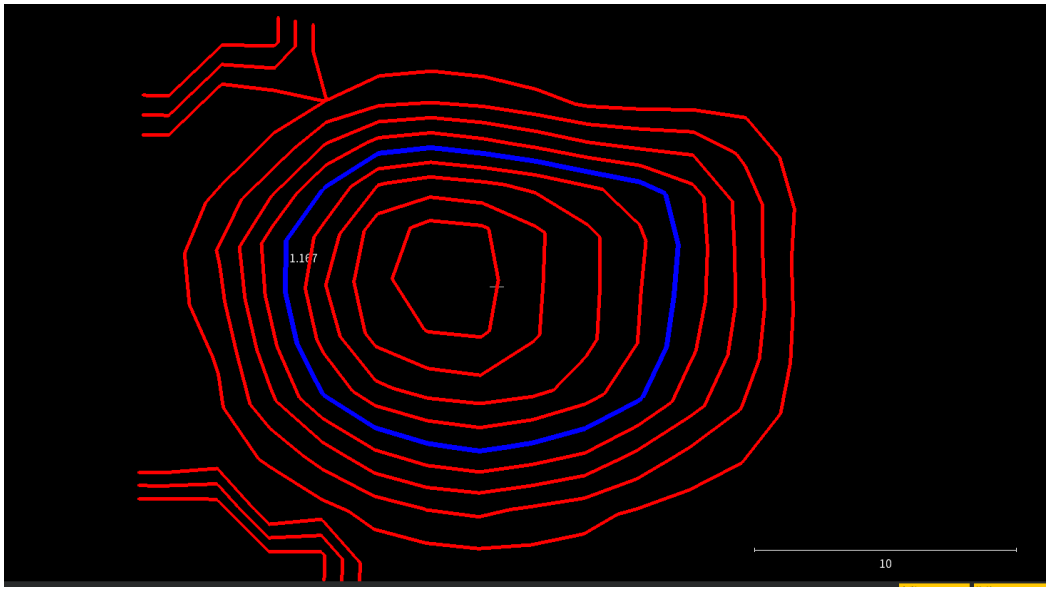



**Figure 204.** Preview of contour lines in custom colors

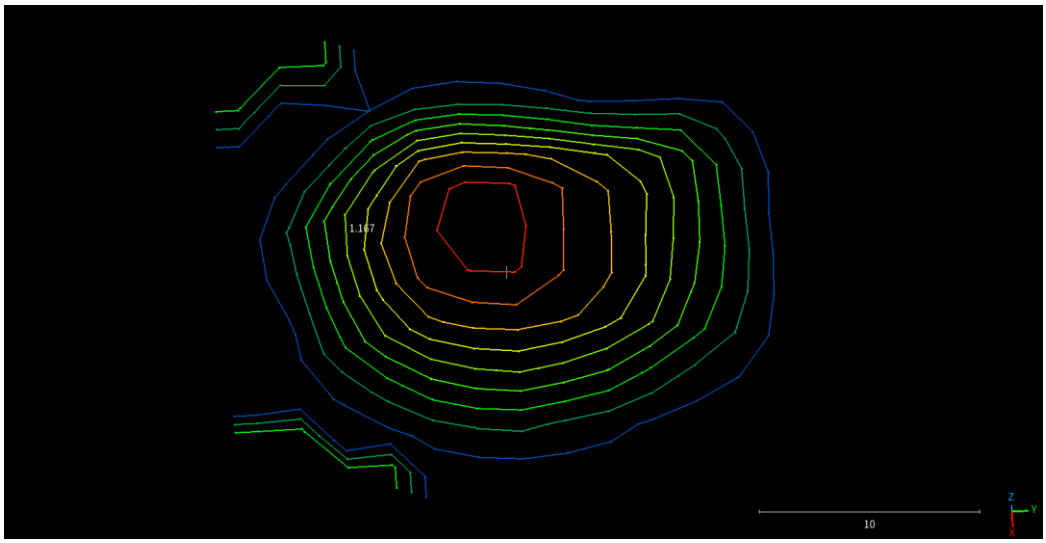



**Figure 205.** Preview of contour lines in point cloud colors

5. If the preview result is satisfactory, click **Save** to generate the contour lines. Then, only the contour lines are displayed in the 3D view area. The contour lines can be exported to a DXF file, which can be imported in other third-party software.

**Parameters:**

- Grid spacing: grid step. You can use this parameter to check whether the grid is too big or too small before generating it.

- Minimum elevation: minimum value of elevation.
- Maximum elevation: maximum value of elevation.
- Contour interval: height difference between two adjacent contour lines.
- Min. number of vertices: minimum number of vertices per contour line, used to remove small contours such as trees and cars.

### 6.5.3 Fill Holes

**Description:**

Fills holes of a triangular mesh through parameter setting to obtain an enclosed mesh.

**Operation procedure:**

1. Select a triangular mesh file under **Objects**.
2. Click on the **Edit** tab, and the following window appears.

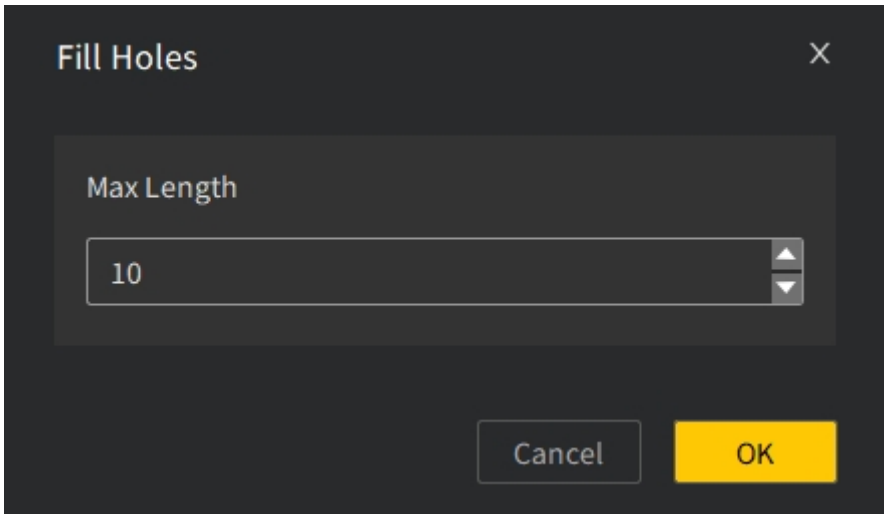



**Figure 206.** Setting the parameter

3. Click **OK** to start filling holes. The following window appears. When the process is completed, the window disappears.

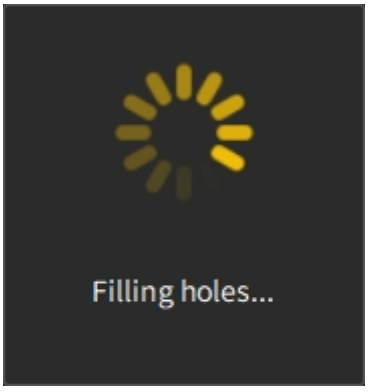



**Figure 207.** Filling holes

The following figure shows a triangular mesh with holes filled.

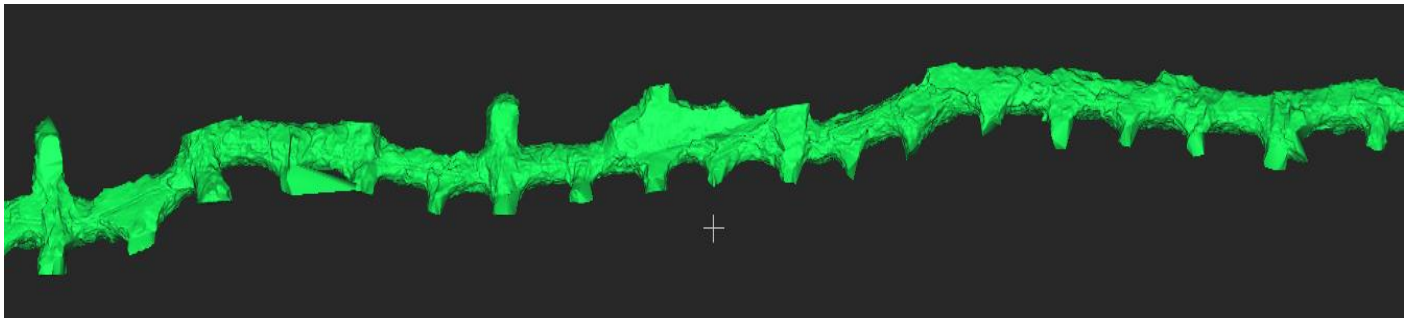


**Figure 208.** Triangular mesh with holes filled

### 6.5.4 Smooth

**Description:**

Adjusts the coordinates of points in the data set and divides polygons on the mesh to smooth the selected mesh and improve the appearance of the mesh and the shape of the cell data set. After smoothing, the data set changes in geometry, not in topology. Mesh smoothing can improve the appearance of is surfaces and remove surface noise, thereby improving the model appearance dynamically.

**Operation procedure:**

1. Select a mesh file under **Objects** to activate the smoothing feature.
2. Click , and the following window appears.
3. Set the high, medium, or low smoothness grade, and click **OK**. The original mesh and the mesh smoothed are shown below. The higher smoothness grade, the higher degree of smoothing, and the more details lost; the lower smoothness grade, the lower degree of smoothing, and the less details lost.

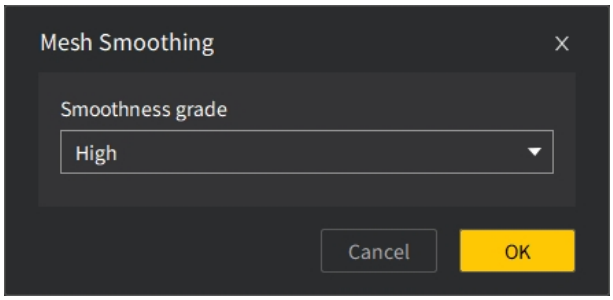



**Figure 209.** Mesh smoothing

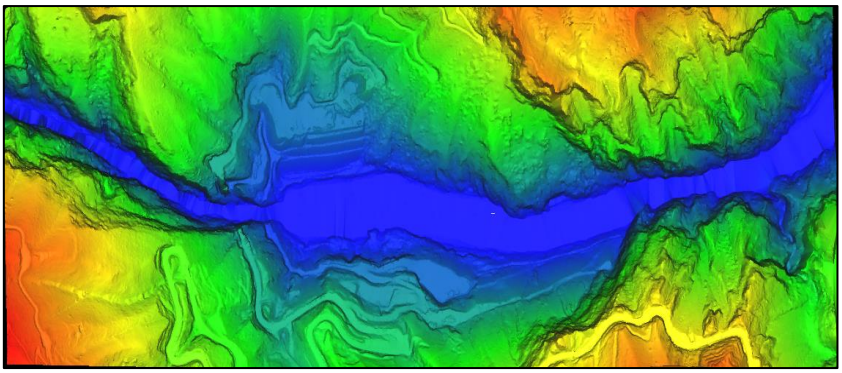


**Figure 210.** Original mesh

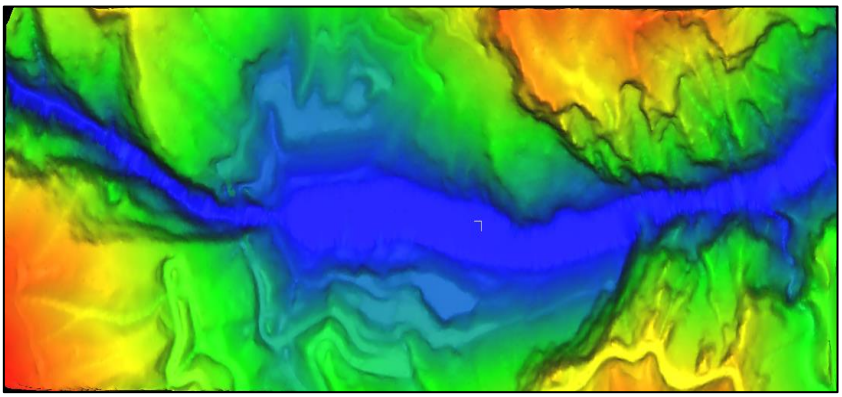


**Figure 211.** Mesh smoothed

### 6.5.5 Sample

**Description:**

Samples the vertices of the mesh according to certain rules to reduce the size of mesh data and avoid redundant data resulting from over-expression. The vertices can be sampled by percentage or grid.

**Operation procedure:**

1. Select a mesh file under **Objects** to activate the sampling feature.
2. Click , and the following window appears.
3. When selecting Percentage, the point cloud points will be optimized according to the set of Sampling percentage. When selecting Grid, the point cloud points will be optimized according to the set grid resolution.

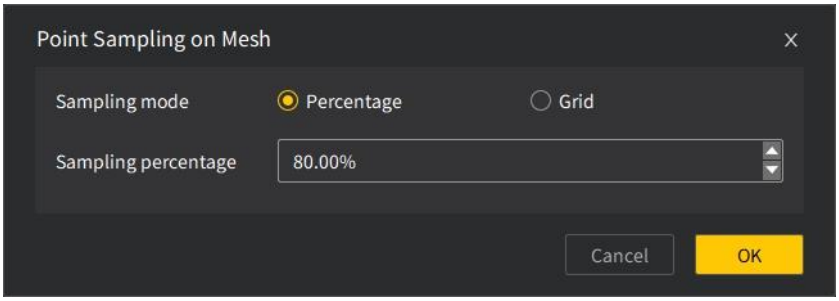
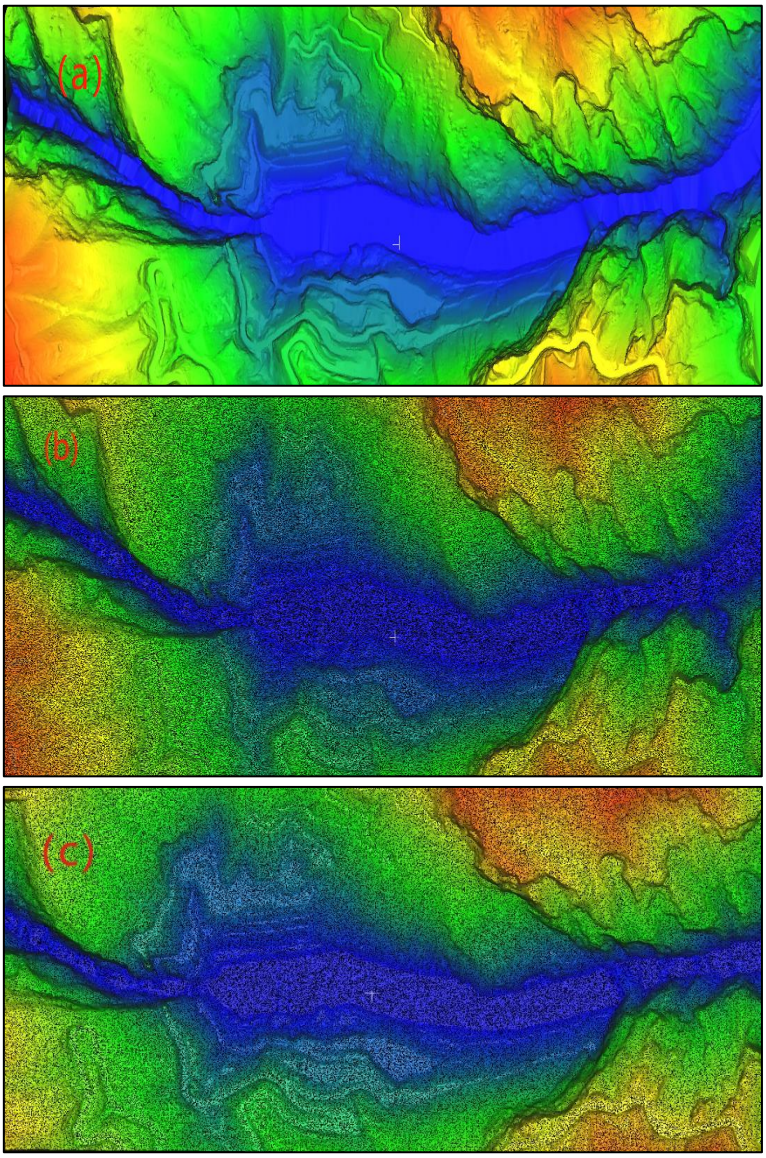



**Figure 212.** Sampling




(a) Original mesh; (b) sampling by percentage; (c) sampling by grid

**Figure 213.** Sampling results

# 6.6 Volume Calculation

## 6.6.1 Enclosed Volume

**Description:**

Constructs a surface triangular mesh from the point cloud of an object to measure its volume. It is

applicable to scenarios such as tanks, mine tunnels, mine pits, and karst caves.

**Operation procedure:**

1. Select a surface triangular mesh file under **Objects**.
2. Click on the **Edit** tab, and the volume calculation result is shown as below.

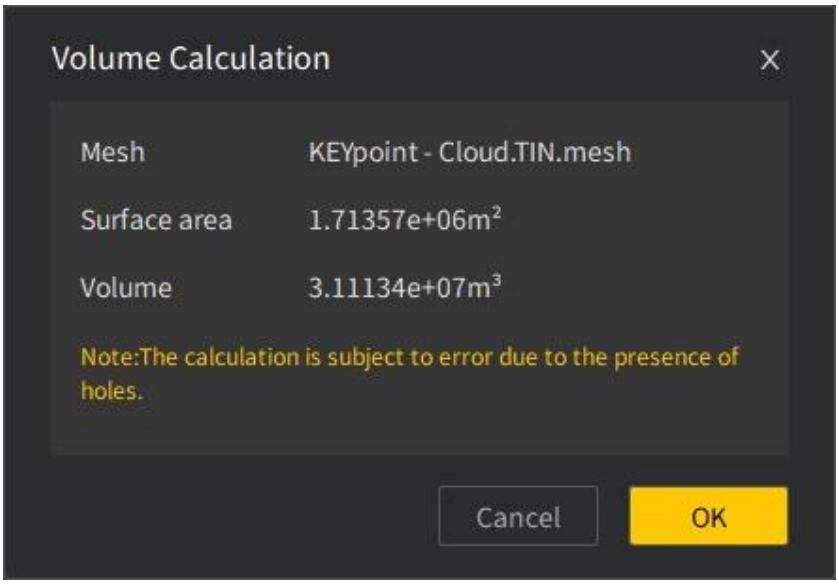



**Figure 214.** Calculation result (with holes)

**Note:**

The enclosed volume calculation works better in scenarios with enclosed volume and may have large errors when used in other scenarios.

### 6.6.2 Grids Volume

**Description:**

Calculates the volume and surface area according to the point cloud. This function can be used in scenarios such as cut and fill, piles, mining areas, caves, cultural relics protection, and vessel displacement calculation.

**Operation procedure:**

1. Select a point cloud file under **Objects** to activate the volume calculation feature.
2. Click , and the following window appears.

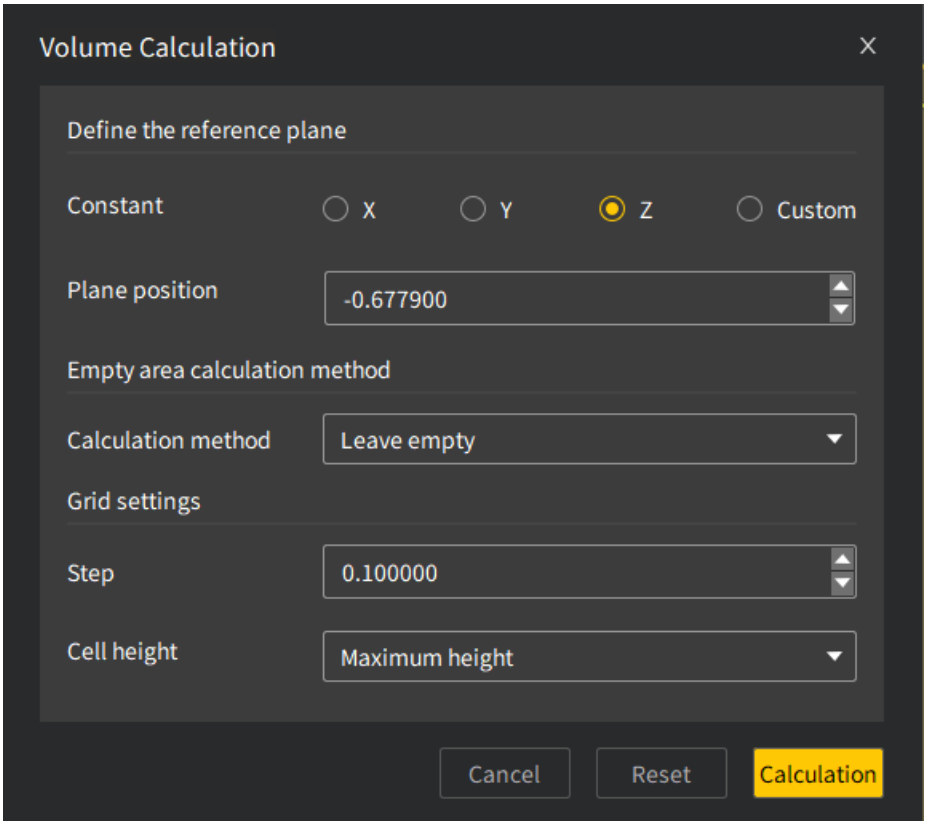



**Figure 215.** Volume calculation

3. Set the constant and plane position. When **Constant** is set, **Plane position** is set to the value of the lowest plane in the corresponding direction by default and can be adjusted based on actual conditions.
4. Select the calculation method from **Leave empty**, **Minimum height**, **Average height**, **Maximum height**, **User specified value**, and **Interpolate**.
5. Set the grid parameters.
   **Step**: Side length of the square, in m. The smaller step, the higher calculation accuracy.
   **Cell height**: Method of calculating the cell height when there are different points on the cell. The options are **Minimum height**, **Average height**, and **Maximum height**.
6. After parameter settings, click **Calculation** to obtain the volume, floor area, added volume, and removed volume.

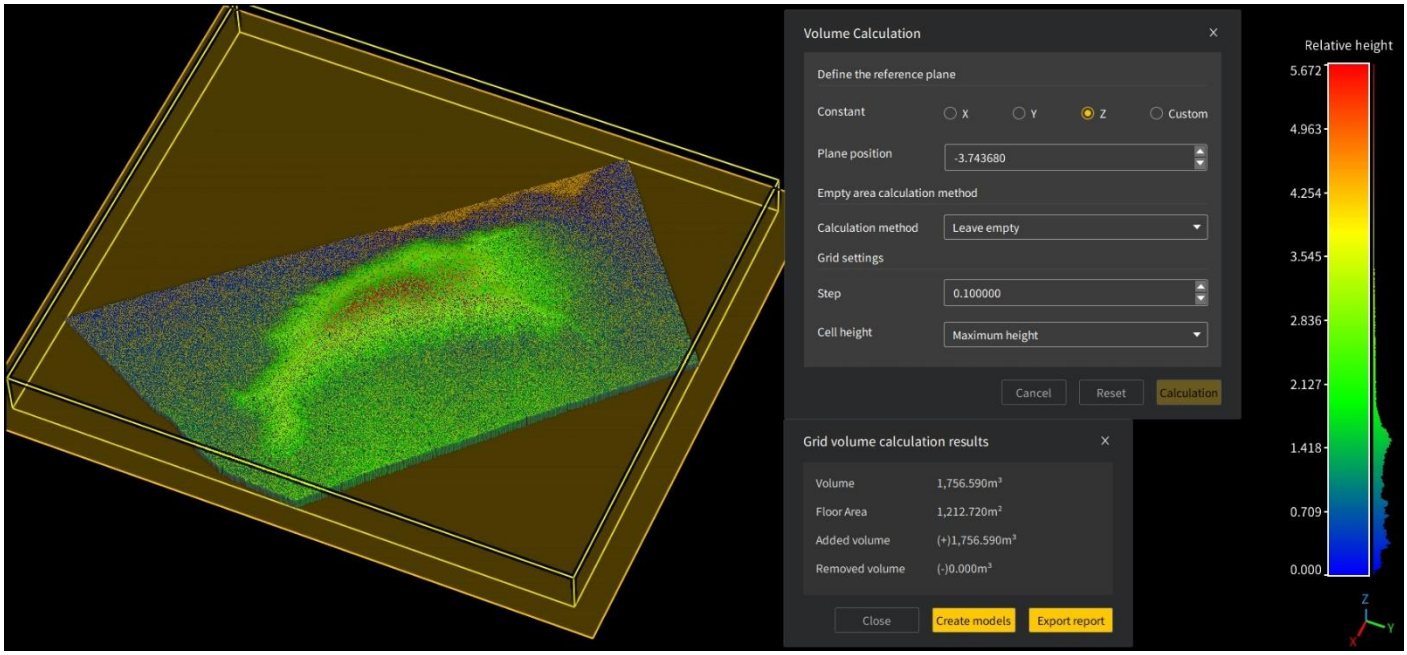


**Figure 216.** Volume calculation results

**Note:**

When **Constant** is set to **Custom**, a projection plane is created by picking points on the point cloud and used as the basis for calculation of the volume, floor area, and more.

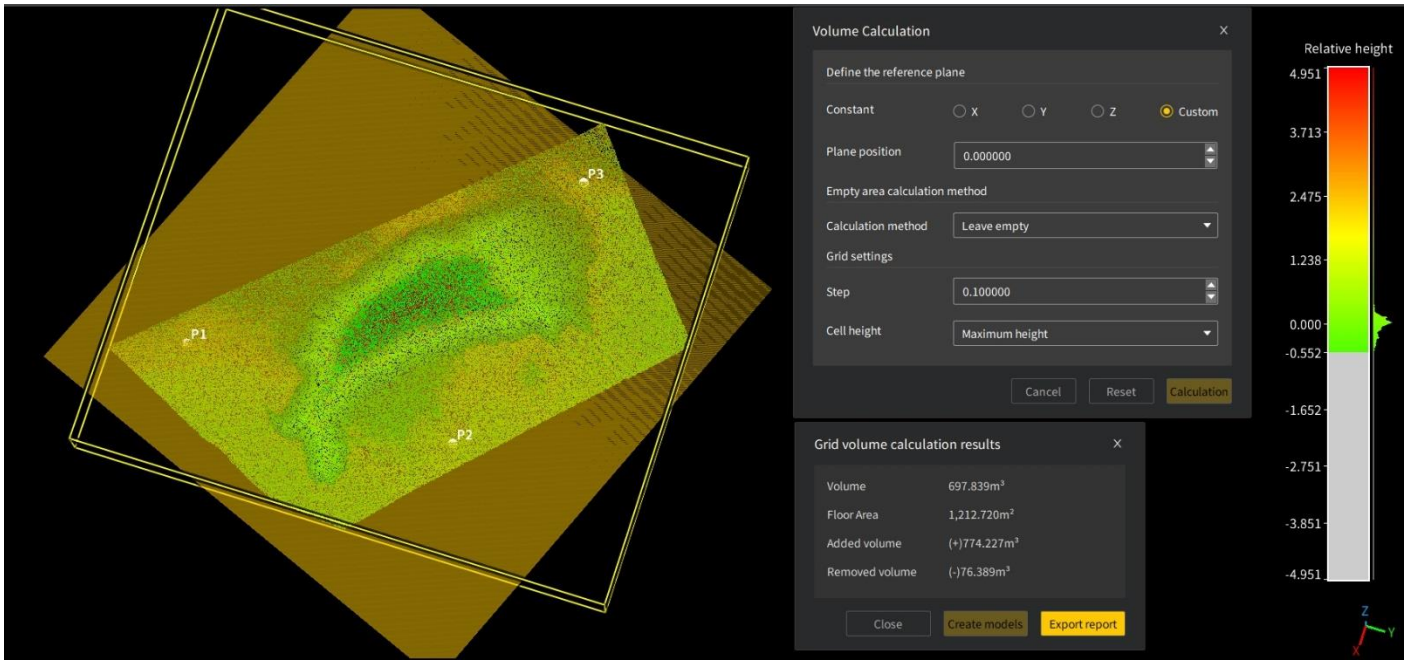



**Figure 217.** Volume calculation results with a custom constant

## 6.6.3 Two-phase Comparison

**Description:**

Calculates the volume difference of two point clouds.

**Operation procedure:**

1. Select two point cloud files under **Objects** to activate the two-phase comparison feature. Click  , and the following window appears.

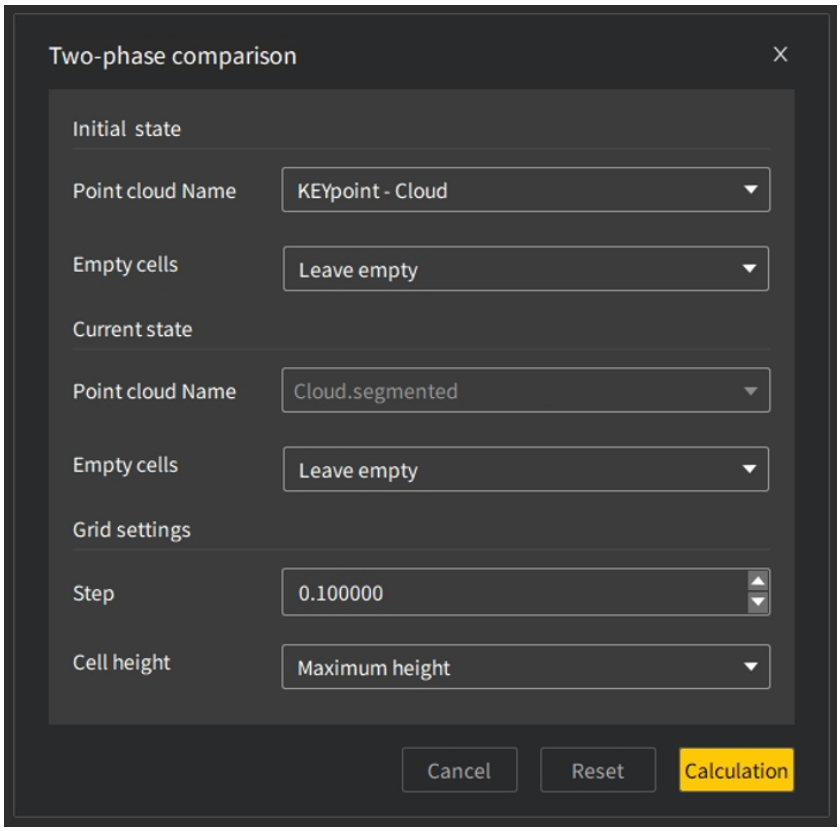


**Figure 218.** Two-phase comparison settings

2. By default, the first file selected is the initial state point cloud file, and the second file selected is the current state point cloud file.
3. Click **Calculation**, and the calculation result window appears. You can change the parameters, and click **Calculation** again. The calculation results are updated on the window.

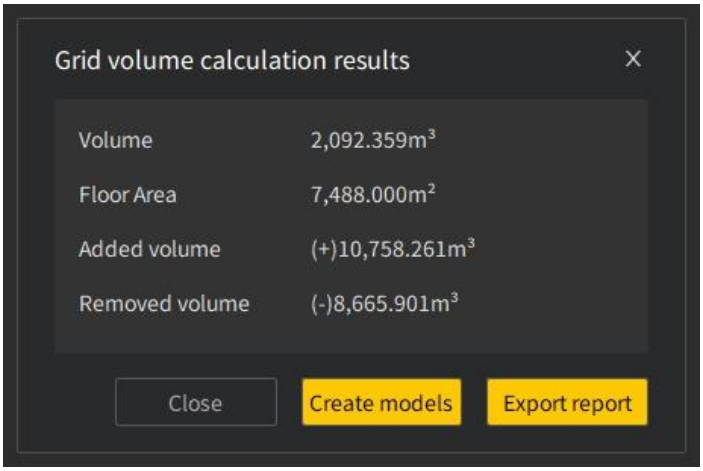



**Figure 219.** Two-phase comparison results

# 7 Display

## 7.1 View

### 7.1.1 Background

**Description:**

Changes the background color of the view area to your preferences.

**Operation procedure:**

Click , select a color, and then click **OK** to apply it. The default background color is black. The display effects before and after changing the background color are shown below.

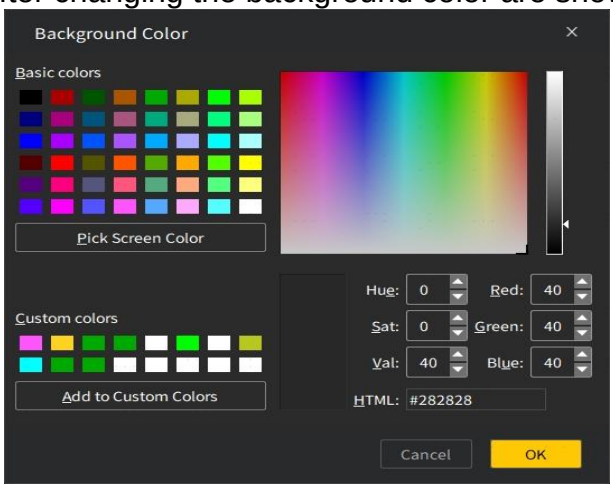



**Figure 220.** Changing the background color

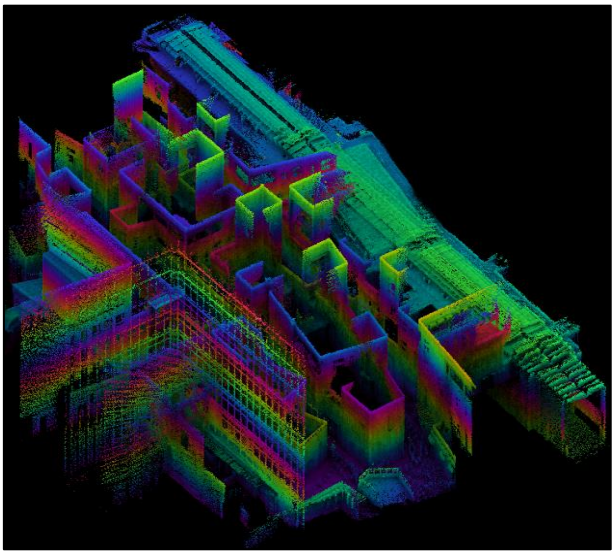


**Figure 221.** Default background color

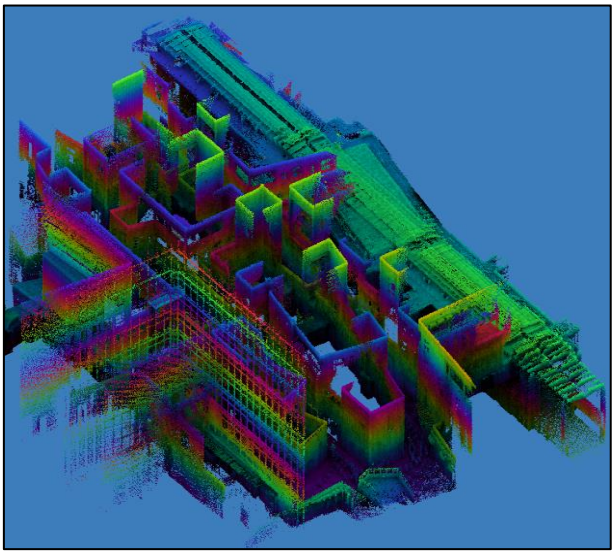


**Figure 222.** Background color changed

### 7.1.2 Direction

Displays views of files opened under **Objects**. Click , and select a view from the drop-down menu.

• 3D View

**Description:**

Displays the data in the 3D view.

**Operation procedure:**

Click **2D** to change it into **3D**, and the data is displayed in the 3D view, as shown below:

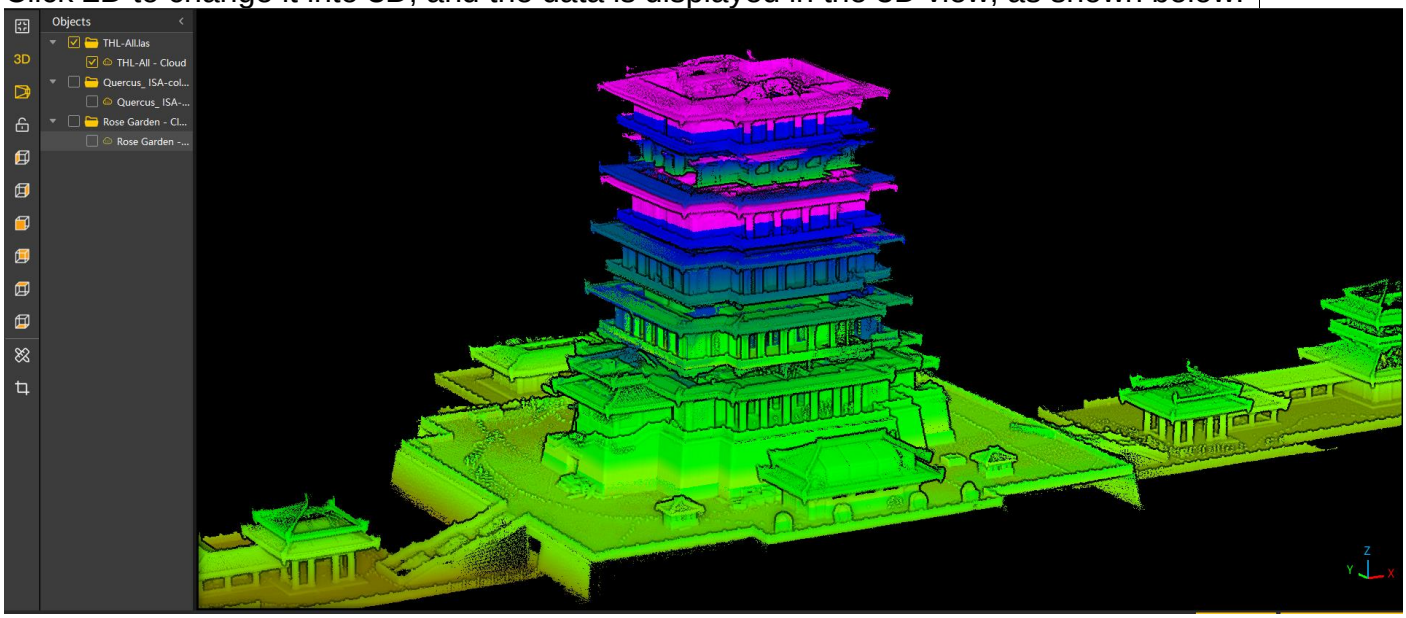



**Figure 223.** 3D view

• 2D View

**Description:**

Displays the data in the 2D view.

**Operation procedure:**

Click **3D** to change it into **2D**, and the data is displayed in the 2D view, as shown below:

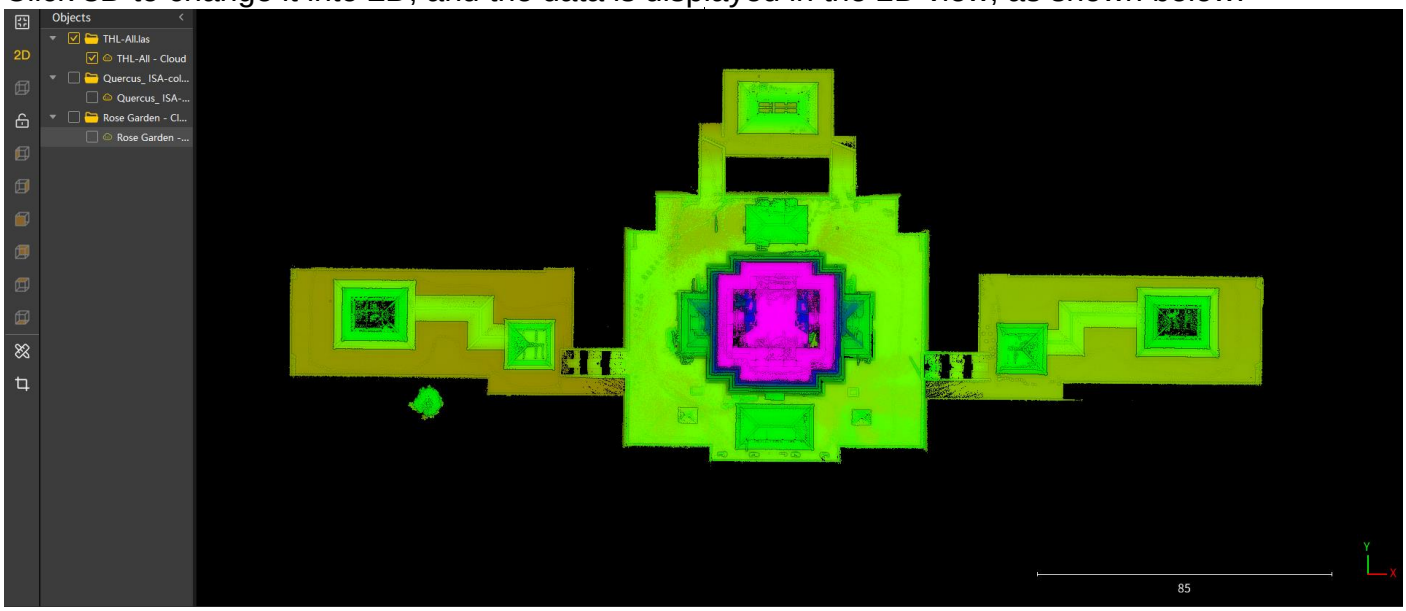



**Figure 224.** 2D view

• Left Side View

**Description:**

Select **Left Side View** to obtain the 3D data on the Y-Z plane from -X to +X direction.

**Operation procedure:**

- Click  to show the following view:

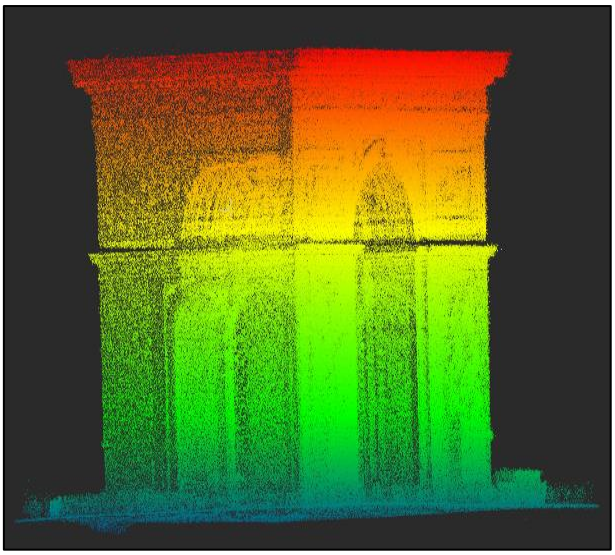


**Figure 225.** Left side view

• Right Side View

**Description:**

Select **Right Side View** to obtain the 3D data on the Y-Z plane from +X to -X direction.

**Operation procedure:**

- Click  to show the following view:

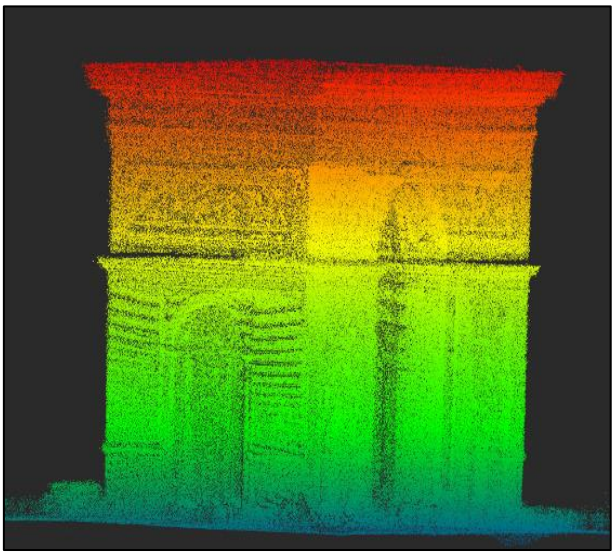


**Figure 226.** Right side view

• Front View

**Description:**

Select **Front View** to obtain the 3D data on the X-Z plane from -Y to +Y direction.

**Operation procedure:**

- Click  to show the following view:

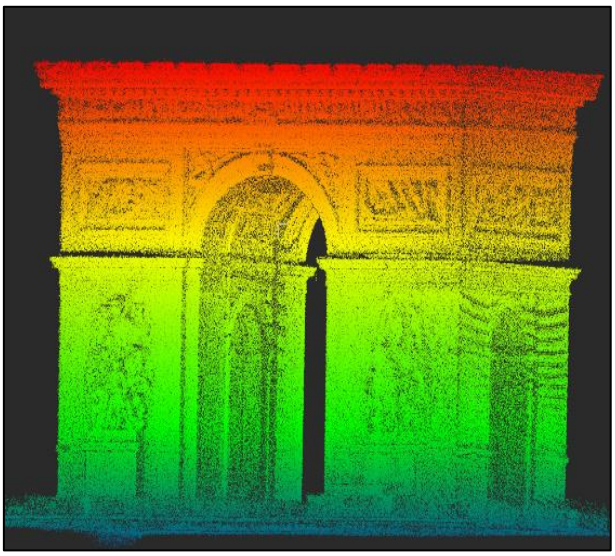


**Figure 227.** Front view

• Back View

**Description:**

Select **Back View** to obtain the 3D data on the X-Z plane from +Y to -Y direction.

**Operation procedure:**

- Click  to show the following view:

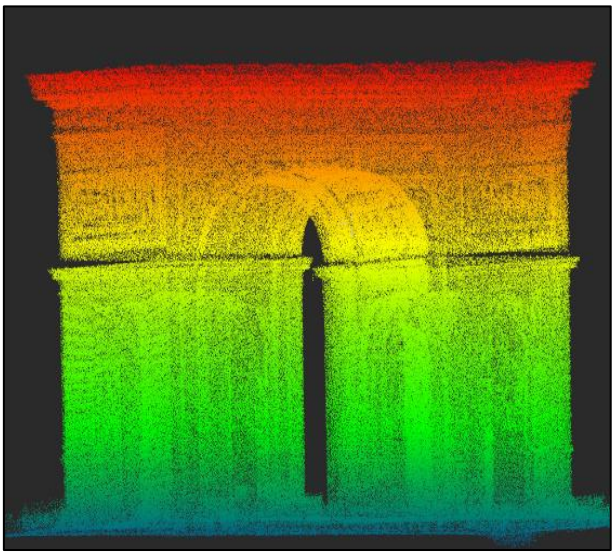


**Figure 228.** Back view

• Top View

**Description:**

Select **Top View** to obtain the 3D data on the X-Y plane from +Z to -Z direction.

**Operation procedure:**

- Click  to show the following view:

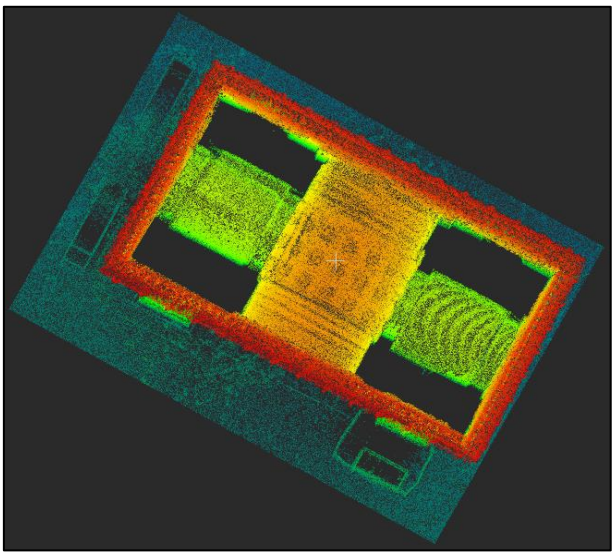


**Figure 229.** Top view

• Bottom View

**Description:**

Select **Bottom View** to obtain the 3D data on the X-Y plane from -Z to +Z direction.

**Operation procedure:**

- Click  to show the following view:

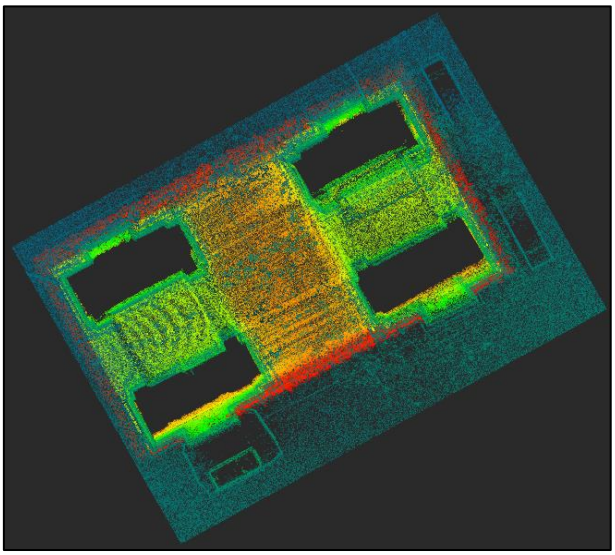


**Figure 230.** Bottom view

• Orthographic Projection

**Description:**

Orthographic projection, also known as orthogonal projection, is a means of representing three-dimensional objects in two dimensions. It is a type of parallel projection where all the projection lines are orthogonal to the projection plane, resulting in the cuboid-shaped view frustum. Regardless of its distance from a virtual camera, the projected object appears the same size on the screen. It is commonly used in blueprinting and computer aided design.

**Operation procedure:**

- Click  , and the point cloud is displayed as follows.

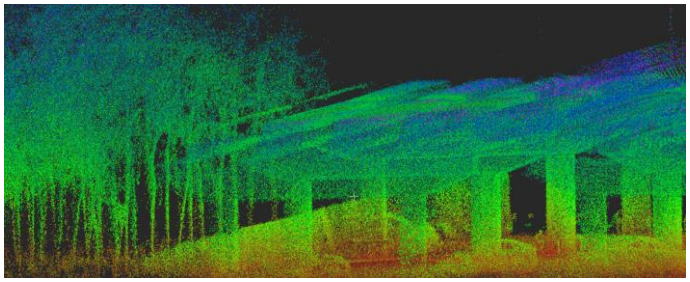


**Figure 231.** Orthographic projection effect

• Perspective Projection

**Description:**

In perspective projection, the projection size of an object is negatively related to its distance from the viewpoint. Its view frustum is shaped like a truncated pyramid. It is widely used for animation, visual simulation, and many other scenarios requiring a true reflection of objects.

**Operation procedure:**

- Click  , and the point cloud is displayed as follows.

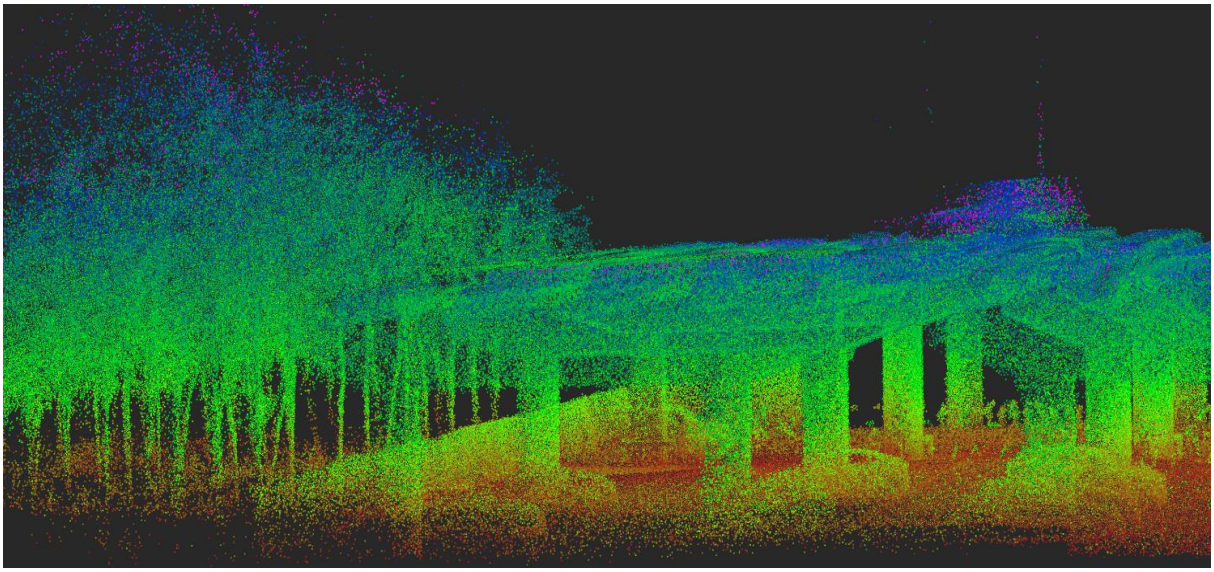


**Figure 232.** Perspective projection effect

**Note:**

The system switches to the orthographic projection effect to ensure the readability of any 2D data in the 3D view area.

### 7.1.3 Clipping Box

**Description:**

Adjusts the dimensions and position of a clipping box to show the points inside the box and hide the points outside the box. The point cloud subset inside or outside the box can be further clipped to form a new cloud. It is often used for denoising or extracting required zones in certain scenarios.

**Operation procedure:**

1. Select a point cloud file under **Objects**.
2. Click the **Display** tab, and select **Clipping Box**. Before any operation, , , , and are disabled, and the initial clipping box completely overlaps with the point cloud bounding box.

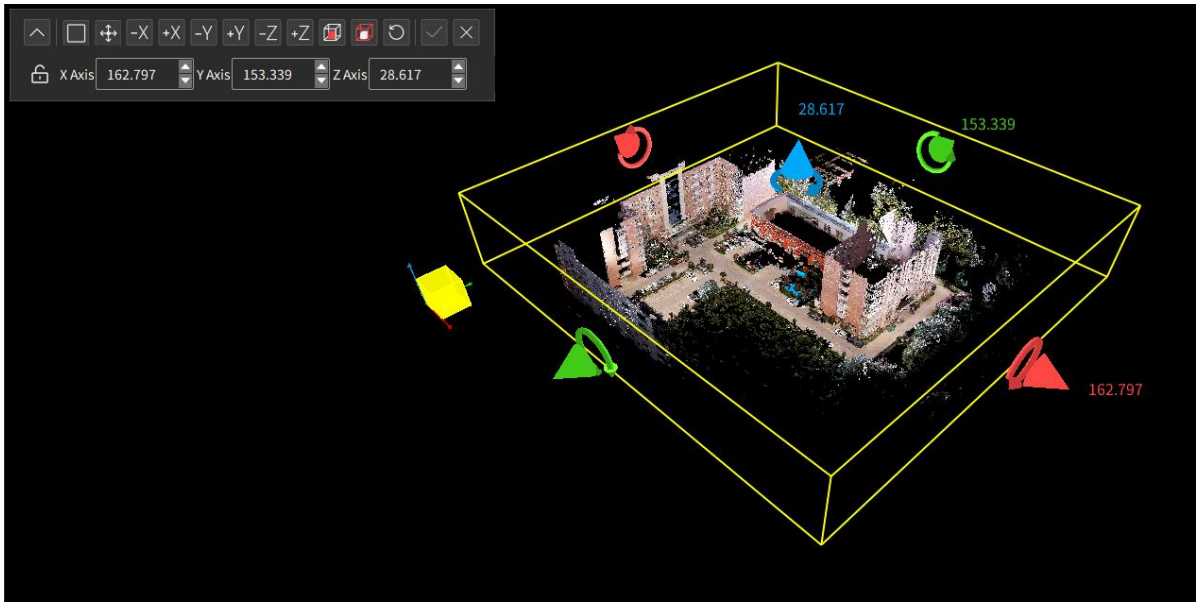



**Figure 233.** Initial clipping box

3. Click to set the box dimensions. After clicking the lock button, only rotation and translation of the cutting box are supported.
4. Click to shift the box by the box length, width, or height in the -X, +X, -Y, +Y, -Z, or +Z direction. Drag the arrow tips to adjust the box face positions, and use the tori to rotate

the box around the arrow axes. Drag to translate the whole box. You can hide points as required through the preceding operations.

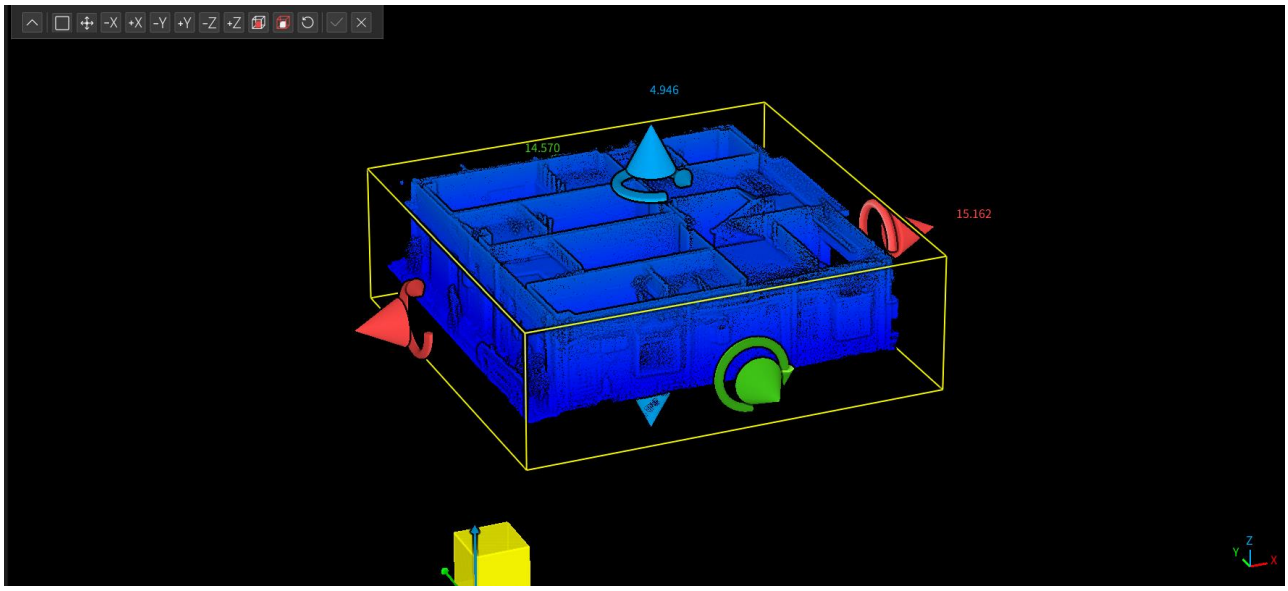



**Figure 234.** Effect of hiding the top of the point cloud

5. Once the clipping box is defined, click or to keep points inside or outside the box. Click to restore the original point cloud if you need before clicking . Click and click to exit the clipping box operation.

**Note:**

The clipping box can work with clipping, registration, coordinate transformation, manual classification, and accuracy verification to hide points blocking the planar target, spherical target, and public features, so you can accurately select information on relevant features and targets. The registration, coordinate transformation, manual classification, and accuracy verification features must be started before you can use them with the clipping box. This requirement does not apply to the clipping feature. When used with other features, the , , and buttons of the clipping box are disabled.

# 7.2 Point Display

With this feature, you can change the point cloud rendering mode and set the color scale, point size, and display range to meet requirements of different scenarios.

## 7.2.1 RGB

**Description:**

Shows the original RGB colors of a point cloud.

**Operation procedure:**

1. Select a file under **Objects**.
2. Click **RGB** on the **Display** tab, and the point cloud is displayed in RGB colors.
3. Click **Settings** to hide, display, or delete the uncolored points. Such points are displayed by default.

**Figure 235.** RGB display

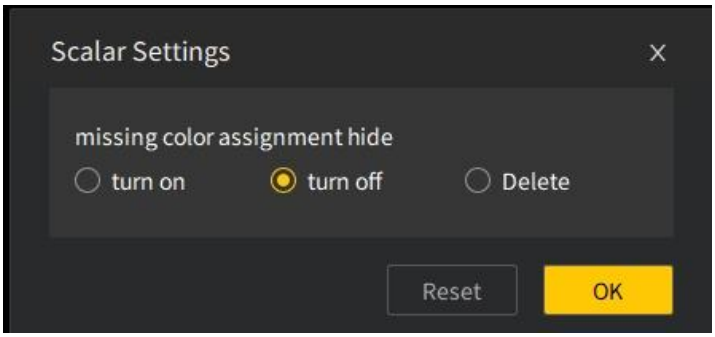



**Figure 236.** Hide uncolored points

### 7.2.2 Elevation

**Description:**

Shows the elevation variation of a point cloud through changes of colors.

**Operation procedure:**

1. Select a file under **Objects**.
2. On the **Display** tab, click **Elevation**, and the point cloud is rendered by elevation.
3. Click **Settings** to change the color scale and display range, and click **OK**. The point cloud is updated accordingly in the 3D view area.

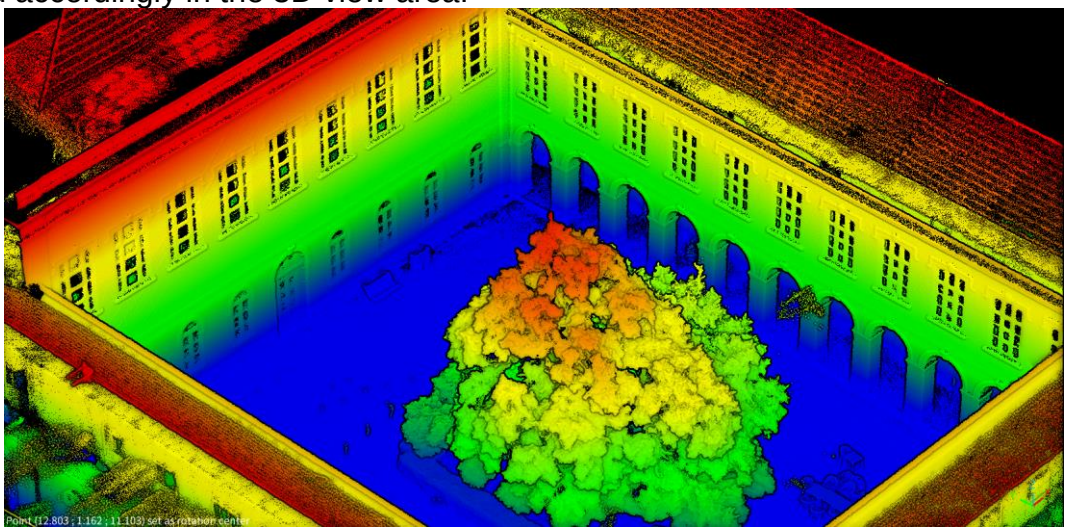


**Figure 237.** Rendering by elevation

### 7.2.3 Intensity

**Description:**

Shows the intensity variation of a point cloud through changes of colors.

**Operation procedure:**

1. Select a file under **Objects**.
2. On the **Display** tab, click **Intensity**, and the point cloud is rendered by intensity.
3. Click **Settings** to change the color scale and display range, and click **OK**. The point cloud is updated accordingly in the 3D view area.

**Figure 238.** Rendering by intensity

### 7.2.4 Time

**Description:**

Shows the time variation of a point cloud through changes of colors.

**Operation procedure:**

1. Select a file under **Objects**.
2. On the **Display** tab, click **Time**, and the point cloud is rendered by time.
3. Click **Settings** to change the color scale and display range, and click **OK**. The point cloud is updated accordingly in the 3D view area.

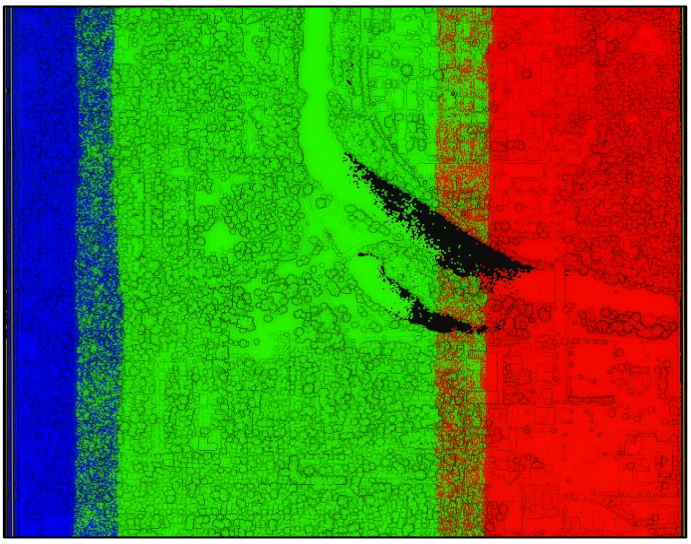


**Figure 239.** Rendering by time

### 7.2.5 Classification

**Description:**

Shows the classification variation of a point cloud through changes of colors.

**Operation procedure:**

1. Select a file under **Objects**.
2. On the **Display** tab, click **Classification**, and the point cloud is rendered by classification.
3. Click **Settings** to change the color scale and display range, and click **OK**. The point cloud is updated accordingly in the 3D view area.

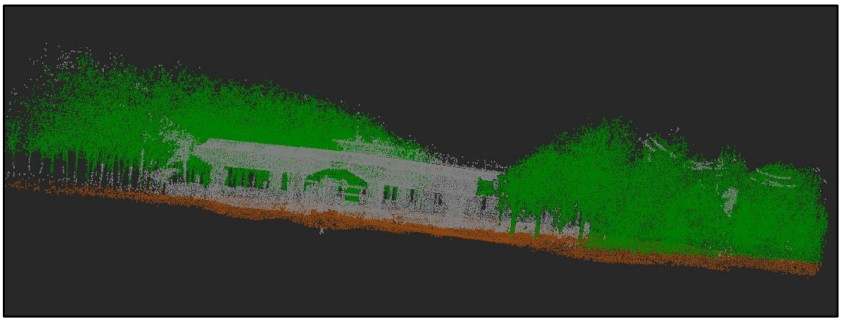


**Figure 240.** Rendering by classification

### 7.2.6 User Data

**Description:**

Shows the user data variation of a point cloud through changes of colors.

**Operation procedure:**

1. Select a file under **Objects**.
2. On the **Display** tab, click **User data**, and the point cloud is rendered by user data.
3. Click **Settings** to change the color scale and display range, and click **OK**. The point cloud is updated accordingly in the 3D view area.

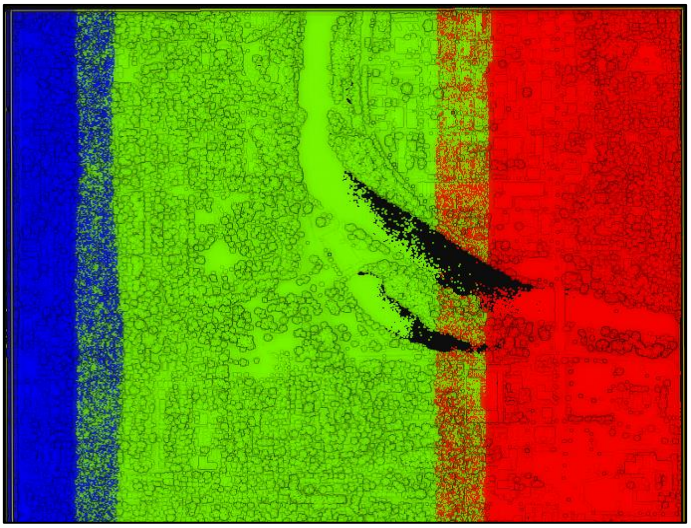


**Figure 241.** Rendering by user data

### 7.2.7 Set Unique

**Description:**

Renders a point cloud with a specified color.

**Operation procedure:**

1. Select a file under **Objects**.
2. On the **Display** tab, click **Set unique**, and the point cloud is rendered in the specified color.
3. The color scale and display range cannot be changed in the **Set unique** mode.

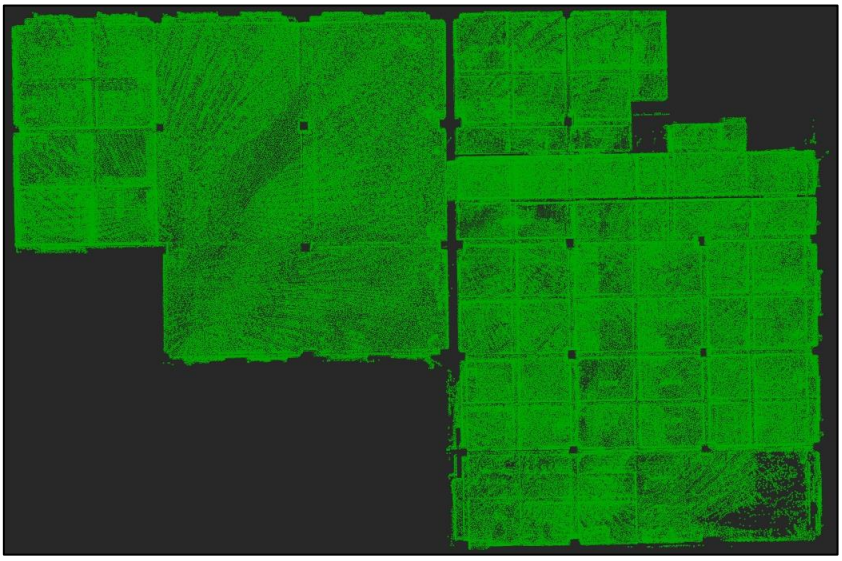


**Figure 242.** Rendering in a specified color

### 7.2.8 Blend

**Description:**

Renders a point cloud by elevation and intensity.

**Operation procedure:**

1. Select a file under **Objects**, and click the **Display** tab.
2. Click **Blend**, and the point cloud is rendered by elevation and intensity.

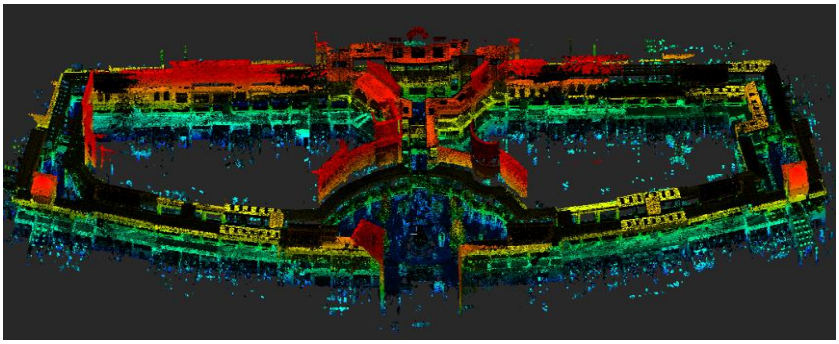


**Figure 243.** Rendering by elevation and intensity

### 7.2.9 Point Size

**Description:**

Sets the point size of any 3D point cloud in the software.

**Operation procedure:**

Click **Point size** and select a proper value from the drop-down list. The following figures show the display effects of different point sizes. The point size is 1 by default and can be set to **Auto** to ensure the display effect when you zoom in or out on the point cloud.

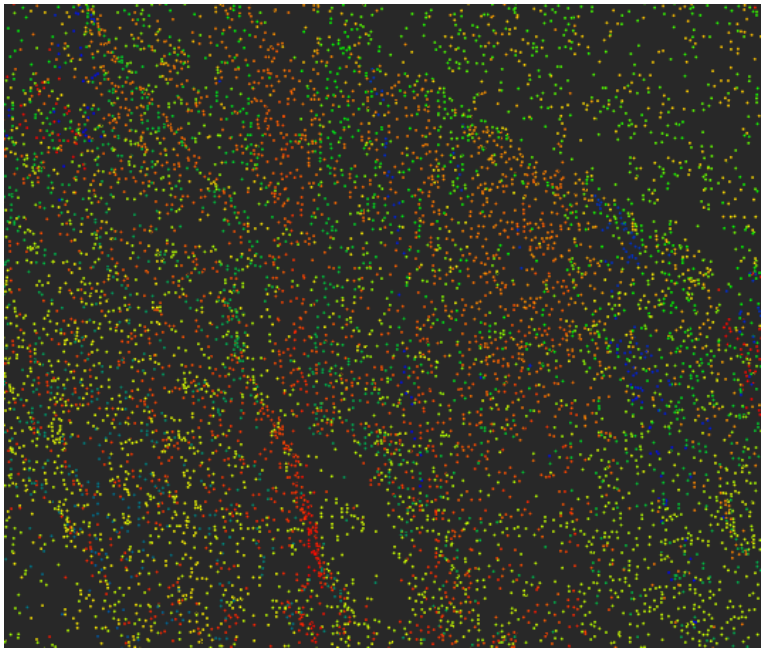


**Figure 244.** Before changing the point size

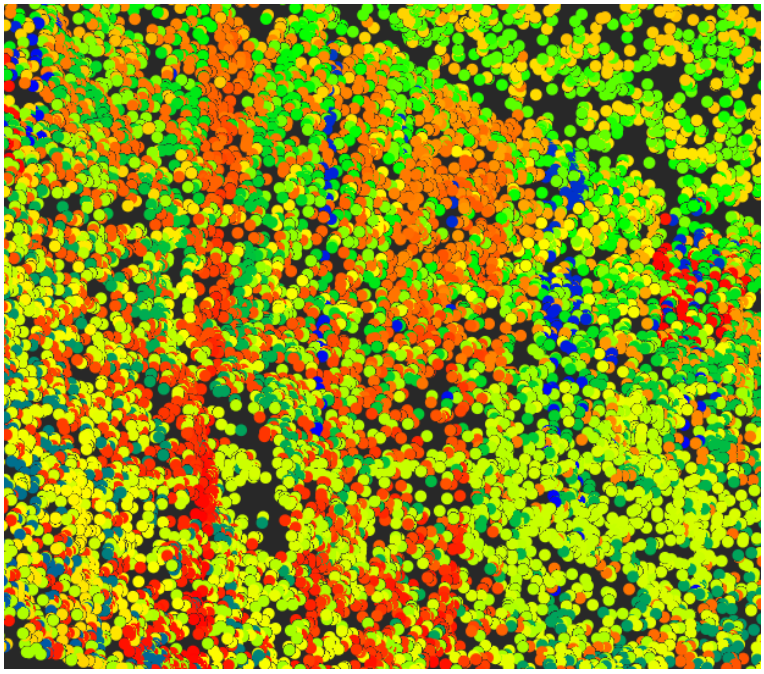


**Figure 245.** After changing the point size

## 7.2.10 Settings

**Description:**

Sets the display scalar and saturation based on the selected rendering mode of the point cloud.

**Operation procedure:**

1. Select a file under **Objects**.
2. Click one rendering mode.
3. Click  to adjust the display scalar and saturation. The following figure shows the **Scalar Settings** window for rendering by time.

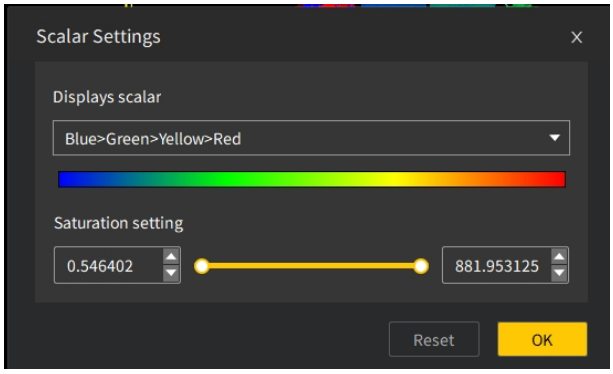



**Figure 246.** Scalar settings for rendering by time

**Note:**

- The color scale changes according to what you select from the **Displays scalar** drop-down list.
- Drag the slider or enter values to adjust the saturation. The point cloud 3D view is updated accordingly.
- By default, the color scale is blue-green-yellow-red, and the range is specified by the upper and lower limits of the corresponding point cloud data.

### 7.2.11 Boundary Reinforcement

**Description:**

Works with other rendering modes to enhance the outline display of point cloud features.

**Operation procedure:**

1. Select a file under **Objects**.
2. On the **Display** tab, click . The following figures show the effects before and after boundary reinforcement.

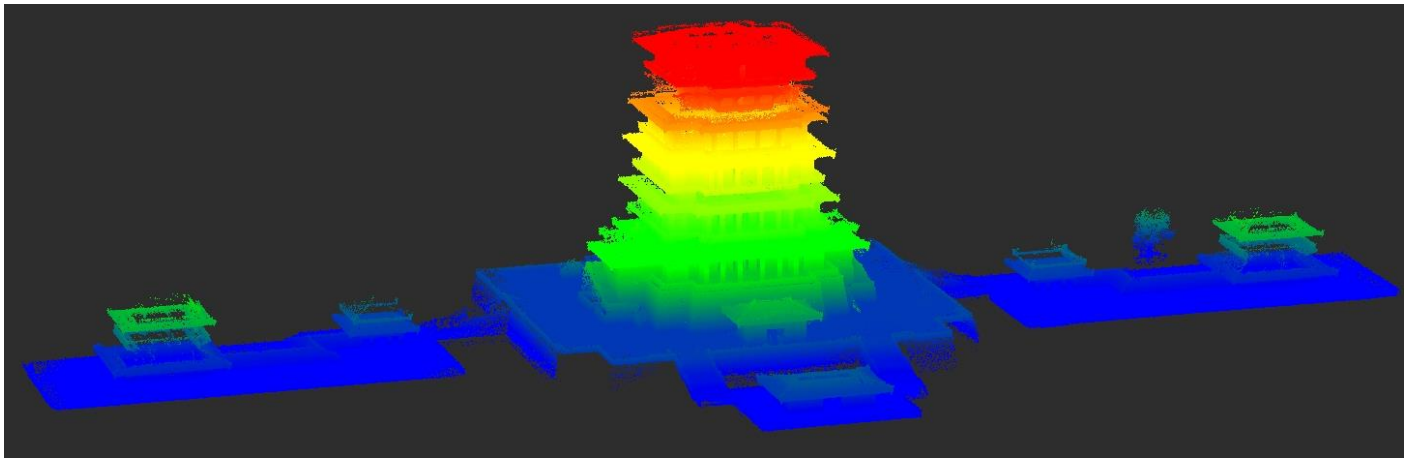


**Figure 247.** Before boundary reinforcement

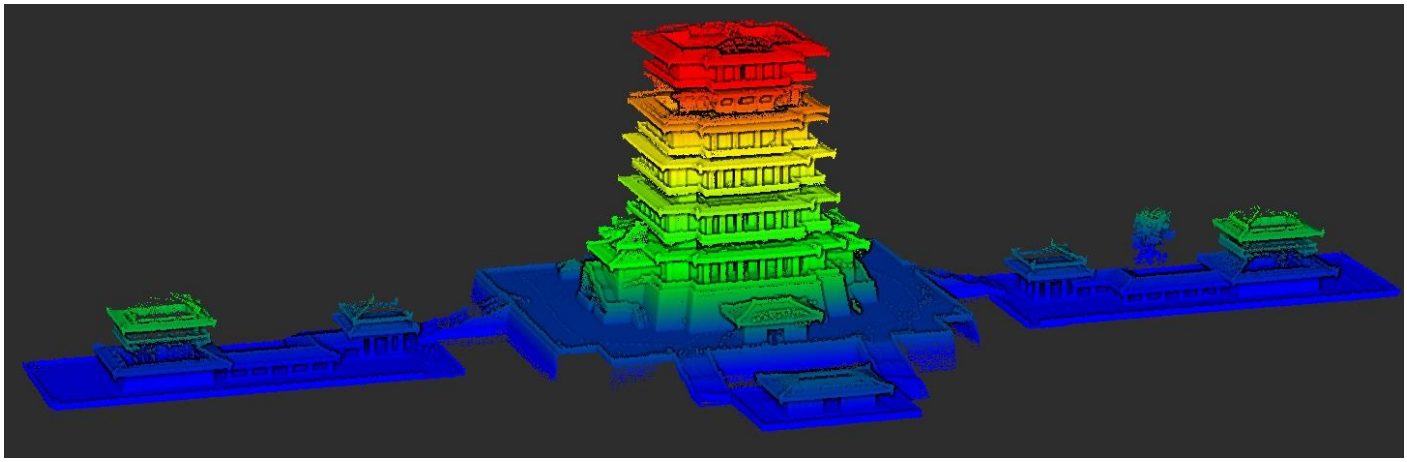


**Figure 248.** After boundary reinforcement

### 7.2.12 X-Ray

**Description:**

Semi-transparent display point cloud, improve the three-dimensional sense of point cloud.

**Operation procedure:**

1. Left mouse click in the project file to select the data;
2. Click the X-ray function button in the menu bar, the point cloud in the view is displayed as a perspective transparent effect.

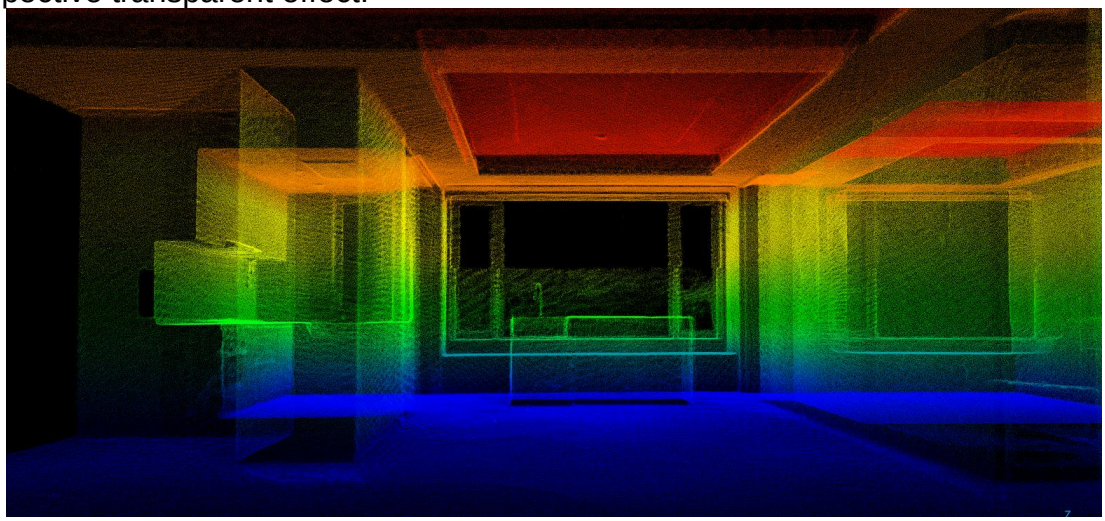


**Figure 249.** X-ray display

### 7.2.13 Light Enhancement

**Description:**

Enhances point cloud visualization to better reflect the features and boundaries.

**Operation procedure:**

1. Select a file under **Objects**.
2. On the **Display** tab, click , and the following window appears.

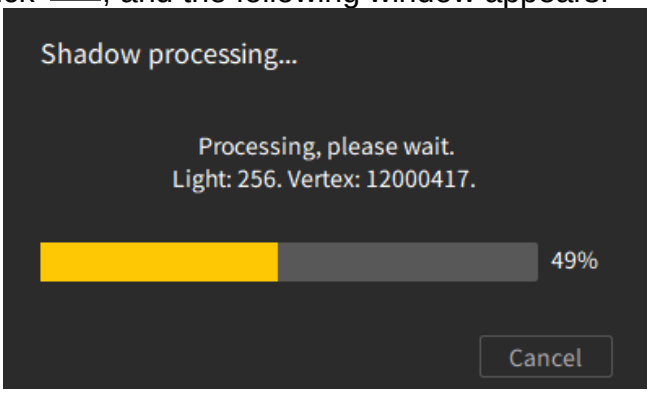



**Figure 250.** Shadow processing progress window

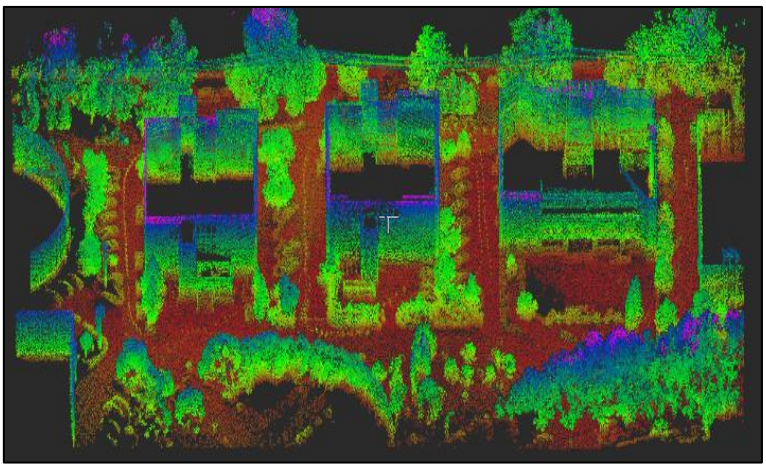


**Figure 251.** Before light enhancement

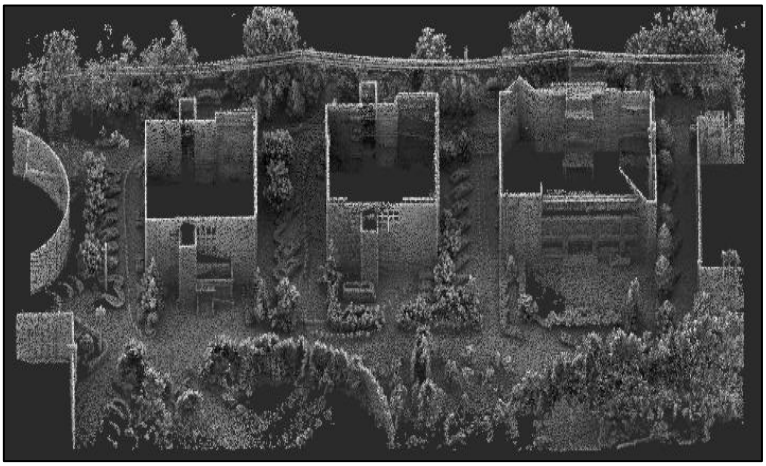


**Figure 252.** After light enhancement

# 7.3 Image Fusion

Integrates the point clouds and panoramic images to generate more realistic scene data.

## 7.3.1 Fusion

**Description:**

Performs fusion of a point cloud with its video data and configuration file.

**Operation procedure:**

1. Select a point cloud file under **Objects**, and the icon  becomes available. It is unavailable if you select multiple or no files. Click the icon, and the following window appears.

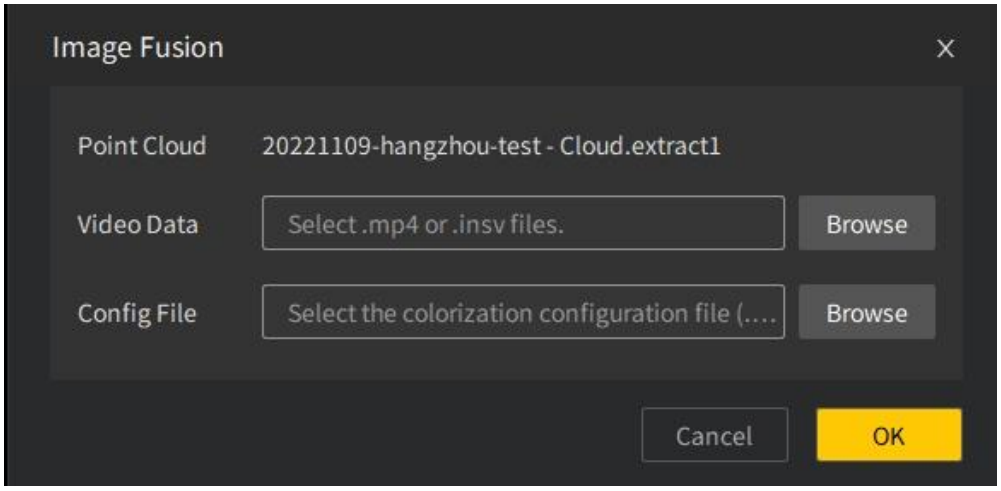



**Figure 253.** Fusion settings

2. Select the video data and configuration file, and click **OK**.
3. The message "Image fusion in progress, please wait" appears. If any required file is missing, the corresponding box becomes red after you click **OK**.
4. If fusion is successful, the result is shown in the view area, and the **Show track points** and **Show image** check boxes become available. If fusion fails, a prompt message appears.
5. Click the icon again, the fusion effect disappears, and the **Show track points** and **Show image** check boxes become unavailable.


**Figure 254.** Fusion result

### 7.3.2 Linkage

**Description:**
Uses the video data and configuration file of a point cloud to show both the point cloud and the image of the same position at the same time.
**Operation procedure:**

1. Select a point cloud file under **Objects**, and the icon  becomes available. It is unavailable if you select multiple or no files. Click the icon, and the data selection window appears.
2. Select the right video data and configuration file, and click **OK**.
3. The message "Image linkage in progress, please wait" appears. If any required file is missing, the corresponding input box becomes red after you click **OK**.
4. If linkage is successful, the result is shown in the view area. Show or hide the point cloud, image, or track points as required. Click the cross in the upper right corner to exit the feature. If linkage fails, a prompt message appears.

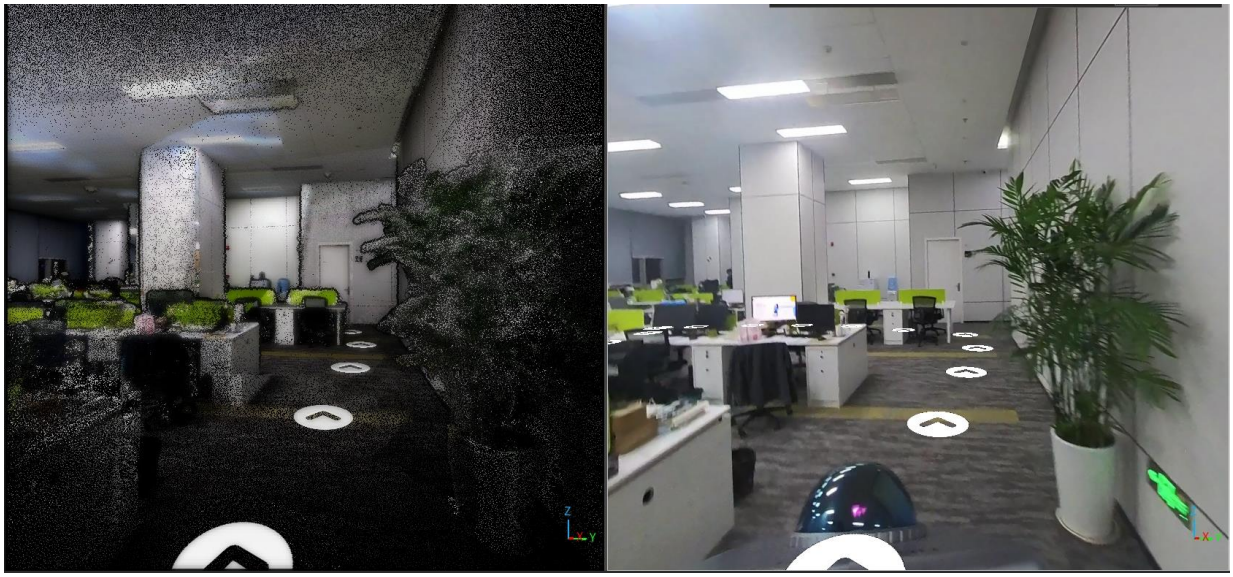


**Figure 255.** Linkage result

### 7.3.3 Panorama Image Export

**Description:**
Output panoramic images and corresponding position information at a fixed distance or time.
**Operation procedure:**

1. Select the point cloud data to export the panoramic image, and then click the function button to pop up the function interface;

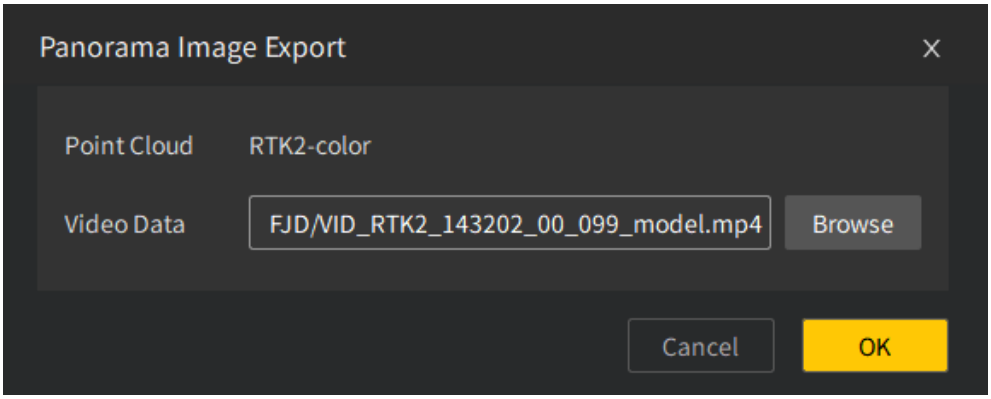



**Figure 256.** Panorama export interface

2. If the point cloud data has already loaded the fjdata configuration file, the video loading interface will pop up directly. Otherwise, the configuration file and video loading interface will pop up. The video loading interface supports 1 mp4 file or 2 insv files.
3. After clicking "OK", the video will be automatically loaded at the bottom of the view, and the export menu will be displayed in the upper right corner;
4. There are two extraction methods, "Extract by Distance" and "Extract by Time Interval". After selecting different extraction methods, the text box will change accordingly. When selecting distance extraction, enter the default value as 1.5m, with a range of values (0, 99); when selecting time extraction, enter the default value as 5s, with a range of values (0.2, 999);

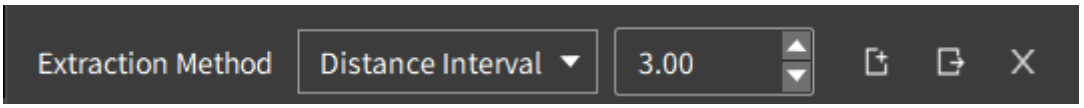



**Figure 257.** Panorama extraction mode settings

5. After entering the relevant parameters, click the "Generate" button to display the corresponding panoramic image information in the lower view area;

**Figure 258.** Panorama preview

6. After confirming the output image information, click the "Export" button to export the panoramic image;

7. After the user completes the data export operation, he can click the exit button to exit the panoramic export function;

# 7.4 Create video

Creates and exports point cloud videos for purposes such as project reporting and promotion.

## 7.4.1 By Track

**Description:**
Creates a video with the track points generated during scanning.
**Operation procedure:**

1. Select a point cloud file under **Objects**.
2. Click  , and the following window appears.
3. Set the frame rate and bit rate.

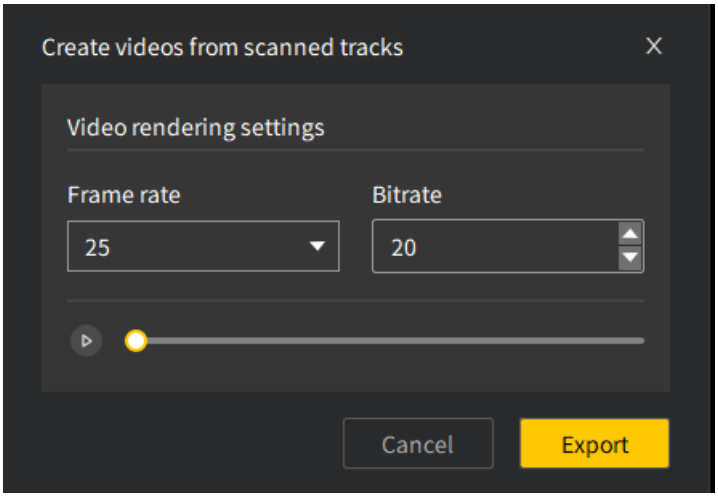



**Figure 259.** Create a video by track

4. Click the play button to preview the effect, and click it again to pause. Drag the slider to change the view image to the corresponding frame.
5. If the effect is satisfactory, click **Export** and configure settings to export the video. The export progress bar is displayed.
6. Completing the export closes the progress bar. The view area remains the condition before **Export** is clicked.

## 7.4.2 By Keyframes

**Description:**
Creates point cloud videos by adding keyframes.
**Operation procedure:**

1. Click  , and the following window appears.
2. Rotate, translate, or zoom in or out on the point cloud to adjust the view. Click **+Add** or hold down the Ctrl+V keys to add keyframes.

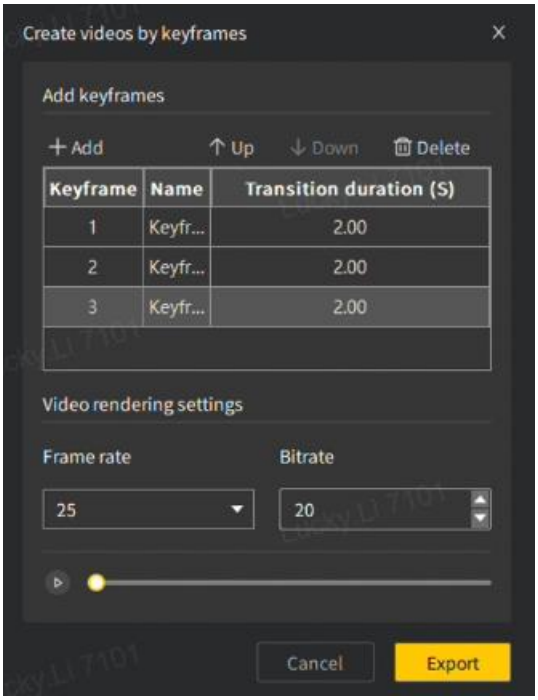



**Figure 260.** Create a video by adding keyframes

3. Click any keyframe from the list to adjust the view to the selected keyframe. Click **Up** or **Down** to move it up or down, or click **Delete** to delete it from the list.
4. The name of each keyframe and the time interval between two consecutive keyframes are editable. The default name is Keyframe*X*, where X refers to the keyframe number; the default time interval is 2s.
5. Set the frame rate and bit rate.
6. Click the play button to preview the effect, and click it again to pause. Drag the slider to change the view image to the corresponding frame.
7. Adjusting keyframe settings pauses the video with the slider returning to the start point.
8. If the effect is satisfactory, click **Export** and configure settings to export the video. The export progress bar is displayed.
9. Completing the export closes the progress bar. The view area remains the condition before **Export** is clicked.

# 8 Drawing

## 8.1 Drawing Management

### 8.1.1 Draw

**Description:**

Activate drawing-related features and enter 2D vector drawing mode.

**Operation procedure:**

1. Select a point cloud or vector file in the project file.
2. Click  to enable drawing related features, you can draw vector graphics.

### 8.1.2 Assisted Drawing

**Description:**

Vector drawing is assisted by point cloud slices and panoramic images.

**Operation procedure:**

1. Select point cloud data, drawn vector objects, or both, to enter the 2D vector drawing function;
2. Click  to open the Point Cloud Assistance Window. You can switch the viewing angle, the point cloud supports rotation, translation, and zooming. By clicking to select a start point and an end point, a sliced plane is created from the point cloud to generate a “quasi-planar” point cloud dataset with adjustable position and slice thickness, which can then be used to assist in drawing.

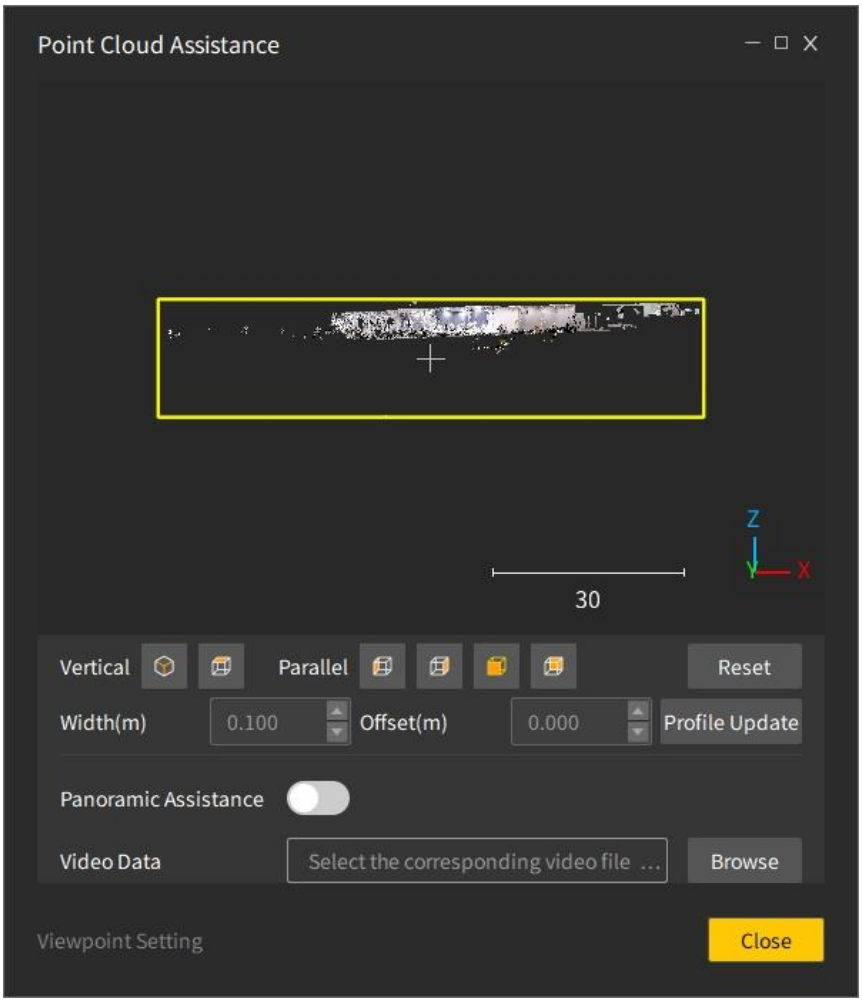



**Figure 261.** Point Cloud Assistance Window

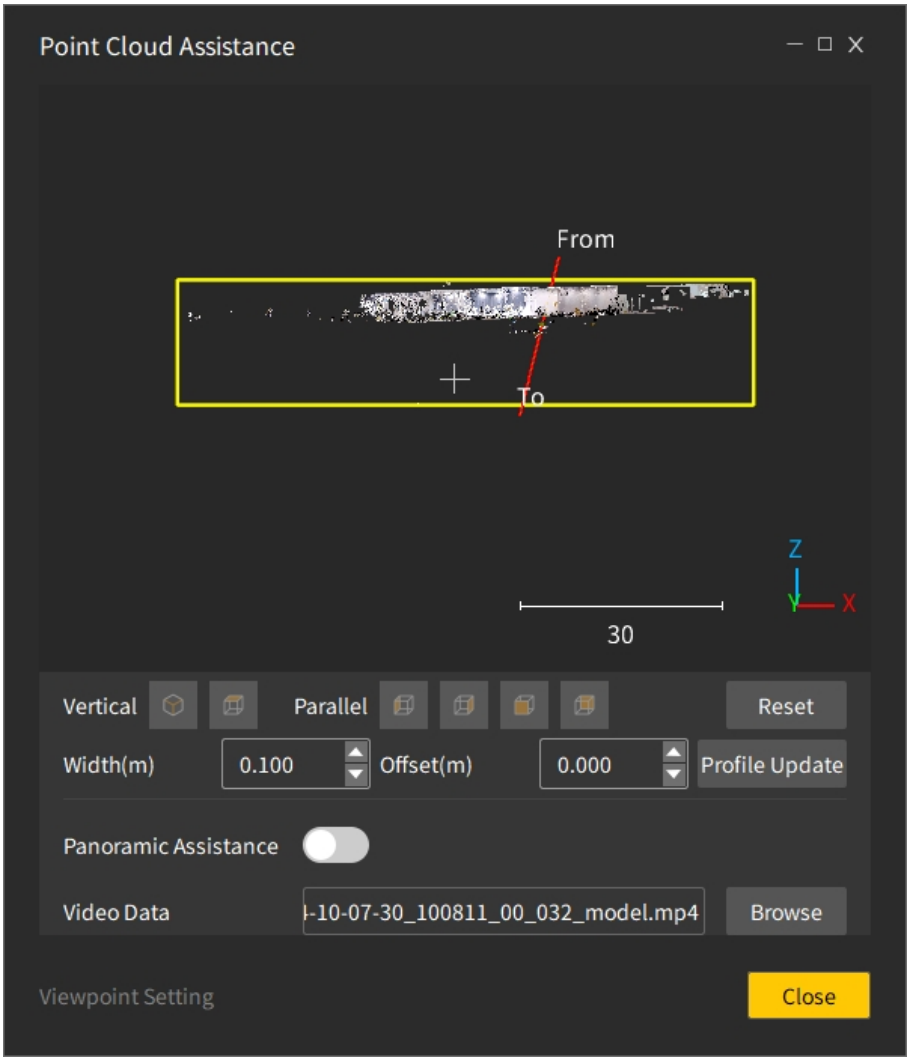



**Figure 262.** Cut a plane

3. Select the image data corresponding to the point cloud, support .mp4/.avi/.insv files, and click the 

, button to enable panoramic assistance. Double-click the track point on the point cloud to switch positions. The panoramic image supports zooming and rotation.

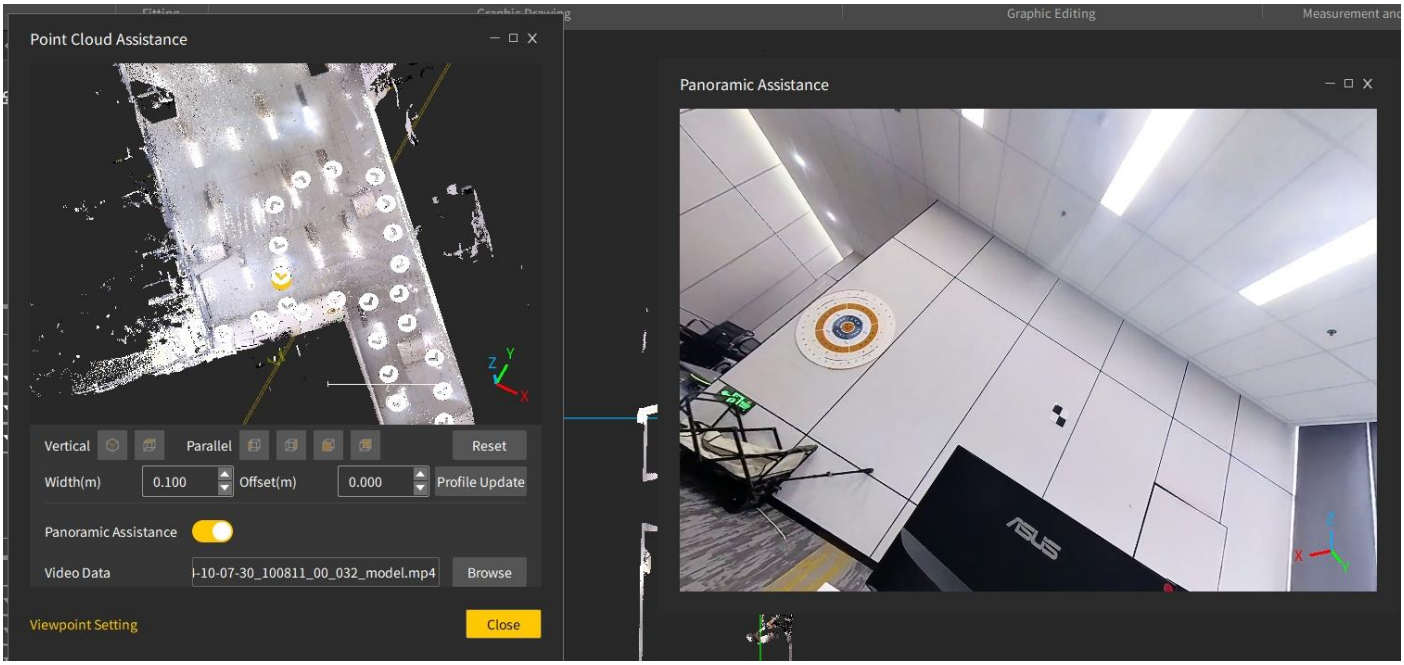



**Figure 263.** Panoramic Assistance

### 8.1.3 Layer

**Description:**

Supports adding and deleting layers, and allowing drawing images on different layers. Each layer supports renaming, locking, unlocking, hiding, showing, and exporting.

You can select a layer to edit the properties of the elements within it.

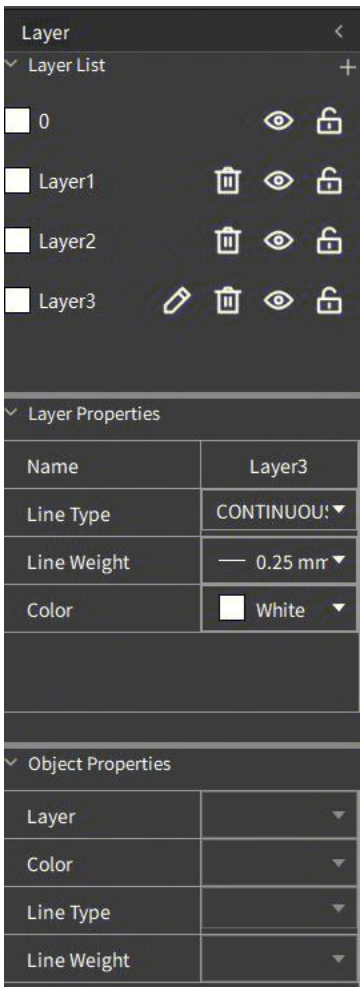



**Figure 264.** Layer

### 8.1.4 Save

**Description:**

Save or update the drawn vector file to the project file's vector directory tree during or after the drawing process.

### 8.1.5 Undo

**Description:**

Undo the most recent drawing action. Multiple undo steps are supported.

### 8.1.6 Redo

**Description:**

Restore the most recently undone action. Multiple redo steps are supported.

### 8.1.7 Delete

**Description:**

Delete the selected vector graphics.

### 8.1.8 Exit

**Description:**

Exits the vector graphic drawing mode. the drawn graphic will be displayed in the 3D view and cannot be edited outside of drawing mode.

### 8.1.9 Extract Contour Line

**Description:**

Automatically fit and extract a 2D vector contour line from a sliced plane of the point cloud.

**Operation procedure:**

1. Select the point cloud in the project and click **Draw** to enter the drawing function.
2. Click **Assisted Drawing** to open the point cloud assistance window. Select the starting point and end point to cut the plane from the point cloud. At this time, the Extract Contour Line function is activated. Click , a progress window will appear. Once the drawing is complete, the progress window will close.
3. Click **Save** to update the drawn vector elements in the project file, and the interface will refresh accordingly.

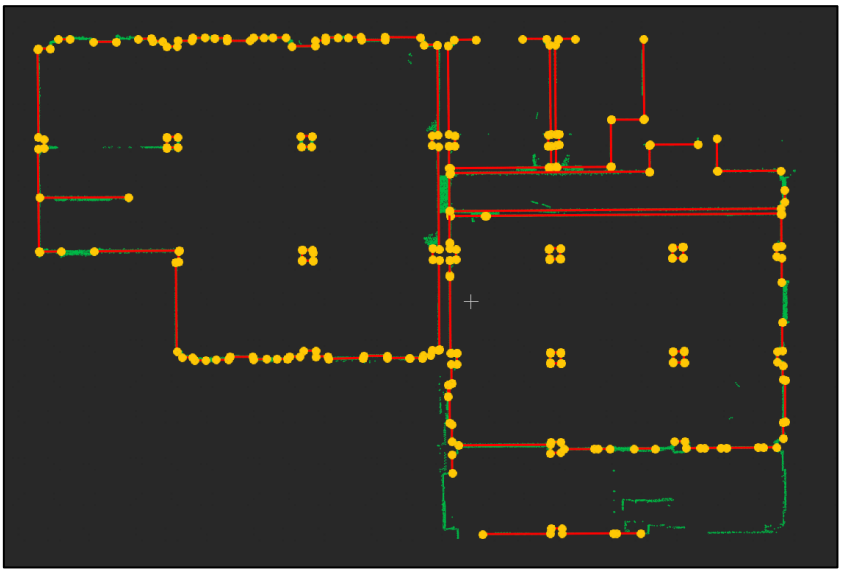


**Figure 265.** Extracted contours

## 8.2 Graphic Drawing

### 8.2.1 Point

**Description:**

Draw points.

**Operation procedure:**

1. Click in the **Graphic Drawing** area, and the icon will be highlighted.
2. Move the cursor into the view area, the cursor changes to point-picking mode.
3. Click the left mouse button to draw points, and you can draw continuously.

### 8.2.2 Straight Line

**Description:**

Draw straight lines.

**Operation procedure:**

1. Click in the **Graphic Drawing** area, and the icon will be highlighted.
2. Move the cursor to the view area, the cursor changes to point-picking mode.
3. Click to select the first point, move the cursor to a new position, and click to select the second point. A straight line is formed.
4. Click again to start drawing the next straight line.

### 8.2.3 Polyline

**Description:**

Draw polylines.

**Operation procedure:**

1. Click in the **Graphic Drawing** area, and the icon will be highlighted.
2. Move the cursor to the view area, the cursor changes to point-piking mode
3. Click to select the first point, move the cursor to a new position, and click to select the second point. Repeat the steps to create a polyline.
4. Right-click to end the drawing. Click again to start drawing the next polyline.

### 8.2.4 Arc

**Description:**

Draw segments.

**Operation procedure:**

1. Click in the **Graphic Drawing** area, and the icon will be highlighted.
2. Move the cursor to the view area, the cursor changes to point-picking mode
3. Left-click three times to specify three points, an arc will be fitted through these points.
4. Click again to start drawing the next arc.

### 8.2.5 Rectangle

Draw vector rectangles using two methods: two-point or three-point

**Two-point Rectangle**

**Description:**

Draws a rectangle using two diagonal points.

**Operation procedure:**

1. In the **Graphic Drawing** area, click the inverted triangle beside the rectangle drawing icon and select **Two-point Rectangle**, or click **Two-point Rectangle** directly. The icon will be highlighted.
2. Move the cursor to the view area, the cursor changes to point-picking mode.
3. Click to select the first point, move the cursor to a new position, and click to select the second point. A rectangle is formed with the line connecting the two points as the diagonal.
4. Click again to start drawing the next rectangle.

**Three-point Rectangle**

**Description:**

Draws a rectangle by setting three vertices.

**Operation procedure:**

1. Click **Three-point Rectangle** in the **Graphic Drawing** area, and the icon will be highlighted.
2. Move the cursor to the view area, the cursor changes to point-picking mode. Click to select the first point, move the cursor to a new position, click to select the second point, move the cursor to another position, and click to select the third point. A rectangle is formed with the three points as vertices.
3. Click again to start drawing the next rectangle.

### 8.2.6 Circle

A circle can be drawn in a 2D view with three points, two points (defining the diameter), or one center point plus the radius.

**Three-point Circle**

**Description:**

Draws a circle by setting three points on the circumference.

**Operation procedure:**

1. Click **Three-point Circle** in the **Graphic Drawing** area, and the icon will be highlighted.
2. Move the cursor to the view area, the cursor changes to point-picking mode.
3. Click to select the first point, move the cursor to a new position, click to select the second point, move the cursor to another position, and click to select the third point. A circle is formed with the three points on the circumference.
4. Click again to start drawing the next circle.

**Two-point Circle**

**Description:**

Draws a circle by specifying the diameter with two points.

**Operation procedure:**

1. In the **Graphic Drawing** area, click the inverted triangle beside the circle drawing icon, and select **Two-point Circle**, or click **Two-point Circle** directly. The icon will be highlighted.
2. Move the cursor to the view area, the cursor changes to point-picking mode.
3. Click to select the first point, move the cursor to a new position, and click to select the second point. A circle is formed with the line connecting the two points as the diameter.
4. Click again to start drawing the next circle.

**Center and Radius**

**Description:**

Draws a circle by setting the center and the radius with two points.

**Operation procedure:**

1. In the **Graphic Drawing** area, click the inverted triangle beside the circle drawing icon and select **Center and Radius**, or click **Center and Radius** directly. The icon will be highlighted.
2. Move the cursor to the view area, the cursor changes to point-picking mode.
3. Click to select the first point, move the cursor to a new position, and click to select the second point. A circle is formed with the first point as the center and the line connecting the two points as the radius.
4. Click again to start drawing the next circle.

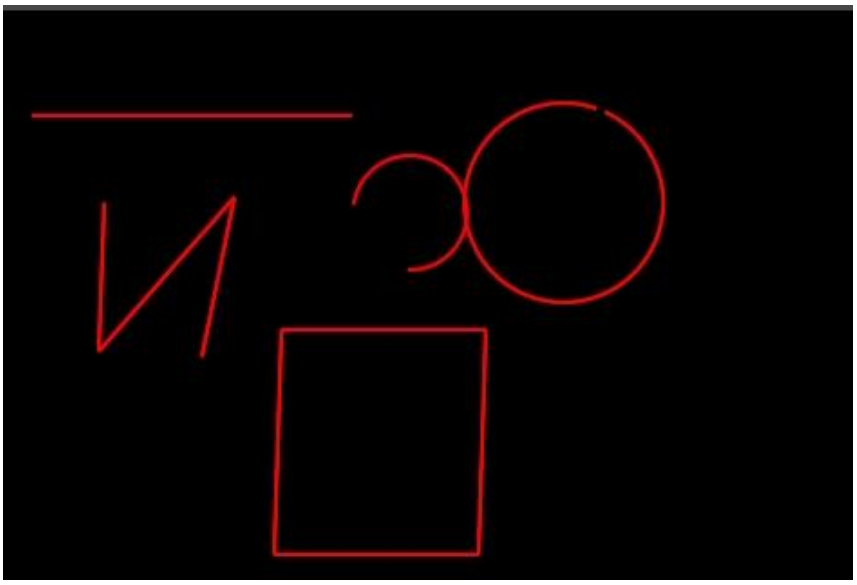


**Figure 266.** Graphic Drawing

### 8.2.7 Ellipse

**Description:**

Draw an ellipse using its major and minor axes.

**Operation procedure:**

1. Click  in the **Graphic Drawing** area, the icon will be highlighted.
2. Move the cursor to the view area, the cursor changes to point-picking mode.
3. Click the left mouse button to determine the center point. When you move the mouse, a dotted line will be displayed between the center point and the current cursor position, and the ellipse will be fitted with the dotted line as the first axis. Click the left mouse button again to determine the length of the first axis. When you move the mouse, a dotted line will be displayed between the center point and the current cursor position, and the ellipse will be fitted with the dotted line as the second axis. Click the left mouse button again to complete the ellipse drawing.
4. When you click the left mouse button again, the second ellipse starts to be drawn.

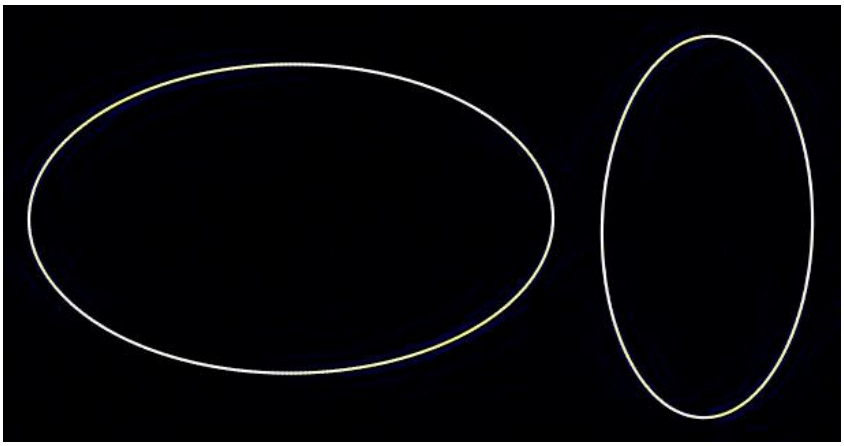


**Figure 267.** Drawing Ellipse

### 8.2.8 Polygon

There are four ways to draw vector polygons in the 2D view: midline point, corner point, two points on an edge, center point, point on an edge, and two points on an edge.

**Regular Polygon Center and Point**

**Description:**

Draw a polygon using its center point and vertices.

**Operation procedure:**

1. Click  in the **Graphic Drawing** area, the icon will be highlighted. A parameter dialog appears to set the number of sides(3-99).

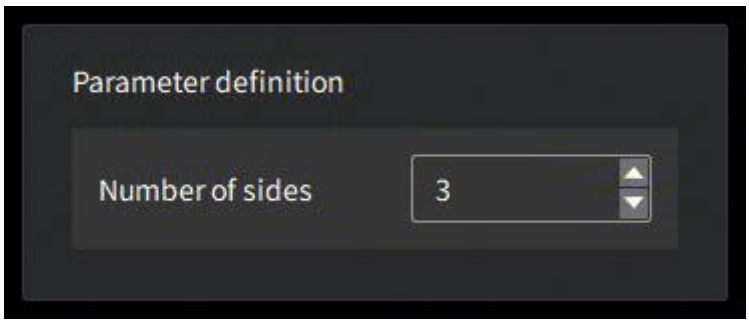



**Figure 268.** Polygon parameter definition window

2. Move the cursor to the view area, the cursor changes to point-picking mode.
3. Click the left mouse button to determine the center point. When you move the mouse, a dotted line will be displayed between the center point and the current cursor position, and a polygon will be fitted with the cursor position as the vertex. Click the left mouse button again to determine the polygon vertex and complete the drawing.
4. When you click the left mouse button again, the drawing of the second polygon begins.

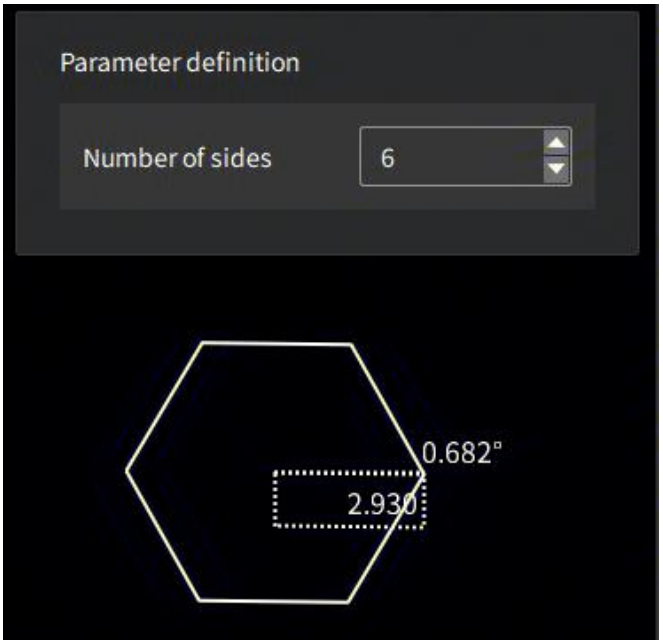



**Figure 269.** Regular Polygon Center and Point

**Regular Polygon Two Point**

**Description:**

Draw a polygon through the center point and the midpoint of one side.

**Operation procedure:**

1. Click in the **Graphic Drawing** area, the icon will be highlighted. A parameter dialog appears to set the number of sides(3-99).Move the cursor to the view area, the cursor changes to point-picking mode.
2. Click the left mouse button to determine the first vertex. When you move the mouse, a dotted line will be displayed between the vertex and the current cursor position, and a polygon will be fitted with the cursor position as the second vertex. Click the left mouse button again to determine the second vertex of the polygon and complete the drawing.
3. When you click the left mouse button again, the drawing of the second polygon begins.

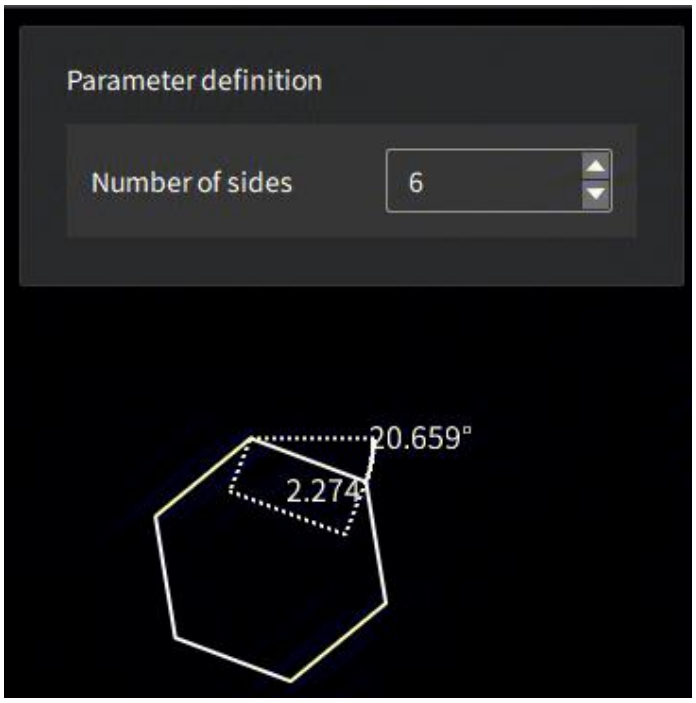



**Figure 270.** Regular Polygon Two Point

**Regular Polygon Center and Line**

**Description:**

Draw a polygon using its center point and vertices.

**Operation procedure:**

1. Click  in the **Graphic Drawing** area, the icon is highlighted. A parameter dialog appears to set the number of sides(3-99).
2. Move the cursor to the view area, the cursor changes to point-picking mode.
3. Click the left mouse button to determine the center point. When you move the mouse, a dotted line will be displayed between the center point and the current cursor position, and a polygon will be fitted with the cursor position as the midpoint of one side. Click the left mouse button again to complete the drawing.
4. When you click the left mouse button again, the drawing of the second polygon begins.

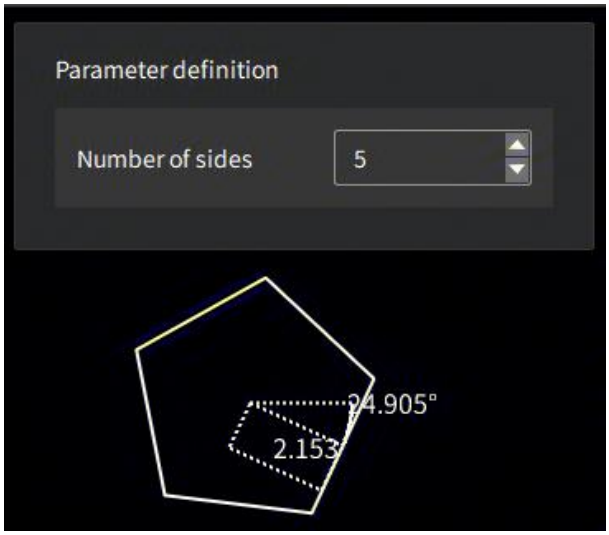



**Figure 271.** Regular Polygon Center and Line

**Regular Polygon Center Two Line**

**Description:**

Draw a polygon by determining the midpoints of two sides.

**Operation procedure:**

1. Click  in the **Graphic Drawing** area, the icon is highlighted. A parameter dialog appears to set the number of sides(3-99).
2. Move the cursor to the view area, the cursor changes to point-picking mode.
3. Click the left mouse button to determine the midpoint of the first side. When you move the mouse, a dotted line will appear between the midpoint and the current cursor position, and a polygon will be fitted with the cursor position as the midpoint of the second side. Click the left mouse button again to complete the drawing.
4. When you click the left mouse button again, the drawing of the second polygon begins.

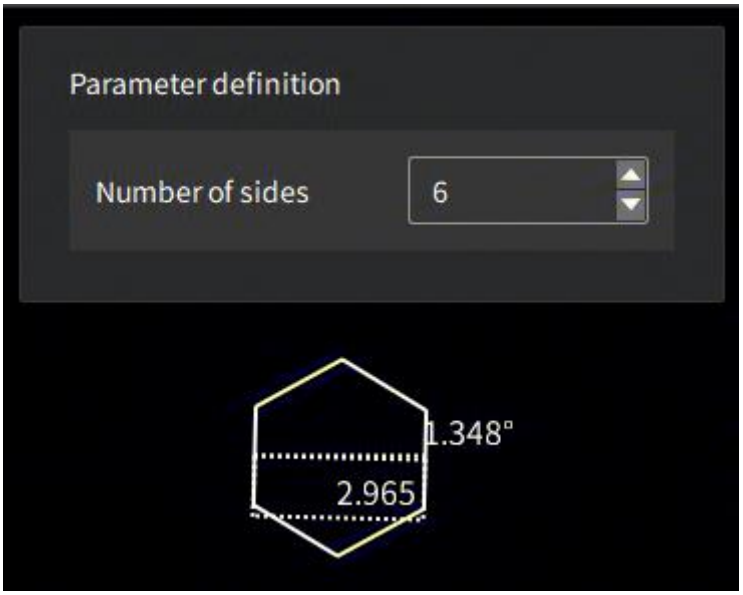



**Figure 272.** Regular Polygon Center Two Line

# 8.3 Graphic Editing

## 8.3.1 Extend

**Description:**

Extends the selected primitive to intersect with an adjacent primitive. It is often used to extend vector lines to make them intersect again after automatic extraction of contours.

**Operation procedure:**

1. On the **Drawing** tab, click to activate the extend feature.
2. When the cursor hovers over a straight line, a polyline, or an arc, the system automatically checks whether their extension lines can intersect with other primitives, and draws dotted extension lines if necessary.
3. Click to execute the extend action. The operation can be performed continuously.
4. Right-click or press the Esc key to exit the feature.

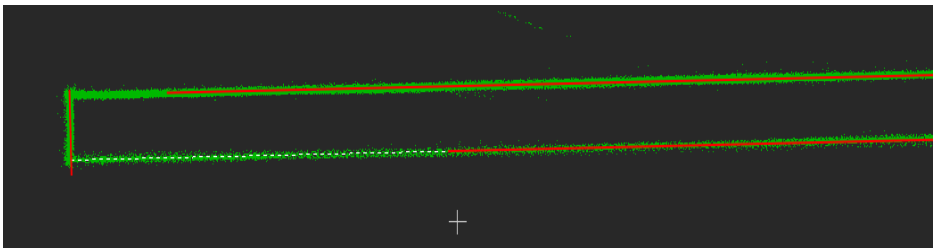


**Figure 273.** Before extension

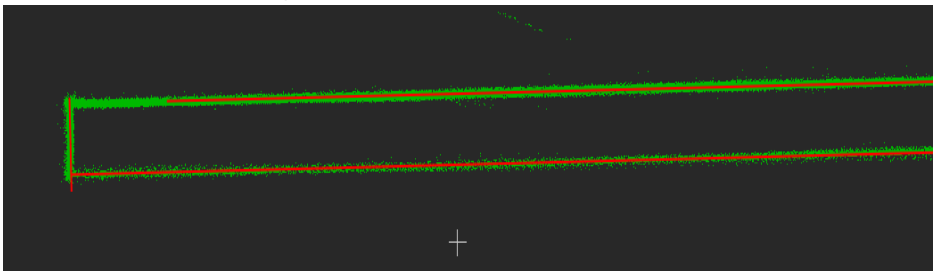


**Figure 274.** After extension

## 8.3.2 Copy

**Description:**

Copies one or more primitives to avoid drawing the same elements repeatedly and improve drawing efficiency.

**Operation procedure:**

1. On the **Drawing** tab, click to activate the copy feature.
2. Click to draw a rectangular area to select one or more primitives.
3. Right-click to end the selection and generate an identical set of primitives with the cursor position as the center. The new set of primitives moves with the cursor.
4. Click again to complete the copy. This operation can be repeated.
5. Right-click or press the Esc key to exit the feature.

**Figure 275.** Primitive copying result

### 8.3.3 Trim

**Description:**

Removes the selected vector lines between endpoints from primitives.

**Operation procedure:**

1. On the **Drawing** tab, click to activate the trim feature.
2. When the cursor hovers over a primitive, the section between the two points closest to the primitive becomes white.
3. Click to execute the trim action. This operation can be repeated.
4. Right-click or press the Esc key to exit the feature.

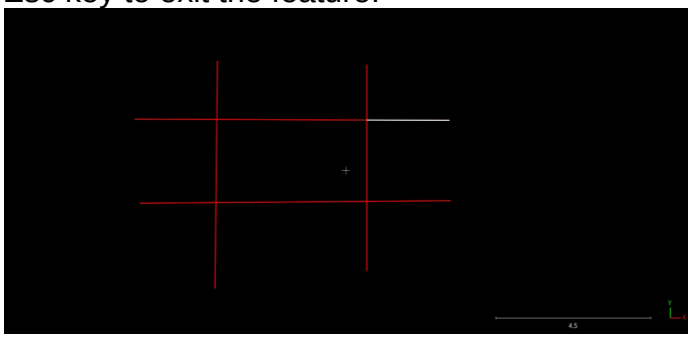


**Figure 276.** Before trim

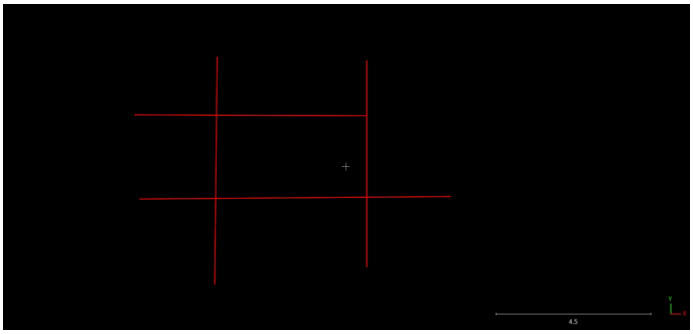


**Figure 277.** After trim

## 8.3.4 Intersect

**Description:**

Connects two lines by creating an intersection automatically.

**Operation procedure:**

1. On the **Drawing** tab, click . Click the first line, and then click the second line. If the two lines can intersect at some point, the two lines are extended by dashed lines until they intersect.
2. Right-click to turn the dashed lines into solid lines.
3. Right-click or press the Esc key to exit the feature.

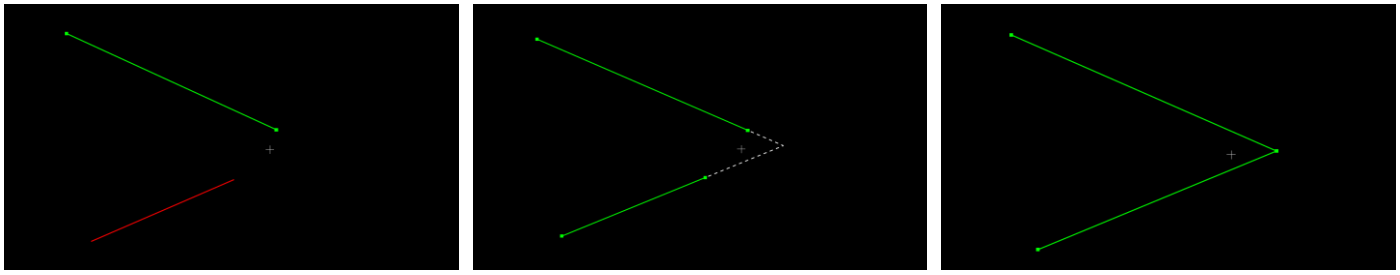


**Figure 278.** Intersection procedure

## 8.3.5 Rotate

**Description:**

Performs a rotation operation on the drawn image.

**Operation procedure:**

1. Select the image you want to rotate (you can select multiple images). On the **Drawing** tab, click , display parameter setting pop-up window.

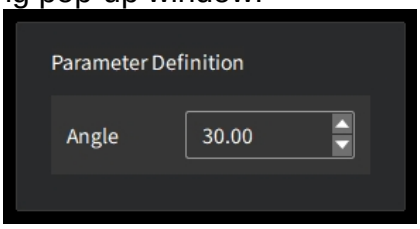



**Figure 279.** Rotation parameter definition pop-up window

2. Left-click to determine the rotation center point. When you move the mouse, a dotted line will be displayed between the starting point and the current cursor position, and the mirrored image will be displayed in real time. Left-click again to complete the rotation.

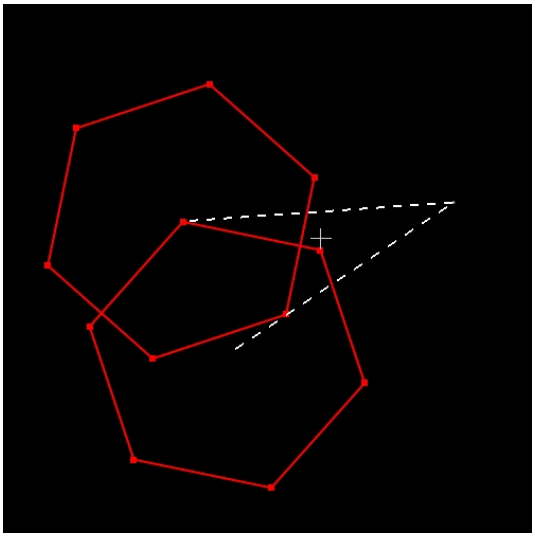


**Figure 280.** Rotation process diagram

### 8.3.6 Mirror

**Description:**

Mirrors the drawn image along the specified axis.

**Operation procedure:**

1. Select the image you want to mirror (you can select multiple images) On the **Drawing** tab, click, the cursor switches to pick mode
2. Left-click to determine the starting point of the mirror axis. When you move the mouse, a dotted line will be displayed between the starting point and the current cursor position, and the mirrored image will be displayed in real time. Left-click again to complete the mirror operation.

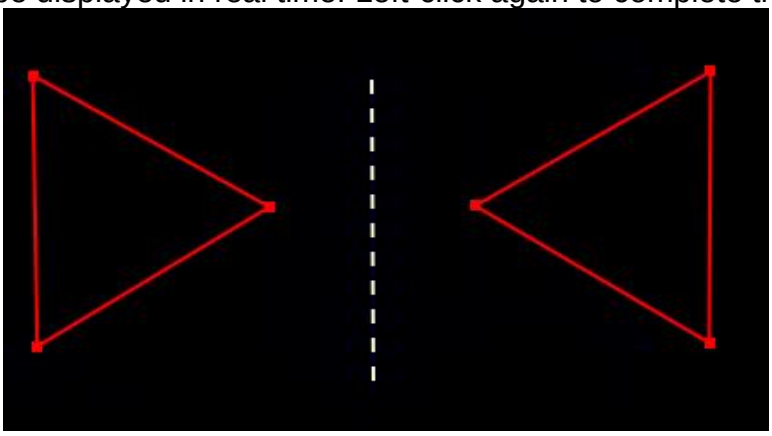


**Figure 281.** Mirroring process diagram

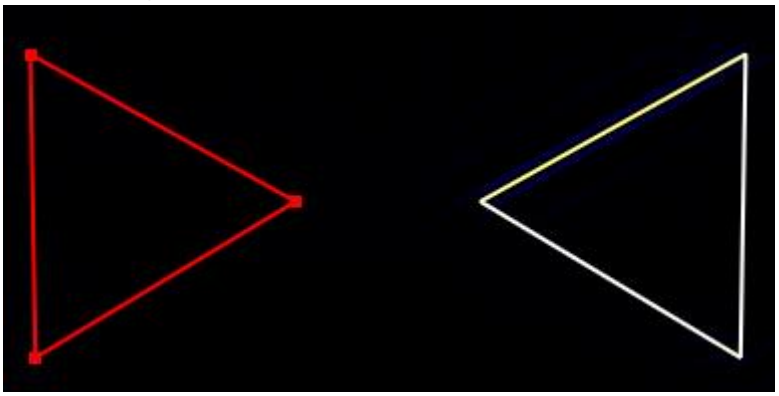


**Figure 282.** Complete mirror drawing

### 8.3.7 Offset

**Description:**

Quickly create parallel lines based on existing elements.

**Operation procedure:**

1. On the **Drawing** tab, click . Switch the mouse to the picking mode and the parameter setting window will be displayed. The polygon mode can offset the entire selected polygon, while the line segment mode only offsets a single line segment. The offset image can be selected in right angle or rounded corner style.

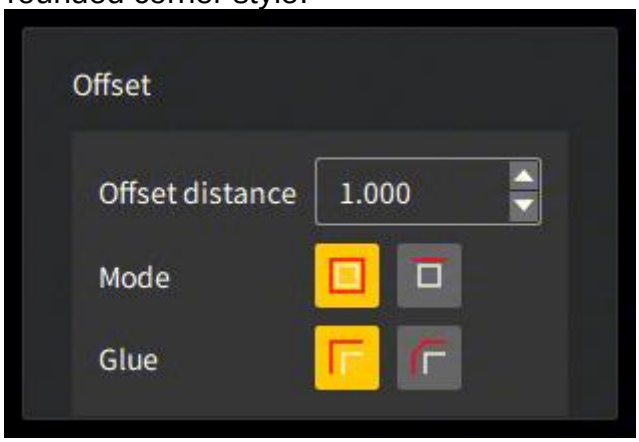



**Figure 283.** Offset parameter setting window

2. After completing the parameter setting, move the mouse to the vicinity of the entity to be offset. The offset image effect will be displayed in real time according to the relative position of the mouse and the image. Lest-click to complete the image offset.

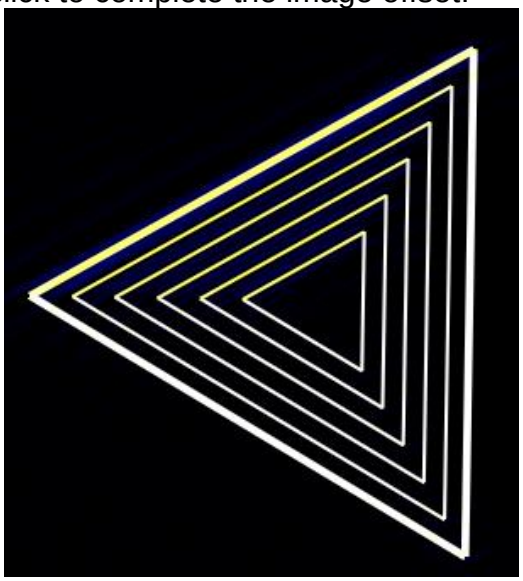


**Figure 284.** Complete offset

# 8.4 Measurement and Dimensions

## 8.4.1 Coordinates

**Description:**

Marks coordinate information of a specific point.

**Operation procedure:**

1. Click the coordinate marking icon, the icon becomes illuminated , and the cursor turns into a crosshair.
2. When the cursor hovers over a primitive, , a snapping indicator will appear
3. Click to show the marker line and X and Y coordinates of the current point. Move the cursor to drag along the marker line and X and Y coordinates.
4. Click again to fix the position of marker data, or right-click or press the Esc key to exit the marking.
5. Once the marking process is completed or canceled, the cursor returns to crosshair mode and can proceed to mark the next point.

## 8.4.2 Length

**Description:**

Marks the length between two points.

**Operation procedure:**

1. Click the length marking icon, the icon becomes illuminated , and the cursor turns into a crosshair.
2. When the cursor hovers over a primitive, a snapping indicator will appear.
3. Click to mark the start point. Move the cursor, and a line forms between the start point and the current position of the cursor.
4. Click again to mark the end point, and the length of the line is shown.
5. Click the marker data and drag to change its position. Before the marker position is confirmed, right-click or press the Esc key to exit the marking.
6. Once the marking process is completed or canceled, the cursor turns back into a crosshair, and you can continue to mark the next length.

### 8.4.3 Angle

**Description:**

Marks the angle between two lines.

**Operation procedure:**

1. Click the angle marking icon, the icon becomes illuminated , and the cursor turns into a crosshair.
2. Click the first line.
3. Move the cursor and click the second line.
4. Right-click to show the angle between the two lines.
5. The cursor turns back into a crosshair, when you can continue to mark the next angle.

### 8.4.4 Area

**Description:**

Marks the area of a planar polygonal area.

**Operation procedure:**

1. Click the area marking icon, the icon becomes illuminated , and the cursor turns into a crosshair.
2. Click to mark the start point. Move the cursor, and a line forms between the start point and the current position of the cursor.
3. Click to mark the second point.
4. Move the cursor, and a triangle is formed with the previous two points and the current position.
5. Move the cursor to mark more points, or right-click to end point selection and measure the area of the zone formed.
6. The area and perimeter are shown in the center of the zone formed.
7. The cursor turns back into a crosshair, when you can continue to mark the next area.


**Figure 285.** Measurement and dimensions

# 8.5 Drawing Settings

## 8.5.1 Set Reference

**Description:**

Define a custom horizontal direction for drawing primitives.

**Operation procedure:**

1. Click the icon for setting a reference line, and the icon becomes illuminated .
2. Move the cursor into the view area to turn it into a hand-style picking pointer.
3. Click to set the first point, move the cursor, and a dashed line is formed between the first point and the current position of the cursor. The line extends outwards infinitely.
4. Click again to set the reference line. Then, the point cloud rotates clockwise until the reference line is parallel to the horizontal direction of the view area, where a reference grid is formed.
5. You can toggle the visibility of custom reference lines via the “Show Reference Line" checkbox.
6. Select **Delete reference line** from the drop-down list to delete the reference line.

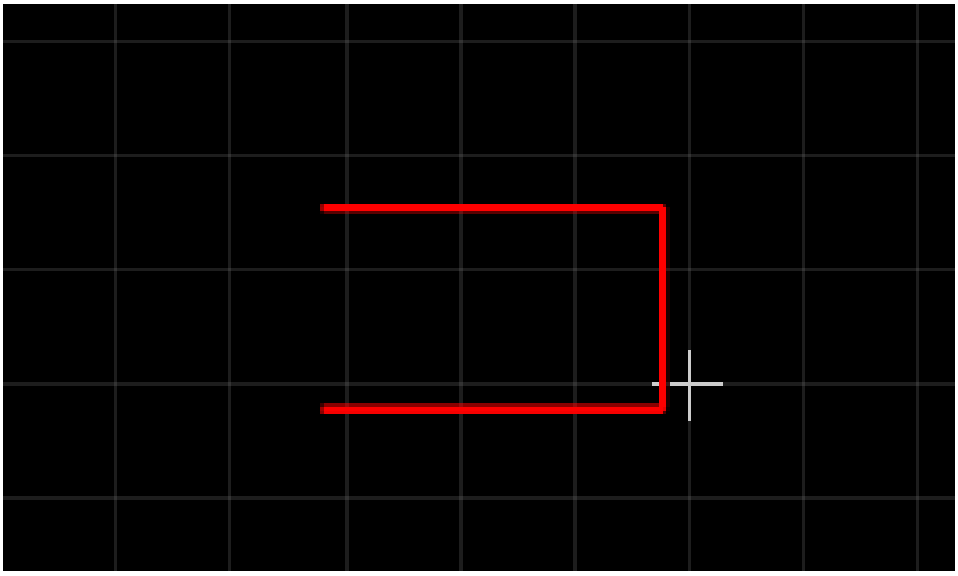


**Figure 286.** Setting a reference line

## 8.5.2 Dimension Settings

**Description:**

Sets the color, font size, unit, and precision of marker data.

**Operation procedure:**

1. Click , and the following window appears.
2. Set the marker line and text colors.
3. Set the font size (default:16; min:6; max:96)
4. Set the precision and unit, and the marker data is updated accordingly.
5. Enable or disable the **Show Unit** option to display unit symbols after the values (enable by default).
6. The **Show Dimensions** box is checked by default, showing marker lines and values. If the box is unselected, marker lines and values will only appear during creation and be hidden afterward.
7. Click **Reset** to restore the default settings.
8. All settings will be save persistently and retained the next time the software is launched.

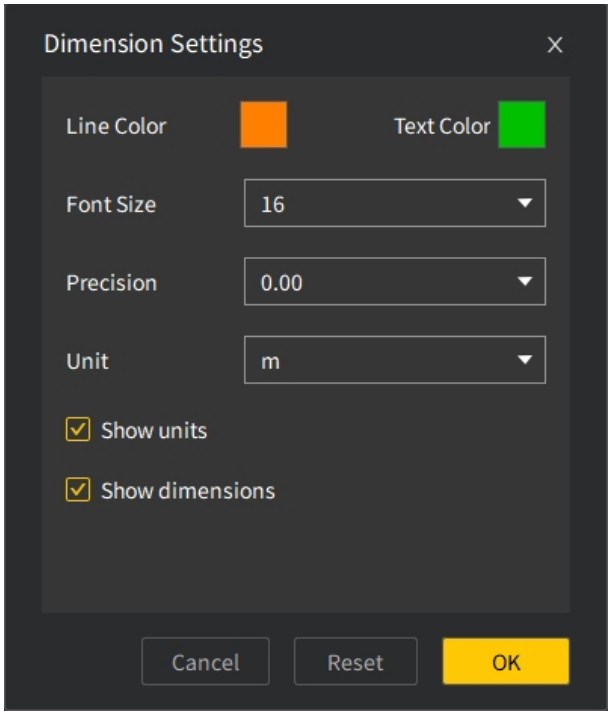



**Figure 287.** Dimension settings

### 8.5.3 Polar Tracking

**Description:**

Toggles polar tracking drawing mode on and off.

**Operation procedure:**

1. Polar tracking is enabled by default, and the increment angle is 30 degrees by default. It can be adjusted manually to assist users in drawing graphics;
2. Uncheck the polar tracking button to turn off the polar tracking function, and fixed-angle rays will no longer be automatically displayed during the drawing process.

### 8.5.4 Object Snap

**Description:**

Enable or disable object snap during drawing.

**Operation procedure:**

1. The **Object Snap** function is enabled by default to assist with graphic drawing.
2. Uncheck the box to disable the feature. The software then does not snap endpoints during drawing.

**Snapping elements:**

- Two endpoints and one middle point of a straight line.
- Every endpoint of a polyline and every middle point of every section line.
- Two endpoints, one middle point, and one center point of an arc.
- Four vertices and one center point of a rectangle, and middle points of the four edges.
- One center point of a circle and middle points of the four reference lines.
- One pedal point of a primitive that is vertical to the line connecting the current cursor position and the last point.
- Any other points on any of the above graphics. The cursor type is different from that for snapping the above elements.

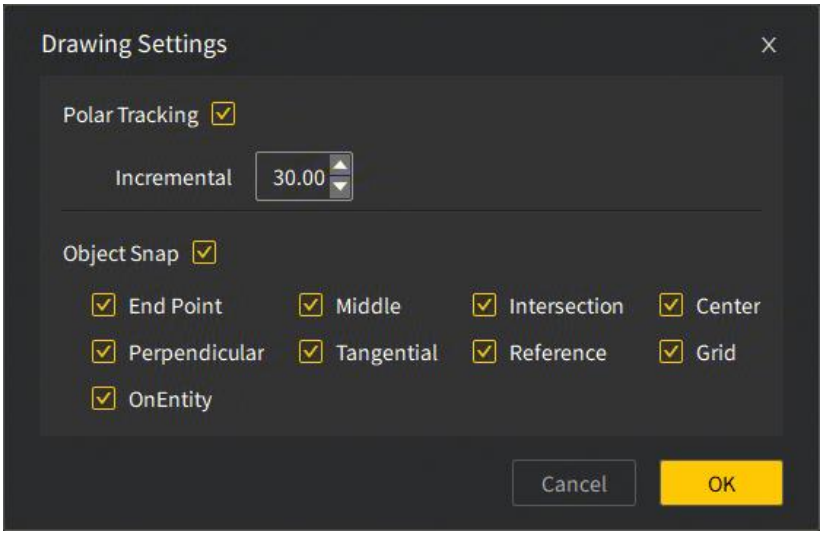



**Figure 288.** Polar Tracking and Object Snap

### 8.5.5 Orthographic Drawing

**Description:**

Restrict the cursor to only horizontal or vertical movements to quickly draw horizontal or vertical lines.

**Operation procedure:**

1. The feature is unavailable when no reference line is set.
2. Once a reference line is set, the orthogonal model becomes active and restricts the next point to lie horizontally or vertically from the previous one.
3. Unchecking the box to disable the feature, so that points can be selected freely.
4. The feature is disabled after the reference line is deleted.

# 9 Forestry

In a traditional forestry survey, the field investigations consume a lot of manpower and materials. Especially when conducted on a large scale, the surveys not only are labor-intensive and time-consuming, but often take the form of small-area sample measurements, which are unsuitable for high-demanding forest survey scenarios. The combined use of the FJD Trion S1 3D laser scanner and the FJD Trion Model point cloud processing software makes a forestry survey more accurate and convenient.

## 9.1 pre-processing

### 9.1.1 Extract Ground Point

**Description:**

Classifies and extracts ground points from the forestry scenario point cloud. It is critical for segmentation by tree.

**Operation procedure:**

1. Select a point cloud file under **Objects**.

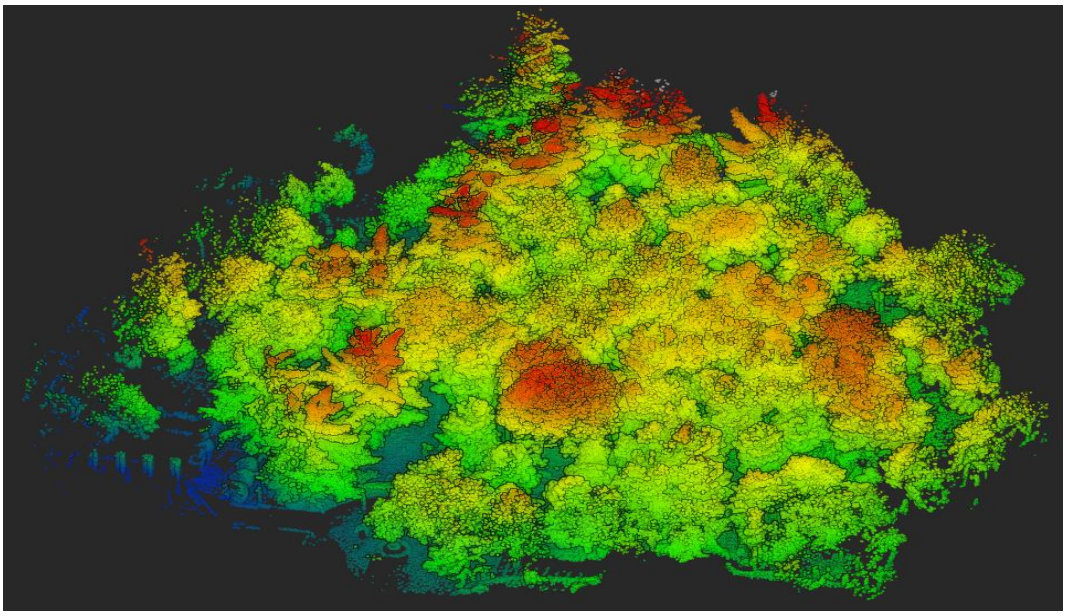


**Figure 289.** Forestry scenario point cloud

2. Click  on the **Forestry** tab, and the following window appears.

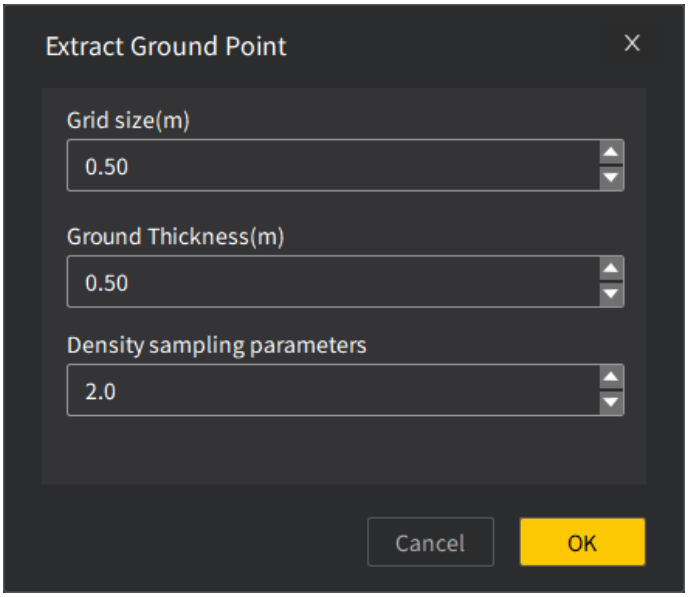


**Figure 290.** Setting parameters

3. Click **OK** to start the extraction of ground points. The following window appears.

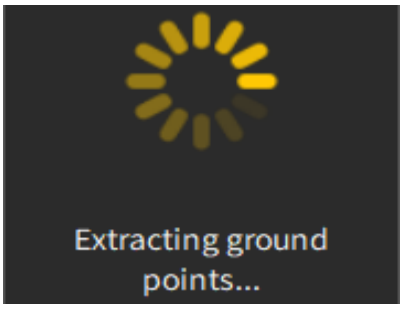



**Figure 291.** Extracting ground points

4. The above window disappears when the extraction is completed.
5. The point cloud with ground points extracted is displayed in the 3D view area. The point cloud is rendered by classification by default.

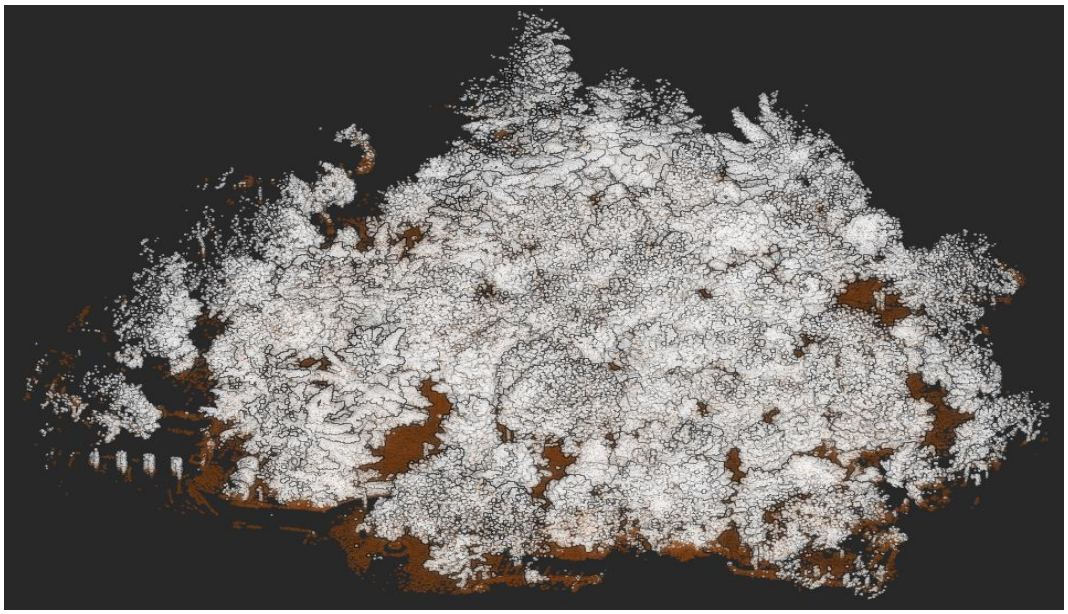


**Figure 292.** Ground points extracted

Note:

- For flat grounds, the grid step can be a little larger for efficient extraction of ground points; while for slopes, a smaller grid step should be used.
- The ground thickness must be set based on the terrain height difference, and can be obtained by the height measurement tool of the software when necessary.

# 9.2 Segmentation

## 9.2.1 Segment by Tree

**Description:**

For scanned data that has undergone ground point extraction, this function calculates the boundaries of single tree point clouds to segment and extract individual trees from the dataset.

**Operation procedure:**

1. Select a point cloud file under **Objects**.
2. Click on the **Forestry** tab, and the following window appears.
3. Click the Single Tree Segmentation icon to open the parameter settings window; Minimum Number of Points: The minimum number of points that constitute a single tree; Minimum Grid

Side Length: Set according to the spacing of the point cloud, typically equal to the average point spacing.

4. For specific forest data, you can adjust parameters in the Advanced Settings panel to achieve better segmentation results.
5. Search radius: Related to trunk thickness and point cloud density. The thicker the trunk and the sparser the data, the larger the radius should be.
6. Trunk height range: the height range interval of the trunk feature points.
7. Trunk Angle Range: The tilt angle of the trunk, referring to the angle between the trunk and the ground.
8. Trunk clustering radius: For scenarios with closely spaced trees, increase the value to group multiple trees as one; decrease the value to separate them more accurately.

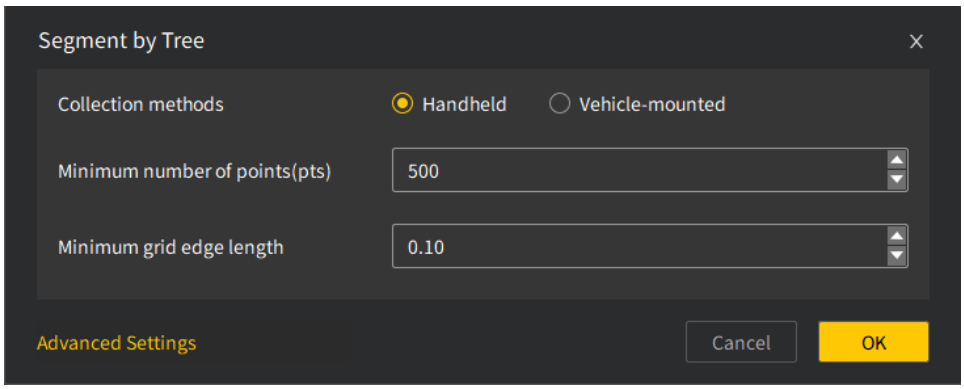



**Figure 293.** Setting parameters

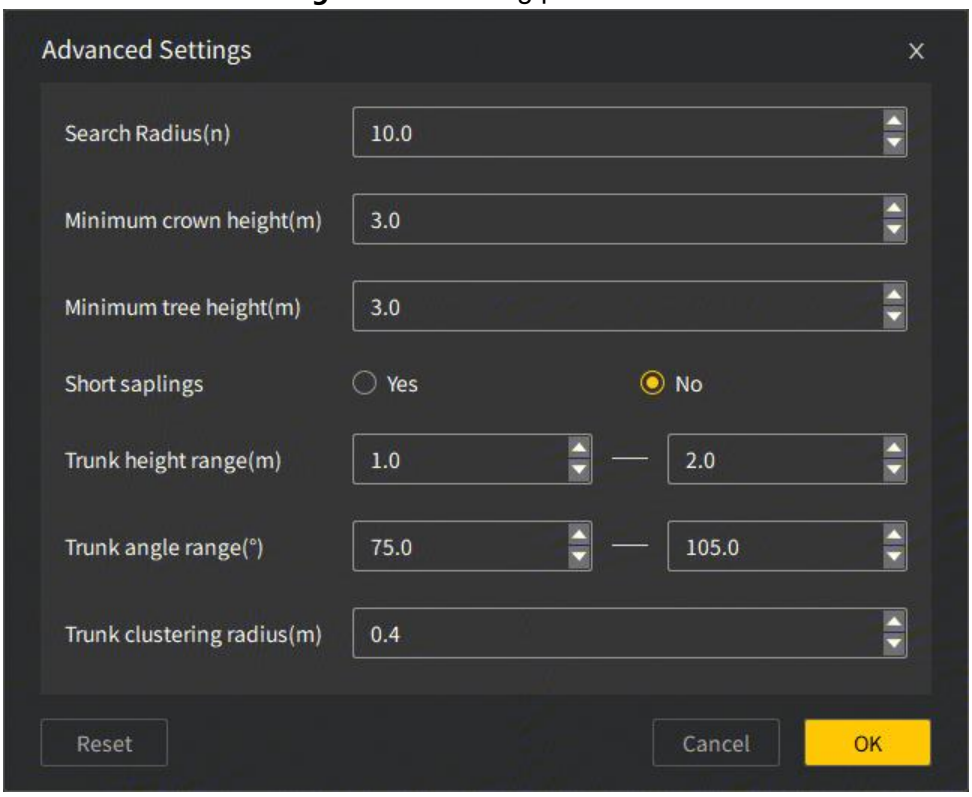



**Figure 294.** Advanced Settings

9. After setting the parameters, click OK to initiate the single tree segmentation process. A processing prompt will appear.

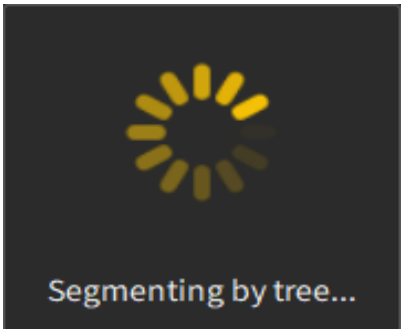



**Figure 295.** Segmenting by tree

10. Once segmentation is complete, the prompt will close automatically. The 3D view will display the segmented results, with each tree rendered using a unique Tree ID. You can enable or disable Tree ID **Display** in the visualization settings.

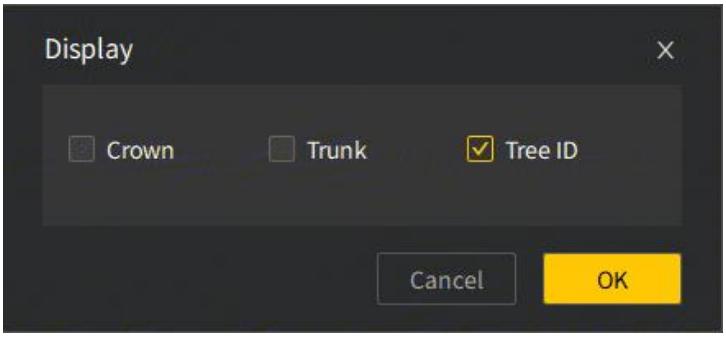



**Figure 296.** Tree ID

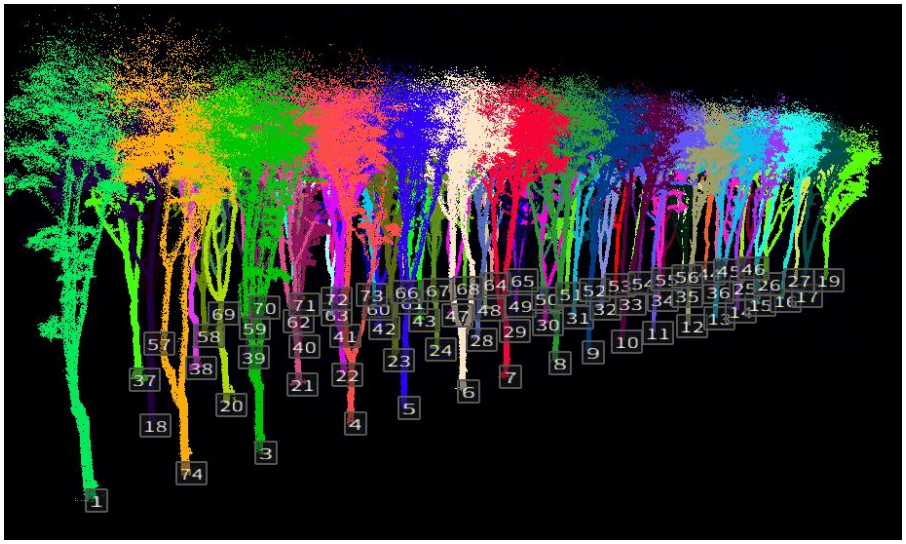


**Figure 297.** Segmentation result

# 9.3 Tree Editing

## 9.3.1 Add Tree

**Description:**

If multiple trees are identified as one tree or trees are not identified, tree segmentation is required to ensure that the right number of trees are identified separately. Trees can be added manually and automatically.

**Operation procedure:**

1. Select one or more post-segmentation point cloud files or folders. Trees can only be added manually if folders are selected.
2. Click , and select **Add Automatically** from the drop-down list.
   a) The cursor turns into a small hand.

b) Click the first point as the reference point, and its data appears in the upper table. In case of wrong selection, click to delete the point and select again.

c) Points selected after the reference point are used for adding trees and their data appears in the lower table. Click to delete them if required.

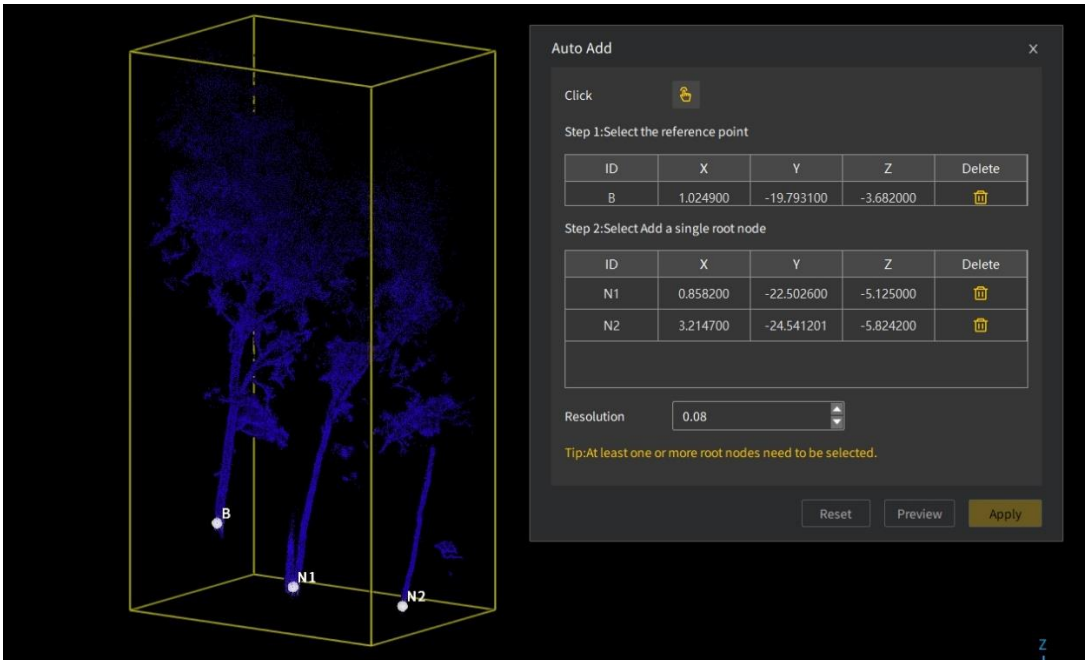



**Figure 298.** Selecting points for adding trees

d) Set the resolution, click **Preview**, and calculation starts. The real-time progress is shown.

e) When calculation is completed, the added trees are rendered in different colors. If the result is satisfactory, click **Apply** to save it. Otherwise, click **Reset** to restore the original point cloud and clear point selection.

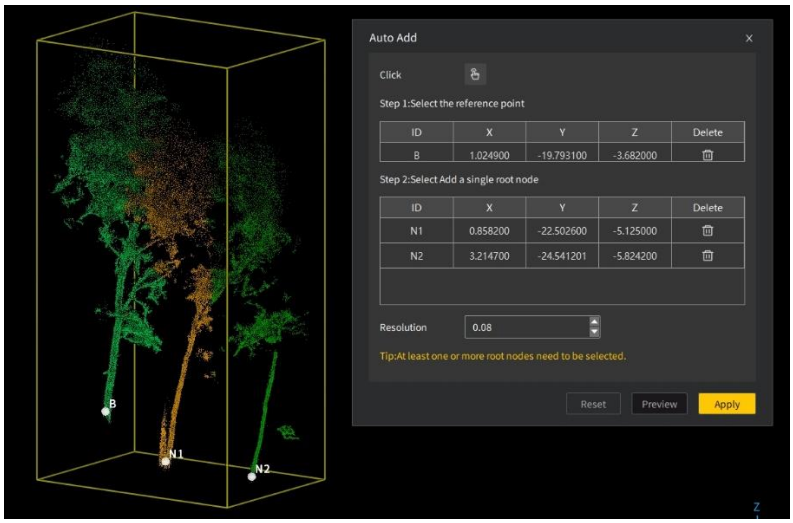



**Figure 299.** Preview of auto add result

3. Click , and select Add Manually from the drop-down list.

a) To add trees manually, use a proper selection tool to select the tree to be added from the point cloud shown in view 1; confirm the selected tree shown in view 2 and click .

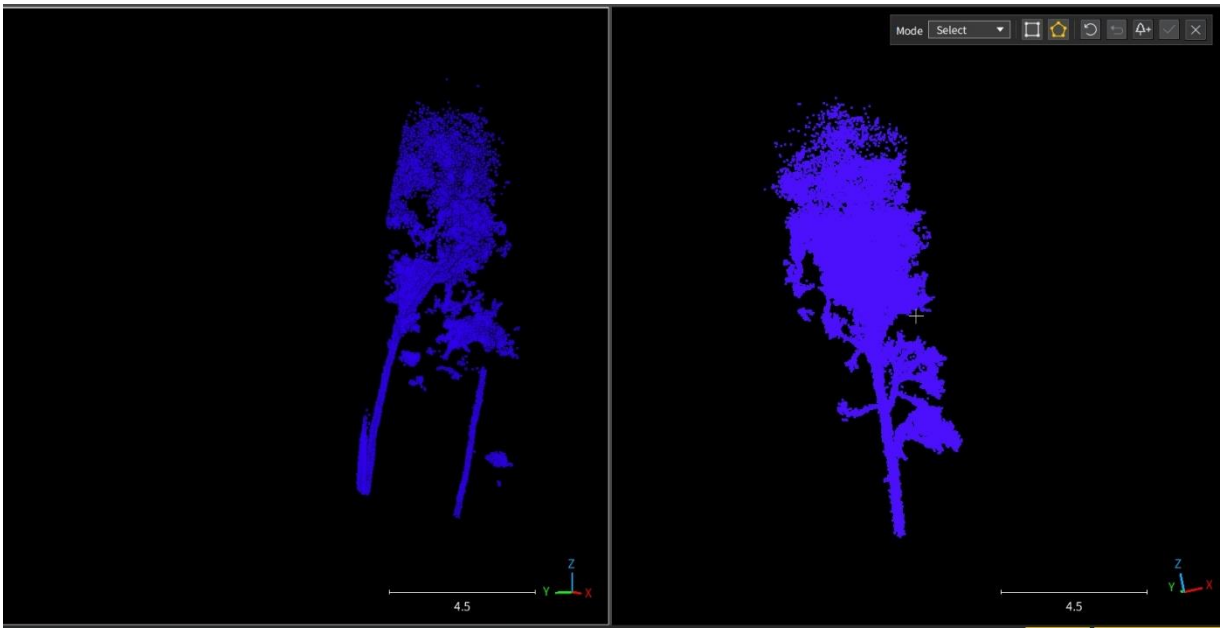



**Figure 300.** Adding a tree manually

b) To cancel the selection, click **Deselect** from the drop-down mode list and select the tree shown in view 2.

c) Click  to save the result. The new tree ID is added to the tree list.

### 9.3.2 Merge Tree

**Description:**

If one tree is identified as two or more trees, tree merger is required to ensure that the right number of trees are identified separately.

**Operation procedure:**

1. Hold down the Ctrl key and the left mouse button to select two or more post-segmentation point clouds under **Objects**.
2. Click  , and the toolbar is shown.
3. Double-click a tree or click its label to select it.
4. Click  to preview the merger effect.
5. Click  to save the result, and continue the merger operation.
6. Click  to exit the feature.

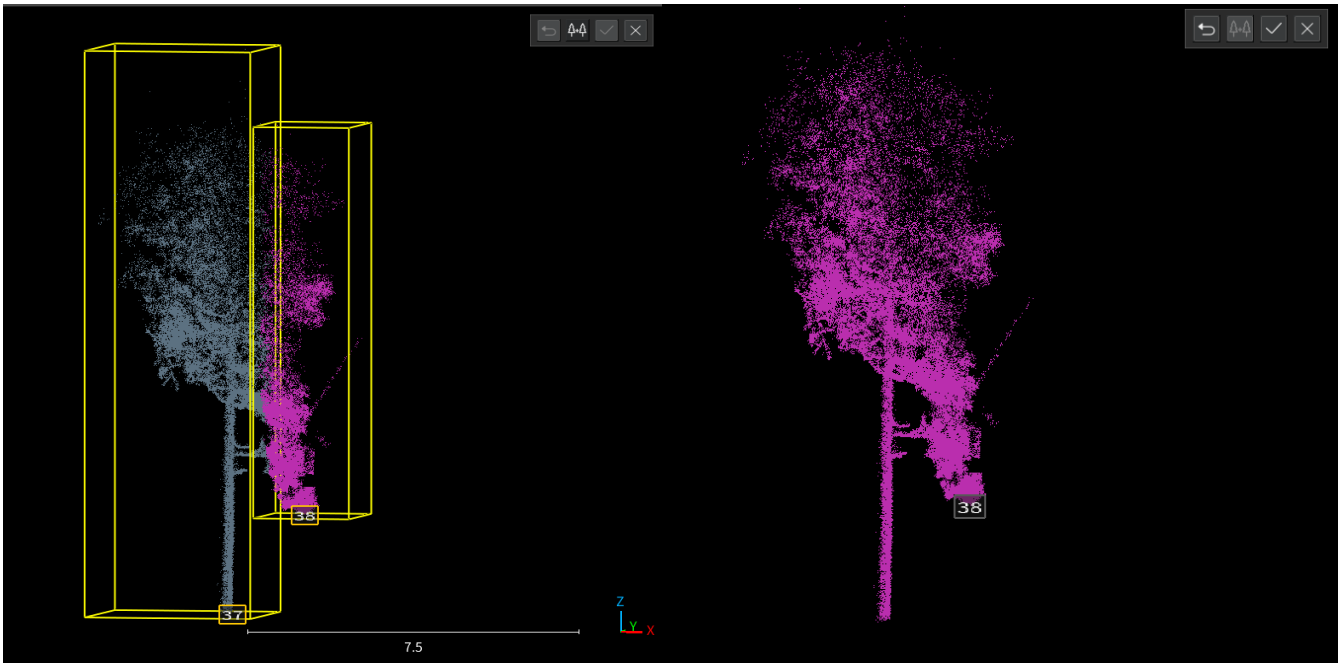



**Figure 301.** Before and after merger

### 9.3.3 Delete Tree

**Description:**

If non-tree objects are identified as trees, tree deletion is required to ensure that the right number of trees are identified separately.

**Operation procedure:**

1. Select one post-segmentation point cloud under **Objects**. Hold down the Ctrl key and the left mouse button for multiple selection.
2. Click , and the toolbar is shown.
3. Double-click a tree or click its label to select it.
4. Click to save the result, and continue the deletion operation.
5. Click to cancel the deletion.
6. Click to exit the feature.

- A point cloud without segmentation by tree performed cannot be edited.
- When you double-click a single tree from the point cloud in the 3D view area, the file with the corresponding tree ID is selected under **Objects**, so that you can edit the data directly.
- Rotation is unavailable when the data is selected in view 1.

## 9.4 Calculation and Analysis

### 9.4.1 Support Pole Removal

**Description:**

Remove the support poles from the segmented single tree point cloud data.

**Operation procedure:**

1. In the project file list, select the single tree segmentation folder or individual segmented tree point cloud data. Multiple single tree datasets can be selected.

2. The support pole removal function is activate. Left-click to open the function window for parameter settings. The default support pole height is 1.5 meters (i.e., the height from the ground to where the support pole contacts the tree). Smaller resolution values result in longer computation times.

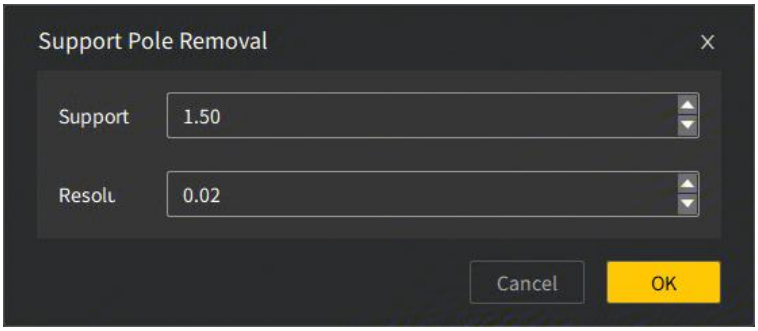



**Figure 302.** Support Pole Removal

3. Click **OK** to start the automatic calculation. The support pole point cloud will be removed, and the single tree point cloud data will be updated accordingly.

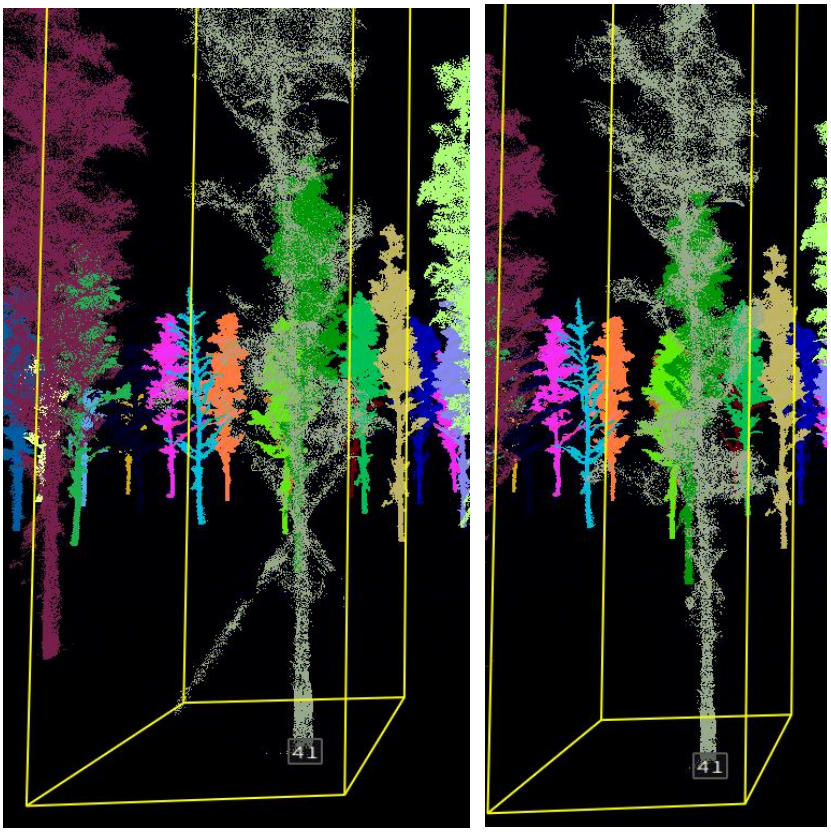


**Figure 303.** Support pole removal result

### 9.4.2 Crown Analysis

**Description:**

Identifies the trunk and crown of a tree point cloud generated after segmentation.

**Operation procedure:**

1. Select one post-segmentation point cloud file or folder under **Objects**. You can select multiple trees point clouds.
2. Click , and the following window appears. Set the accuracy, which is high by default. The higher the accuracy, the more accurate the trunk and crown identification will be.

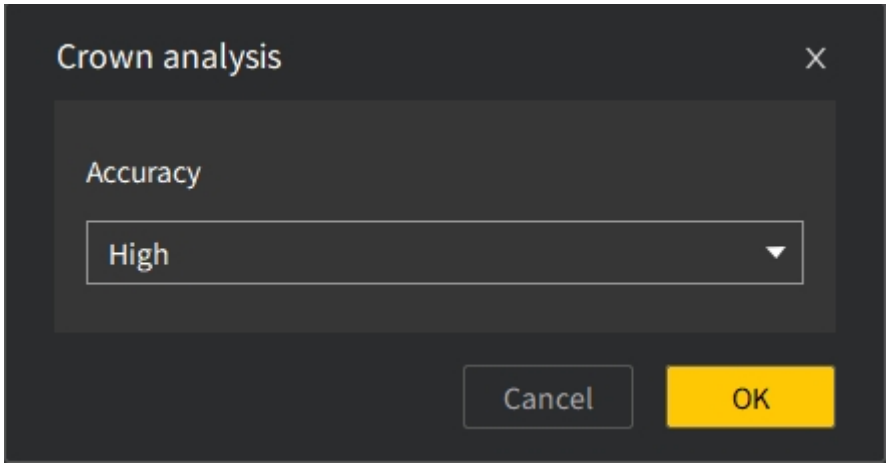



**Figure 304.** Crown analysis window

3. Click **OK**, and the calculation in-progress window appears.
4. Completing the calculation closes the progress window. The icon becomes activated. Click , and select or deselect **Crown** and/or **Trunk** to show or hide the crown and/or trunk. The **Crown** and **Trunk** are selected by default.
5. The **Crown** and **Trunk** are selected by default when you close or restart the software.

### 9.4.3 Property Calculation

**Description:**

Calculates the tree parameters such as coordinates, tree height, DBH, crown width, and other crown parameters as required, based on the accurate segmentation result.

**Operation procedure:**

1. Select a point cloud folder named with "segment" at its end or the required tree point cloud under **Objects**.
2. Click , and the following window appears. Select the parameters to calculate. Click **Settings** to set the DBH and ground diameter measurement locations and the minimum and maximum DBH as required.
3. Click **OK**, and calculation starts. A message pops up when calculation is completed.

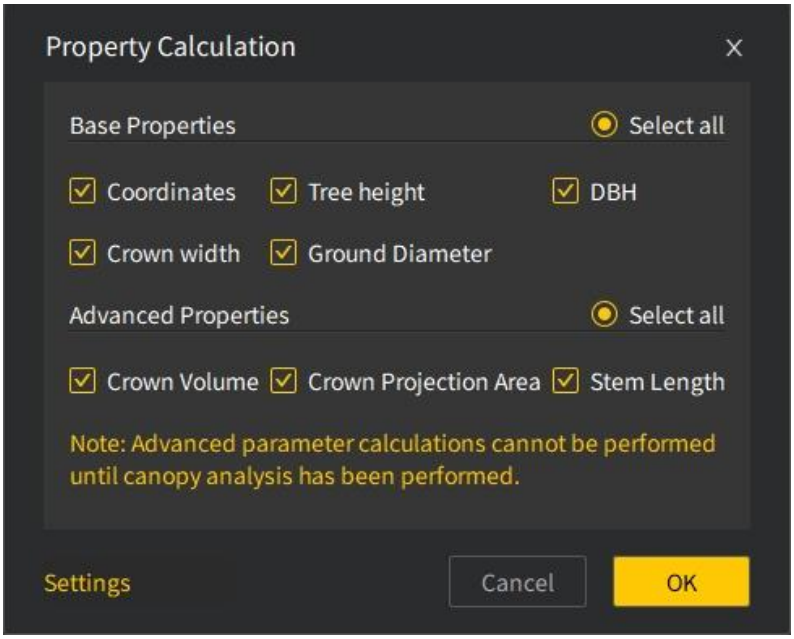



**Figure 305.** Selecting parameters to calculate

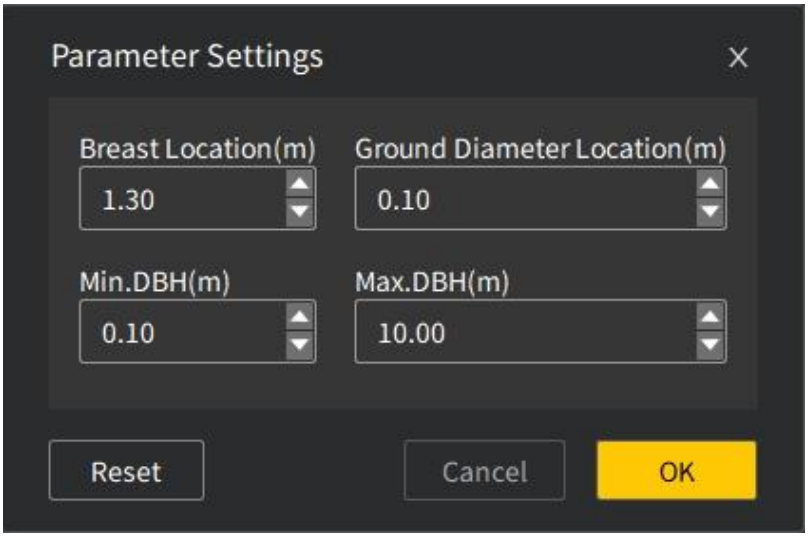



**Figure 306.** Set parameters

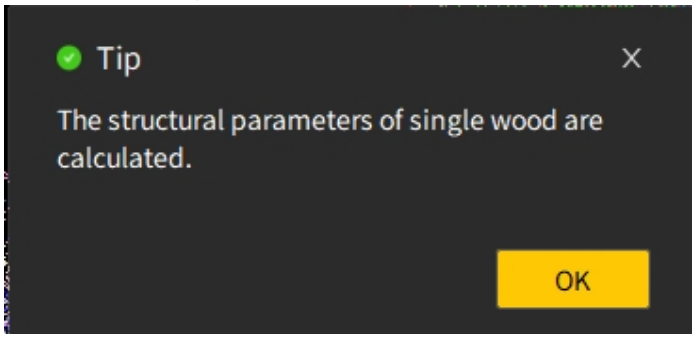



**Figure 307.** Prompt for completed calculation

### 9.4.4 Storage Capacity

**Description:**

The storage capacity is the total volume of all standing trees in the forest. Select different species to calculate the storage capacity with different structural parameters.

**Operation procedure:**

1. Select the required point cloud file under **Objects**.
2. After selecting the data, the volume function can be activated, and the user can choose the appropriate volume calculation method according to the existing structural parameters; only the diameter parameter supports the one-variable volume table method, and when the parameters include diameter and tree height, the two-variable volume table method is supported to calculate the volume;
3. First, choose the country. Currently, China, Japan, and other countries are supported. China supports tree species such as broad-leaved trees, cypress trees, and eucalyptus trees using one-variable and two-variable volume methods. Only the diameter parameter supports one-variable volume table calculation, and when the parameters include both diameter and tree height, it supports two-variable volume table calculation. Japan supports tree species such as Aizu Niigata, Akita broad-leaved trees, and Akita fir, using the Yamamoto-style calculation method.

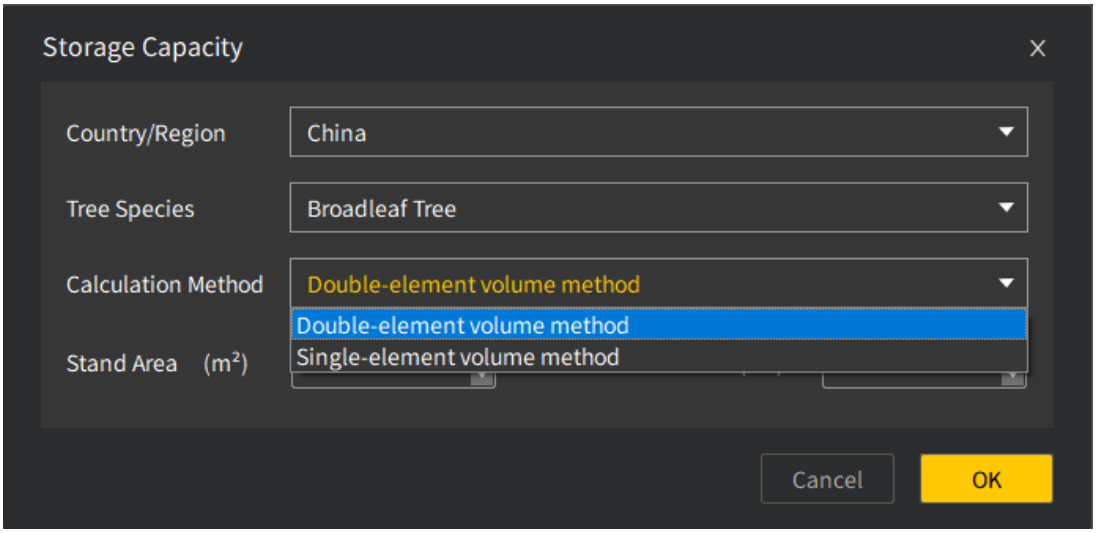



**Figure 308.** Storage capacity calculation

4. Select another country's calculation method to automatically switch to "custom formula" and the calculation method does not support switching; After entering the forest area and standard land area, define the calculation formula, and display the prompt message "Please enter the accumulation calculation formula" in the text box before starting to input; After the user starts to input, hide the prompt, and after completing the formula input, the user can verify the correctness of the input formula by inputting relevant information;
5. Users can enter the values of diameter and tree height. After completing the input, replace the relevant parameters in the text box and click the "Verify" button for calculation and verification.
6. The software automatically saves the last input formula, and does not clear it when it is opened again after exiting the software. The verification value is cleared.
7. When the formula input is abnormal (incorrect, non-standard, etc.) and the calculation cannot be executed correctly, the final display of the calculation is 0.
8. Single wood structure attribute D is diameter, H is tree height, case sensitive.
9. The calculation result is displayed in the result output box. After the user confirms that it is correct, he can click "OK" to calculate the accumulation amount of the selected single log;

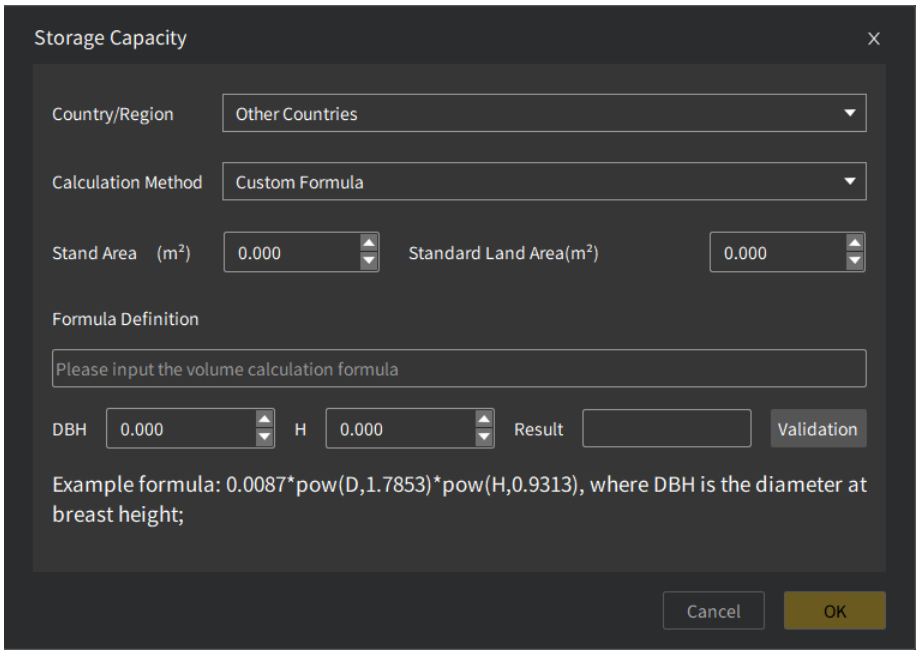



**Figure 309.** custom formula

10. Users can switch the calculation method and enter the forest area and the standard land area;
11. After setting the relevant parameters, click the "Calculate" button to enter the calculation process, and the result window will pop up when the calculation is completed, displaying the accumulated amount;
    Figure 46. Volume display
12. After completing the calculation, right-click on the split single wood point cloud data - structural properties to view the volume information, or export it through the reporting function;

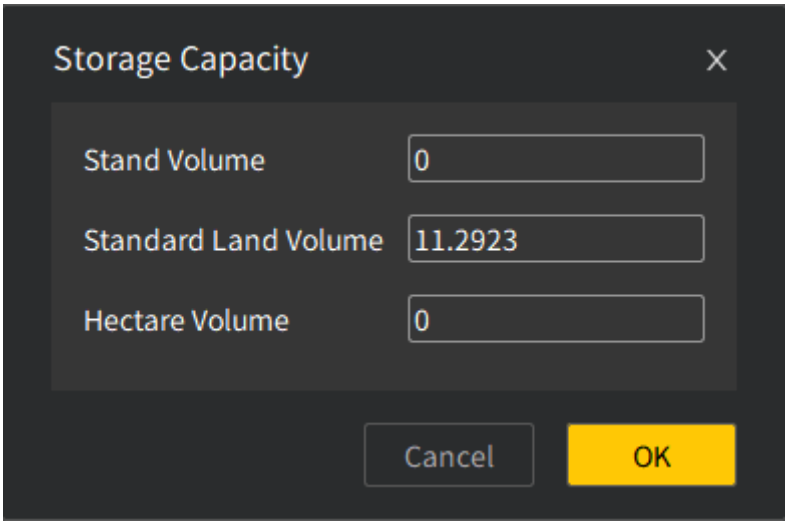



**Figure 310.** Tree storage capacity

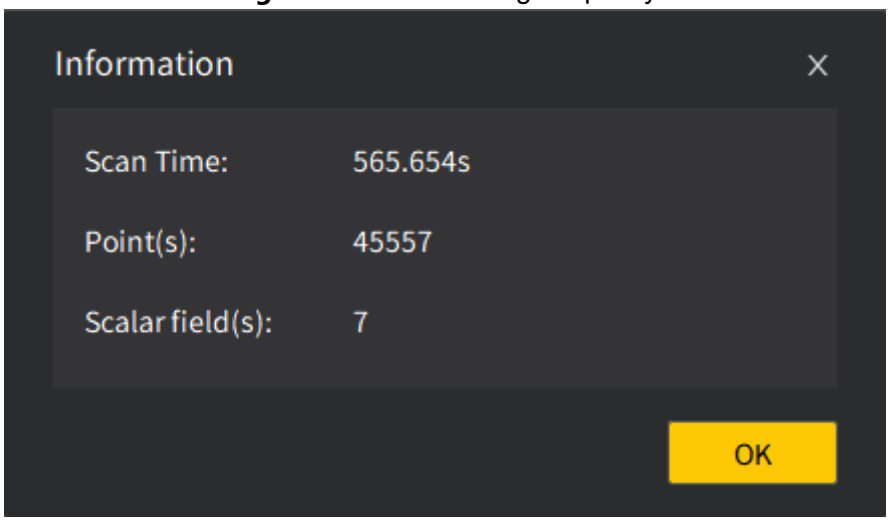



**Figure 311.** Information

### 9.4.5 Carbon Storage

**Description:**

The carbon content of a single tree is calculated by selecting tree species based on the calculated single tree volume.

**Operation procedure:**

1. Select the folder or the single wood segmentation data after the forestry scanning data is segmented, and click the carbon storage function button;
2. First, the tree species type is selected, and after the tree species type is selected, the carbon content is linked to follow the change;

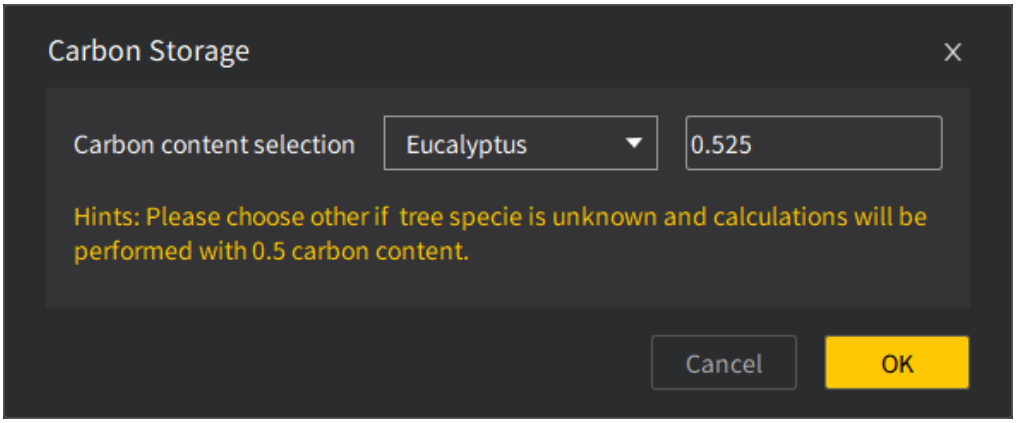



**Figure 312.** Calculation of carbon stocks

3. If the tree species does not exist in the preset tree species list, you can choose another tree species. At this time, the system will use the default carbon content of 0.5 for calculation;
4. After entering the calculation, check whether the calculation object contains the accumulated amount calculation value; if it is detected that the selected object does not contain the accumulated amount, an error pop-up window will be displayed with the prompt: Carbon storage calculation failed, please calculate the accumulated amount and try again!
5. If the selected objects contain the amount of accumulation, the objects are calculated; at the same time, for the case where the amount of accumulation is not included, the amount of accumulation can be directly calculated as 0;
6. After the calculation is completed, it will be updated in the engineering document, and the carbon storage calculation result of the single wood structure parameter will be added.

### 9.4.6 Species Identification

**Description:**

Identifies tree species in a point cloud with panoramic images to perform forestry surveys with a higher accuracy.

**Operation procedure:**

1. In the project file, select the folder or single tree point cloud data obtained from forestry segmentation to activate the tree species identification function.
2. Click  , and the following window appears. After selecting the corresponding image data, click "**Next**" to enter the image-point cloud linkage process. A waiting popup window will appear. The system will detect whether a matching configuration file is loaded. If the configuration is correctly loaded, it will enter the image linkage calculation process, close the original window, and display a progress bar.

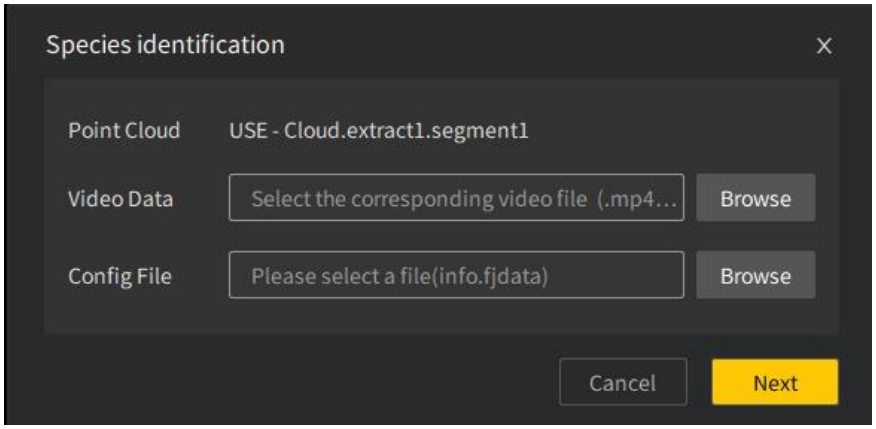



**Figure 313.** Species identification configuration window

3. If the configuration file is not loaded, a warning popup window will appear, prompting the user to manually load the .fjdata configuration file.
4. Once the linkage computation is complete, the waiting popup window will automatically close, and the identification window will open. The point cloud and image linkage view will be updated. Upon entering the function, View 1 will not display the trajectory by default, while View 2 will. When entering the annotation interface, single tree labels will be displayed in the default state.

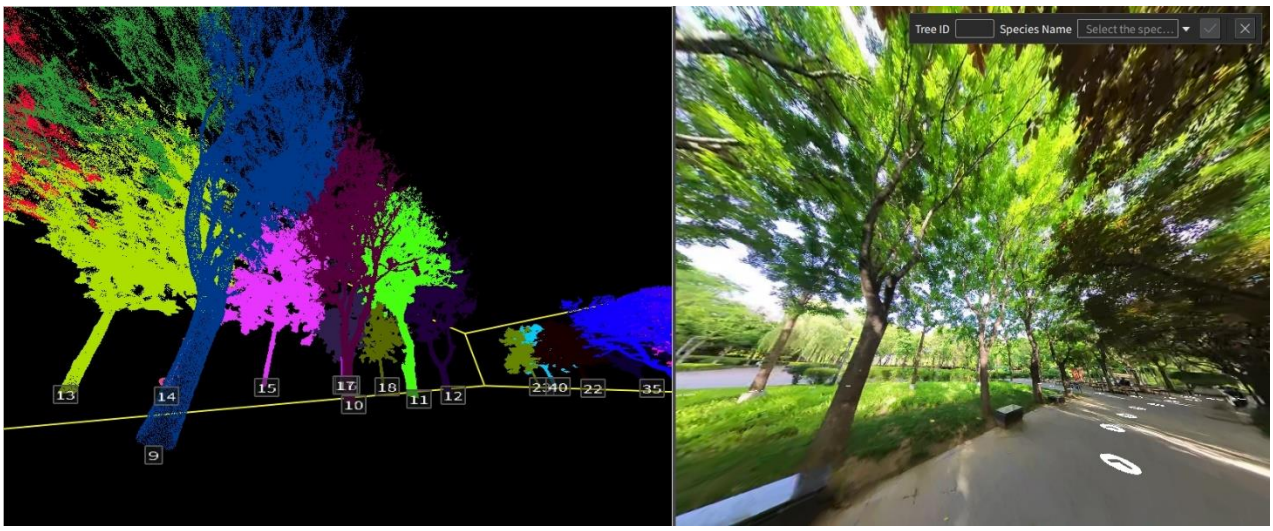



**Figure 314.** Showing the point cloud and the panoramic image

5. You can rotate and zoom to locate the image corresponding to a specific Tree ID. Left-click a point cloud label to select it. Once selected, the Tree_ID will be updated in the window, and the selected single tree label will turn yellow to indicate selection. If the selected point does not belong to any single tree, the ID field will not update. Clicking another point will overwrite the previous selection.
6. After confirming the selection, the user can assign a tree species via a drop-down menu that supports fuzzy search.


**Figure 315.** Setting the species

7. If the required species is not available in the preset list, the user can choose “Custom” to open a popup window for adding a new species. Preset species with codes cannot be modified. Once the custom species is defined, the popup window will close, and the new species will be selected.
8. Click the check icon to save the settings, and the label for that single tree will turn green to indicate it has been annotated. The process can then continue for the next tree.
9. Click the **Close** button or cross icon to exit the feature.

# 9.5 Application

## 9.5.1 Single Tree Distribution Map

**Description:**

Output tree distribution vector diagrams derived from forestry scanning point cloud data.

**Operation procedure:**

1. In the project file list, left-click to select the point cloud data for which you want to generate a single-tree distribution map. First, use the **Property Calculation** function to process the data.
2. After the calculation is complete, the single-tree distribution map function icon will be activated. Clicking it will open a parameter settings pop-up window, where you can manually import a boundary file or use coordinates to select the range, set the filtering method, and specify the minimum spacing.
   - Range File: supports **dxf**, **dwg**, **shp** file formats.
   - Coordinate Information: Uses the maximum area of the point cloud by default;
   - Minimum Tree Spacing: Filters based on the distance between trees, default is 5 meters.
   - Structural Attributes: Filter based on tree attributes.

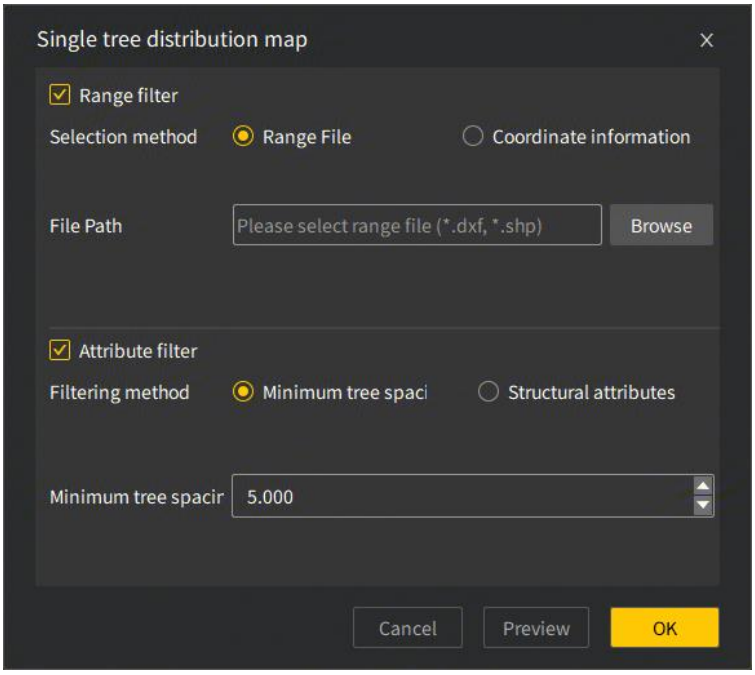



**Figure 316.** Parameter setting pop-up window

3. After completing the settings, click **"Preview"** to generate and view the vector map, then click **"OK"** to save the vector map and close the settings pop-up window.
4. The vector map can be exported in **dxf**, **dwg**, or **bin** file formats.

### 9.5.2 Report

**Description:**

Outputs structural and overall parameters of trees derived from forestry point cloud data. Calculated attributes of segmented individual trees can be exported based on user-defined filtering conditions.

**Operation procedure:**

1. In the **Project** file list, left-click to select the point cloud data for which the report is to be generated.
2. Click to open the report window, which automatically loads the calculated attribute parameters of the individual trees. The table displays up to 10 rows and 8 columns. If the number of entries exceeds the limit, horizontal and vertical scroll bars will appear.

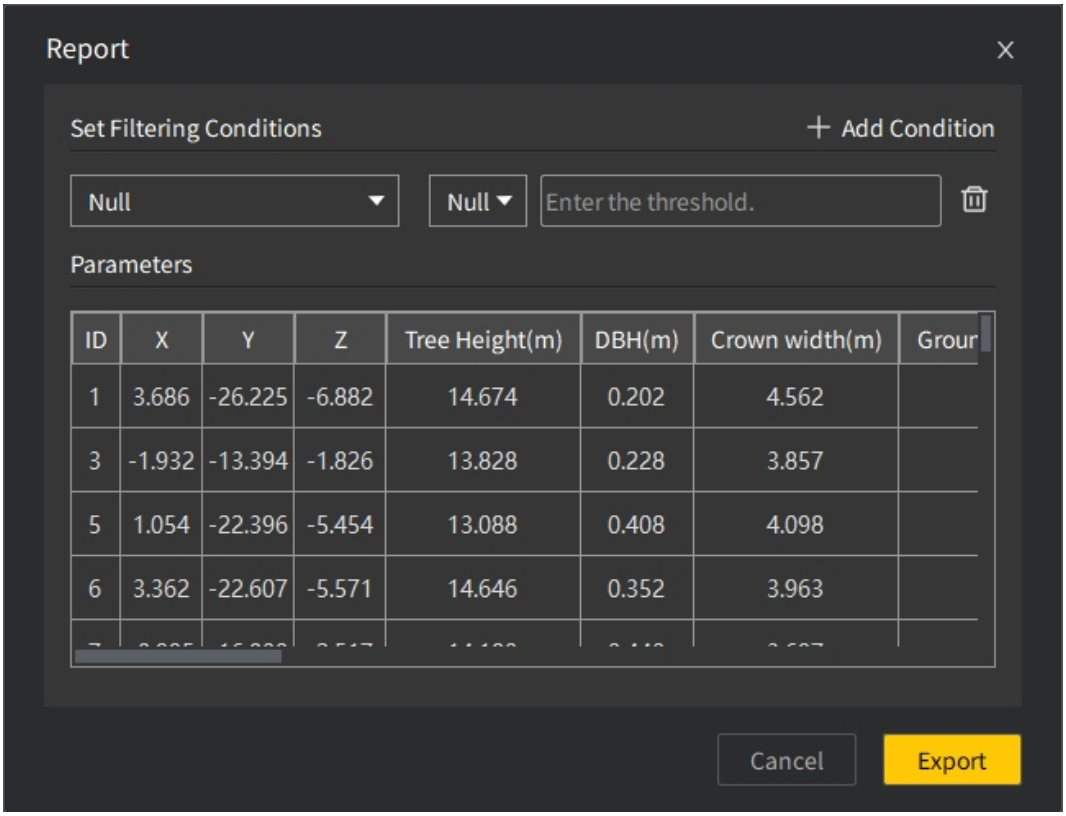



**Figure 317.** Report window

3. You can set filtering conditions to screen the output parameters. All structural attributes supported by the current version can be selected, along with logical operators and threshold values. The list below will automatically update according to the conditions. You can also sort in ascending or descending order using the sorting icons in the table and select the data to be exported via checkboxes;
4. Click **Export**, and the following window appears. Enter the survey area, survey size, surveyor, and export location. Then click OK to export the report. Supported export formats are doc and docx. The default file name is: “PointCloudDataName_ForestrySurveyReport_SystemTime.” Users can rename the file.

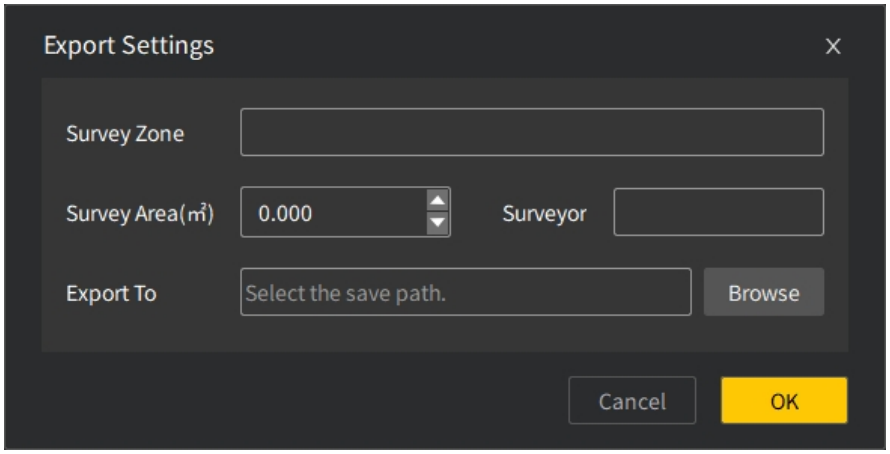



**Figure 318.** Export settings

# 10 Site Survey

Creates a site survey map covering information such as vehicles, buildings, and vegetation based on a site base map and exports a survey report for reference and archiving.

## 10.1 Base Map

**Description:**

Exports an orthographic image or the current view as the base map.

**Operation procedure:**

1. Select a point cloud file under **Objects**.
2. Click , and the following window appears.
3. Select one from the four **Display** options, which are **RGB**, **Height grid values**, **Intensity**, and **Top view**. The default option is **RGB**.

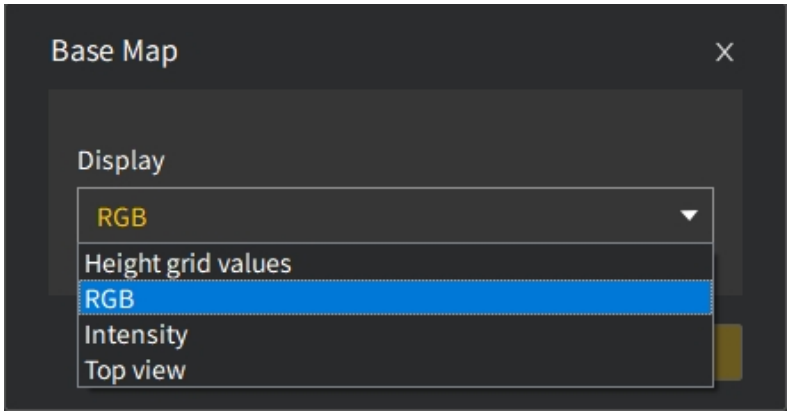



**Figure 319.** Base map setting

4. Click **Preview**, and check whether the result is satisfactory.
5. If satisfactory, click **OK** to start the base map generation process. Once the generation is completed, the base map is shown and the original window disappears.
6. A new file named *original point cloud name*_map*X* where X is a number starting from 1 appears at the same level as the original point cloud file under **Objects**. The number of any new file generated is not impacted if any file is deleted in the list.

## 10.2 Draw

**Description:**

Draws ground features such as vehicles and buildings.

**Operation procedure:**

1. Select a base map file under **Objects**.
2. Click , and the following window appears.

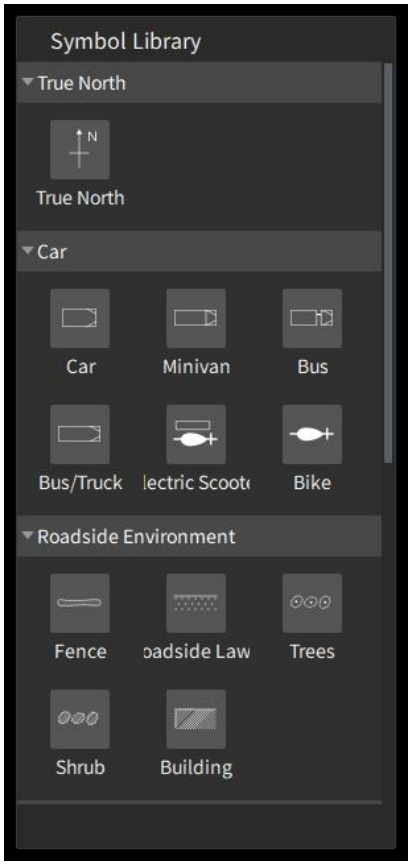



**Figure 320.** Symbol library

3. The 2D view is displayed by default, and can be zoomed in or out or rotated. Select an element from the symbol library, and drag it to the desired place in the view area.

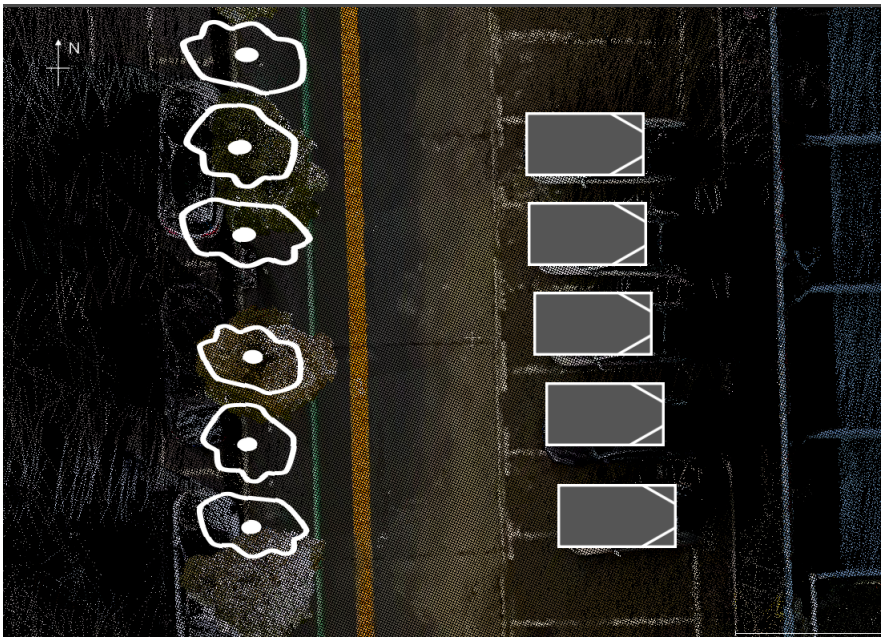


**Figure 321.** Drawing a base map

4. If you select an element, the eight points of the element are shown. Move the cursor to any angular point, and the cursor becomes the diagonal scaling pattern. Move the cursor to the middle point of any horizontal line, and the cursor becomes the vertical scaling pattern. Move the cursor to the middle point of any vertical line, and the cursor becomes the horizontal scaling pattern. Move the cursor to the rotation button, and the cursor becomes the rotation pattern, when you can drag the mouse to rotate the element. Move the cursor to any of the remaining points, and you can drag the mouse to translate the element.

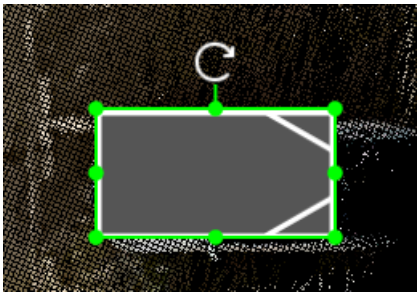


**Figure 322.** Adjusting the symbol

5. To delete an element, select the element, and press Delete or Backspace.
6. Click  to save any changes. If you click  before you save any changes, the message "Changes not saved. Are you sure you want to exit?" appears. After drawing is completed, click  to exit the feature.

## 10.3 Marking

**Description:**

Marks with text and images on a drawing.

**Operation procedure:**

1. Select a base map under **Objects**.
2. Click **Text Marker**, and select a proper place to display the text edit box, whose size can be adjusted based on the text entered. After you enter the text, the text is displayed on the base map.
3. Click **Image Marker**, and add only one image each time on the base map. The details field is optional.

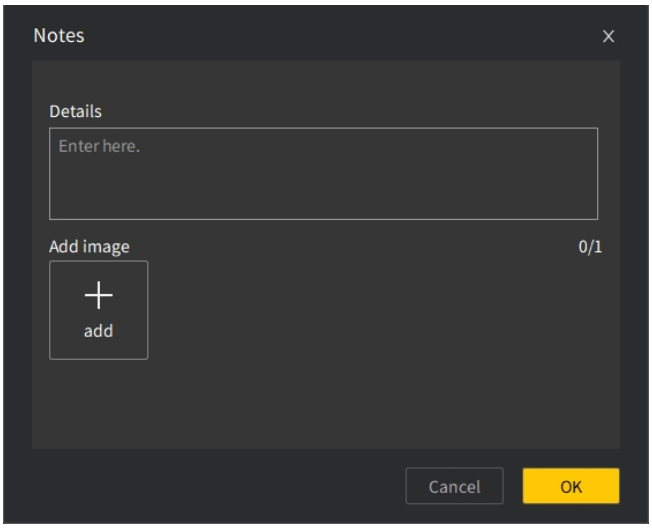



**Figure 323.** Image marker

4. The added image appears as a child node below the current object. When you move the cursor to the image marking place, the thumbnail image is shown.
5. To delete a text marker or image marker, select the text marker or image marker in the view area, and press Delete or Backspace.

## 10.4 Measure

**Description:**

Measure the length, angle, area, and perimeter, and obtain coordinates of a point on the drawing.

**Operation procedure:**

1. Select a base map under **Objects**.
2. Click  to obtain and display the coordinates on the base map. If you want to measure the length, angle, area, or perimeter, click the corresponding icon.

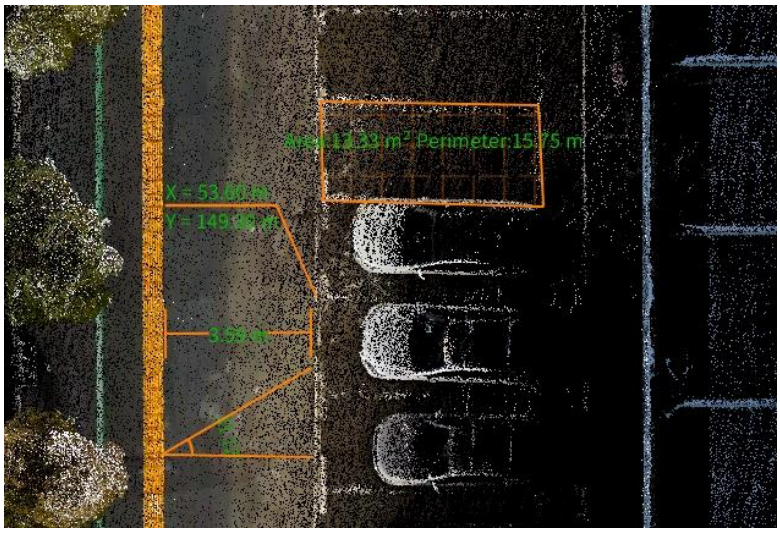



**Figure 324.** Measurement

# 10.5 Report

**Description:**

Exports the drawing.

**Operation procedure:**

1. Select a base map under **Objects**, and click **Report**.
2. Enter the parameters. The drawing time is the system time obtained automatically, which cannot be changed in the current window. Change it in the exported report if you want to.

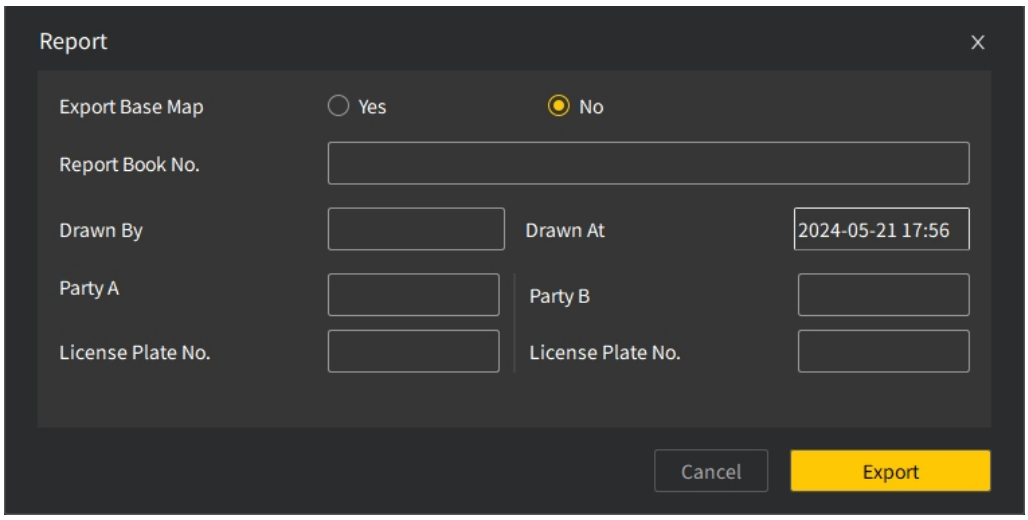



**Figure 325.** Exporting the report

3. Click **Export** and select the export location.
4. If no data writing is allowed in the selected path, a message "No data writing allowed in the path. Select again." appears.
5. The export in-progress window appears during exporting, and a message appears when the export is successful.

# 11 Section Analysis

The section analysis module for tunnel design mainly deals with reference section line design based on point cloud and 3D model data, actual section extraction, over-excavation analysis, and report generation.

Broken Chain: Broken Chain refers to a phenomenon in tunnel design. When a certain part of the tunnel line is interrupted due to certain reasons (such as geological conditions, design changes, etc.), it is called broken chain. In the process of tunnel design and construction, it is necessary to analyze and handle broken chains to ensure the safety and stability of the tunnel.

## 11.1 Design Import

### 11.1.1 Design Axis Line

**Description:**

Create axis based on tunnel parameters.

**Operation procedure:**

1. Click the function icon to open the axis import function window, enter the parameters to create the axis;
2. First, create a broken chain parameter to define the starting mileage and ending mileage; if there is no broken chain, you can directly click Next to create a flat curve;
3. When selecting the flat curve type, the analytical method can be activated, and there are two methods to choose from: intersection method and line element method;
4. When the user selects the intersection method, there are three types: starting point, intersection point, and endpoint. The definition elements of different types of points are also different. The definition elements of the starting point are: starting mileage, x coordinate, y coordinate; the definition elements of the intersection point are: number, radius, x coordinate, y coordinate, easing curve length Ls1, easing curve length Ls2, easing curve parameter A1, easing curve parameter A2; the definition elements of the endpoint are: x coordinate, y coordinate.
5. When the user selects the line element method, there are five types: straight line, circular curve, incoming easing curve, outgoing easing curve and incomplete easing curve. The definition elements of different types of points are also different. Definition elements of a straight line: start mileage, end mileage, start x-coordinate, start y-coordinate, start azimuth; Definition elements of a circular curve: start mileage, end mileage, start x-coordinate, start y-coordinate, radius, turn, start azimuth; Definition elements of an easing curve: start mileage, end mileage, start x-coordinate, start y-coordinate, end radius, turn, start azimuth; Definition elements of an incomplete easing curve: start mileage, end mileage, start radius, end radius, turn, start azimuth; Definition elements of an incomplete easing curve: start mileage, end mileage, start x-coordinate, start y-coordinate, start radius, end radius, turn, start azimuth;

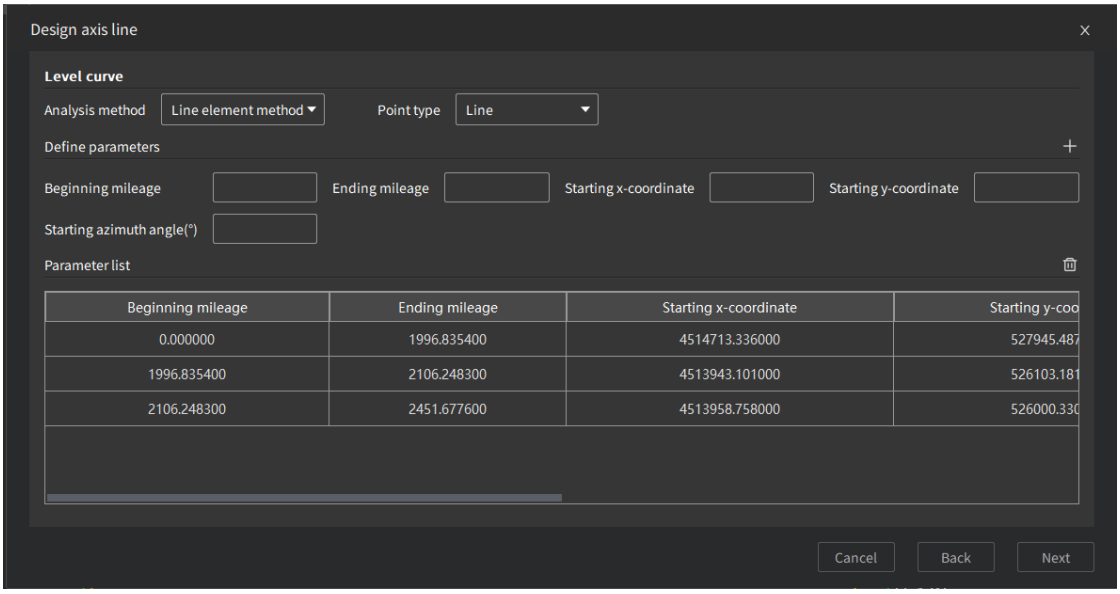



**Figure 326.** Design axis line

6. After selecting the curve type and parsing method, you can enter the parameters in the text box below, and click the + button in the upper right corner to load the input information into the data list;
7. For the data in the data list, you can select it and delete it through the delete button in the upper right corner. After the flat curve is created, the vertical curve is created;
8. When the curve type is switched to a vertical curve, the definition parameters and data list follow the option transformation. The definition parameters of the vertical curve are mileage, elevation and radius;
9. Users can enter the parameters in the text box below, and click the + button in the upper right corner to load the input information into the data list;
10. For the data in the data list, you can select it and delete it by clicking the delete button in the upper right corner.
11. After completing the relevant curve creation, click the OK button, and the interface will pop up the waiting prompt box of "Design file creation, please wait!"
12. After creating the design axis, close the waiting prompt box and load the design axis under the engineering vector node;

### 11.1.2 Design Section

**Description:**

Generate the section at the specified mileage based on the point cloud and the design axis.

**Operation procedure:**

1. Click the function icon to open the section import function window, where you can create a section by defining parameters;
2. The interface displays the interface created by parameters. Define the parameters of line type, starting point x, starting point y, ending point x, ending point y, and radius. Among them, line type includes two types: straight line and arc.
3. The user selects the line type and enters the relevant definition parameters. After completing the input, click the + button in the upper right corner to load the input information into the data list.

4. For the data in the data list, you can select it and delete it by clicking the delete button in the upper right corner;
5. Click the Confirm button, and the interface will pop up the waiting prompt box of "Design file creation, please wait!"
6. After completing the creation of the design section, close the wait for prompt box, update it in the view, and only display the section creation result;

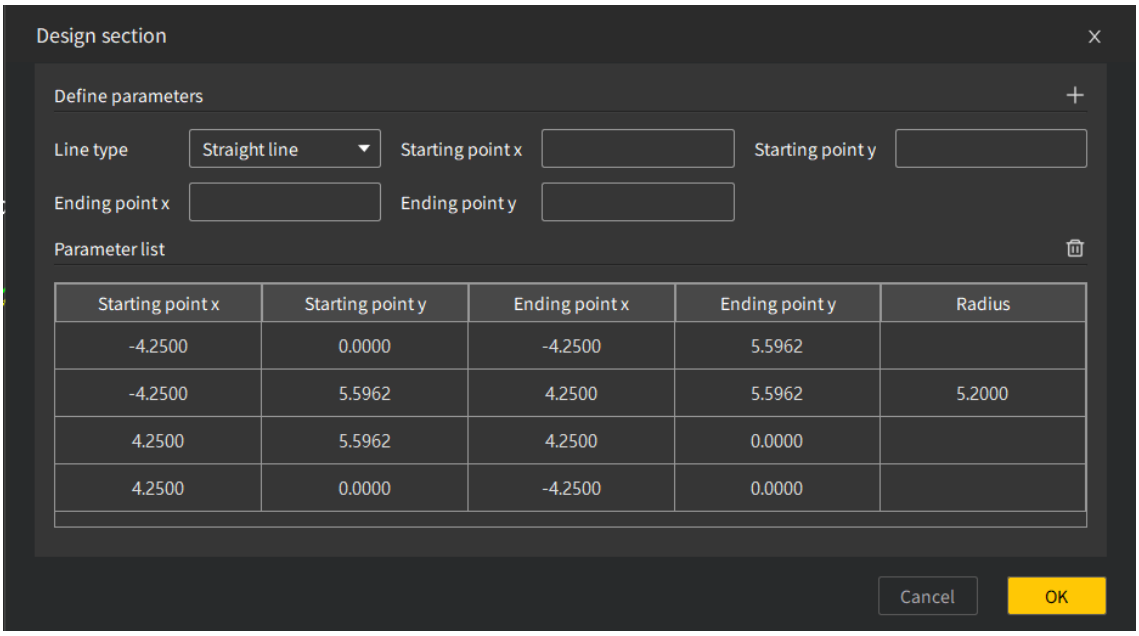



**Figure 327.** Design section

# 11.2 Analysis Comparison

## 11.2.1 Section Generation

**Description:**

Generate section files based on point cloud data.

**Operation procedure:**

1. In the project file column, left-click to select the point cloud object to be analyzed, and click the section generation function icon;
2. After clicking the section generation function icon, open the section generation function window. The user selects the corresponding design axis of the comparison point cloud, and the design axis drop-down menu loads the axis information by default;
3. Next, set the generation parameters. The starting mileage and ending mileage are defaulted to the starting and ending mileage in the design file. The default value of the section thickness is 0.02m, and the mileage interval is 5m;
4. It is also a section output that supports specific mileage, and can output up to 10 mileages;
5. After setting the parameters, click "OK" to create the section and display the waiting window;
6. Automatically close the pop-up window after completing the section creation, and create a section file as a sibling file and hang it under the analyzed point cloud. By default, only newly created data is displayed

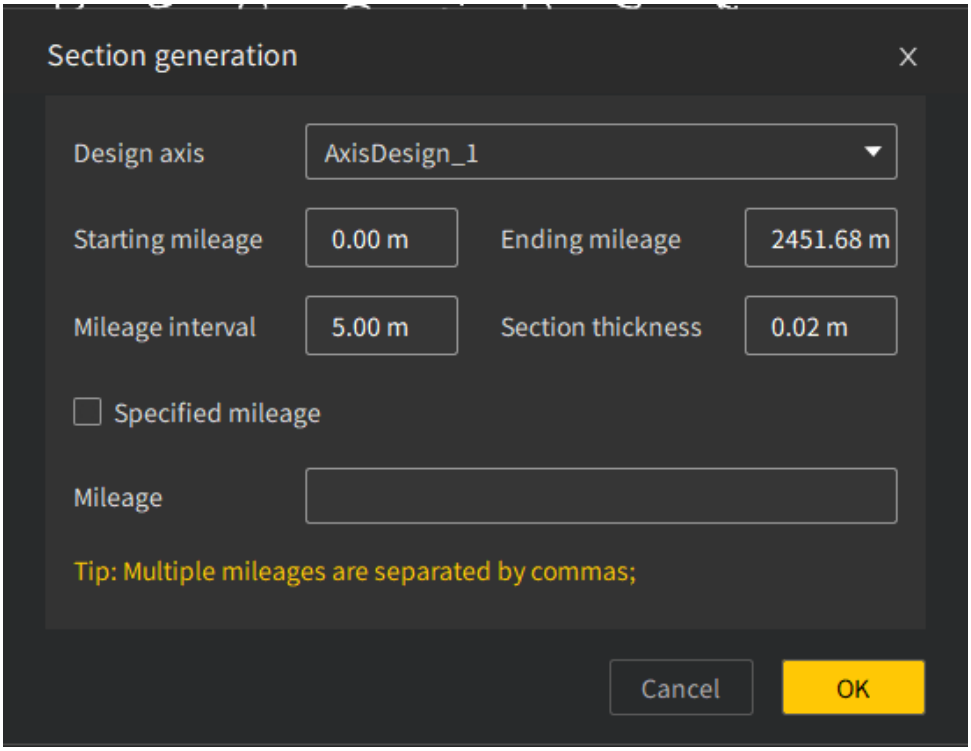



**Figure 328.** Section generation

## 11.2.2 Section Comparison

**Description:**

Compare the cross-sectional data with the design file.S

**Operation procedure:**

1. Select the cross-sectional data to be compared and analyzed in the project file, and support multiple selection by Ctrl + left mouse click;
2. After selecting the section file, click Section Analysis;
3. After clicking the section analysis function icon, open the section analysis function window, select the reference section, and the drop-down list of the reference section is to import the design file;

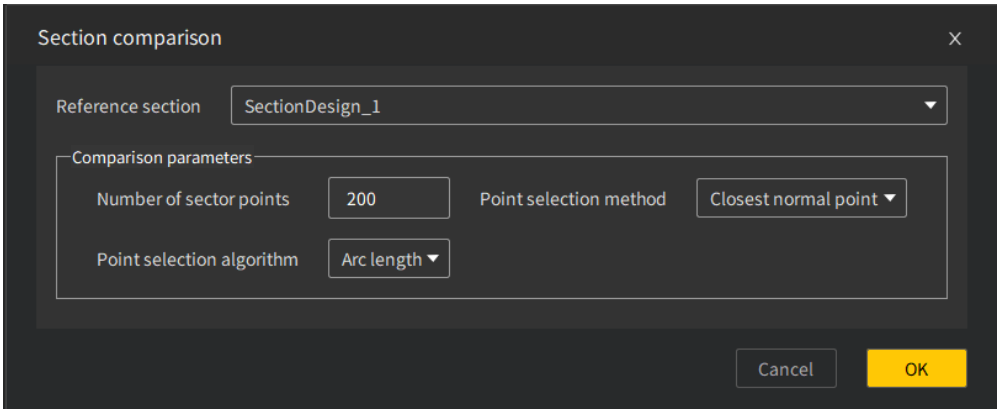



**Figure 329.** Section comparison

4. Then set the comparison parameters, and click "OK" to enter the cross-sectional analysis process after completing the parameter setting;
5. Display the calculation waiting prompt box (need to display the calculation and analysis progress, such as 5/10, completed calculation/total calculation).

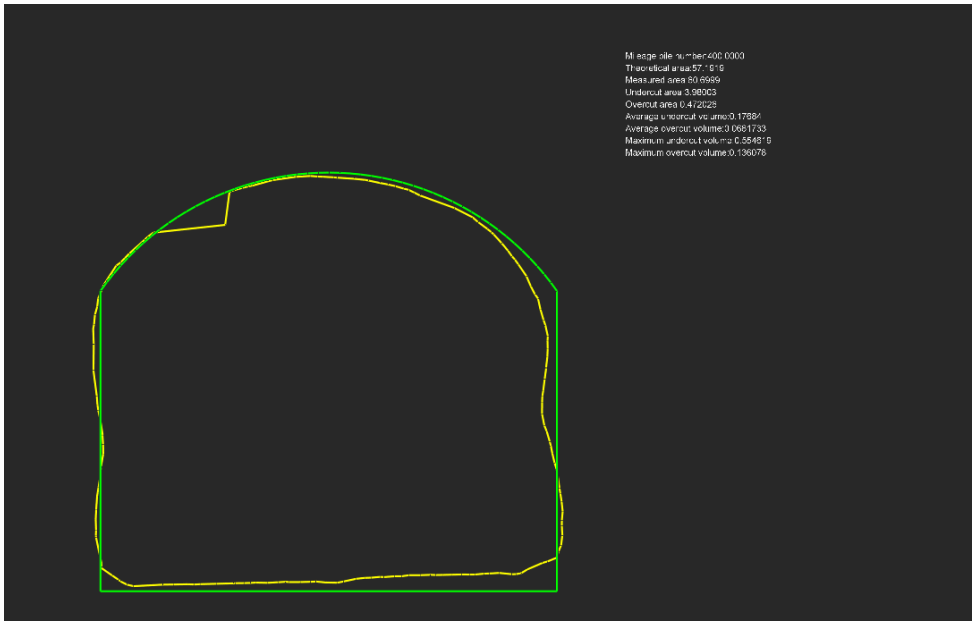


**Figure 330.** Section comparison

### 11.2.3 Result Annotation

**Description:**

Mark the cross-sectional results.

**Operation procedure:**

1. After selecting the section analysis result in the project engineering document, the result marking function is activated and available;
2. Click the result annotation function icon to open the annotation settings window, and the mouse in the view area changes to the annotation style;
3. There are two annotation methods: manual annotation and automatic annotation. When selecting manual annotation, some users click the mouse to determine the result annotation point, and the result annotation cannot be performed in the non-point cloud area. The specific annotation style: over-mined data: annotated with red (255,0,0) 16-point font; under-mined data: annotated with blue (0,0,255) 16-point font, and the annotation information is arranged perpendicularly to the cross-sectional vector.

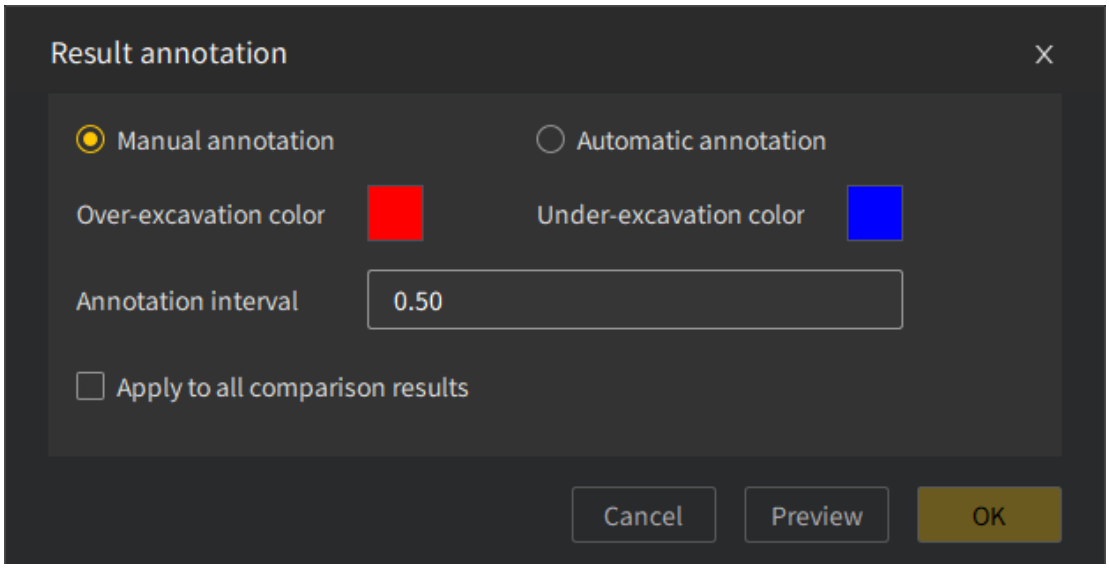



**Figure 331.** Result annotation

4. When marking an error, you can use the Ctrl + Z shortcut combination to undo the operation. After the marking is completed, the user can choose whether to apply it to all comparison results;
5. After selecting the application, perform the same result labeling on other data according to the current analysis result position;
6. If the user selects automatic annotation, the annotation will be performed in ascending order of the x-coordinate within the height range of the filter by default, and the annotation interval shall be determined by the input;
7. If Apply to all comparison results is not checked, only the currently selected result will be annotated. If Apply to all comparison results is checked, all annotations will be annotated with the same annotation trajectory.
8. When modifying the marking parameters and marking again, clear the existing marking results and re-mark them;
9. After completing the annotation, click "OK" to apply the annotation result and exit the result annotation function;

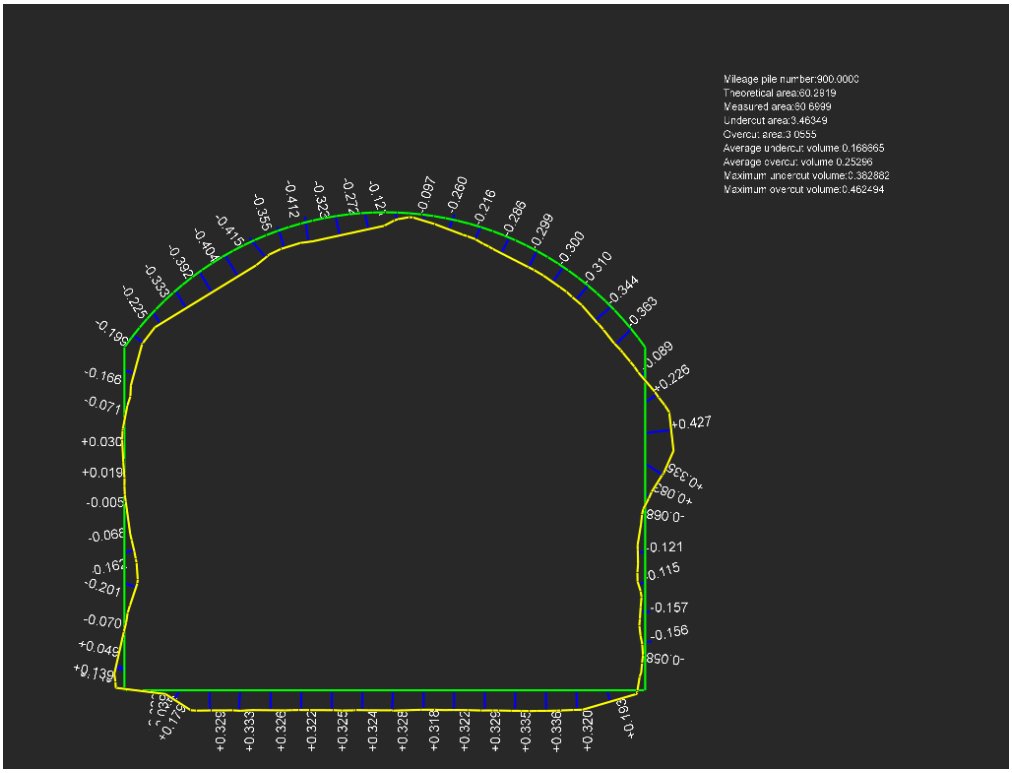



**Figure 332.** Result annotation

## 11.2.4 Report Output

**Description:**

Output section analysis report.

**Operation procedure:**

1. After selecting the section analysis result in the project engineering document, the result report function is activated and available;
2. Click the report function icon to open the function window, enter the project name, measurement time (default generation time, format is 2024/04/11), measuring equipment, surveyor and reviewer, the above are not required;

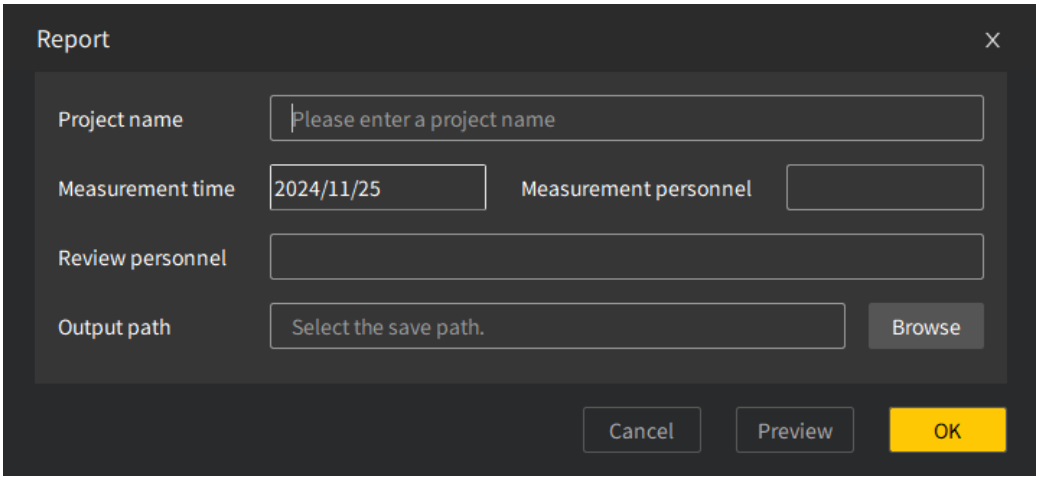



**Figure 333.** Report

3. Finally, after browsing and selecting the output location, you can click to preview the report content and style, and export the supported format as: docx;
4. After confirming, click "Close" to close the preview window, and finally click "OK" to save the report output;
5. Display a waiting box when entering the save process, close the pop-up window after saving, and display the save success prompt;

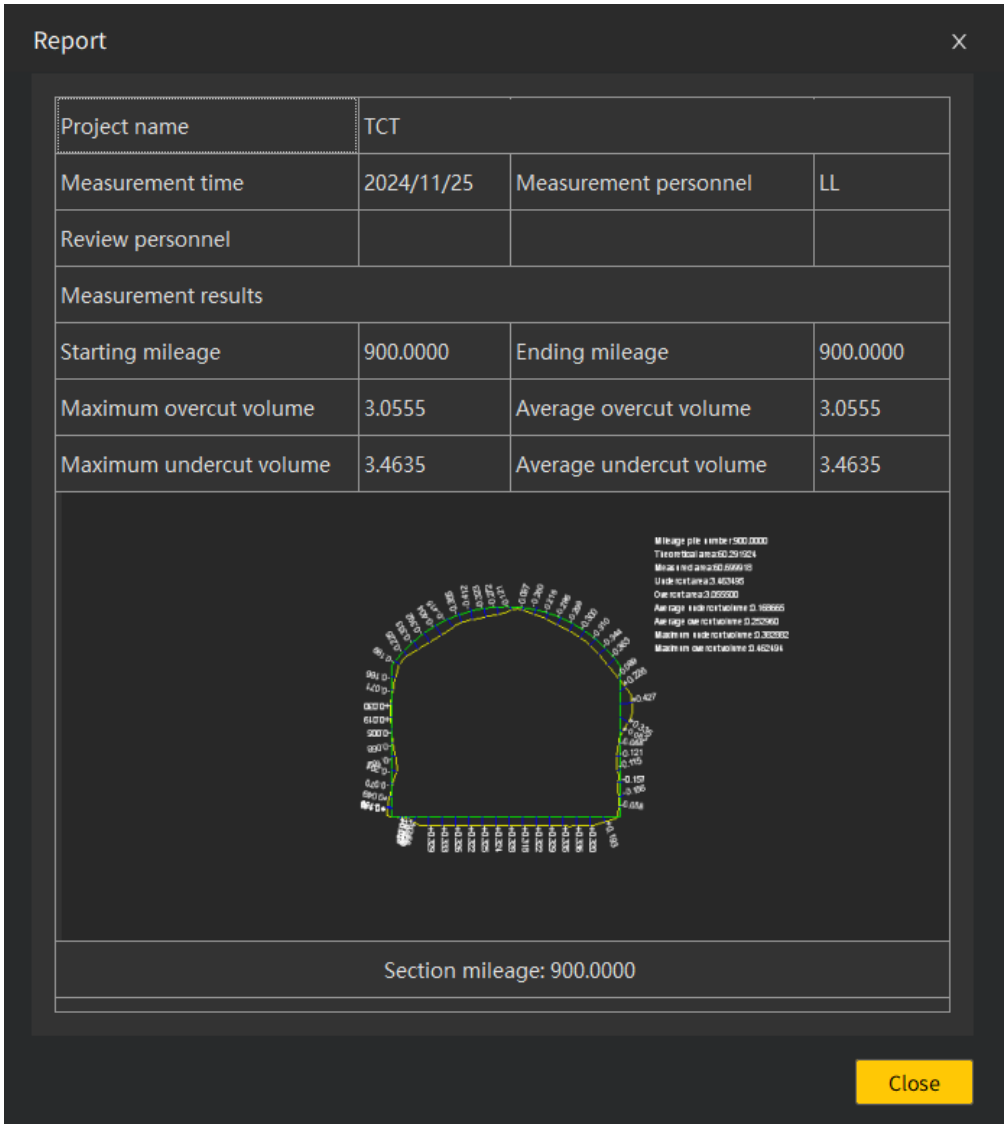



**Figure 334.** Report View

# 12 Gaussian Modeling

## 12.1 Reality Modeling

### 12.1.1 Reality Modeling

**Description:**

Create Gaussian models from point clouds and panoramic videos.

**Operation procedure:**

1. In the project file list, left-click to select the point cloud data. The Reality Modeling icon will become available;
2. Click the icon to open the reality modeling popup window. Select the scanning scenario, camera type, image data, and set the model save path.
3. When using the built-in camera, you need to select the original project data file (.fjdslam or .fjdslam.tgz). If you using an external camera, you need to select the original video file (.insv);

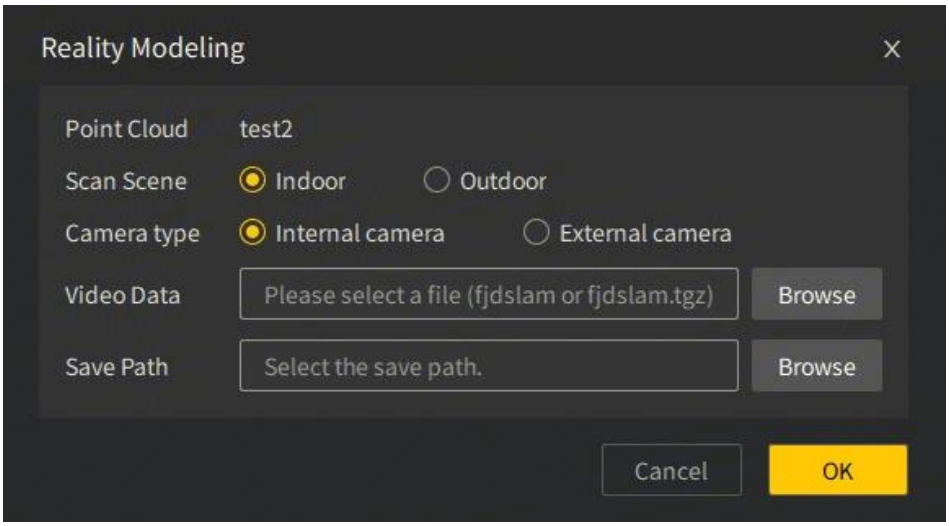



**Figure 335.** Reality modeling pop-up window

4. After the settings are completed, click **OK** to automatically calculate and generate a 3D Gaussian model (.ply);
5. In addition to using the mouse to move the view, you can also use the WASD keys on the keyboard to control the view to move forward, backward, left, and right, use U/O to rotate the view counterclockwise and clockwise, use I/K to adjust the pitch of the view, and use Q/E to move the view vertically.


**Figure 336.** 3D Gaussian Model

### 12.1.2 Load

**Description:**

Imports and displays a 3D Gaussian model.

**Operation procedure:**

1. Click the function button , select the local 3D Gaussian model file (.ply), click OK and import it into the project.

### 12.1.3 Return To Starting Point

**Description:**

Return to the initial scanning position.

**Operation procedure:**

1. In the project file list, left-click to select the 3D Gaussian model data. The function button will become available.
2. Click the button to return to the starting point of the scanning trajectory (if the .fjdata file is missing, it will not be possible to return to the starting point).

## 12.2 Image Fusion

### 12.2.1 Linkage

**Description:**

Select the Gaussian model, match the corresponding point cloud data and configuration file to perform matching calculations between the point cloud and the Gaussian model, so that the Gaussian model and point cloud at the same location can be displayed simultaneously, making it easier for users to view data. Allows the selected Gaussian model to be matched with corresponding point cloud data and configuration files, enabling simultaneous display of both datasets at the same position. This helps you better visualize and compare data.

**Operation procedure:**

1. In the project file list, left-click to select the 3D Gaussian model data. The function button becomes available;
2. Select the corresponding point cloud data used to generate the Gaussian model and click OK to start the matching computation;

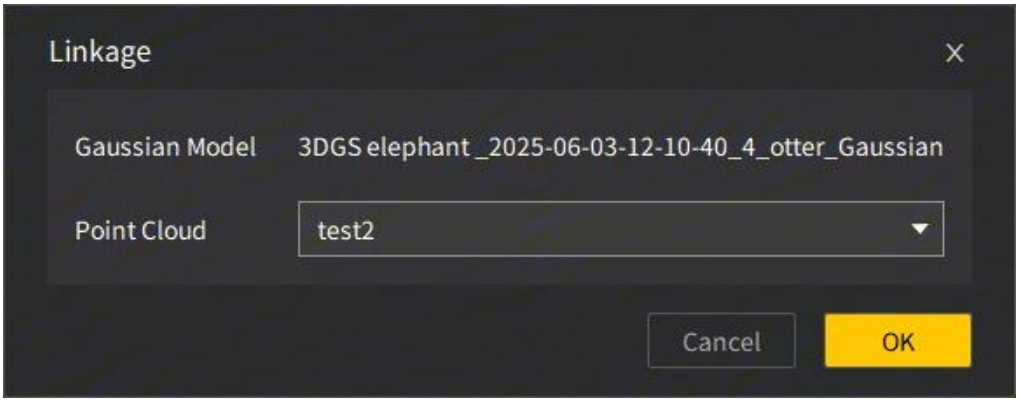



**Figure 337.** Linkage pop-up window

# 12.3 Video Creation

## 12.3.1 By Track

**Description:**

Supports creating video files of the Gaussian model by following the scanning trajectory.

**Operation procedure:**

1. After selecting a single Gaussian model, the function of creating videos by track is activated;

2. Click to display the settings pop-up window, select the video rendering frame rate, preview and play, and click Export after confirmation.

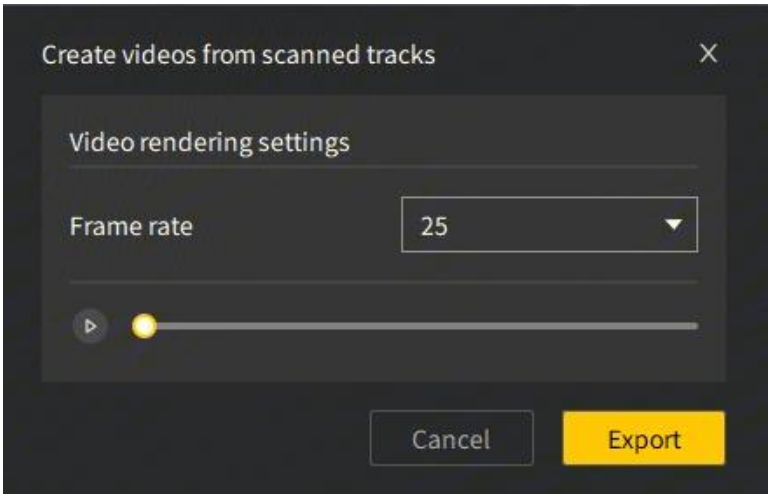



**Figure 338.** By track

## 12.3.2 By Keyframes

**Description:** Supports creating video files of the Gaussian model based on user-defined keyframes.

**Operation procedure:**

1. After selecting a single Gaussian model, the Create Video by Keyframes function becomes available.

2. Click to display the setting pop-up window. First, add keyframes, then select the video rendering frame rate. You can preview the animation before confirming and exporting the video.

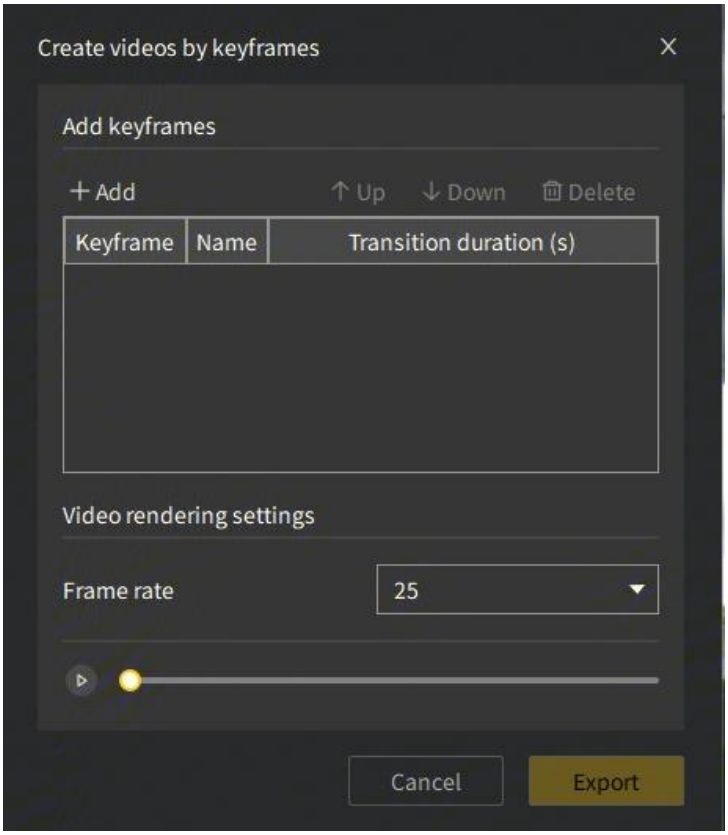



**Figure 339.** By Keyframes

# 13 Appendix

## 13.1 Key Terms

1. Lidar: A system that emits laser beams and receives echoes to obtain 3D information of the target. Source: GB/T 14950-2009 *Terms of Photogrammetry and Remote Sensing*, 4.150.
2. Light Detection and Ranging (LiDAR): An active measurement technology that, using a fixed or mobile platform, emits laser beams to obtain information such as 3D coordinates and reflection intensity of the surface of objects. Source: CH/T 3020-2018 *Technical Regulations for Real Scene 3D GIS Data Collection Using LiDAR*, 3.2.
3. Point cloud: A collection of points distributed in a 3D space in a discrete and irregular way. Source: T/CAGIS 5-2021 *Vehicle-Mounted LIDAR Mobile Mapping System*, 3.4.

## 13.2 High-performance Graphics Mode Settings

1. Download and install the latest graphics driver.
2. Restart the computer, and right-click the desktop and select **NVIDIA Control Panel**.

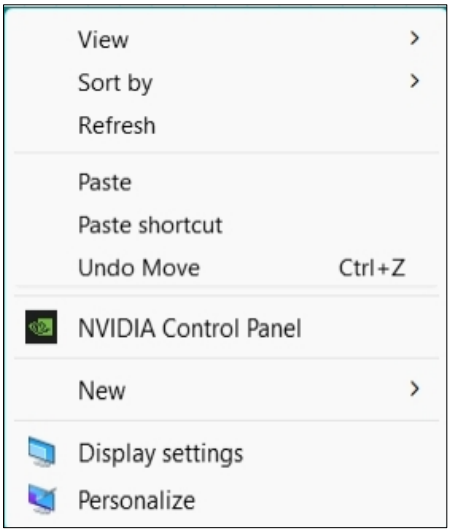



**Figure 340.** NVIDIA control panel

3. Choose **Manage 3D settings** > **Program Settings** and add "FJD Trion Model.exe" to the high-performance graphics mode list, and then click **Apply**.

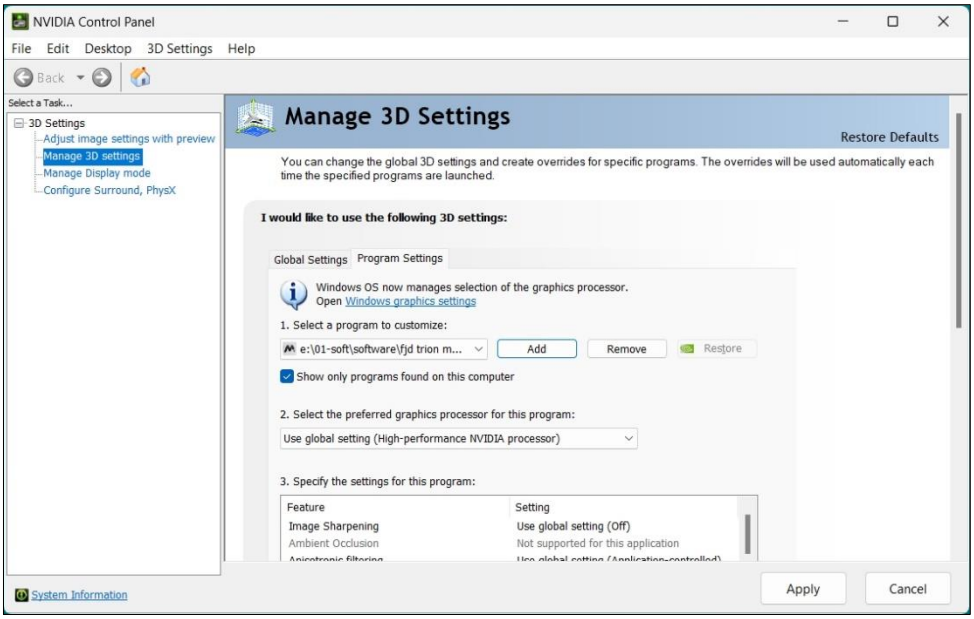



**Figure 341.** Graphics mode settings

# 14 Troubleshooting

1. The language switch (between Chinese and English) does not take effect after the software is restarted.

**Solution**: This may appear when the software is installed on C drive. Right-click it and select **Run as administrator** to make the language switch take effect.

2. The **Edit** toolbar is gray and inactive.

**Solution**: Select the point cloud file to edit under **Objects** and the **Edit** toolbar becomes active.

3. The software cannot be started.

**Solution**: Download and install the software from the official website, and then contact the dealer for an activation code.