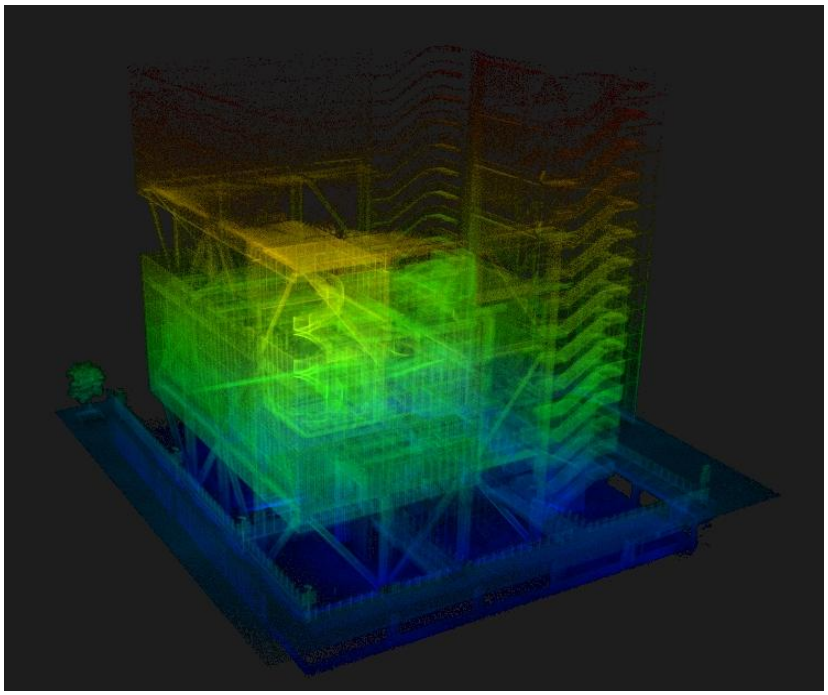
FJDTrion™



FJD Trion Model

Point Cloud Processing Software

User Manual

January 2024 | Rev.1.5

Copyright Notice:

Read Before Use:

Operate in strict accordance with this software user manual.

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.

Contents

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. Usually, such point cloud data includes a 3D point cloud and an optional 2D image set. Each point in the point cloud includes its 3D coordinates and other properties, such as intensity and surface normals. A point cloud can be visualized, rotated, translated, zoomed in, and zoomed out in a 3D view area. The visualization effect can be enhanced in different ways, for example, by changing the point size, intensity, colors, and visual angles.

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.

FJD Trion Model supports the following features:

Point cloud mapping	Supports offline reconstruction of point cloud data collected by the FJD Trion S1 or FJD Trion P1 laser scanner.
Accuracy verification	Supports plane and elevation accuracy verification, providing users with more accurate and reliable data. Analyzes the point cloud quality by use of horizontal and vertical slices.
View	Sets display colors of the point clouds and changes background color and view angles, providing good user experience.
Point cloud processing	Preprocesses raw point clouds with features such as point cloud segmentation, thinning, clustering and SOR filter to satisfy different scenario requirements.
Mesh processing	Creates triangular meshes from point clouds, and provides mesh smoothing, and mesh sampling, improving data operation fluency.

Measurement	Carries out measurements of point, length, height difference, angle, and area, ensuring data accuracy.
Classification	Carries out classification in multiple indoor and outdoor scenarios to automatically extract walls, floor, ceiling, grounds with different slopes, and trees in point clouds.
Volume calculation	Measures volume of piles such as coal piles, granaries, and construction earthwork.
Two-dimensional vector graphic drawing	Draws two-dimensional vector graphics based on point cloud slices to reversely restore the plane drawings of the scanned objects.
Forestry data processing	Obtains tree parameters and exports reports.

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 GTX 1650 Ti 4G
- Hard disk drive: 64 GB
- Screen resolution: 1680 × 1050 or above

Minimum configuration:

- Processor: Intel® Core™ i5-2300 CPU @ 3.10 GHz (or an AMD CPU with equal performance) or above
- Memory: 8 GB
- Graphics card: Graphics 630
- Hard disk drive: 32 GB
- Screen resolution: 1680 × 1050 or above

2. Software:

- Operating system: Windows 8 or above (Windows 11 recommended)

2.2 Download

Download the FJD Trion Model package at:

https://www.fjdynamics.com/product/trion-s1-3d-lidar-scanner.html

2.3 Installation

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

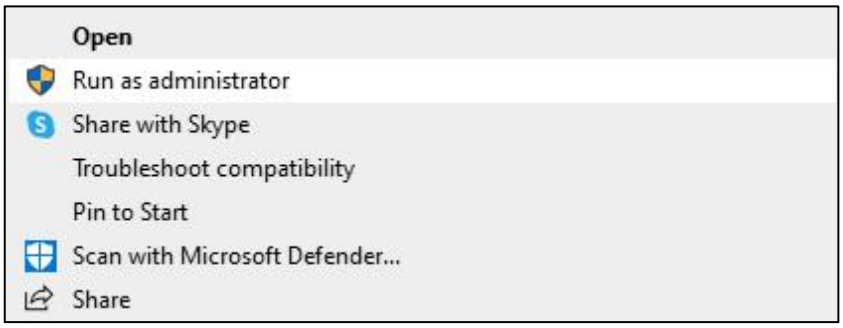


Figure 1. Running the program

2. Select the setup language from "简体中文","English", "繁體中文", and "日本語", and click **OK**.

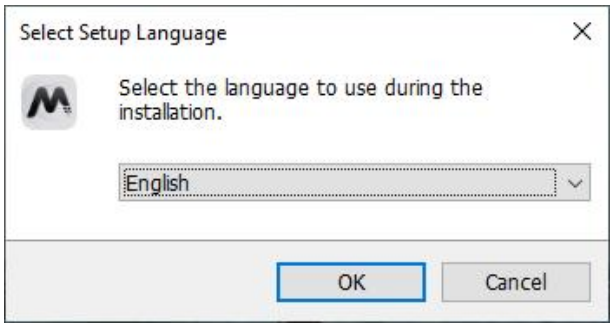


Figure 2. Selecting the setup language

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.

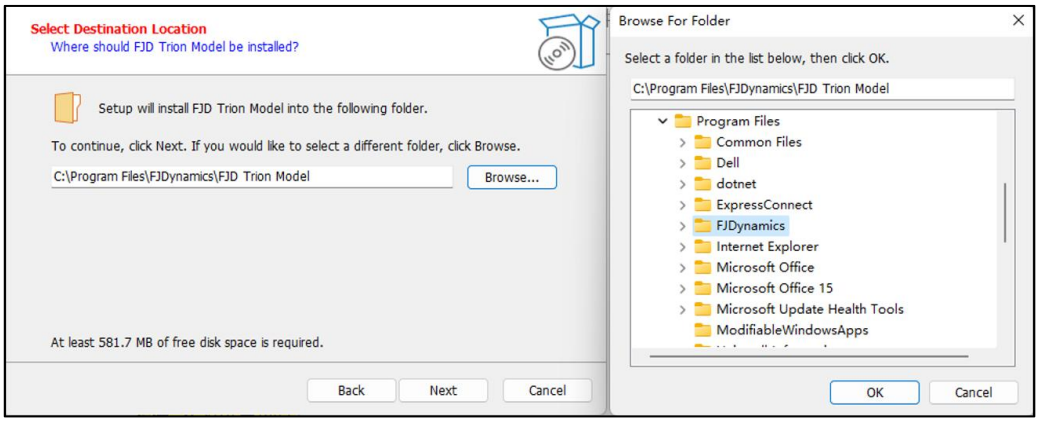


Figure 4. Selecting the installation location

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

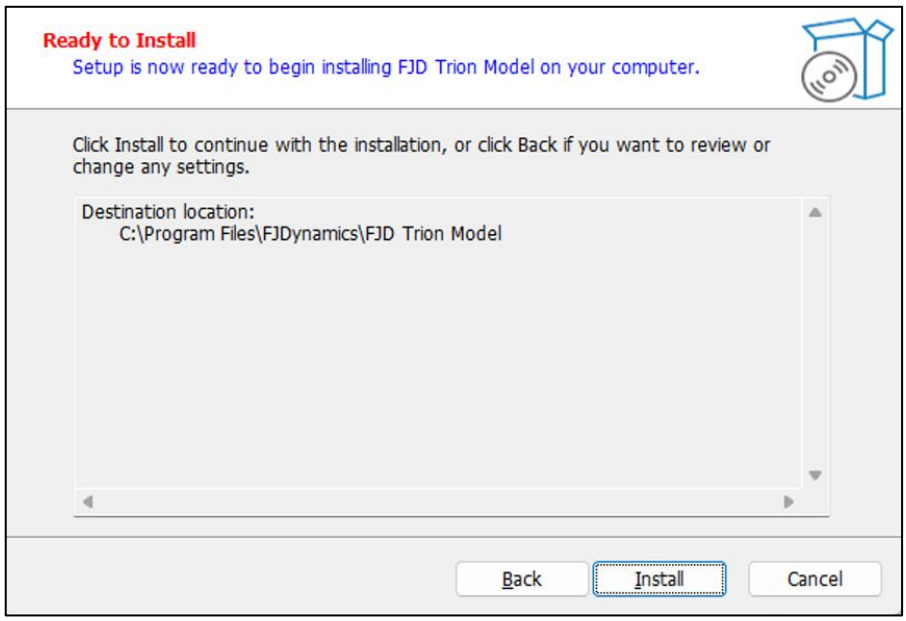


Figure 5. Installing FJD Trion Model

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 6. Uninstalling FJD Trion Model

You can also go to the software installation directory and click unins000 to uninstall it.

2.6 Activation

FJD Trion Model is a LiDAR point cloud processing software developed by FJDynamics, and can be used only after activation. You can activate it by activation code or encryption lock based on your actual needs. Go to the **License setting** window, choose **Activation code** to activate by activation code, or choose **Encryption Lock** to activate by encryption lock, as shown in Figure 7. When the software is activated successfully, the **License management** window is displayed as Figure 8 or Figure 9.

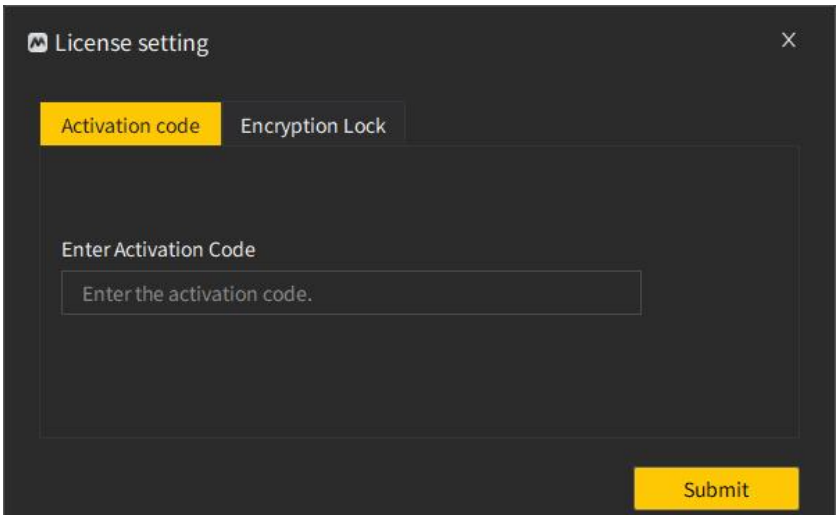


Figure 7. License setting

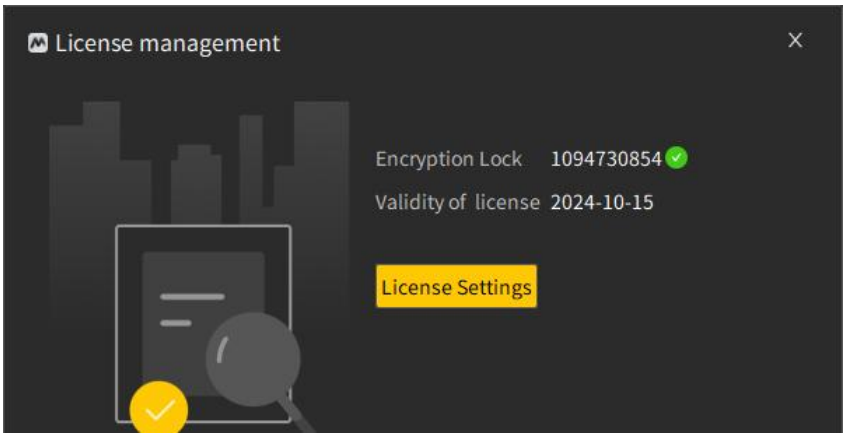


Figure 8. Successful activation by encryption lock

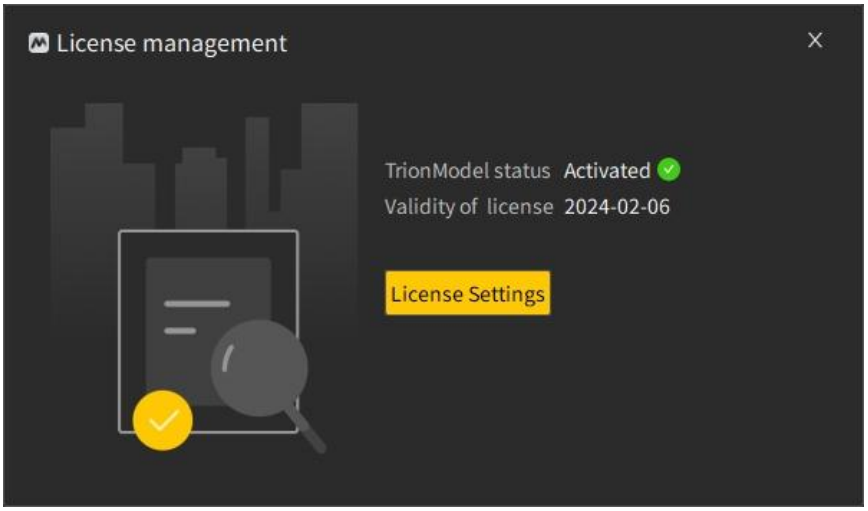


Figure 9. Successful activation by activation code

Note: FJD Trion Model detects the encryption lock in real time, so always insert it to the USB port during use.

When the license expires, contact FJDynamics technical support staff.

2.7 Language

FJD Trion Model supports simplified Chinese and English currently, and you can select the language as needed.

Choose **Help** > **Language**, and select the language you prefer.

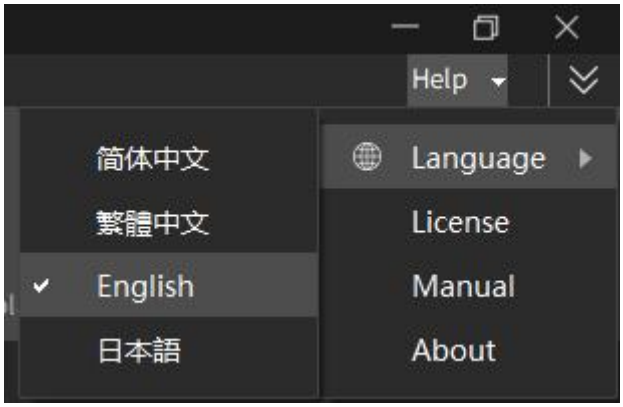


Figure 10. Selecting the language

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.

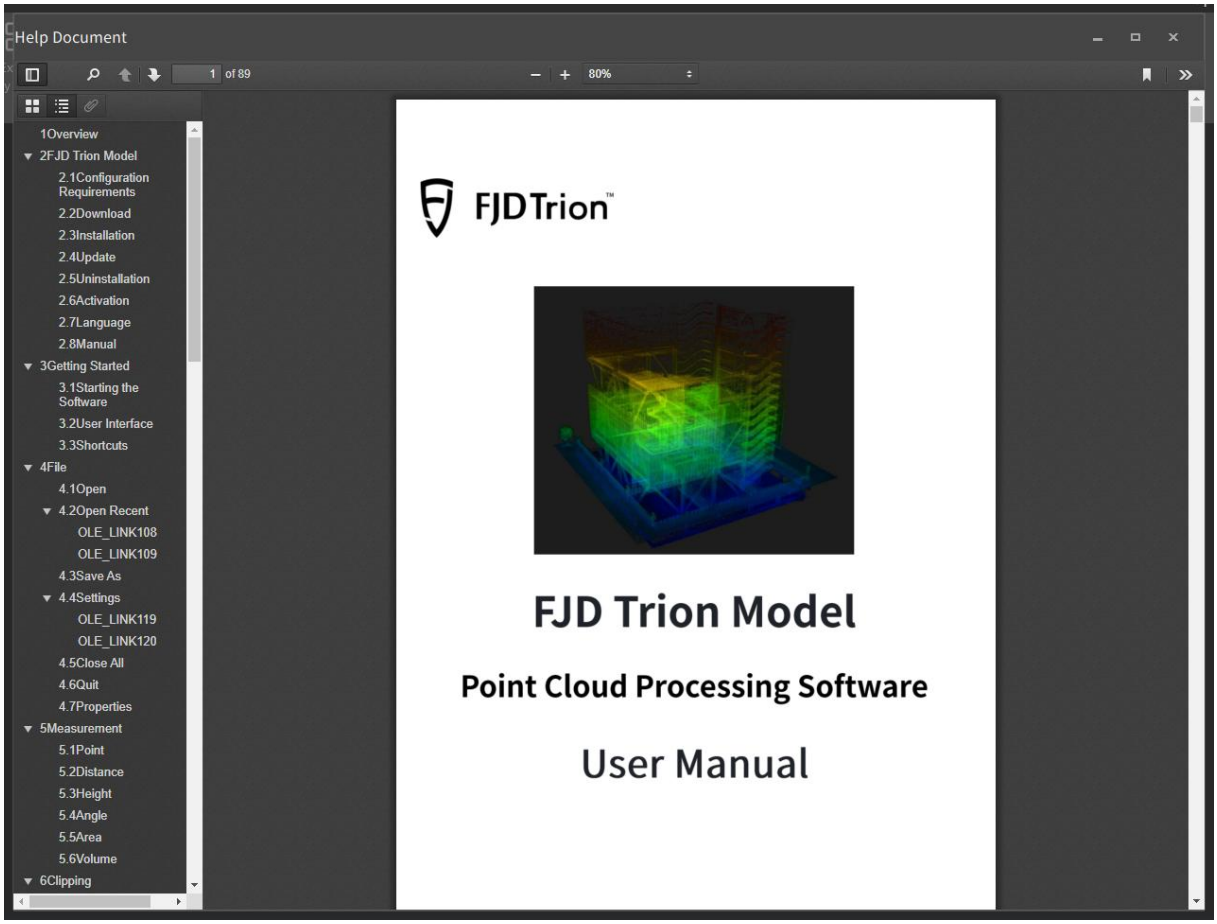


Figure 11. Manual

3 Getting Started

3.1 Starting the Software

Double-click to start FJD Trion Model.

3.2 User Interface

The main user interface is shown below.

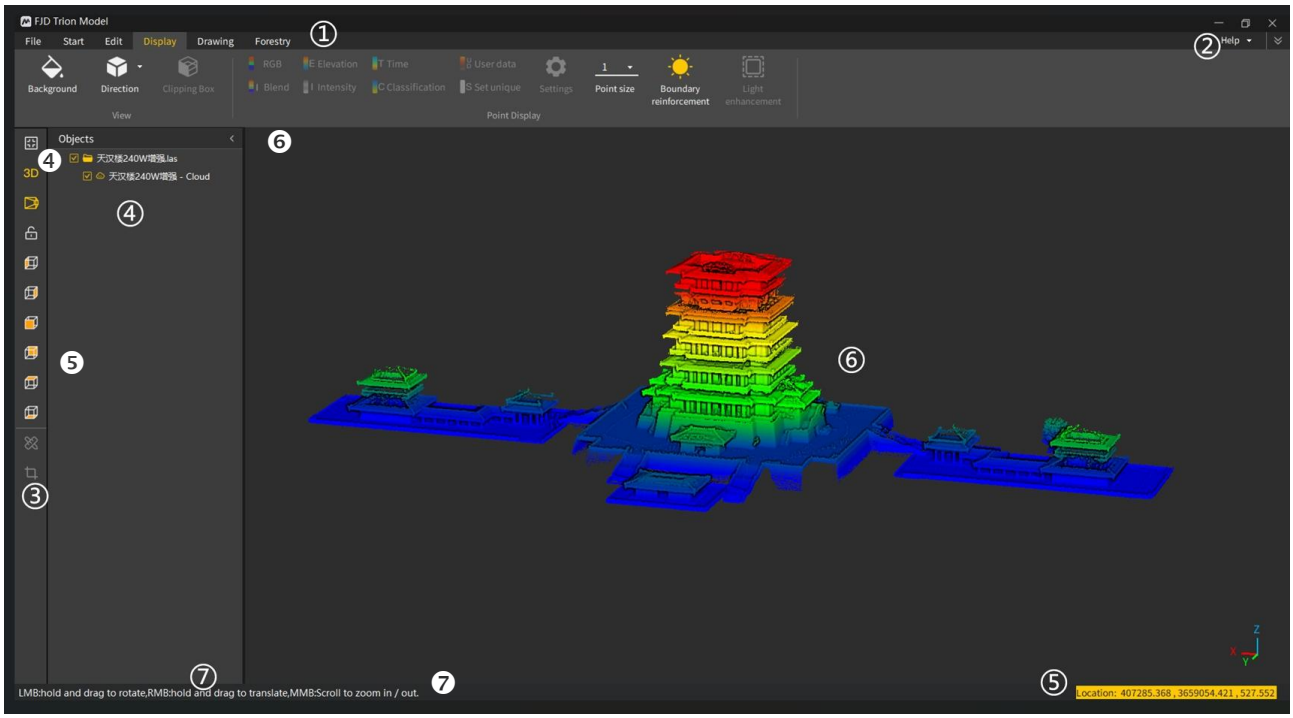


Figure 12. User interface

The following parts are included:

1	Menus	**2**	Help	**3**	Main toolbar
4	Project files	**5**	Properties	**6**	View area
7	Bottom bar				

3.3 Shortcuts

Shortcut	Description
Ctrl+O	Opens a file.
Ctrl+S	Saves the file.
Ctrl+click project files	Selects 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

Description:

Opens a project file and displays it in the 3D view area.

Operation procedure:

1. Click to browse the files. Shortcut: Ctrl+O.

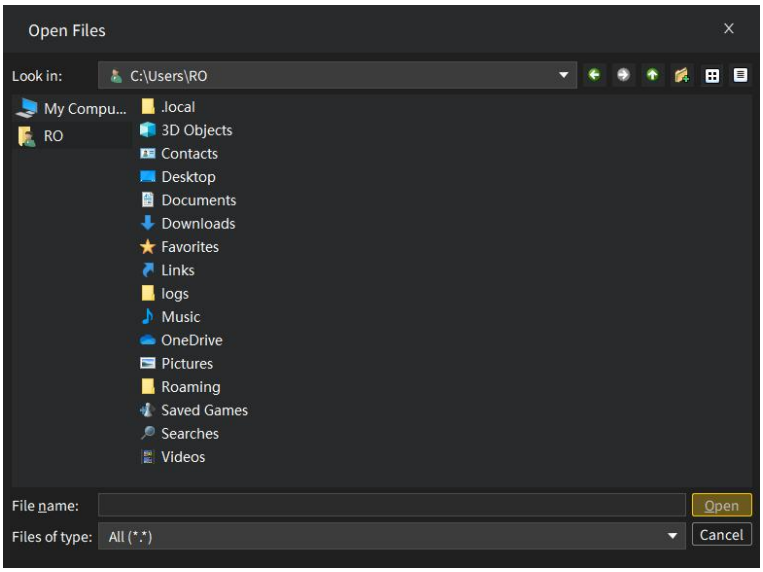


Figure 13. Opening a file

2. Select the target file, and click **Open**.

4.2 Open Recent

Description:

Displays directories of recent files, so that you can view the recent operation records.

Operation procedure:

- Choose **File** > **Open Recent**, and select the file to open.

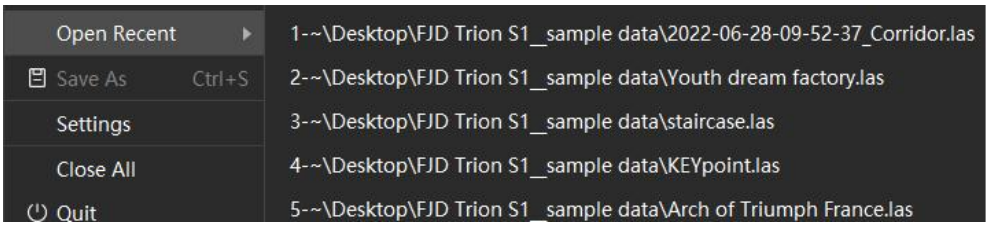


Figure 14. Viewing recent files

- Click **Clear Menu** to clear the historical records.

4.3 Save As

Description:

Saves the current entities.

Operation procedure:

1. Choose **File** > **Save As** to save the file. Shortcut: Ctrl+S.

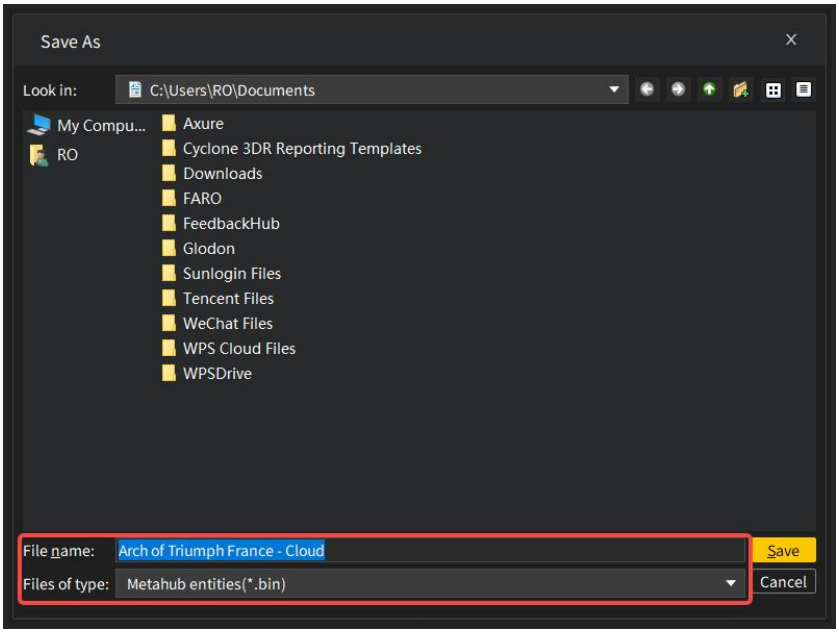


Figure 15. Saving a file

2. Set the storage path, file name, and file type.
3. Click **Save**.

4.4 Settings

Description:

Sets the units, labels, classes and other parameters. The unit settings include the units of length, area, radius, volume, and angle, the number of decimal digits, and whether the unit label is shown after the numerical value.

Operation procedure:

1. Choose **File** > **Settings**, and a popup window appears.

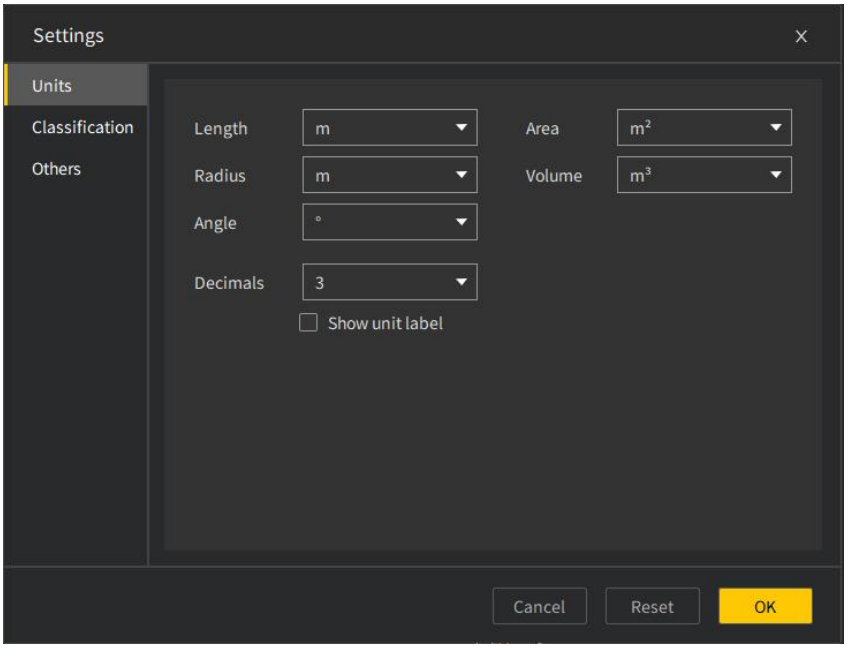


Figure 16. Settings

2. Choose **Units** to set the units of measurement and the number of decimal digits. The figure below shows the different displays when **Show unit label** is selected or not.

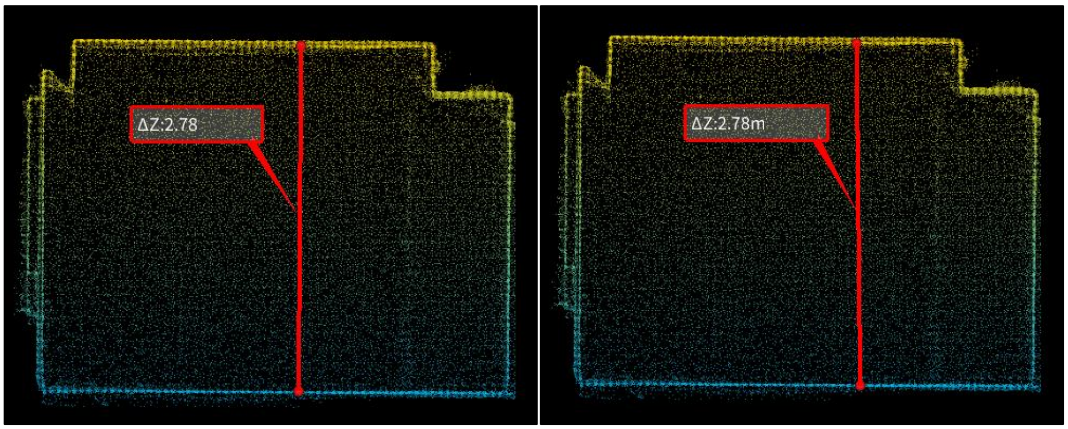


Figure 17. Display of unit

3. The **Classification** tab shows the class ID, class name, and color. Click **+ Add** to add a class, double-click any class name with ID above 15 to modify it, or click any color box to change the color of the corresponding class.

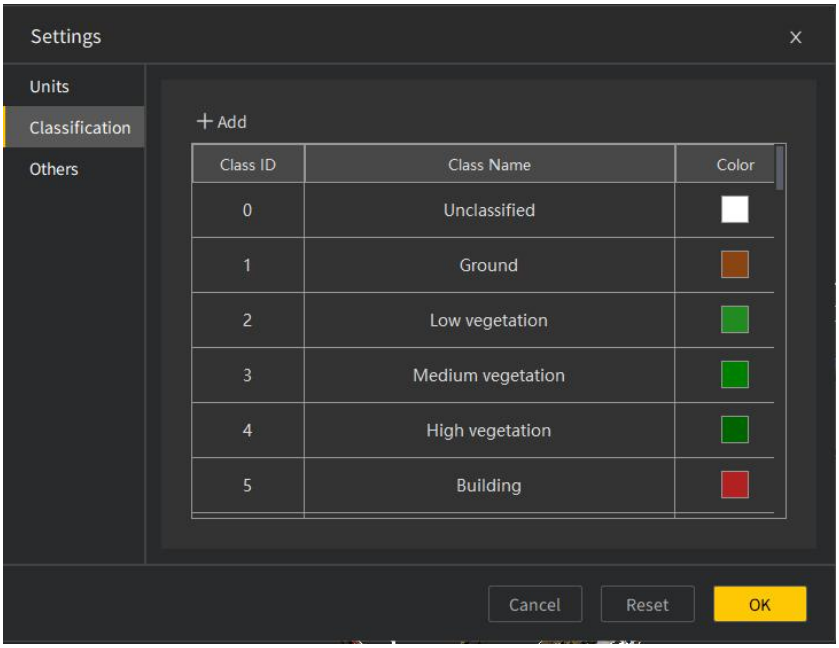


Figure 18. Classification settings

4. Choose **Others** to set the font size of prompts, point type, whether to show the middle screen cross, and display brightness and contrast.

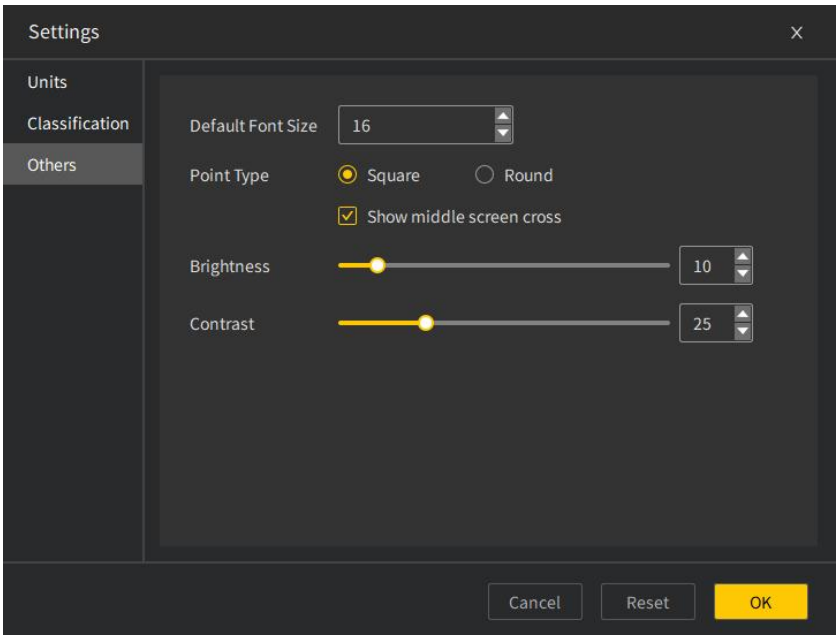


Figure 19. "Others" settings

The figure below shows different point types. The round point type is shown on the left, and the square point type is shown on the right.

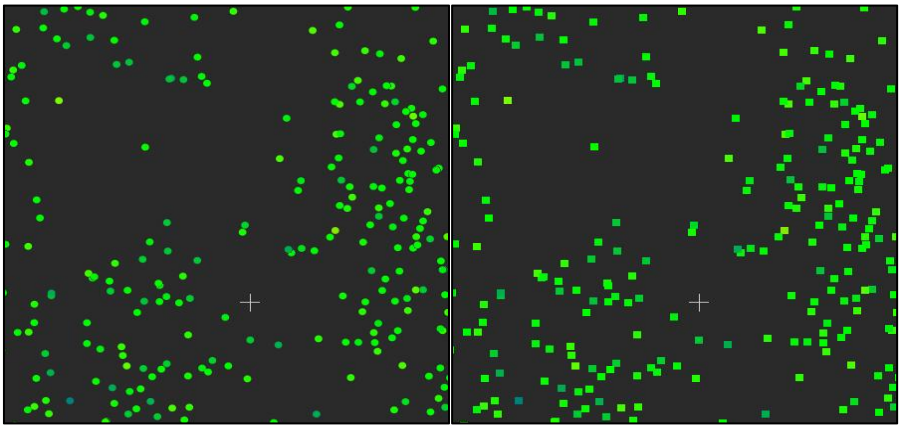


Figure 20. Point types

5. To complete the settings, click **OK**. To restore the default settings, click **Reset**.

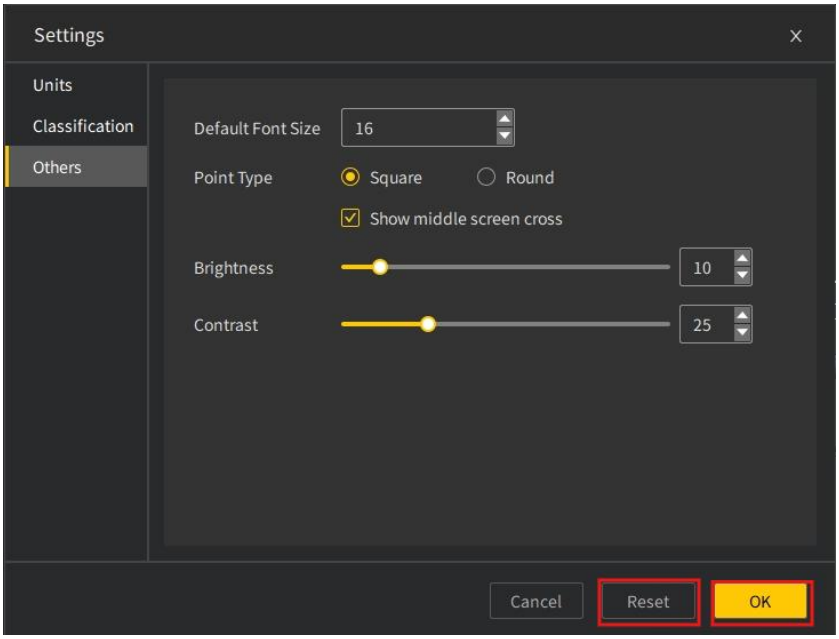


Figure 21. Restoring default settings

4.5 Close All

Description:

Removes all loaded entities, and restores the software to the default status.

Operation procedure:

1. Choose **File** > **Close All**, and a popup window appears.

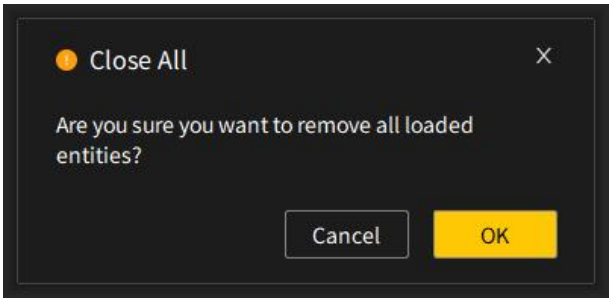


Figure 22. Confirmation window

2. Click **OK** to remove all loaded entities and restore the software to the default status, or click **Cancel** to cancel this operation.

4.6 Quit

Description:

Quits the application.

Operation procedure:

1. After all data is processed and saved, choose **File** > **Quit** or click ☒ in the upper right corner of the application window to quit.

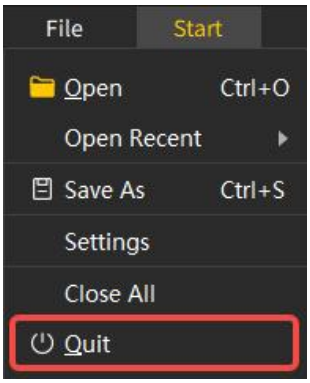


Figure 23. Quitting the application

2. A popup window appears to ask you to confirm this action, to prevent you from quitting the application accidentally or losing data. Click **OK** to quit. To quit directly afterwards, select **Never ask again**.

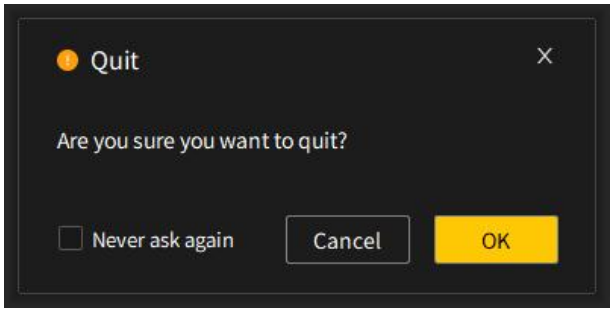


Figure 24. Confirmation window

5 Measurement

Description:

Obtains the required measurements in point clouds or meshes, including point properties, distance, height difference, angle, and area. When a point cloud or mesh is selected under **Objects**, click , and a measurement toolbar appears.

5.1 Point

Description:

Selects a point and displays its properties. This tool is activated by default and available for point clouds and meshes.

Operation procedure:

1. Select a point cloud or mesh under **Objects**.
2. Click to show the measurement toolbar.
3. Click , select a point, and a label as shown below appears, showing the coordinates of the point.

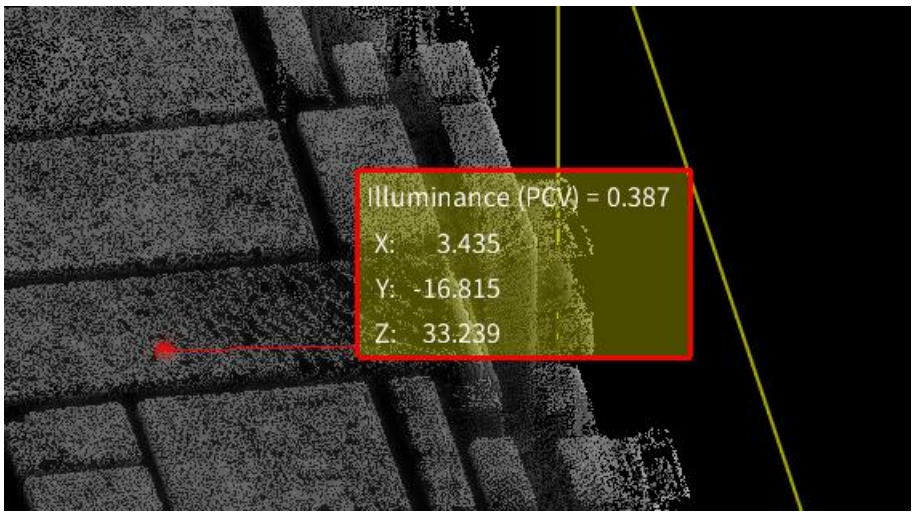


Figure 25. Display of point properties

5.2 Distance

Description:

Measures the distance between two points. Click to select the points. In the 2D view, the measurements represent the planar distance, while in the 3D view, the measurements represent the distance between the points in the space. This tool is available for point clouds and meshes.

Operation procedure:

1. Select a point cloud or mesh under **Objects**.
2. Click  to show the measurement toolbar.
3. Click  , select two points, and a label as shown below appears, showing the distance between the points selected. The following figures show the distance measured between two points selected and the distance measured among multiple points selected.

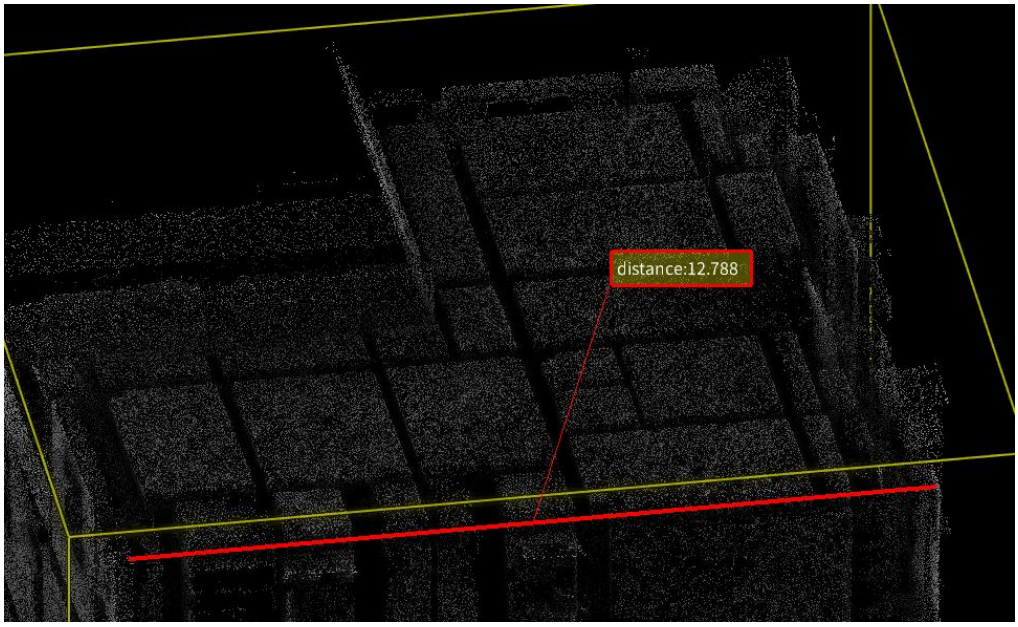


Figure 26. Measuring the distance between two points

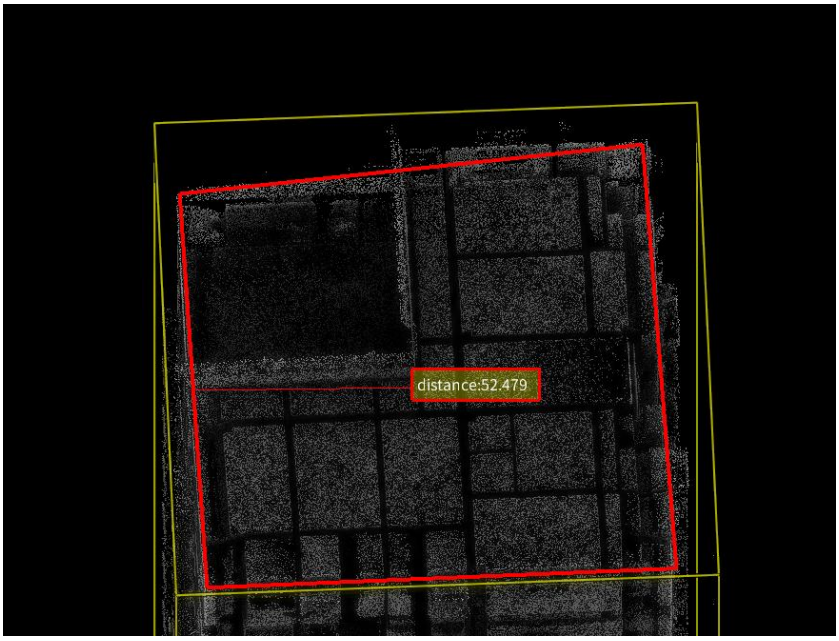


Figure 27. Measuring the distance among multiple points

5.3 Height

Description:

Measures the height difference between two points. Click to select the points. This tool is available for point clouds and meshes.

Operation procedure:

1. Select a point cloud or mesh under **Objects**.
2. Click to show the measurement toolbar.
3. Click , and select two points.
4. A straight line connecting the two points appears, with a label showing the height difference.

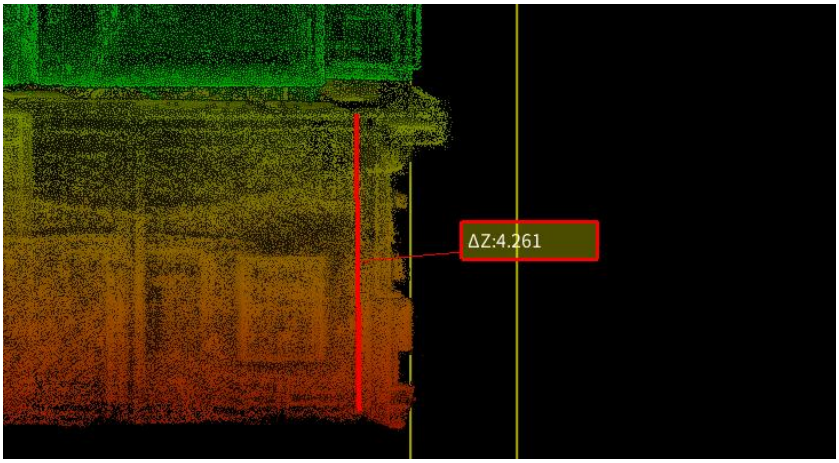


Figure 28. Measuring the height difference

5.4 Angle

Description:

Measures the angles between the selected three points. Click to select the points. This tool is available for point clouds and meshes.

Operation procedure:

1. Select a point cloud or mesh under **Objects**.
2. Click to show the measurement toolbar.
3. Click , and select the first point.
4. Select the second point.
5. Double-click to select the third point and end the selection, and the measured angles appear in the label, as shown in the figure below.

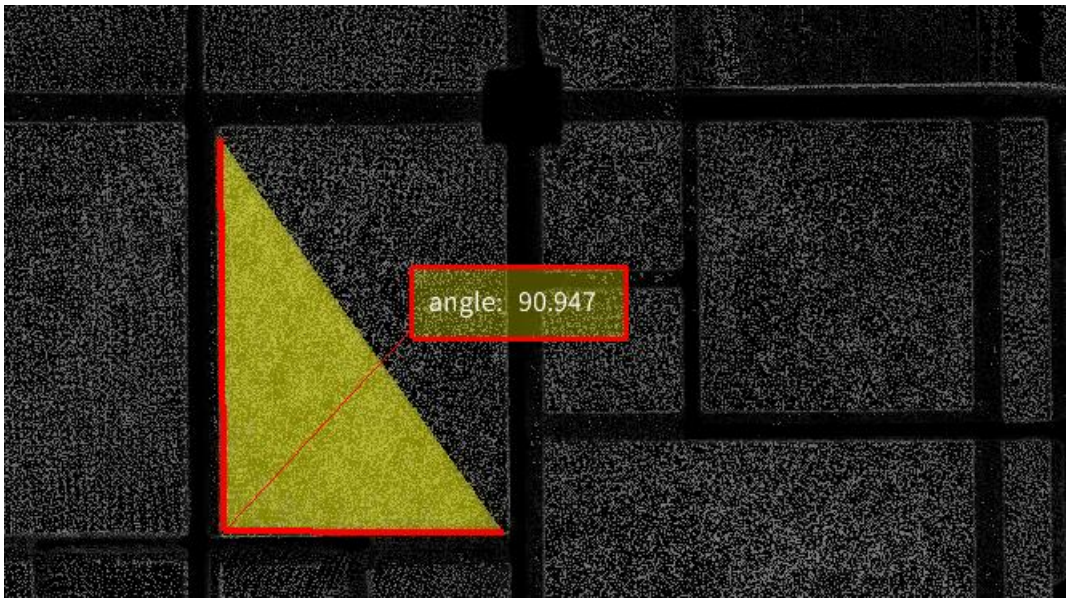


Figure 29. Measuring angles

5.5 Area

Description:

Measures the area covered by a triangle. Click to select points and draw a triangle. This tool is available for point clouds and meshes.

Operation procedure:

1. Select a point cloud or mesh under **Objects**.
2. Click to show the measurement toolbar.

3. Click , and select three points. The figure below shows the area measurement in a point cloud.

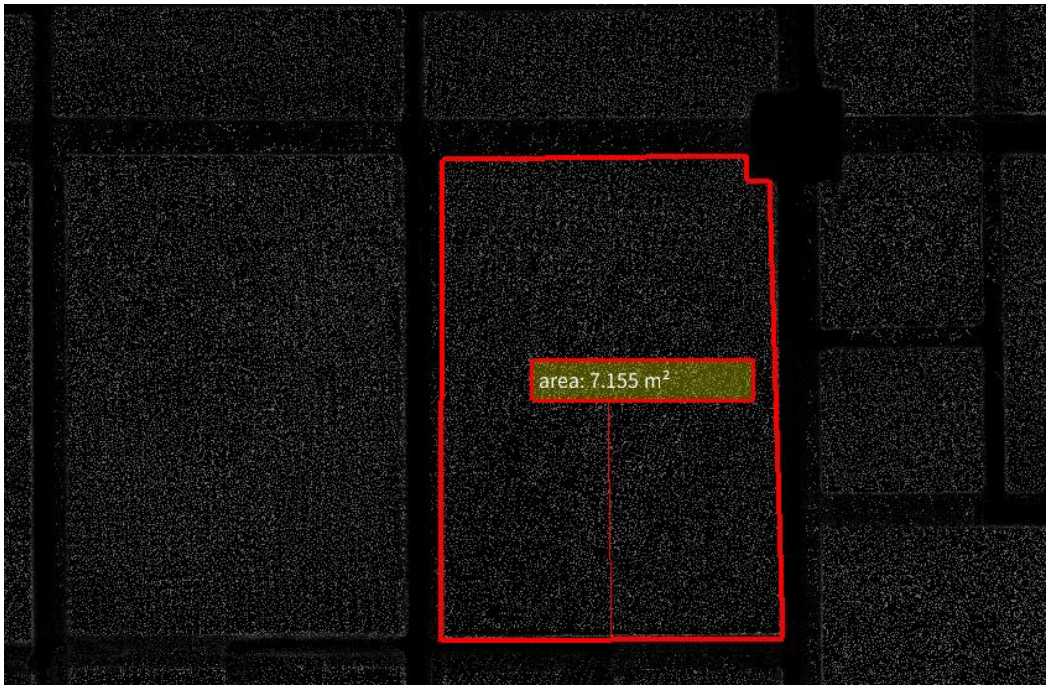


Figure 30. Measuring the area

6 Clipping

6.1 Cropping

Description:

Selects and keeps one or more parts of a point cloud and deletes the remaining part.

Rectangular selection and polygonal selection are supported.

Operation procedure:

1. Select a point cloud under **Objects**.
2. Click , and a toolbar appears. Click to select a rectangle area, or click to select a polygon area.

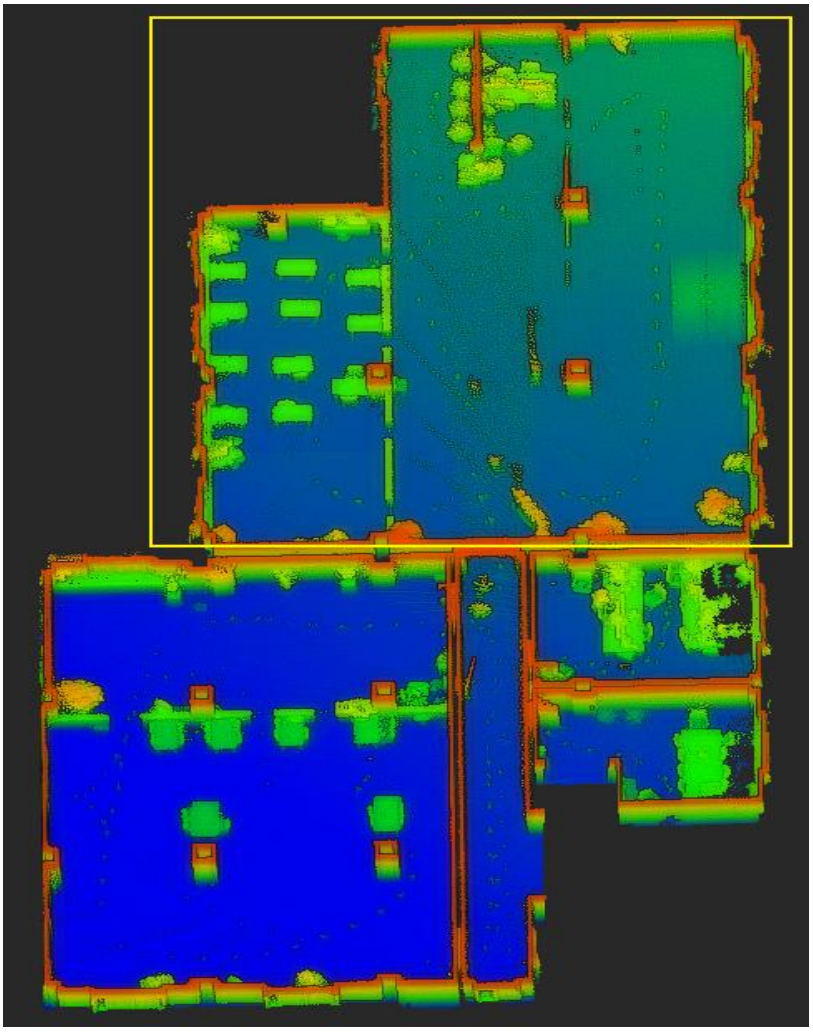


Figure 31. Point cloud before cropping

3. Click to keep the points falling inside the selected area, or click to keep the points falling outside the selected area.

4. Click to apply the result, or click to exit the cropping operation and restore the initial state. To undo the action, click to restore the initial state. The following figure shows the result of cropping out the selected area.

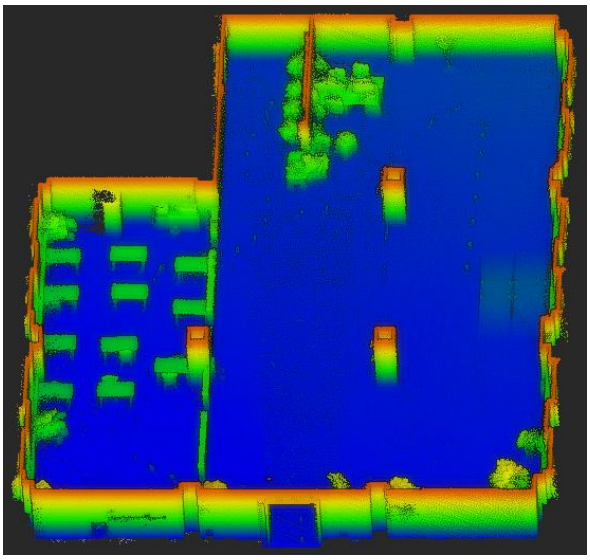


Figure 32. Cropping result

6.2 Segmentation

Description:

Segments a point cloud into multiple clouds. Rectangular selection and polygonal selection are supported.

Operation procedure:

1. Select a point cloud under **Objects**.
2. Click , and a toolbar appears. Click to select a rectangle area, or click to select a polygon area.

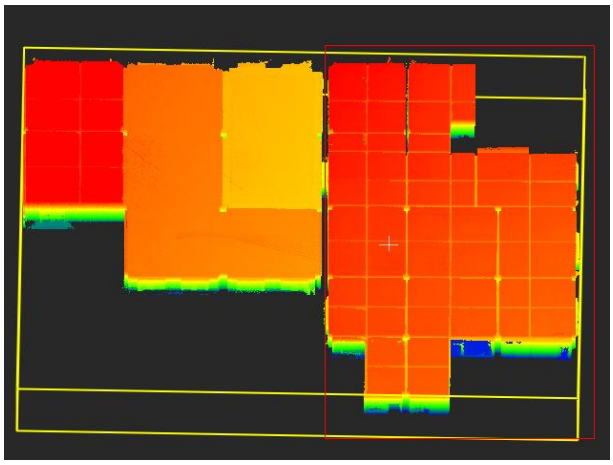


Figure 33. Selecting the target area

3. Click to show the points falling inside the selected area.

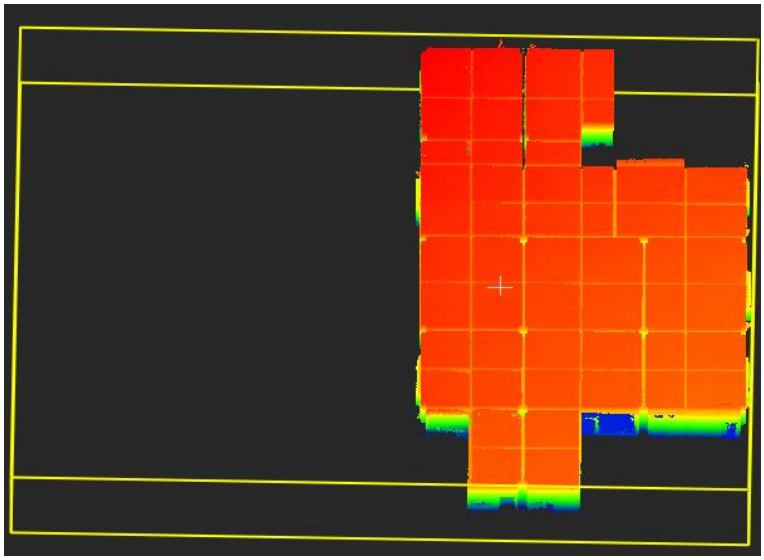


Figure 34. Showing the points inside the selected area

4. Click to show the points falling outside the selected area.

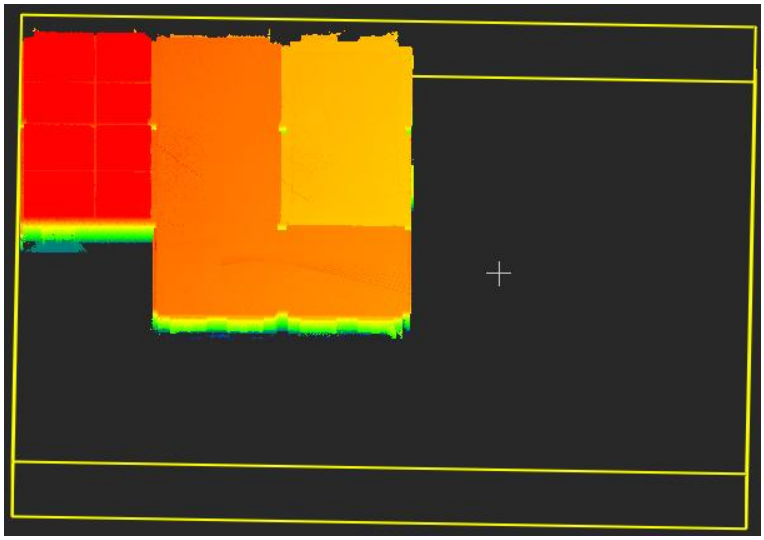


Figure 35. Showing the points outside the selected area

5. Click to apply the result, or click to exit the segmentation operation and restore the initial state. To undo the action, click to restore the initial state.

Note:

When selecting a polygonal area, press Ctrl+Z to cancel the last point you select. This action is repeatable and also supported by the polygonal selection tool in forestry editing and manual classification.

7 Start

7.1 Data Resolving

7.1.1 Point Cloud Mapping

Description:

Maps the raw data downloaded from FJD Trion S1 and FJD Trion P1.

Operation procedure:

1. Click  on the **Start** tab, and the following window appears. Based on the data type, select the device model, mapping mode, and scanning scene, and click **Next**. Or, click **Cancel** to cancel mapping.

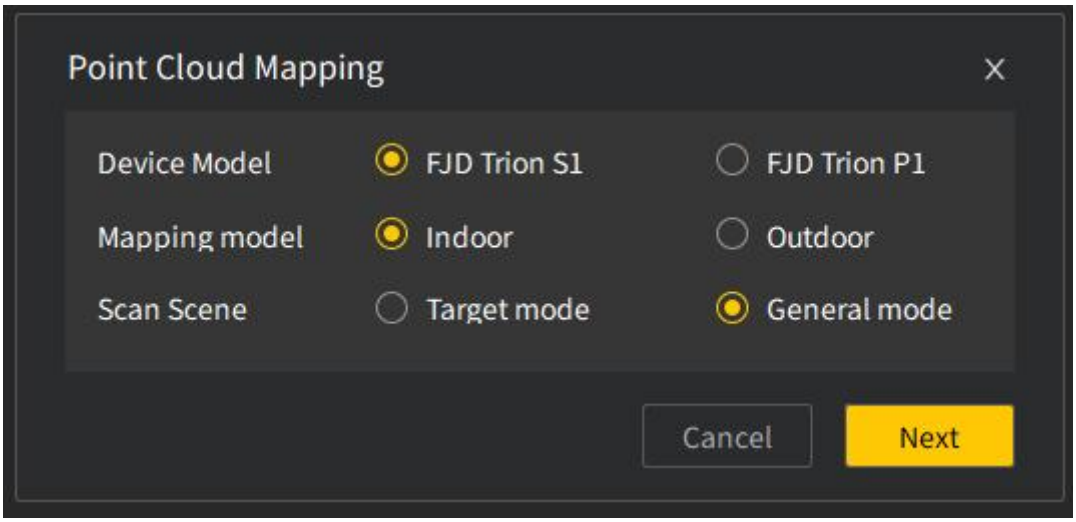


Figure 36. Point cloud mapping

2. Enter the project name, select the raw data file path and the project file path, and set the reconstruction range and RTK fusion. To cancel mapping, click **Cancel**. To go back to the previous screen, click **Back**.

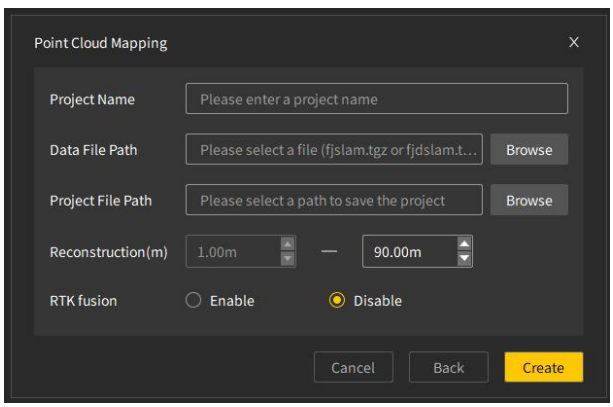


Figure 37. Parameter settings

3. Click **Create** to start data parsing. The **Show pointcloud** box is selected by default. Deselect the box to not display the point cloud.

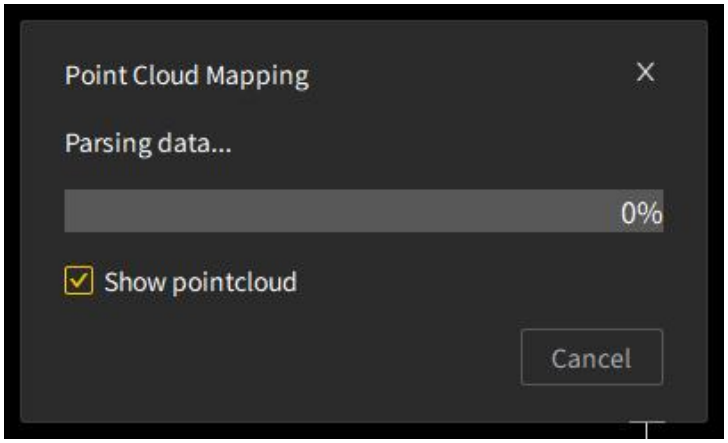


Figure 38. Data parsing progress

4. The mapping progress window appears after the data parsing is completed, and disappears when the progress is 100%. The file list under **Objects** is updated, and the mapping result is shown in the 3D view area.

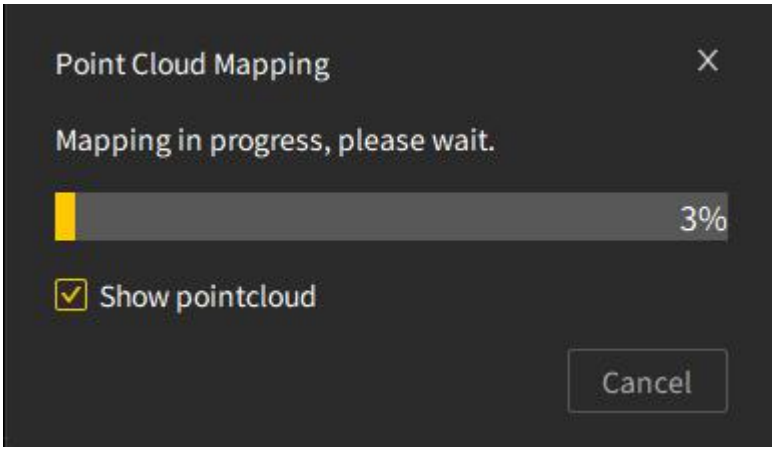


Figure 39. Mapping progress

Parameters:

- Reconstruction: The default range is 1 m to 90 m.
- RTK fusion: Enable the function when mapping outdoors with few features but a good GNSS signal.

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.

7.1.2 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.

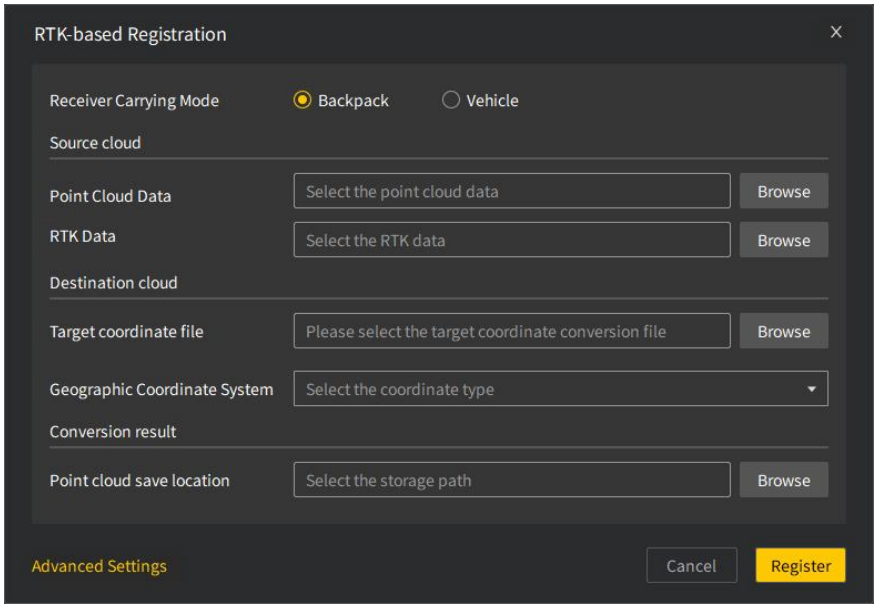


Figure 40. RTK-based registration

2. Select the receiver carrying mode, point cloud data, RTK data, geographic coordinate system, and the point cloud storage location. Two point cloud data file formats are supported: .las and .fjslam.tgz.

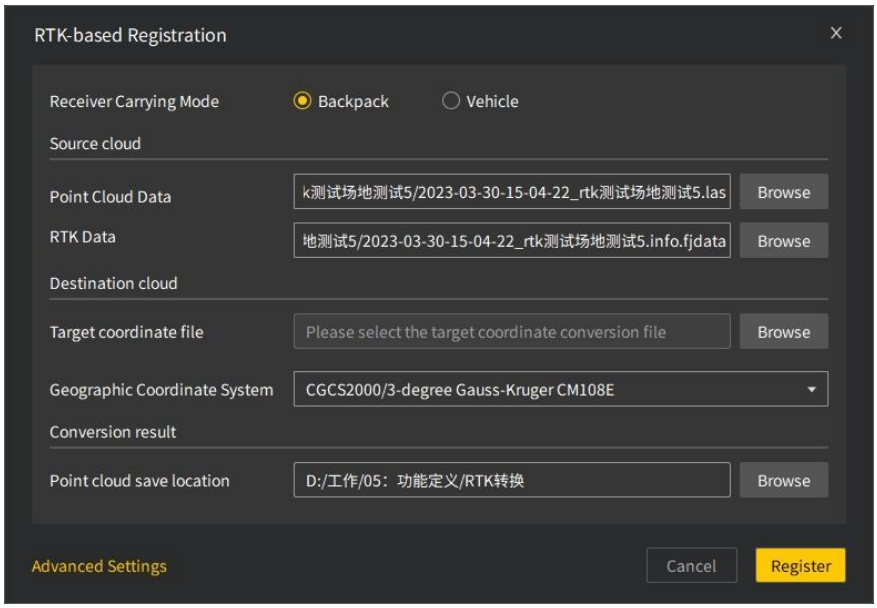


Figure 41. Registration settings

3. Select **Custom coordinate system** and set the ellipsoid (XIAN80, WGS84, CGCS2000, BJ54, GRS80, BESSEL, Krass, or Intl1910) and projection mode (Gaussian or UTM projection), when the geographic coordinate system list does not contain the desired coordinate system.

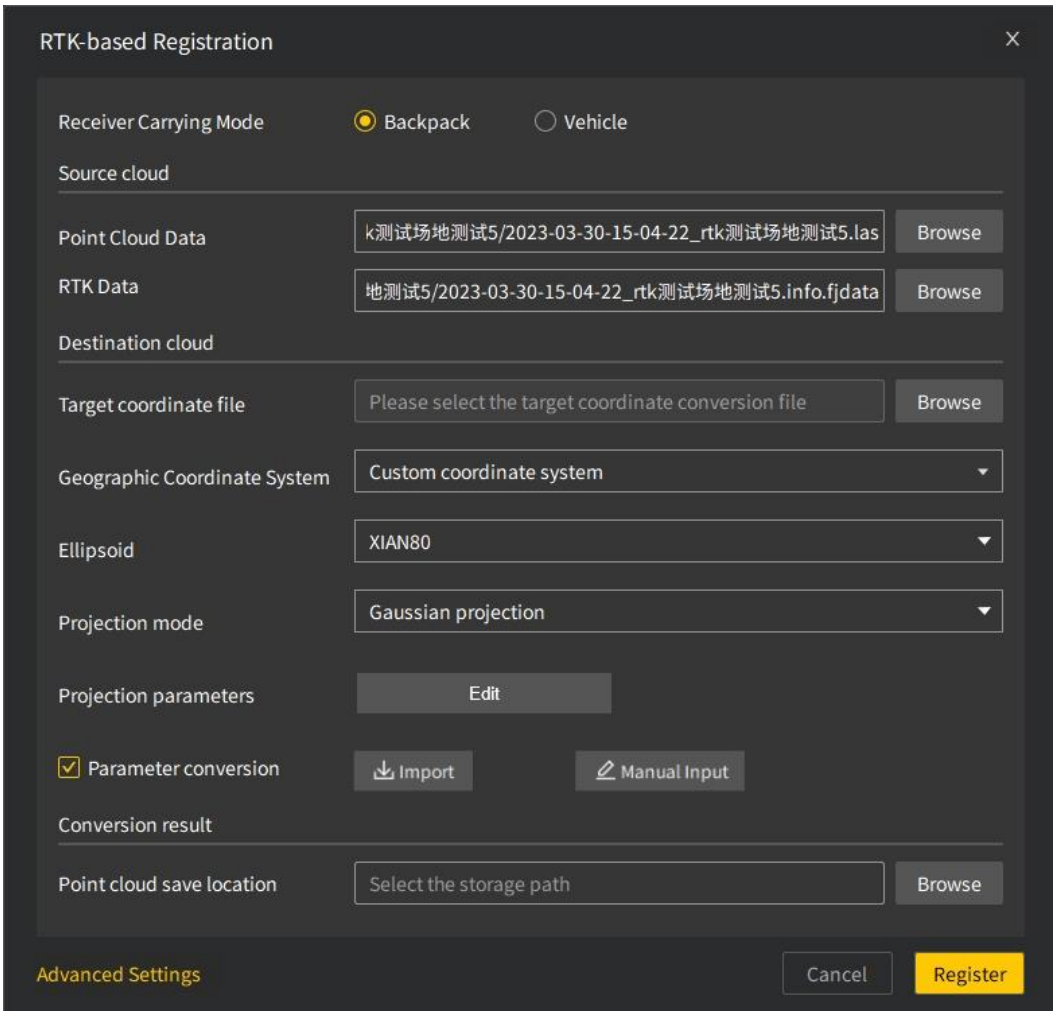


Figure 42. Custom coordinate system

4. Set projection parameters and seven parameters. Projection parameters include the central meridian, origin latitude, false easting, false northing, and scale factor. Seven parameters are the translation parameters (Dx, Dy, and Dz), rotation parameters (Rx, Ry, and Rz), and scale (m) which can be imported in an .xlsx or .txt file or entered manually.

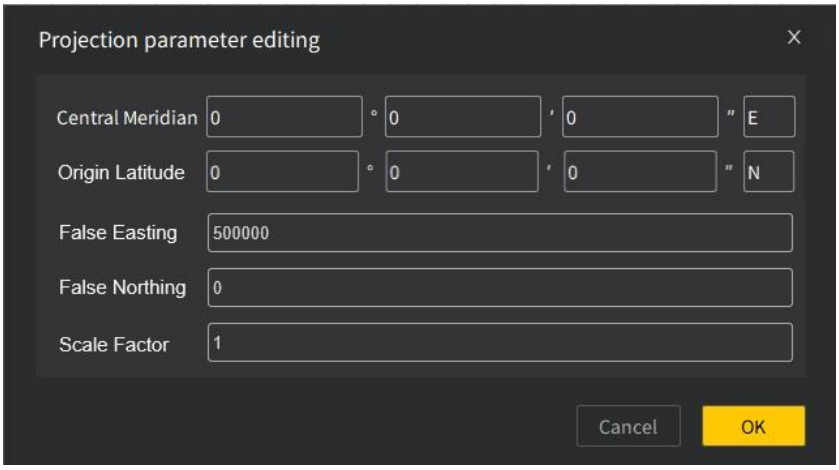


Figure 43. Setting projection parameters

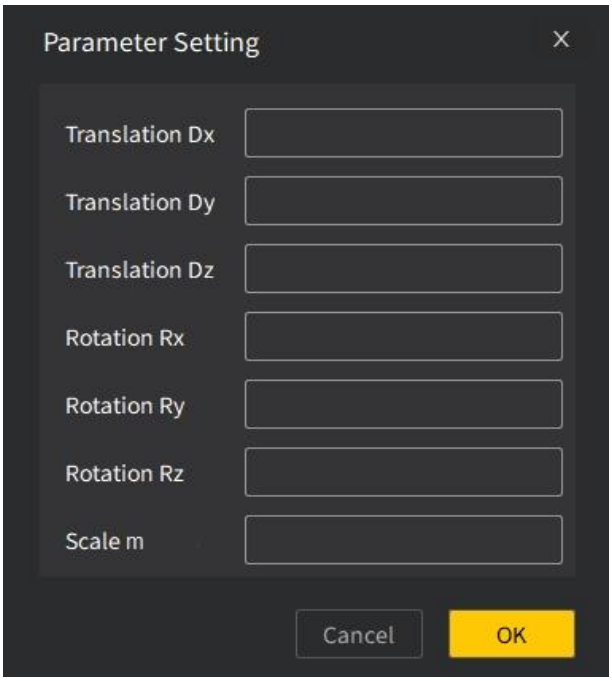


Figure 44. Setting seven parameters

5. Click **Register** to start the registration. The settings window disappears and the following window appears.

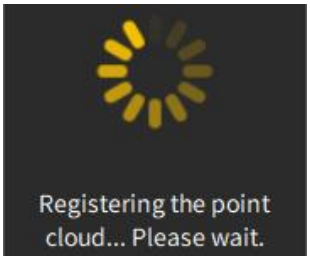


Figure 45. Registering the point cloud

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.

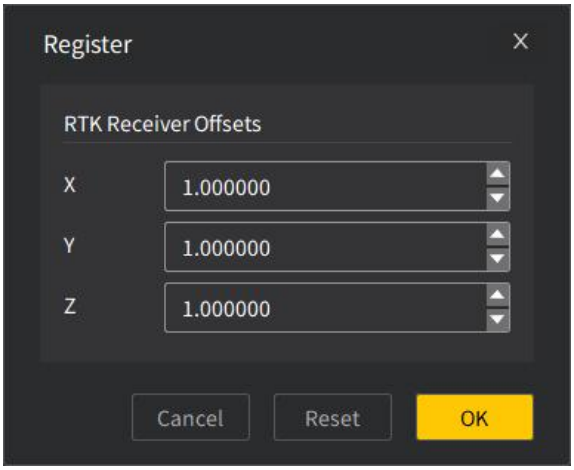


Figure 46. Advanced settings

7. All required fields on the **RTK-based Registration** window must be filled in. Otherwise, the empty input box becomes red after you click **Register**.

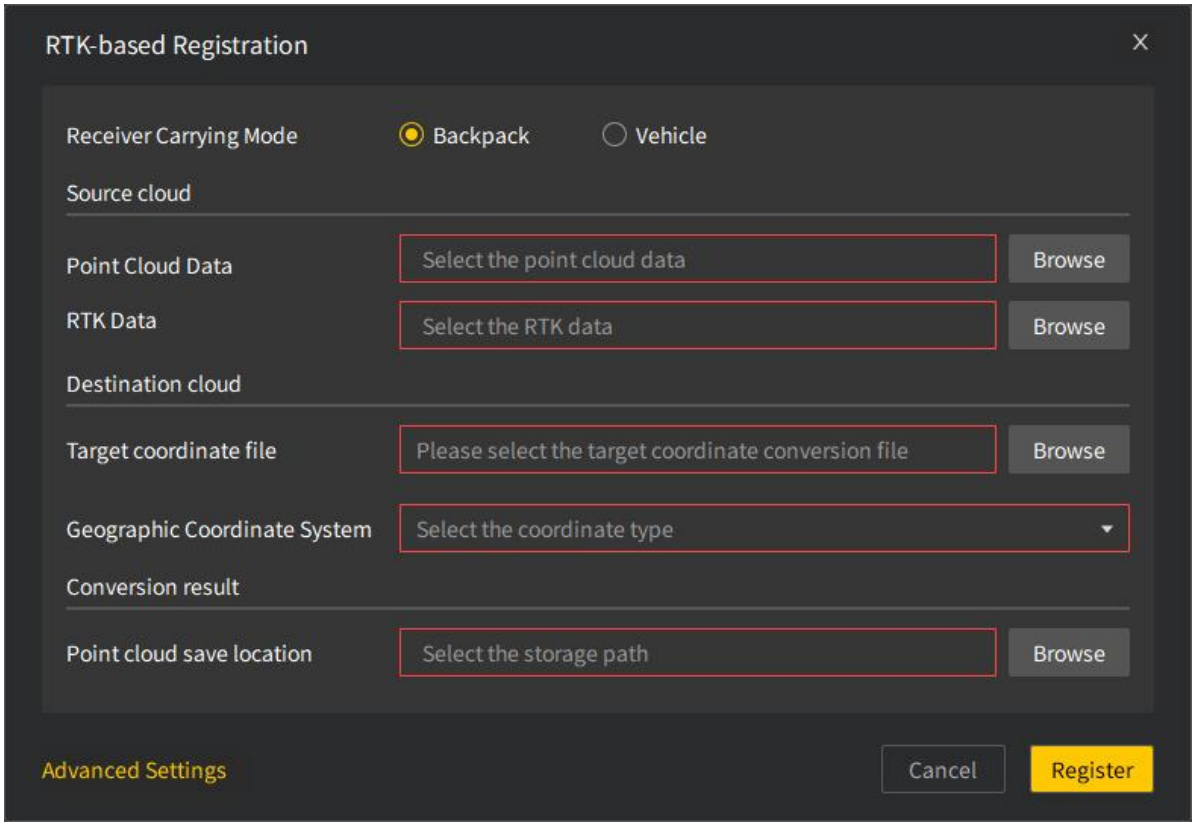


Figure 47. Empty required fields

Note:

- When a point cloud data file is selected, the system automatically searches for any RTK data file in the same folder, and fills in the data path into the RTK data input box if an RTK data file is found.

- When you select a compressed file of point cloud data downloaded from a laser scanner, the RTK data input box becomes disabled, and the system automatically decompresses the file and obtains the RTK data.
- You can use fuzzy search to select a geographic coordinate system, making the registration much easier.
- If the geographic coordinate system list does not contain the target coordinate system that you need, you can use the RTK software provided by FJDynamics to create a target coordinate file (*.fjdrtk) and import it for coordinate system transformation.

7.1.3 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 device model, point cloud name, configuration file, and image file, and click **Next**.

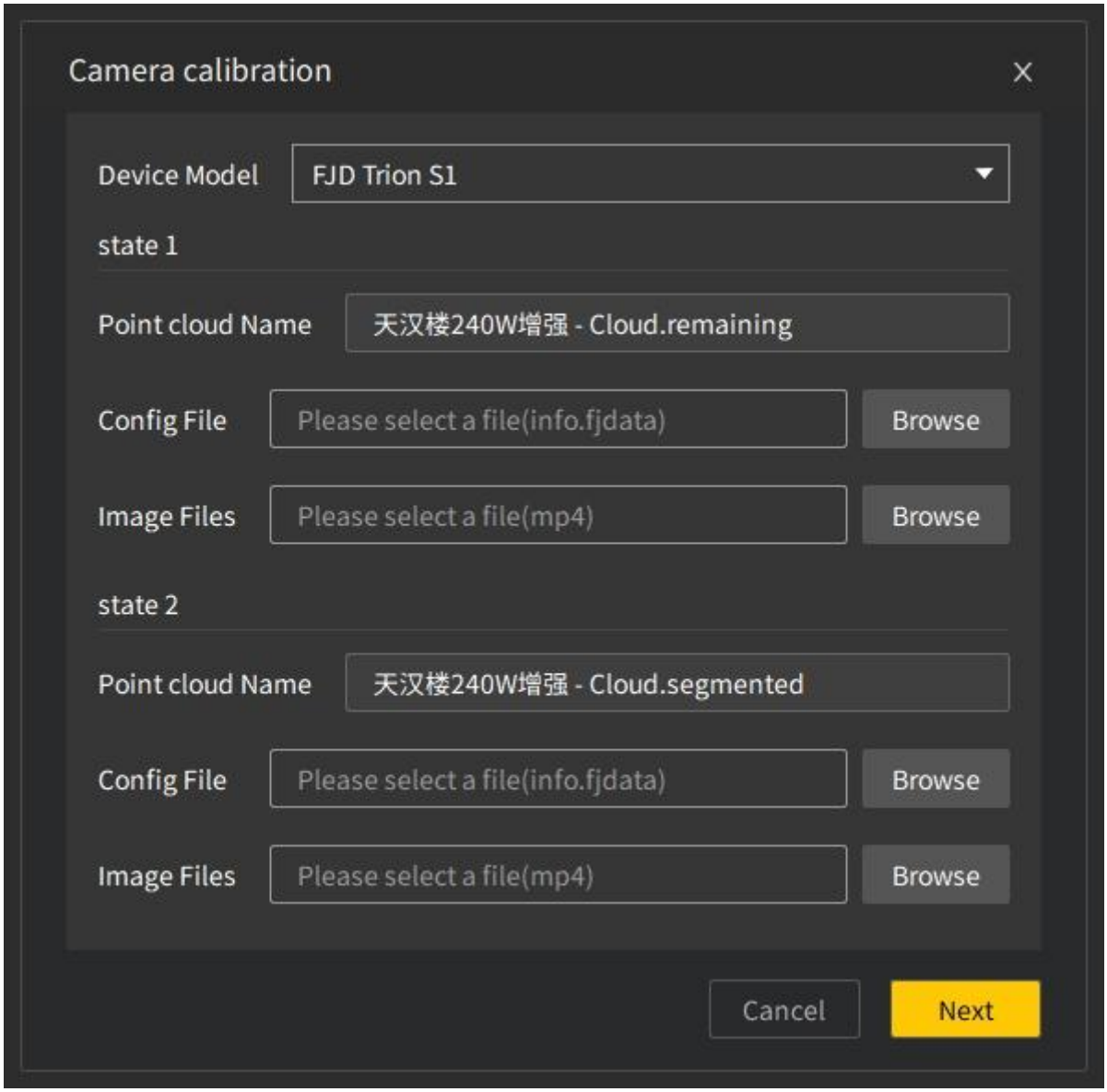


Figure 48. Camera calibration window

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**.

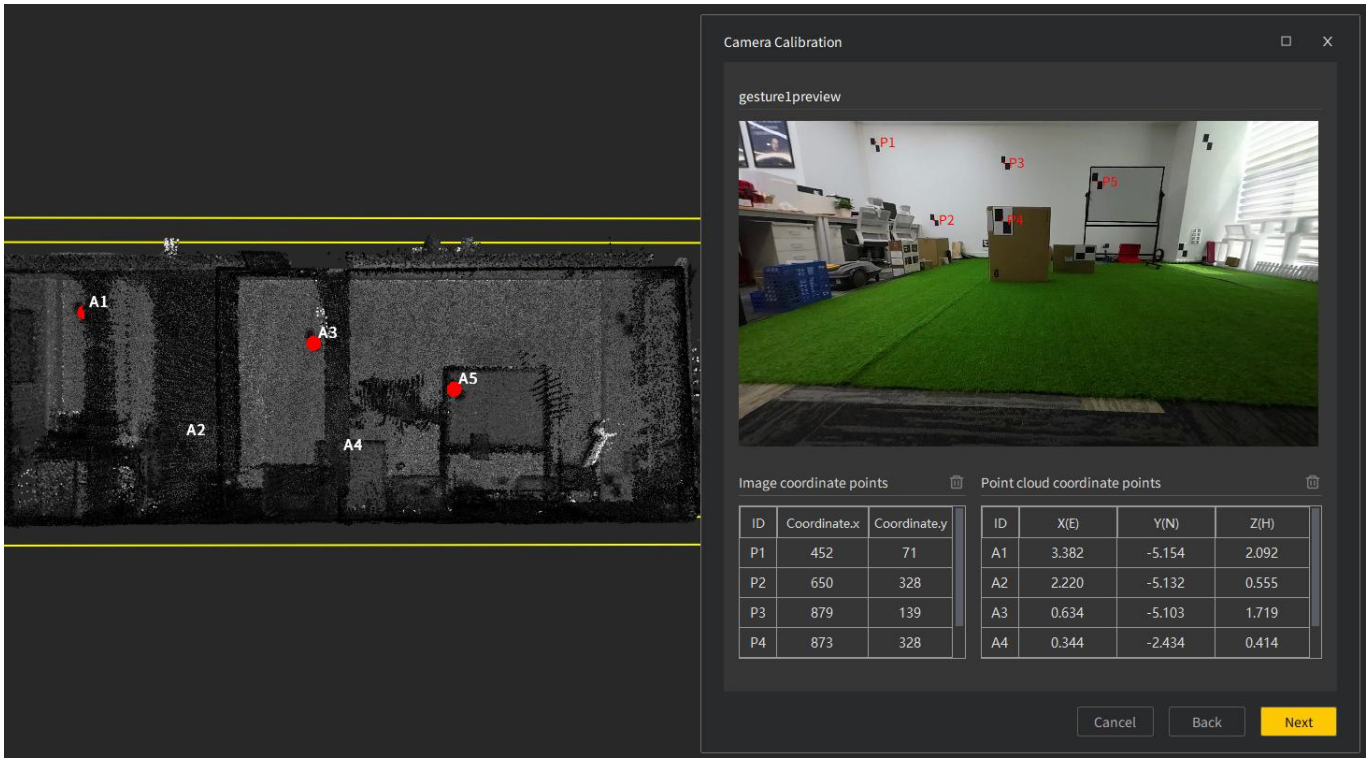


Figure 49. Picking target points

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

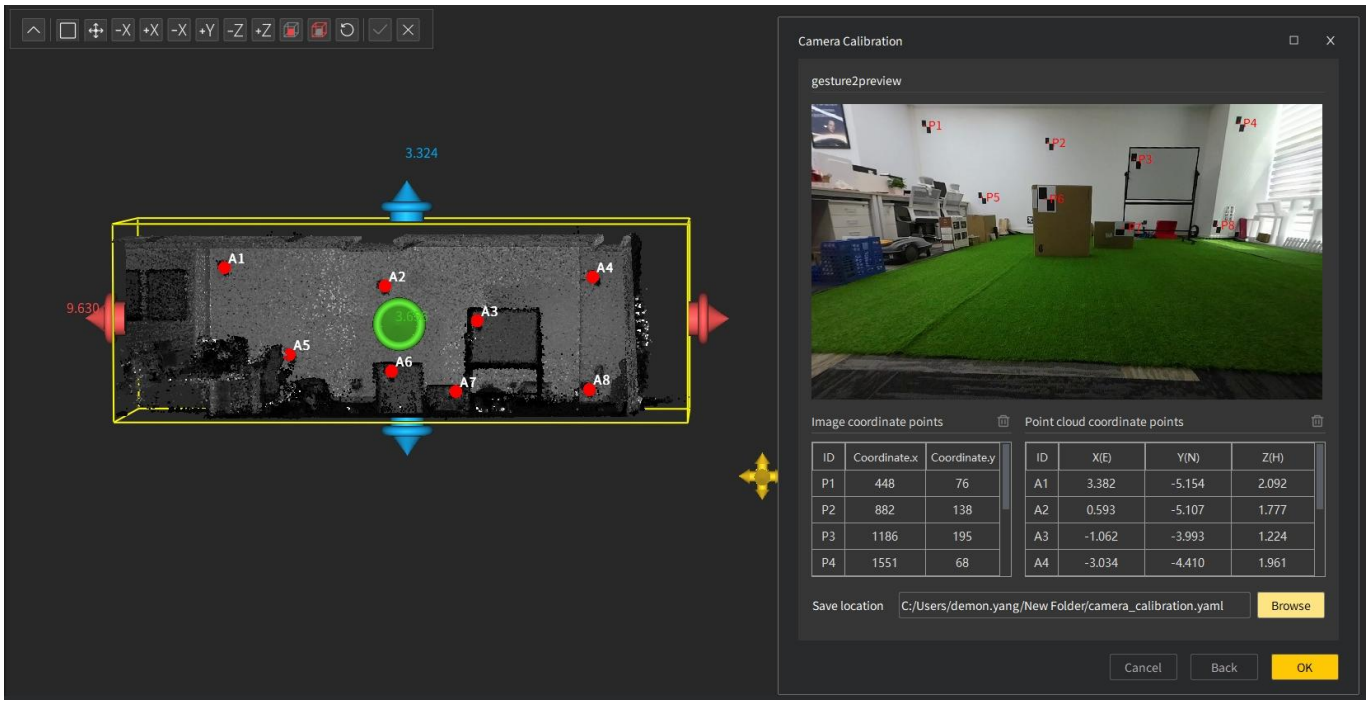


Figure 50. Selecting the storage path

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.

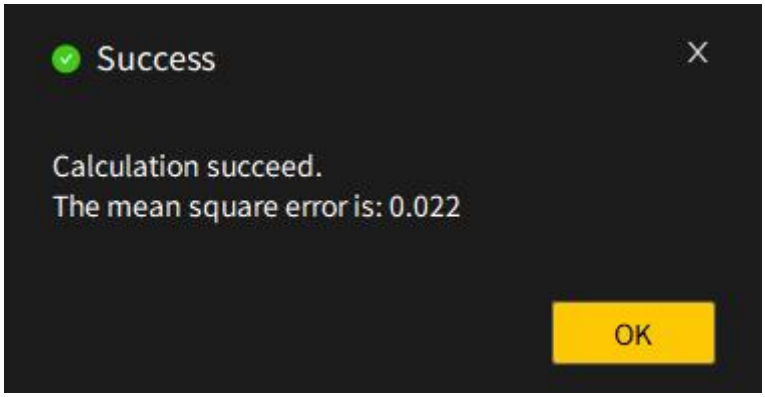


Figure 51. Successful calculation message

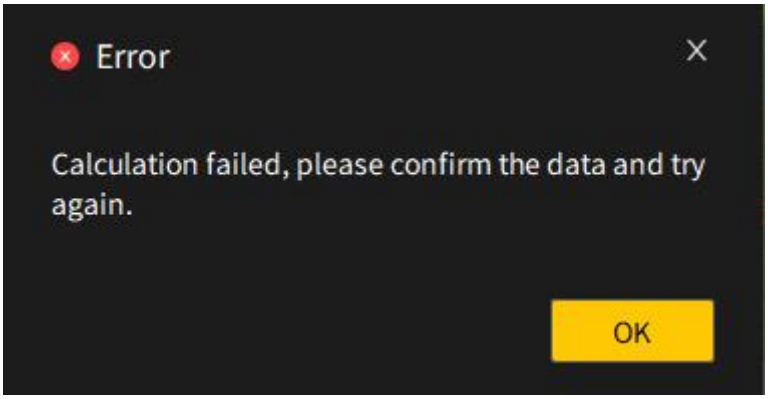


Figure 52. Unsuccessful calculation message

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.

7.1.4 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.

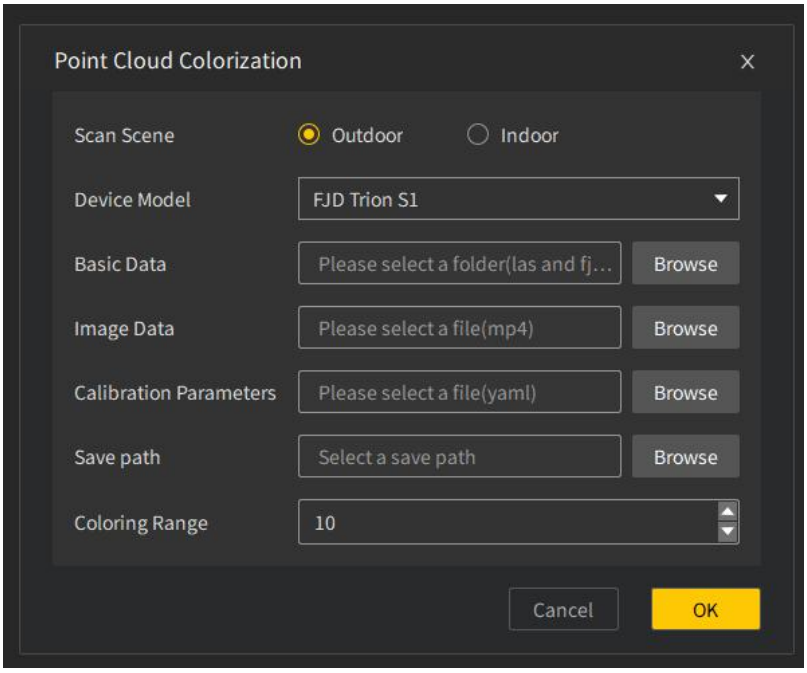


Figure 53. Setting colorization parameters

2. Select the scanning scenario, device model, corresponding files for the basic data (formats: las and fjdata), image data (format: mp4), and calibration parameters (format: yaml), saving path, and coloring range. If any required field is empty, the input box becomes red after you click **OK**.

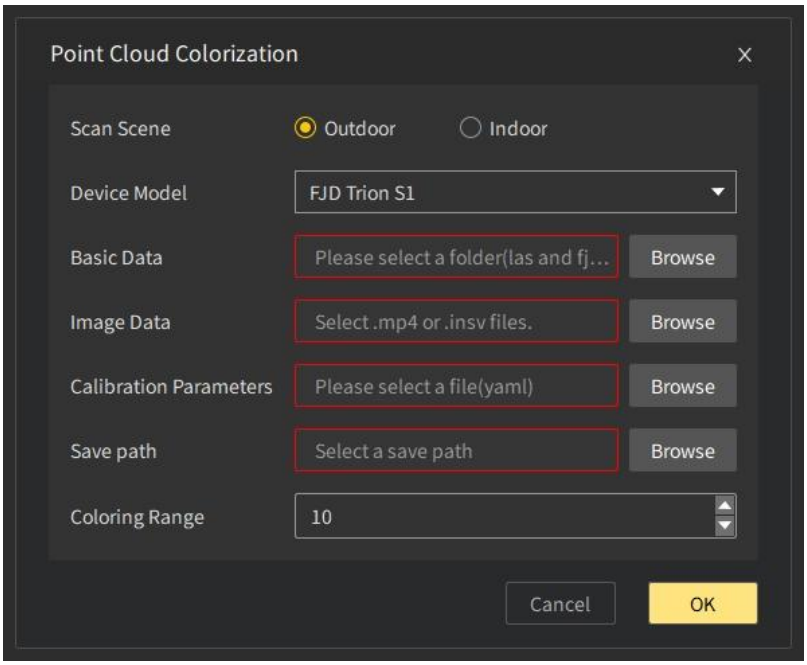


Figure 54. Empty required field

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

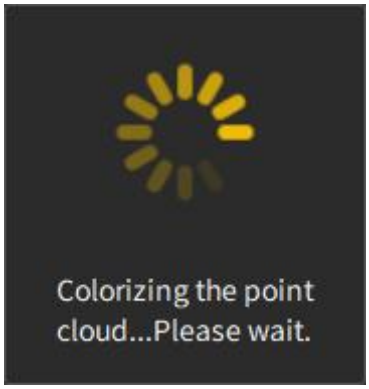


Figure 55. Colorizing the point cloud

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

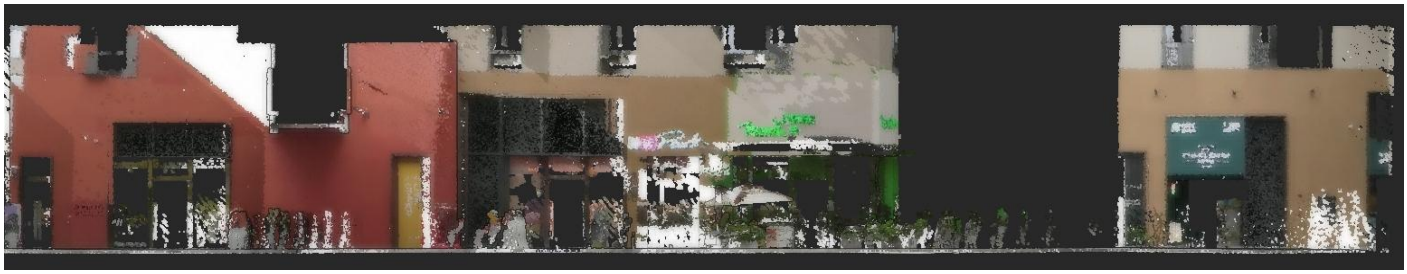


Figure 56. A colorized neighborhood point cloud

Figure 57. A colorized commercial street point cloud

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

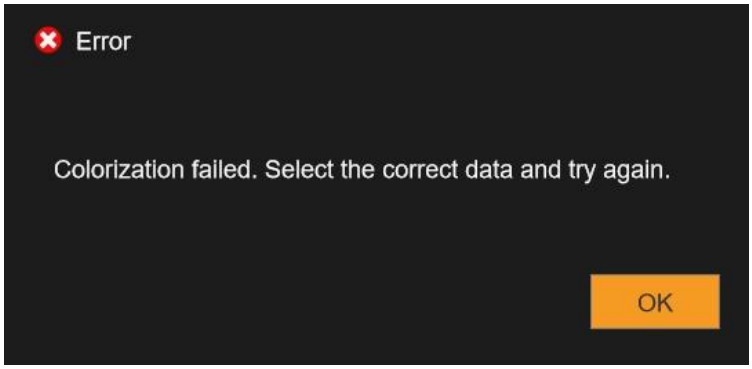


Figure 58. Unsuccessful point cloud colorization

7.2 Point Processing

7.2.1 Moving Object Removal

Description:

Removes abnormal points caused by moving people or vehicles during scanning to ensure better point cloud display. This solves problems of abnormal point cloud data and too much noise when scanning crowded areas.

Operation procedure:

1. Click on the **Start** tab, and the following window appears. Select the point cloud data and configuration files, and set the removal grade.

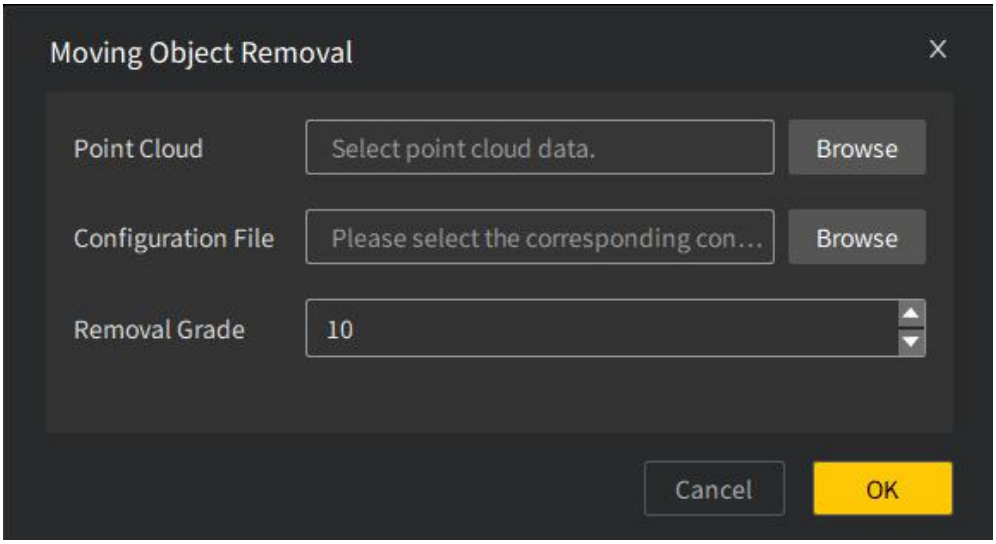


Figure 59. Moving object removal window

2. Click **OK**, and the removal progress window appears. The higher grade you set, the longer time the removal takes and the better result you will get.

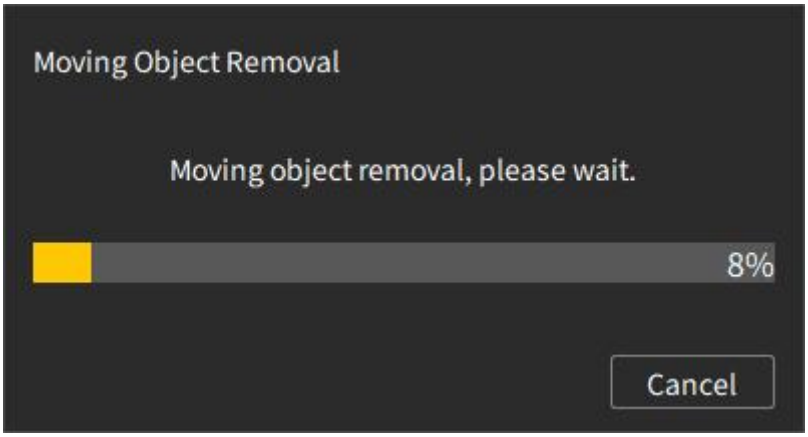


Figure 60. Moving object removal progress

3. When the process is completed, the new point cloud is displayed in the 3D view area. The following figures show a point cloud before and after moving object removal.

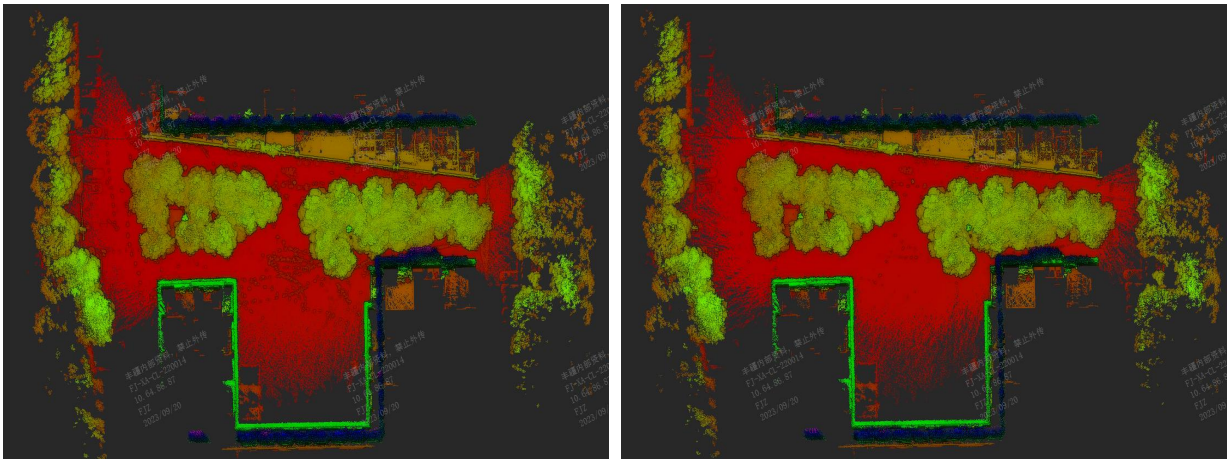


Figure 61. Before and after moving object removal

7.2.2 Rectification

Description:

Adjusts the normals of points on different planes according to the normal relationship of coordinate axes, to reduce the impact of irregular points on the representation of point clouds and improve the point cloud registration precision in certain scanning scenarios.

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 62. 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 under **Objects**, 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.

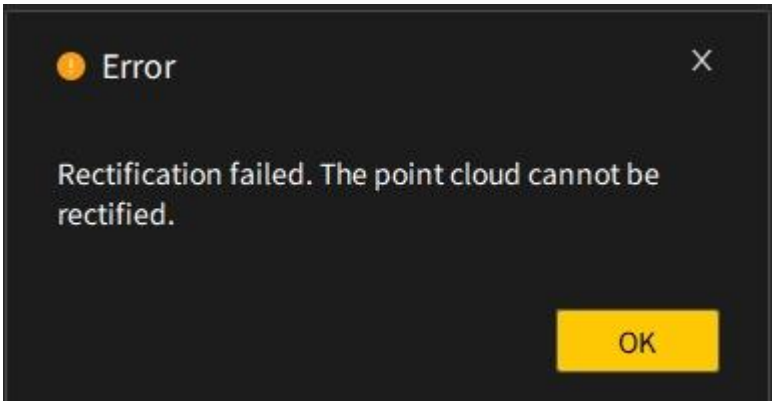


Figure 63. Rectification error prompt

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

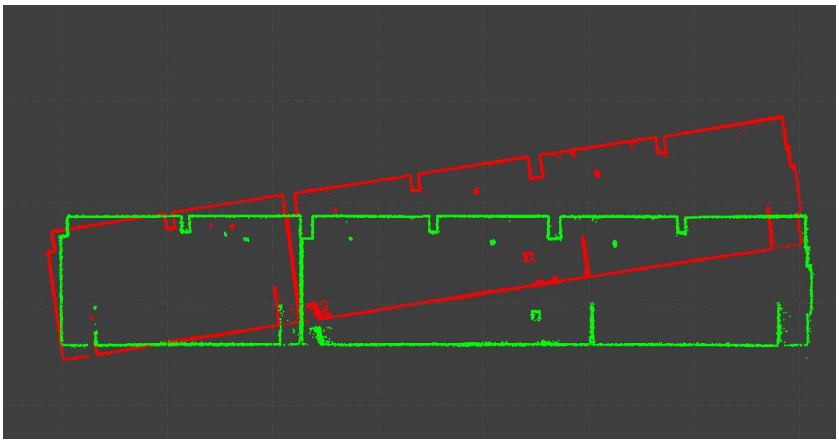


Figure 64. Effects before and after point cloud rectification

7.2.3 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**.

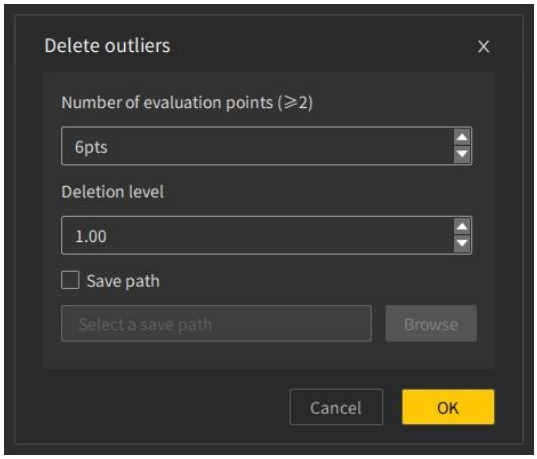


Figure 65. Outlier deletion screen

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.

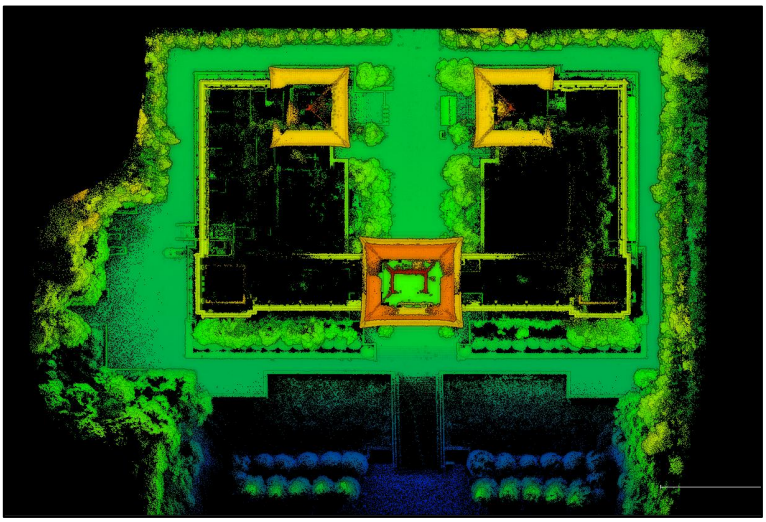


Figure 66. Before outliers are deleted

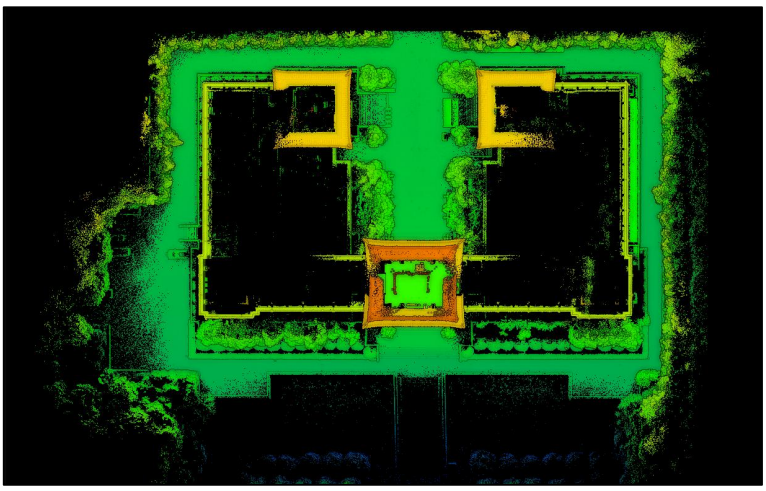


Figure 67. After outliers are deleted

7.3 Registration

7.3.1 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. The following is the procedure of registration when the control points are picked by clicking.

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.

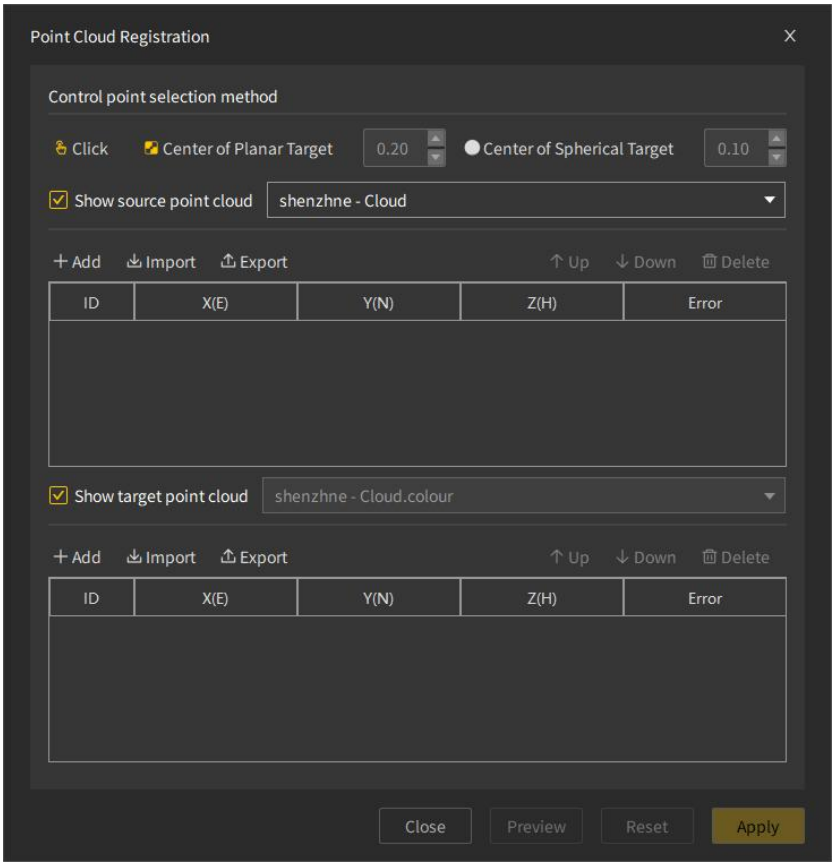


Figure 68. Point cloud registration window

2. 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.

3. To help better pick points in the source point cloud, you can hide the target point cloud by deselecting the **Show target point cloud** checkbox. After picking the control points in the source point cloud, select their corresponding points in the target point cloud, as shown below.

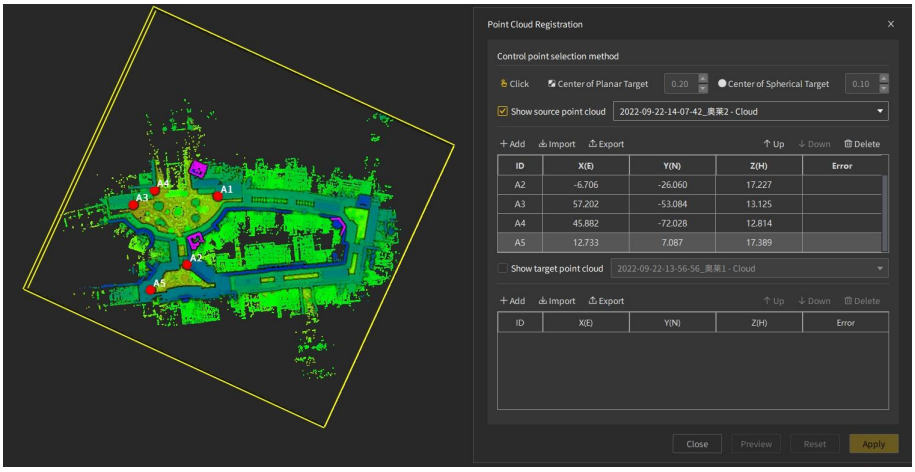


Figure 69. Picking points in the source point cloud

4. When the same number of control points are selected in the source and target point clouds and the number is greater than three, the errors are calculated automatically. You can modify the control point coordinates and click **Preview** to refresh the display.

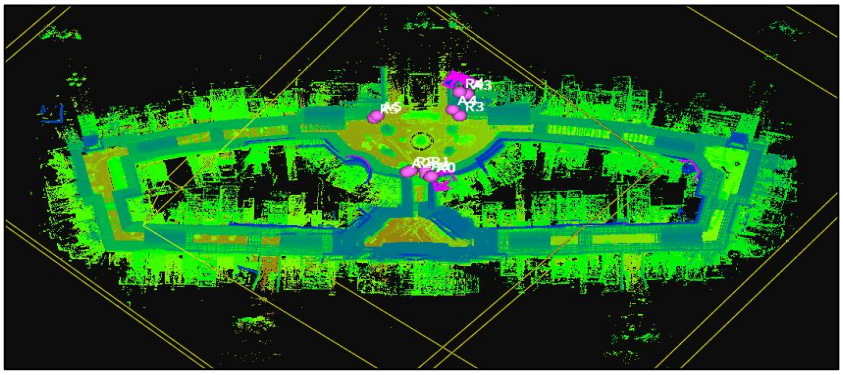


Figure 70. Registration preview

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

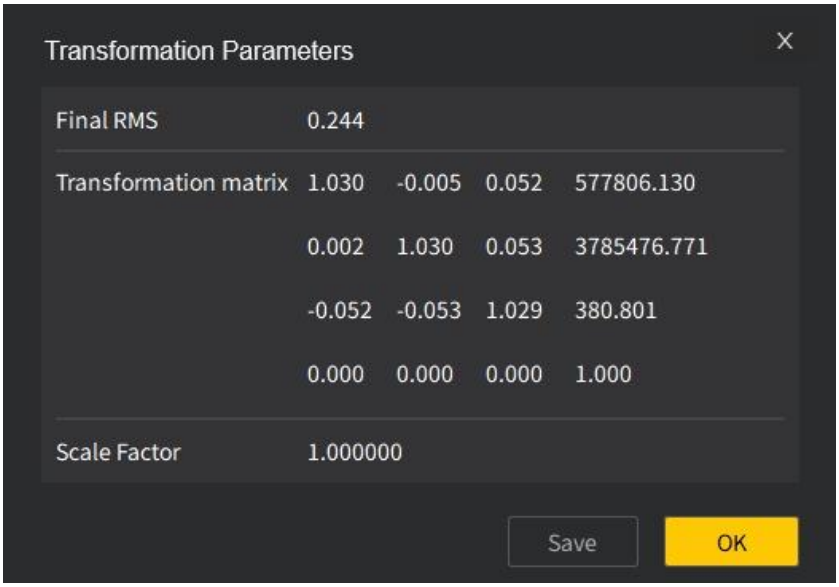


Figure 71. Transformation parameters

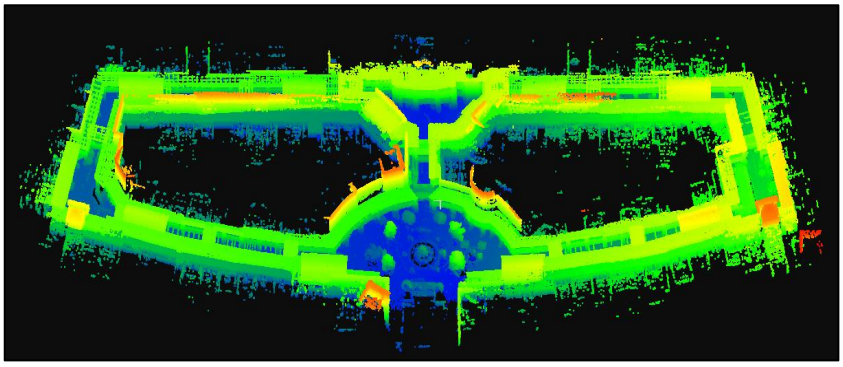


Figure 72. Registration result

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.

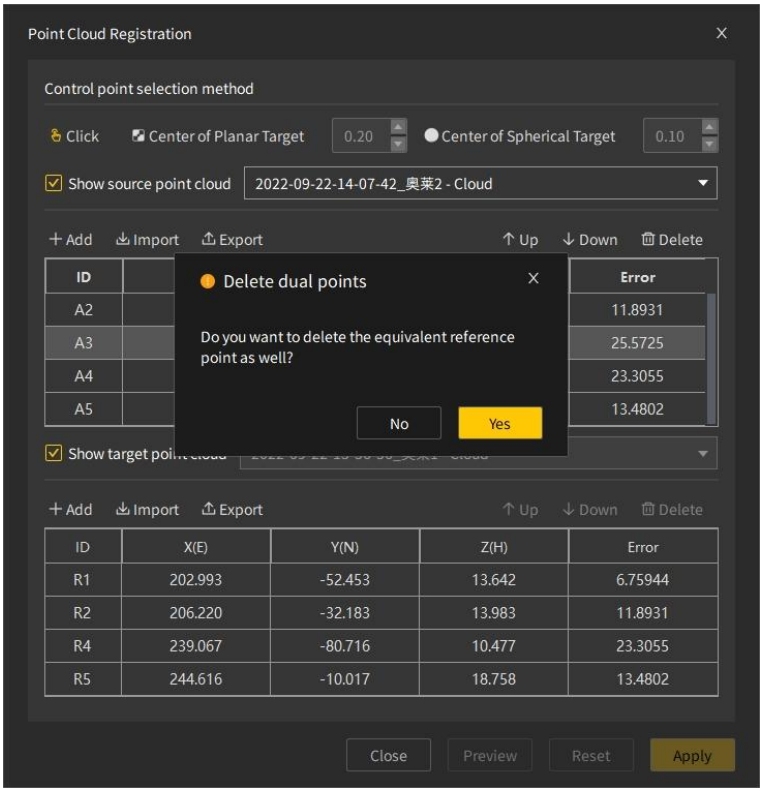


Figure 73. Prompt for deleting the control point in the source point cloud

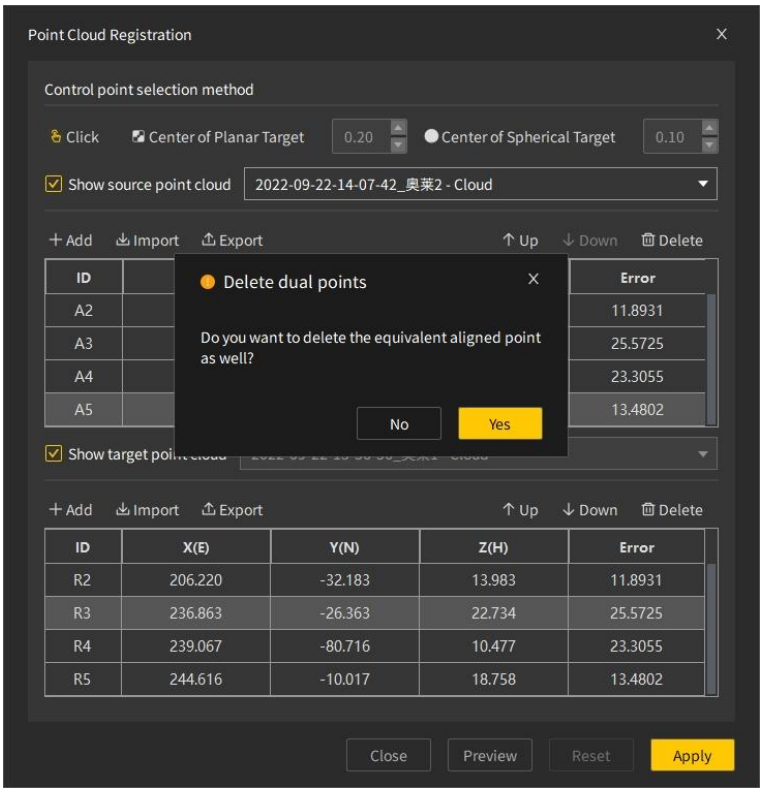


Figure 74. Prompt for deleting the control point in the target point cloud

7.3.2 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.

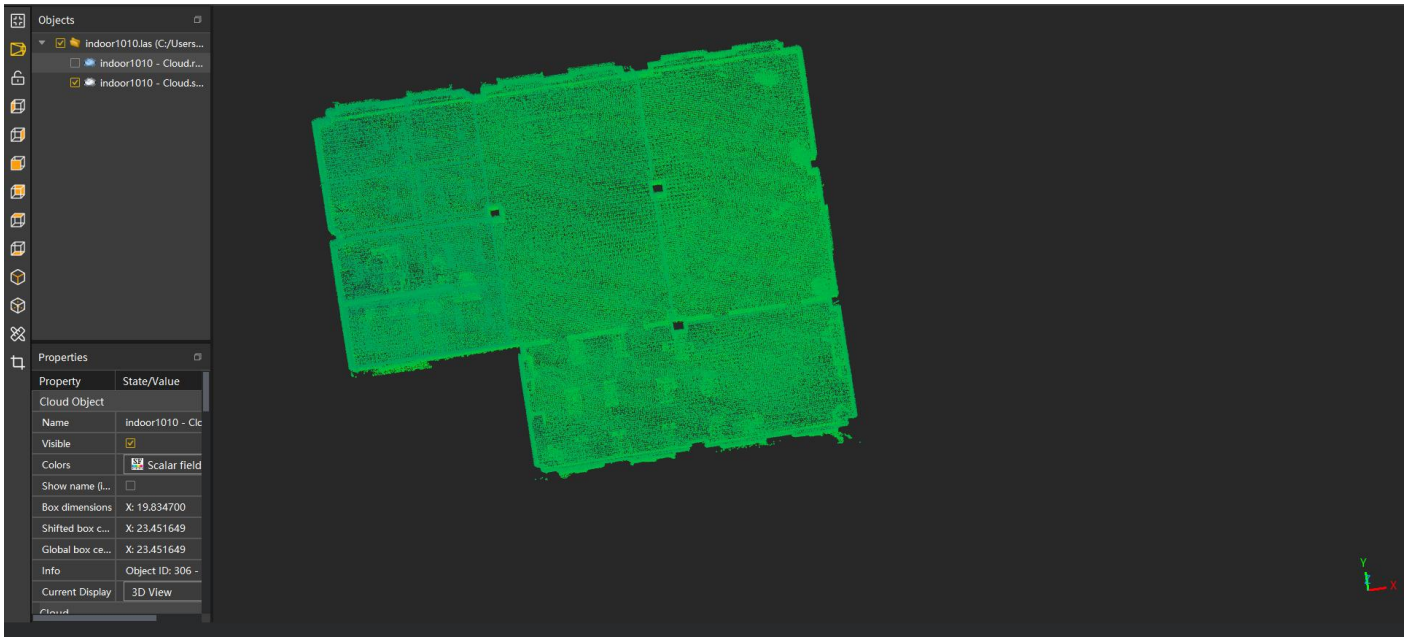


Figure 75. Point cloud 1

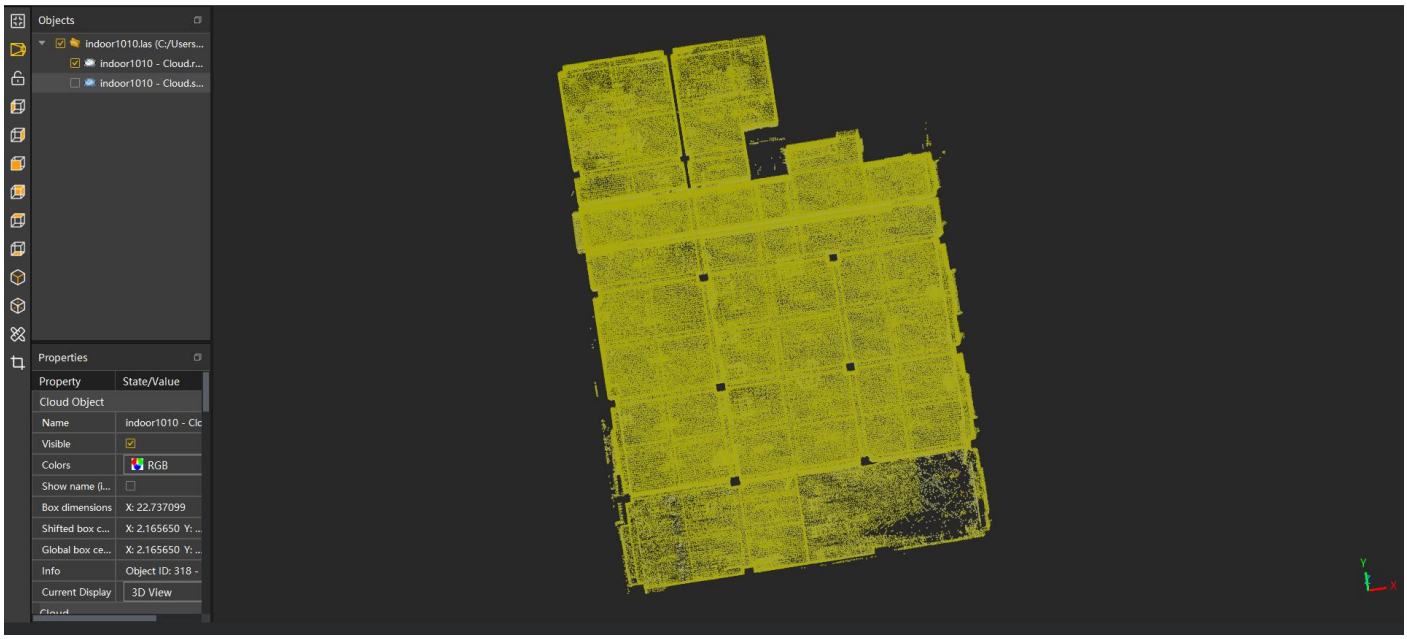


Figure 76. Point cloud 2

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.

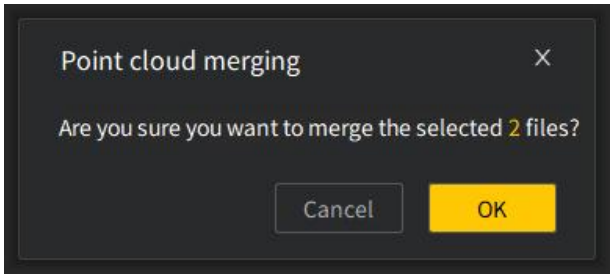


Figure 77. Merging confirmation window

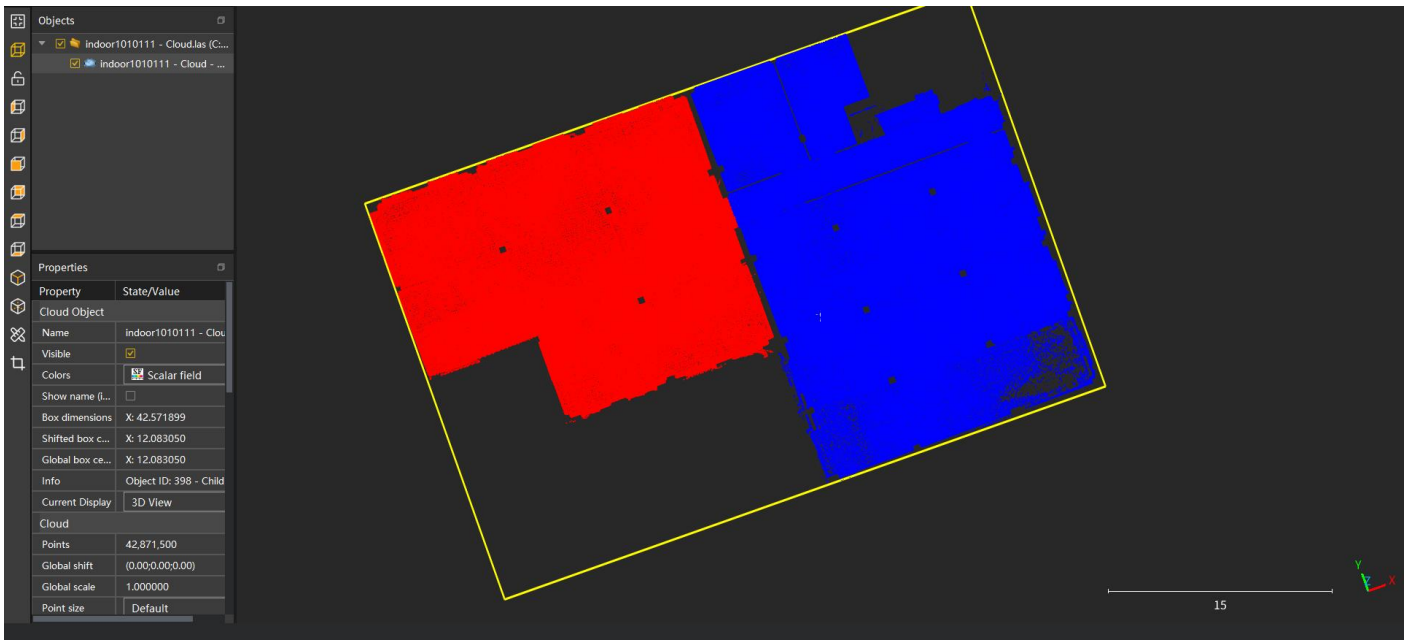


Figure 78. Point clouds merged

7.4 Transformation

7.4.1 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. Select **Unknown matrix**, and calculate the matrix through selected points.

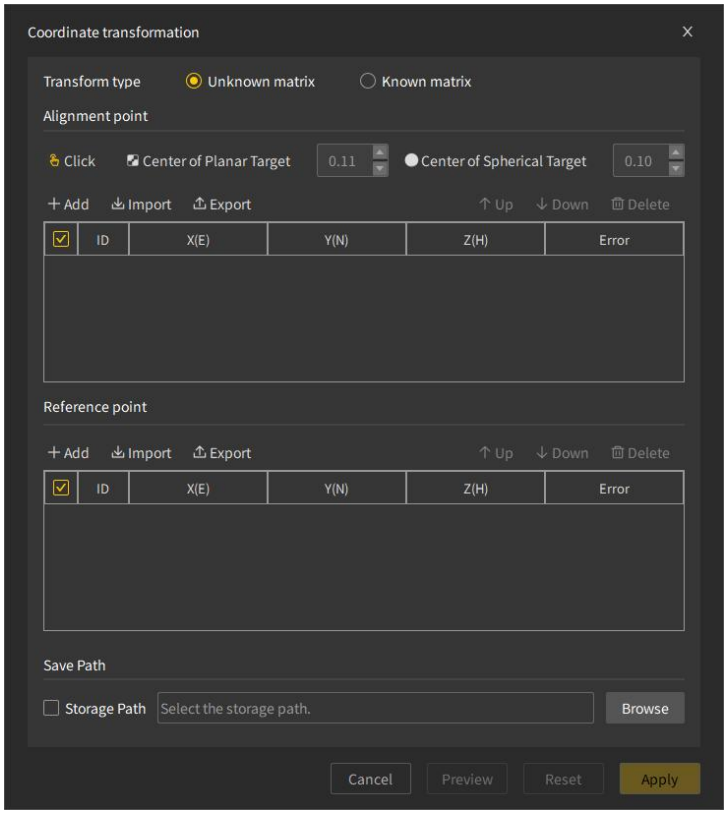


Figure 79. Coordinate transformation window

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

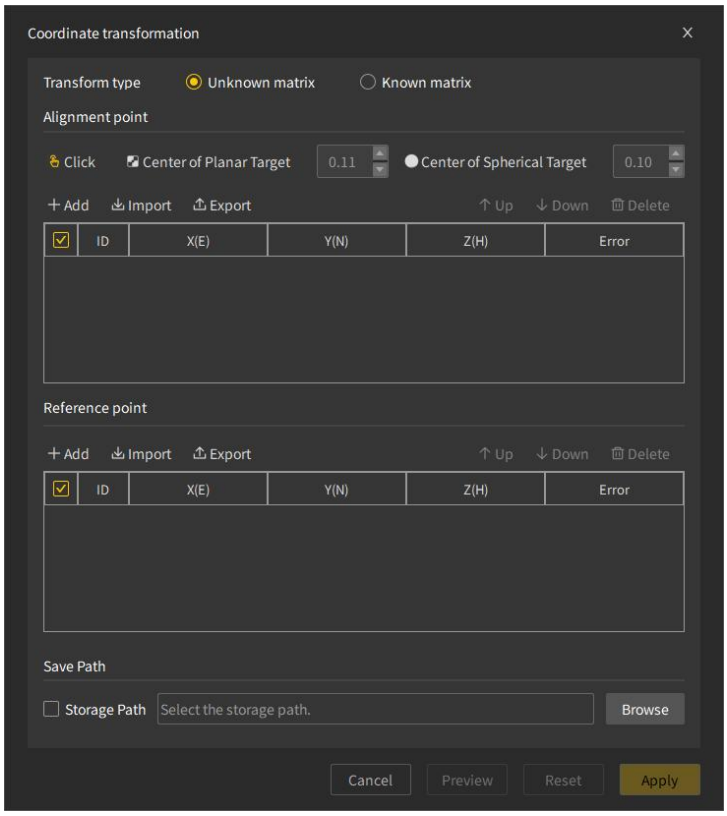


Figure 80. Control points picked by clicking

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.

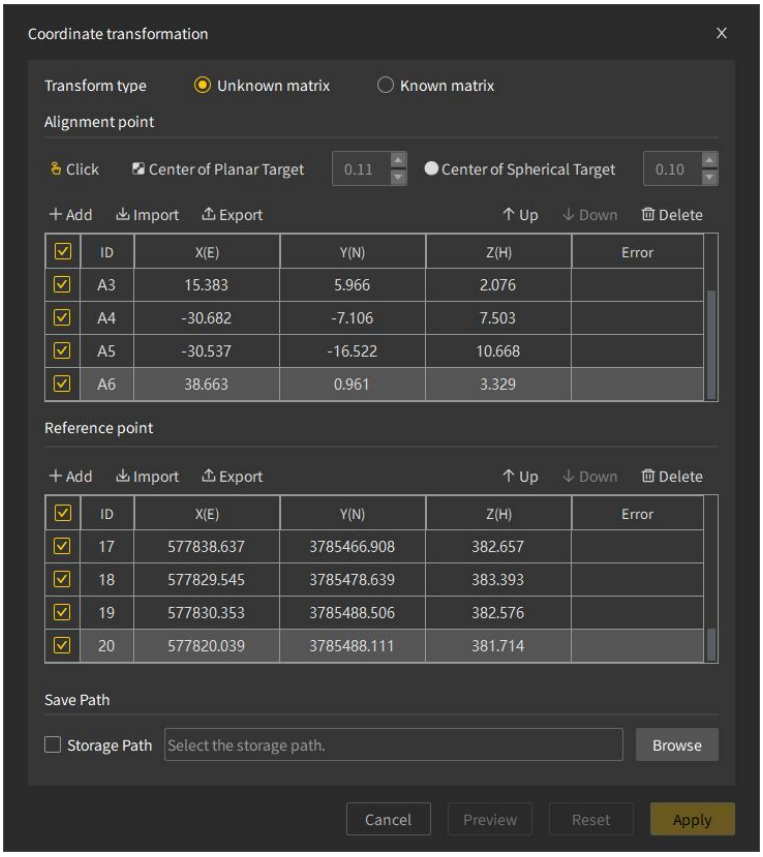


Figure 81. Entering the actual coordinates

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 errors 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.

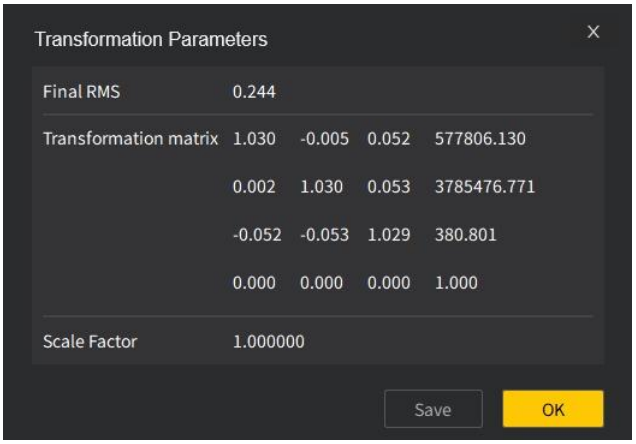


Figure 82. Transformation parameters

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.

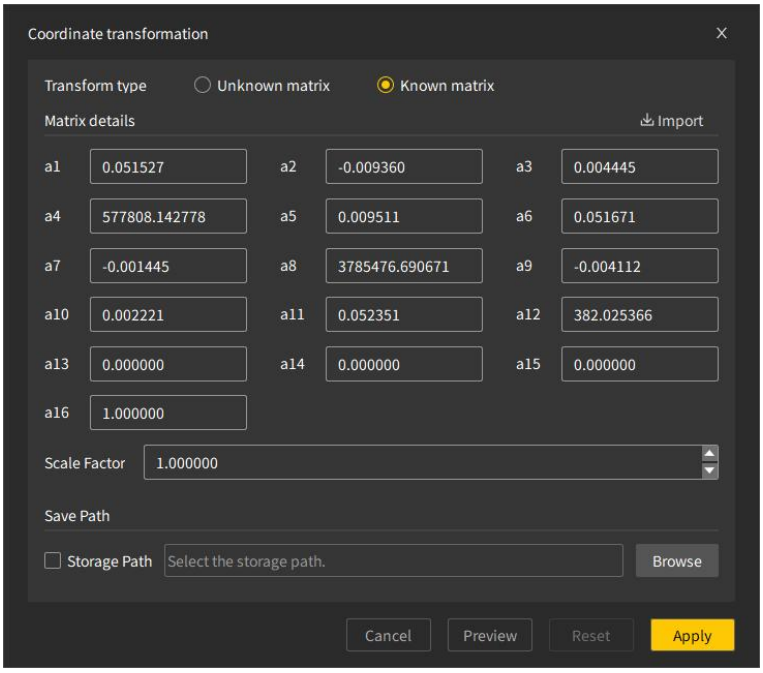


Figure 83. Known matrix transformation

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.

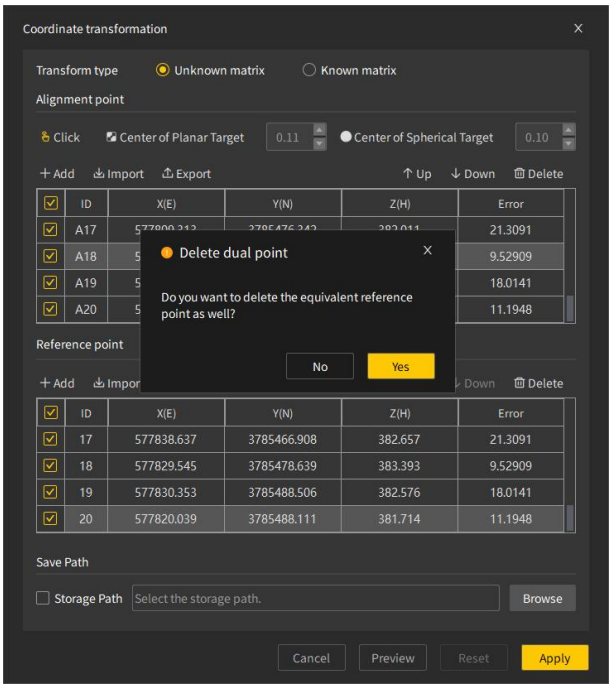


Figure 84. Prompt for deleting an alignment point

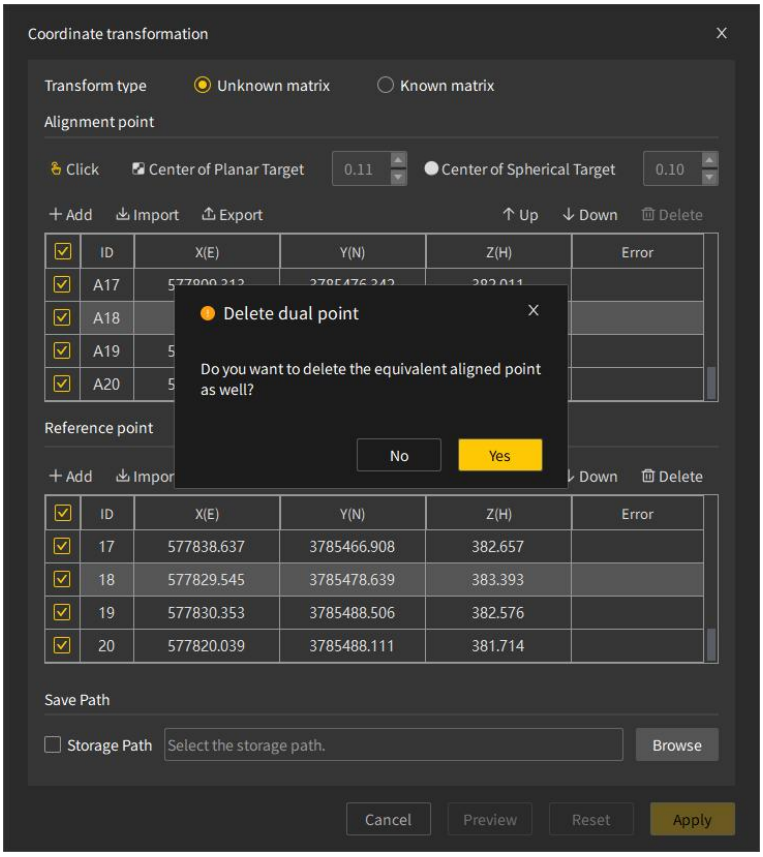


Figure 85. Prompt for deleting the actual coordinates

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.

7.4.2 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.

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.

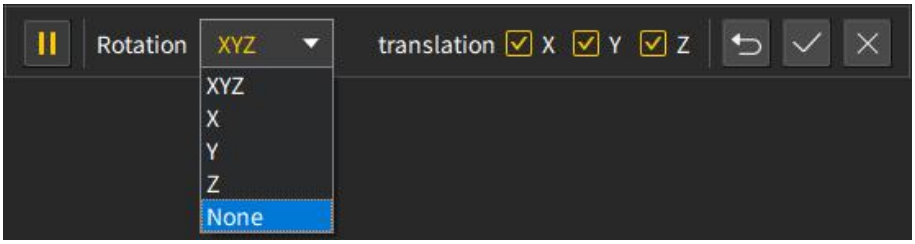


Figure 86. Selecting a rotation mode

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 87. Pausing the translation/rotation

Figure 88. 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 89. 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 90. Pausing the translation/rotation

Figure 91. 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).

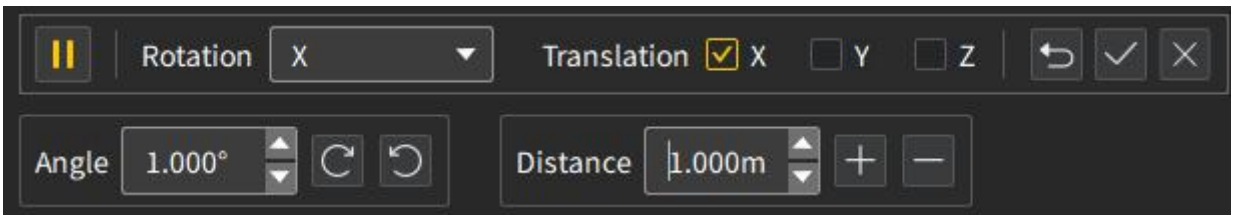


Figure 92. Accurate translation/rotation

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.

7.4.3 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, or .xlxs 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.

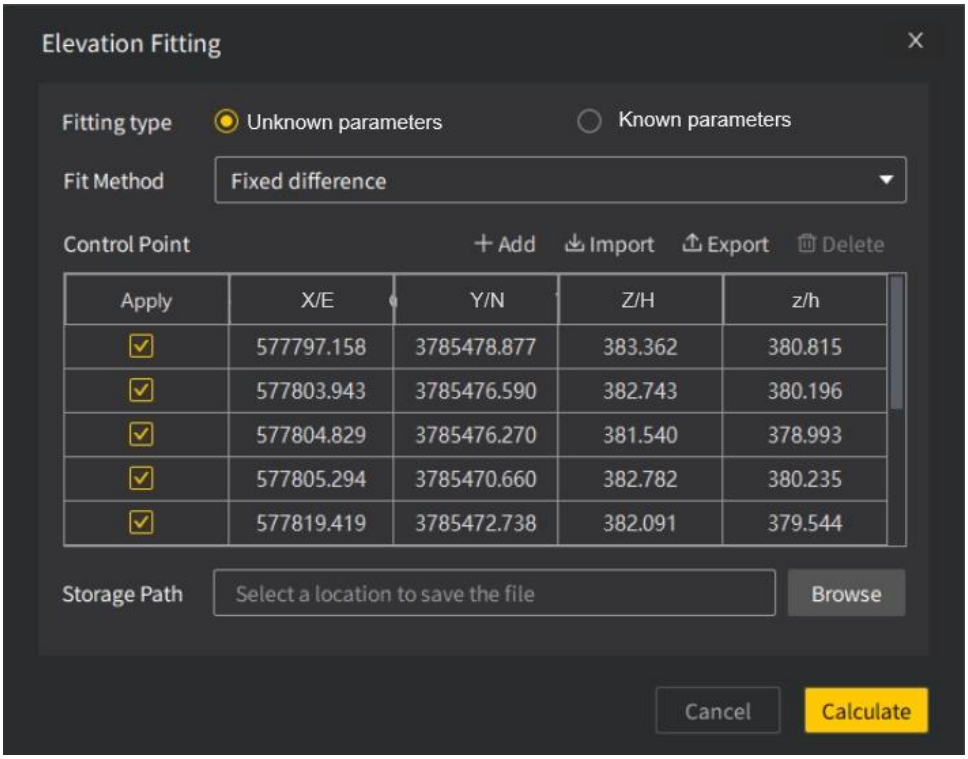


Figure 93. Fitting with unknown parameters

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

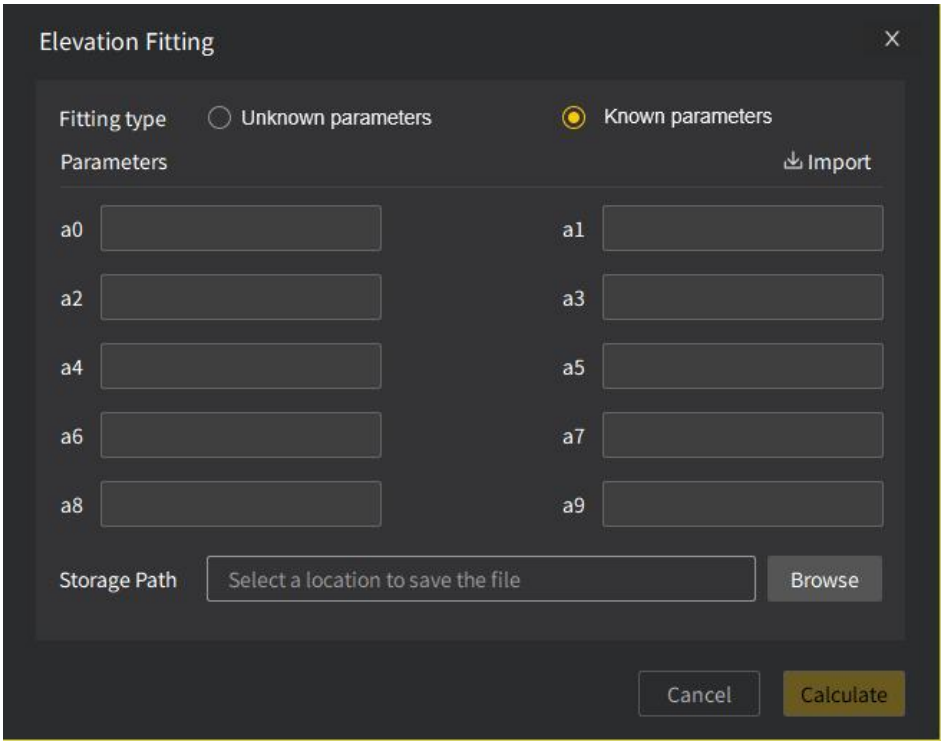


Figure 94. Fitting with known parameters

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.

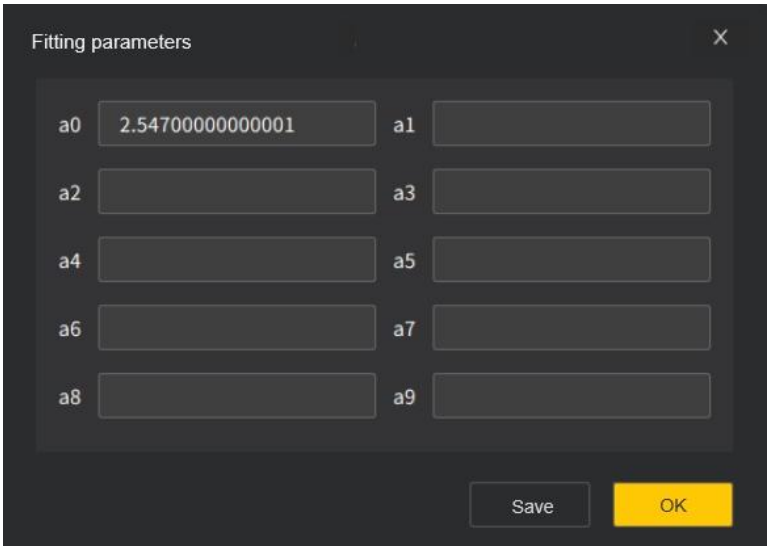


Figure 95. Fitting parameters

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.

7.5 Quality Analysis

7.5.1 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.

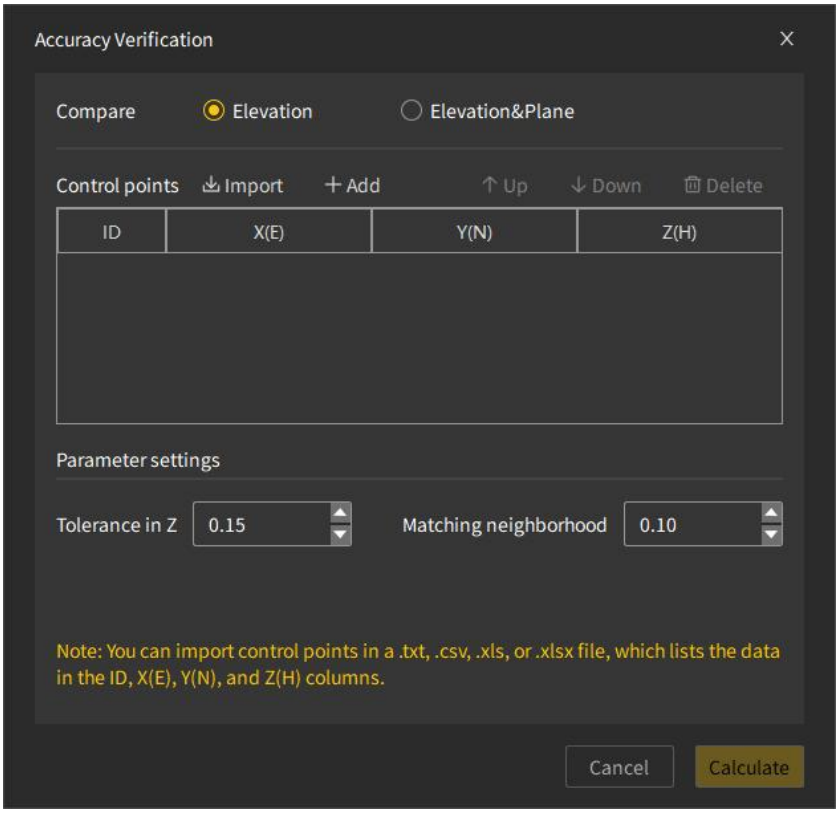


Figure 96. Elevation accuracy verification window

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.

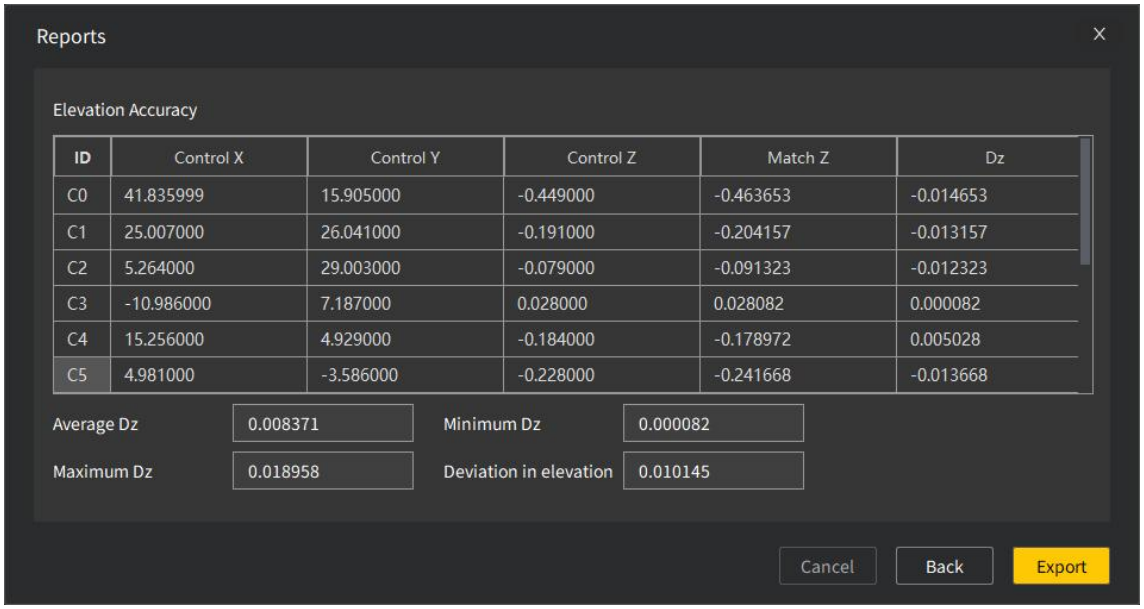


Figure 97. Elevation accuracy verification result window

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.

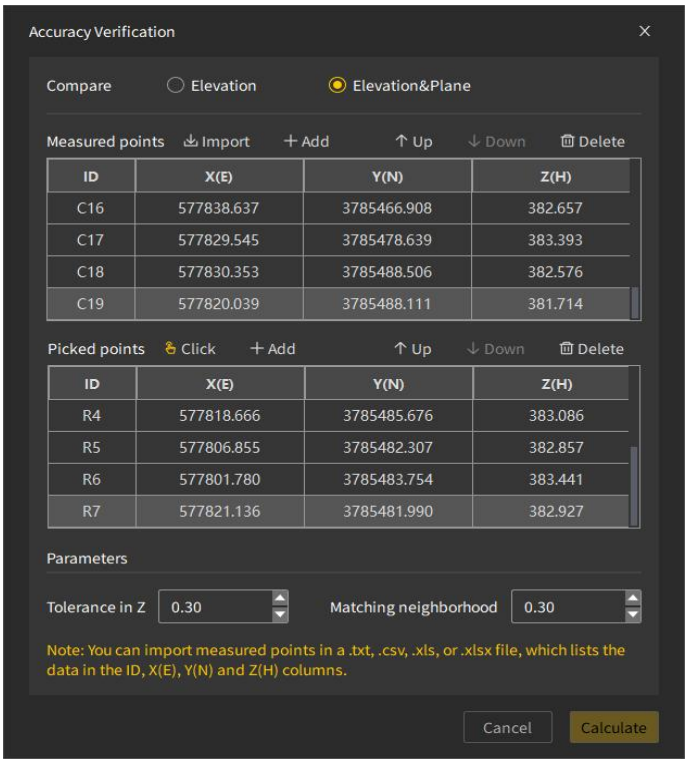


Figure 98. Elevation & plane accuracy verification window

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.

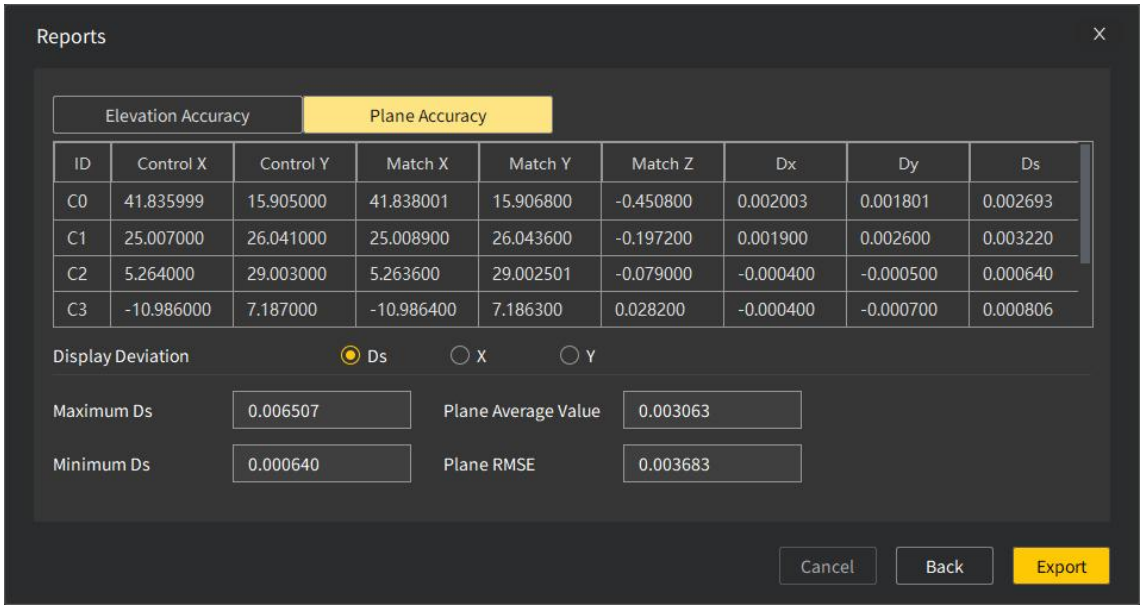


Figure 99. Elevation and plane accuracy verification result window

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.

7.5.2 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).

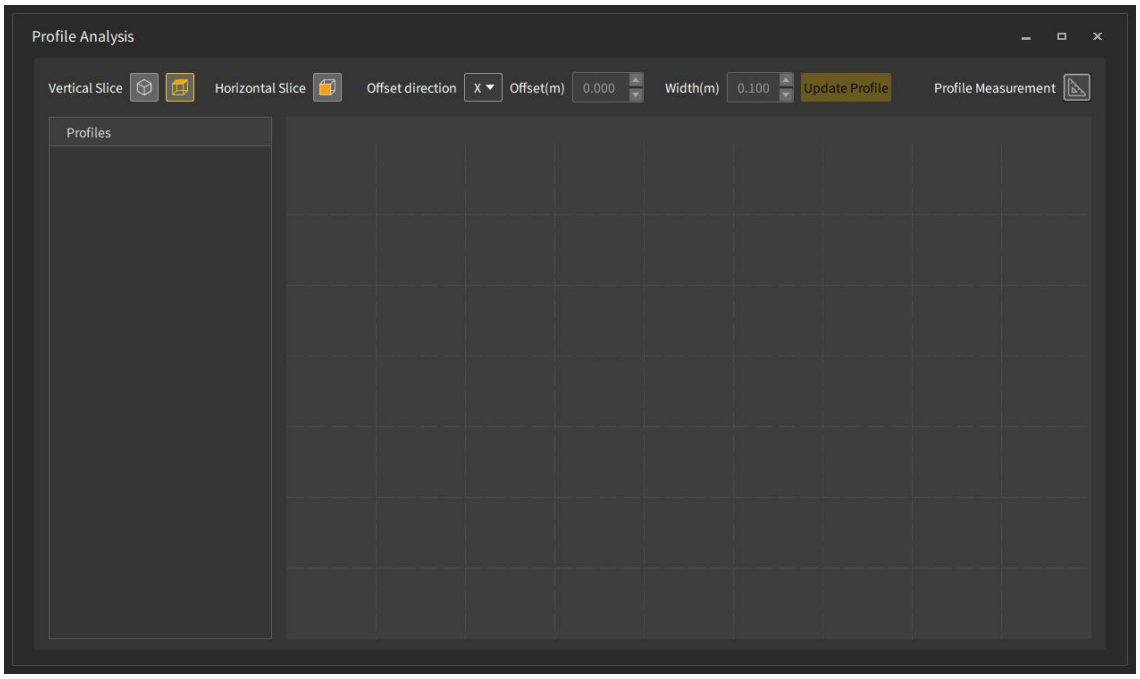


Figure 100. Profile analysis window

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.

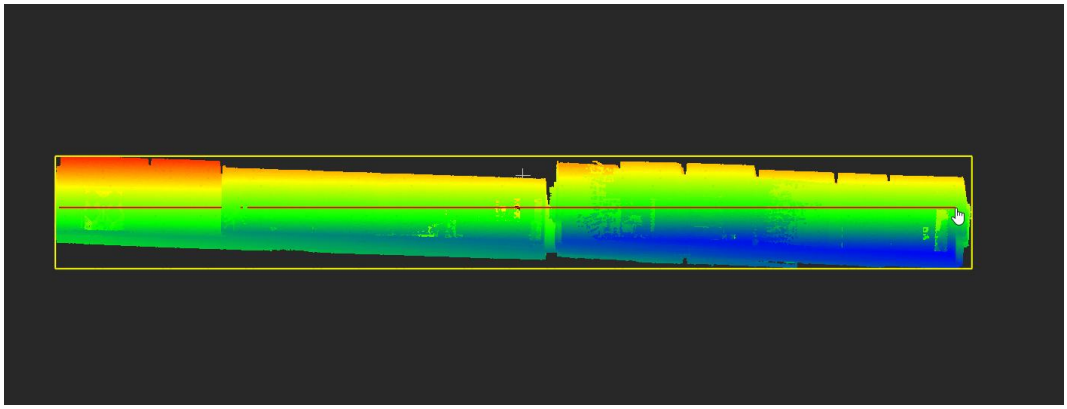


Figure 101. Creating a horizontal slice profile

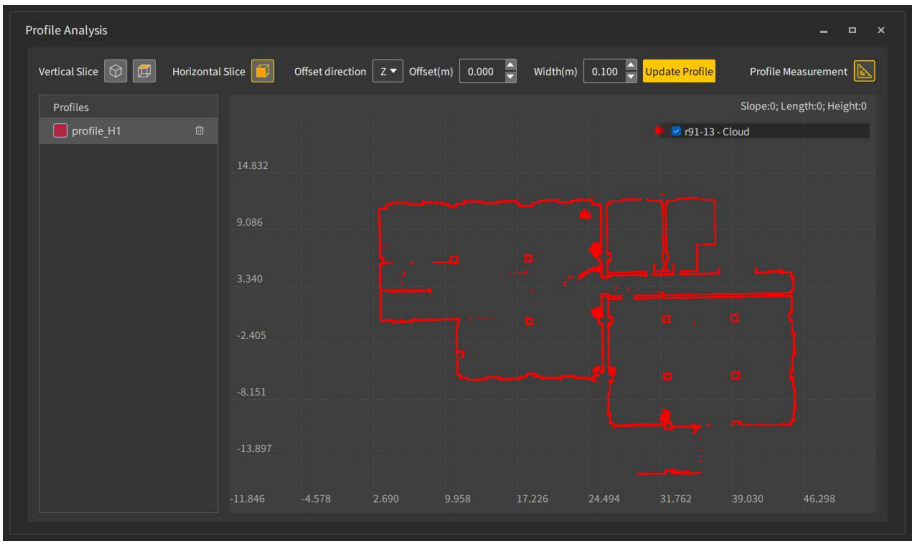


Figure 102. Profile display window

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.

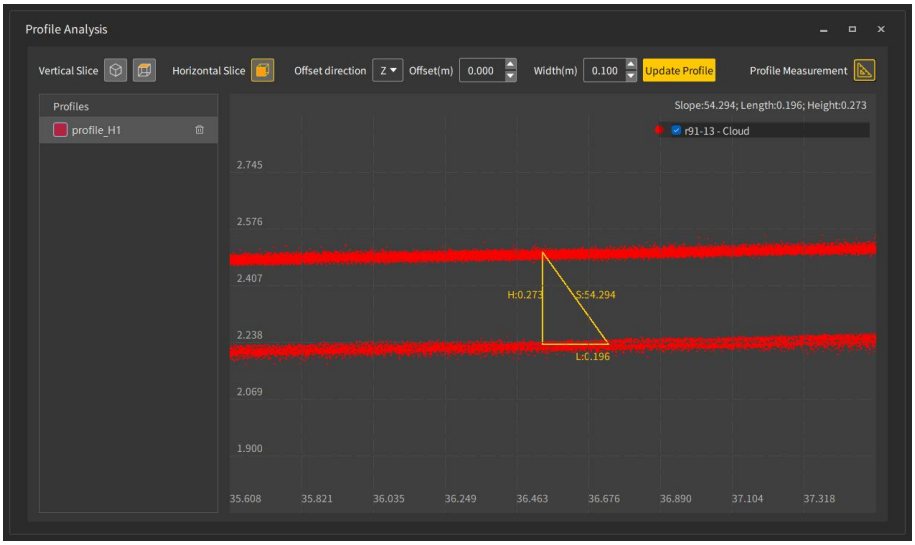


Figure 103. Profile measurement window

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.

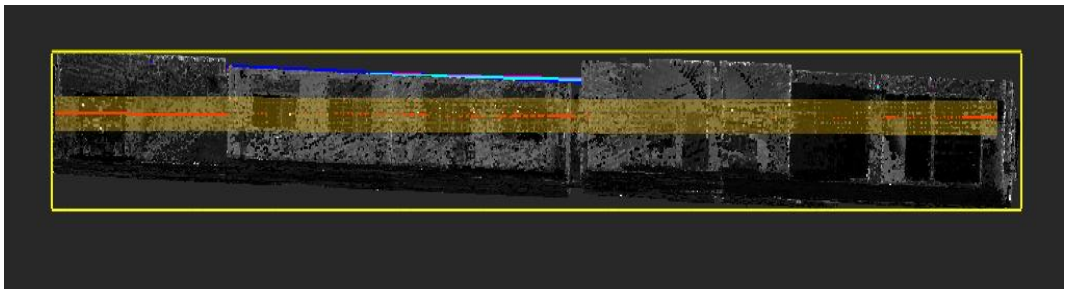


Figure 104. Adjusting the profile width

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

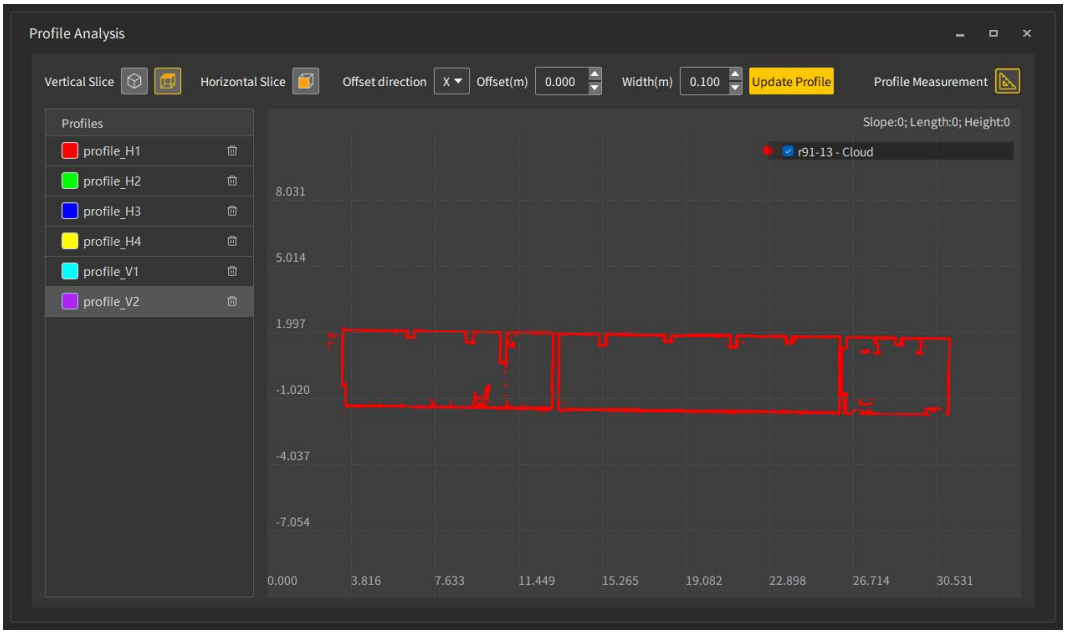


Figure 105. Multiple profiles created from the same point cloud

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.

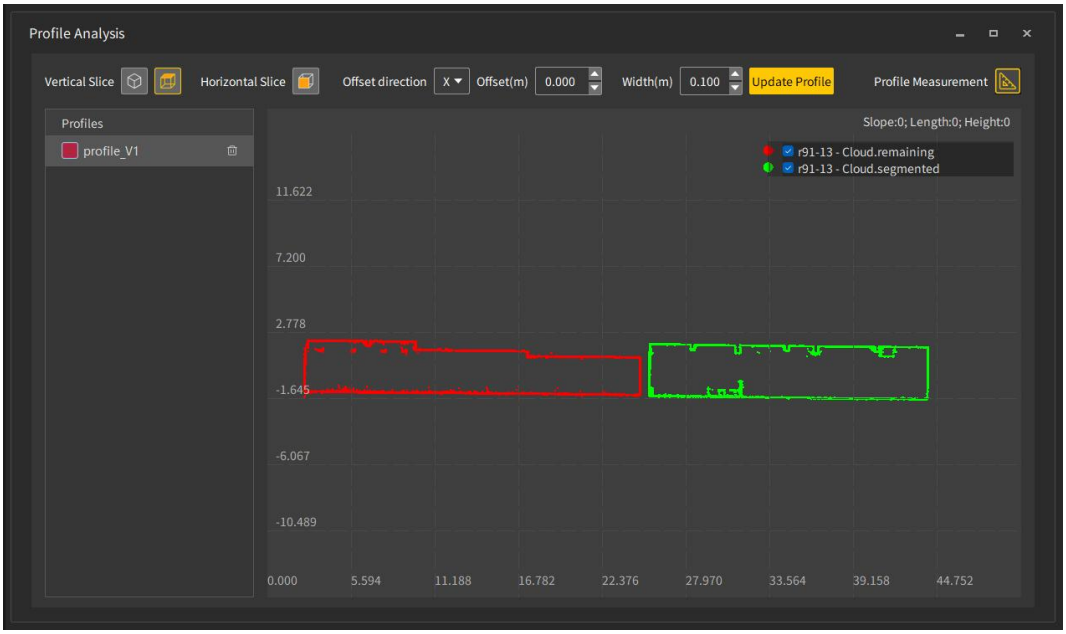


Figure 106. Profile created from two point clouds

Note:

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

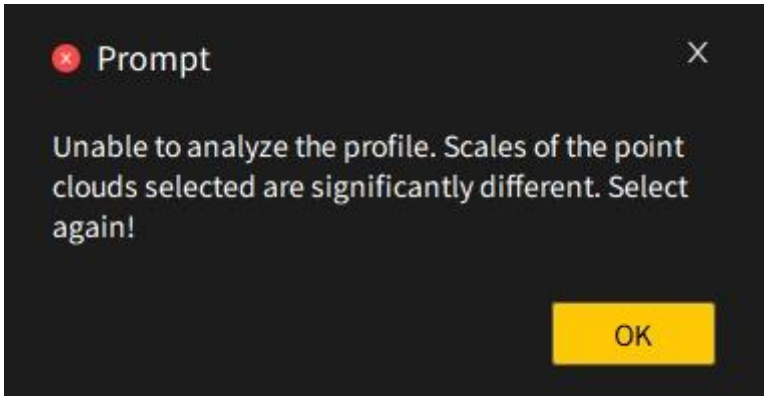


Figure 107. Prompt for unavailable profile analysis

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

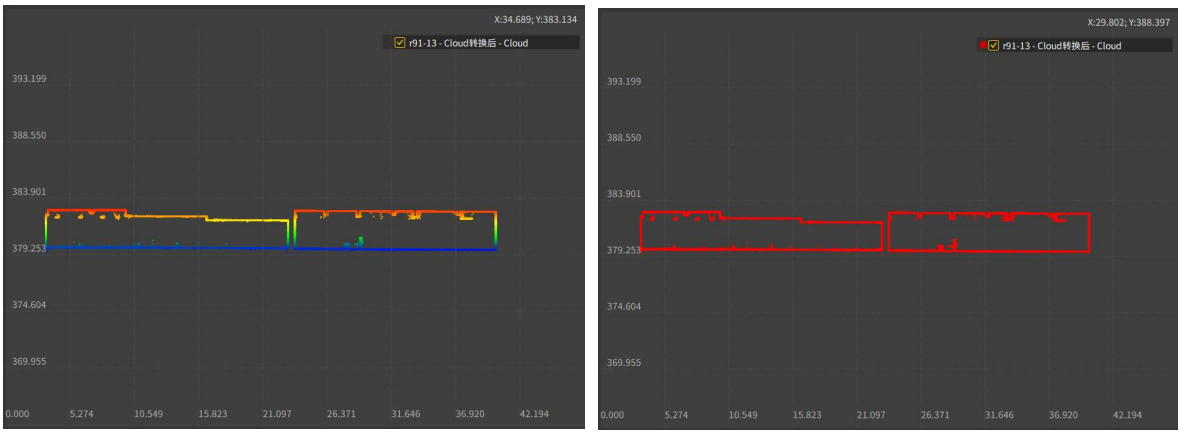


Figure 108. Profile after and before rendering

7.5.3 Density Measurement

Description:

Obtains the number of points per unit area. It shows the spatial distribution and concentration of points to evaluate the device performance and applicable scenarios.

Operation procedure:

1. Select a point cloud file under **Objects**.
2. Click the **Start** tab, and select . The following window appears.

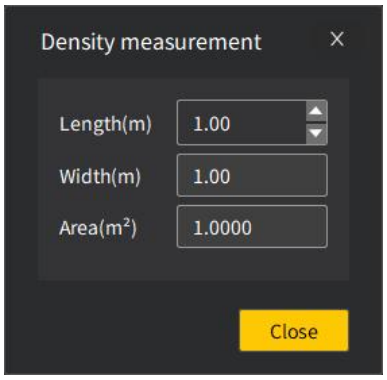


Figure 109. Density measurement

3. When the hand cursor appears, click the point cloud to obtain the density of the corresponding area.

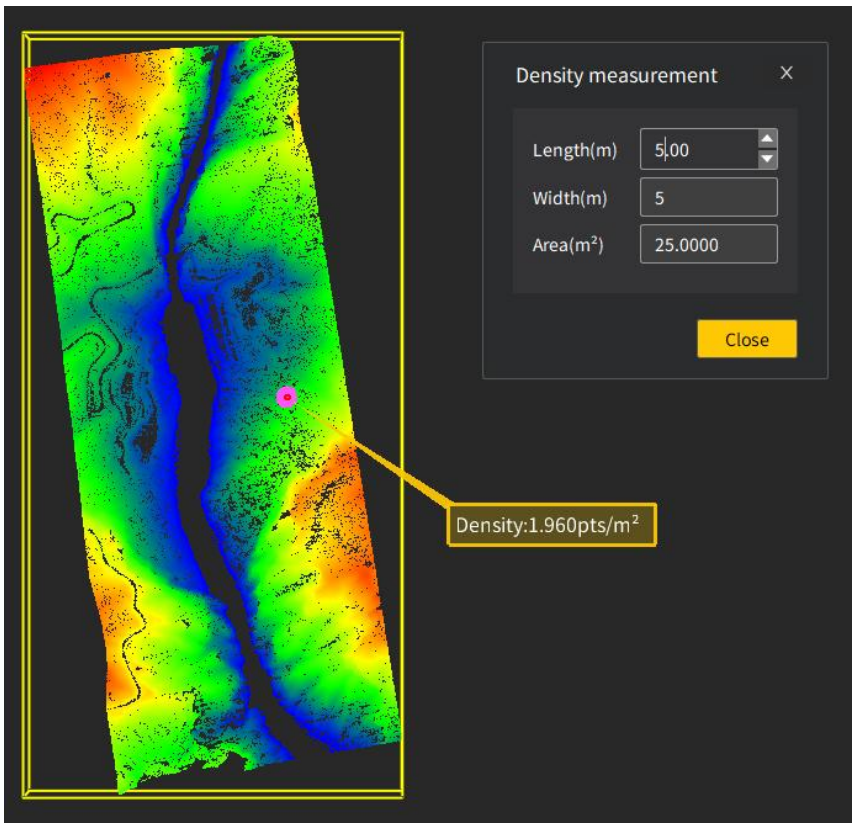


Figure 110. Density measurement result

4. Click **OK** or the exit button in the upper right corner to exit the feature.

Note:

- Density measurement is unavailable when you click the area of the view other than the point cloud.
- When you change the length, both the width and the area are updated. The width is always the same as the length.

7.6 Project Templates

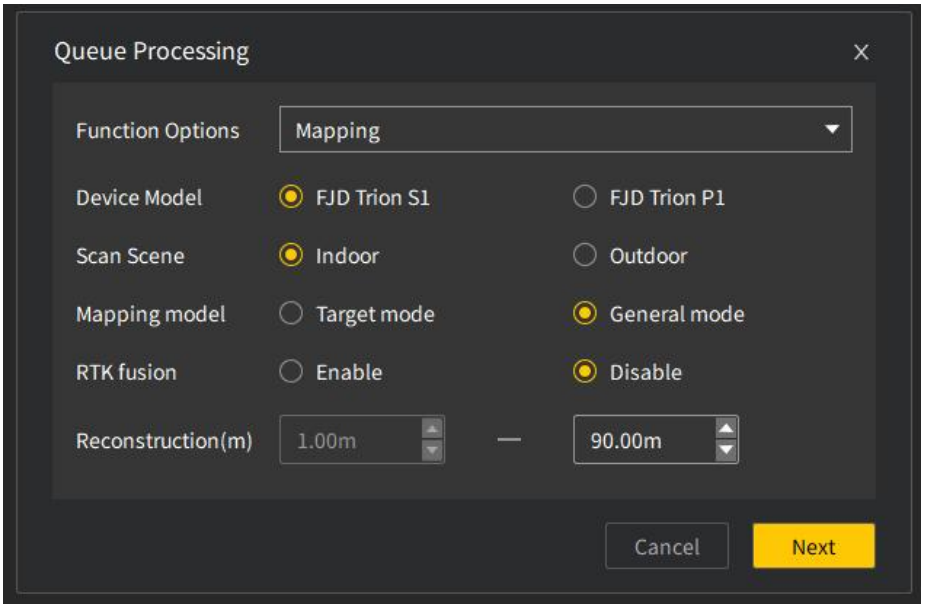
7.6.1 Queue Processing

Description:

Allows processing multiple compressed files at the same time. It is currently available for 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**.



2. Select no more than 10 compressed files for mapping.

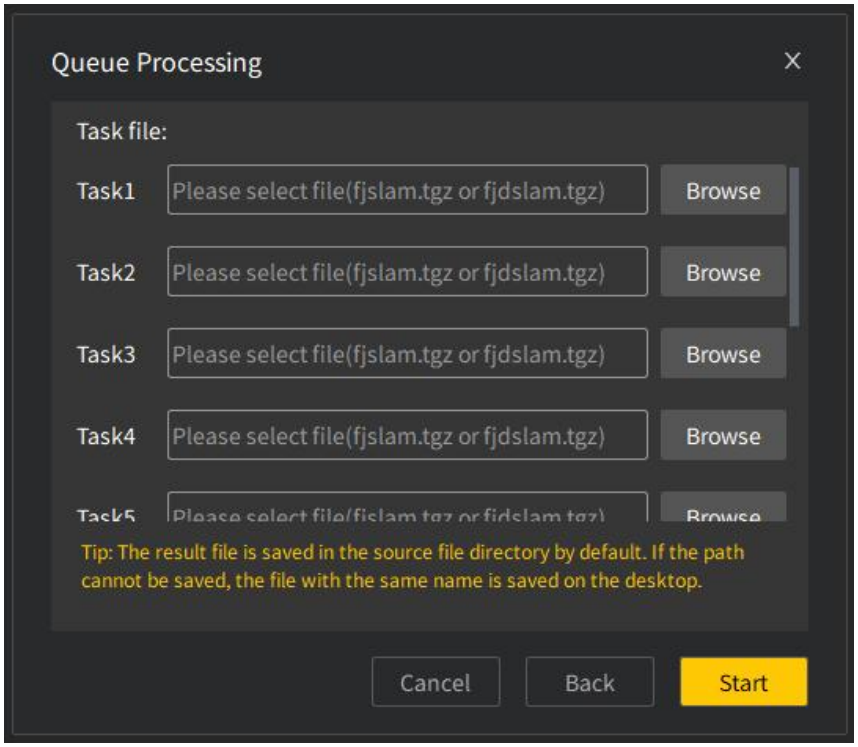


Figure 111. Queue processing for mapping

3. Click **Start** to start the mapping. The parameter setting window disappears and the progress window appears. The progress bar may show the status of completed, in progress, or failed.

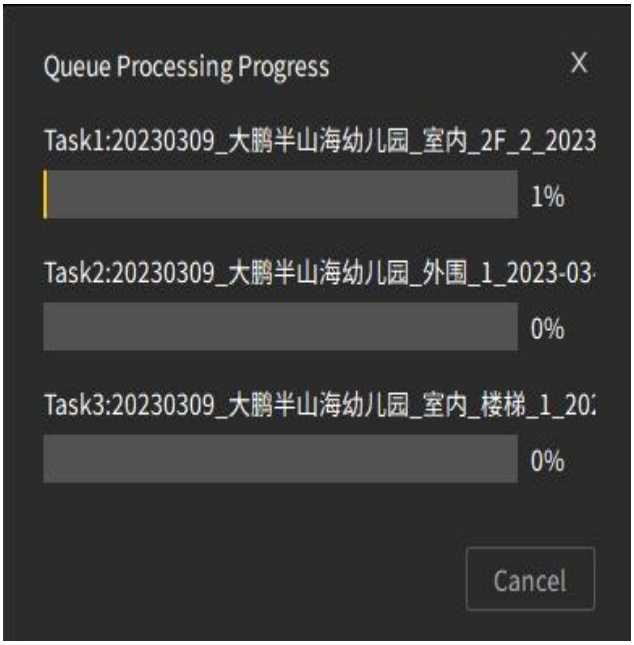


Figure 112. Queue processing progress

4. When all tasks are completed, the progress window disappears and the window for selecting data to load appears. Select data, and click **OK**.

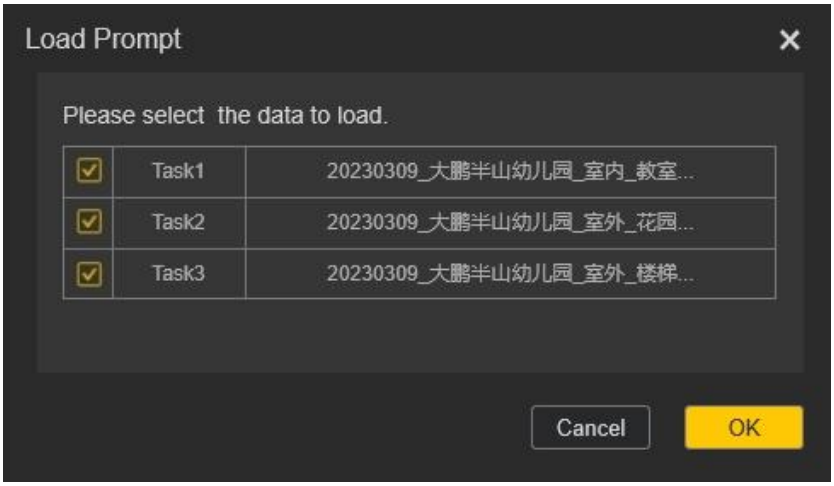


Figure 113. Selecting data to load

Note:

- A maximum of 10 tasks can be processed at a time.
- Ensure that there is sufficient memory available before queue processing.

8 Edit

8.1 Sampling

8.1.1 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**.

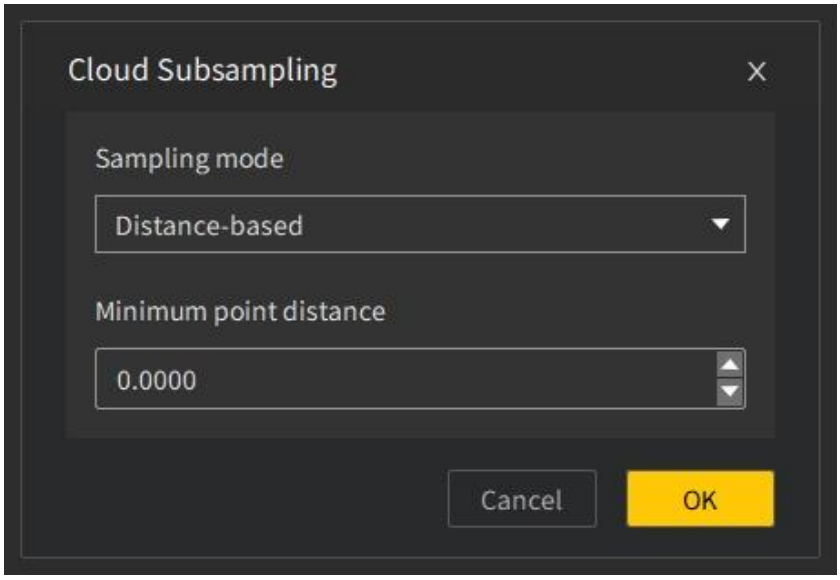


Figure 114. Cloud subsampling

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.

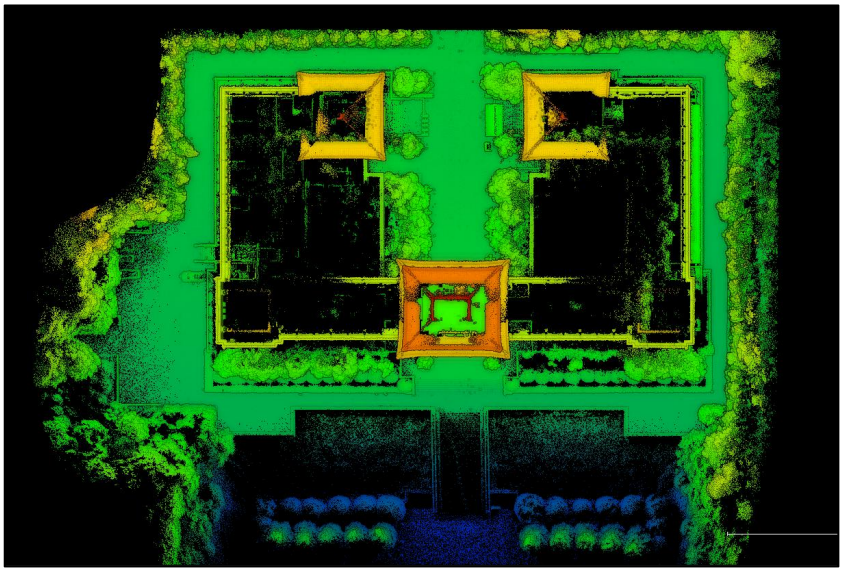


Figure 115. Raw data

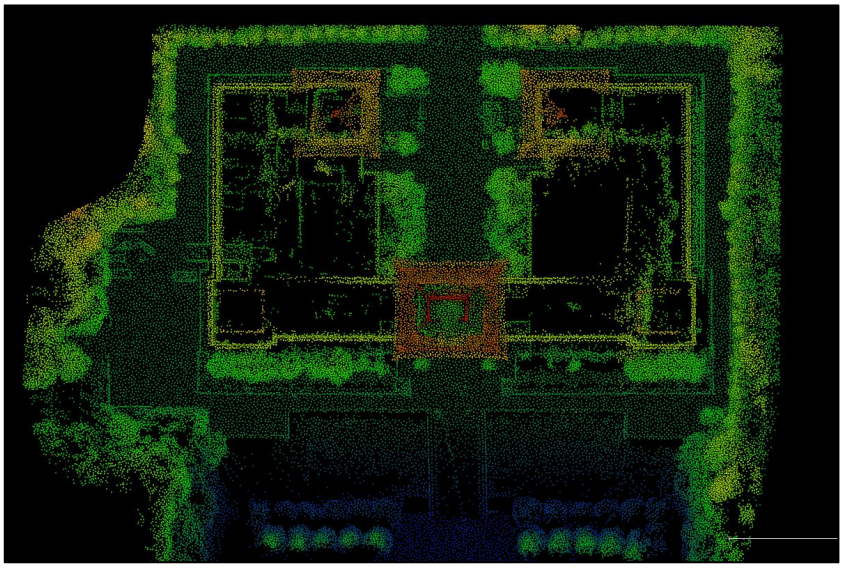


Figure 116. Distance-based subsampling

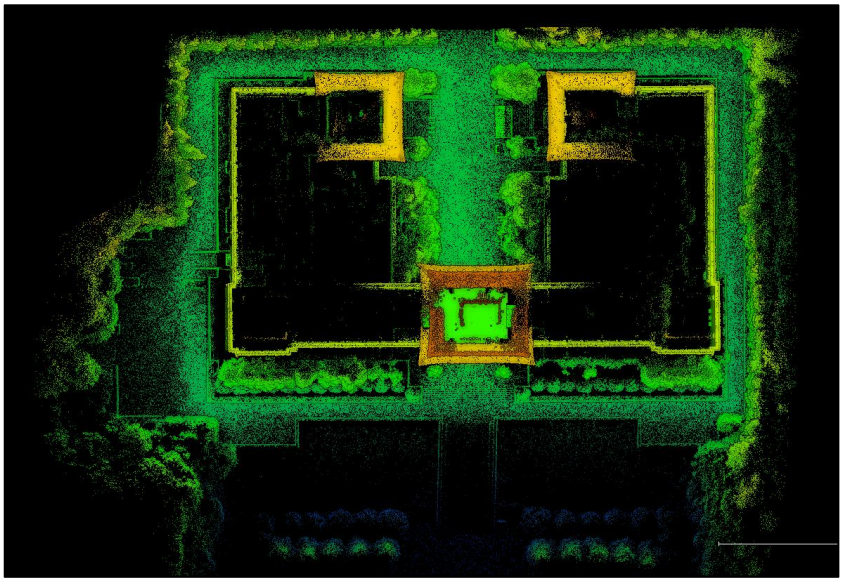


Figure 117. Density-based subsampling

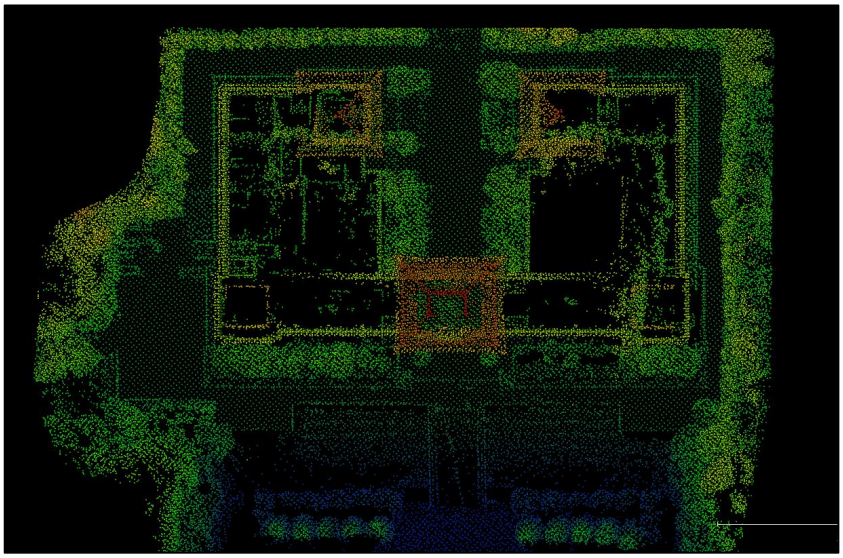


Figure 118. Spatial structure-based subsampling

8.2 Segment

8.2.1 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.

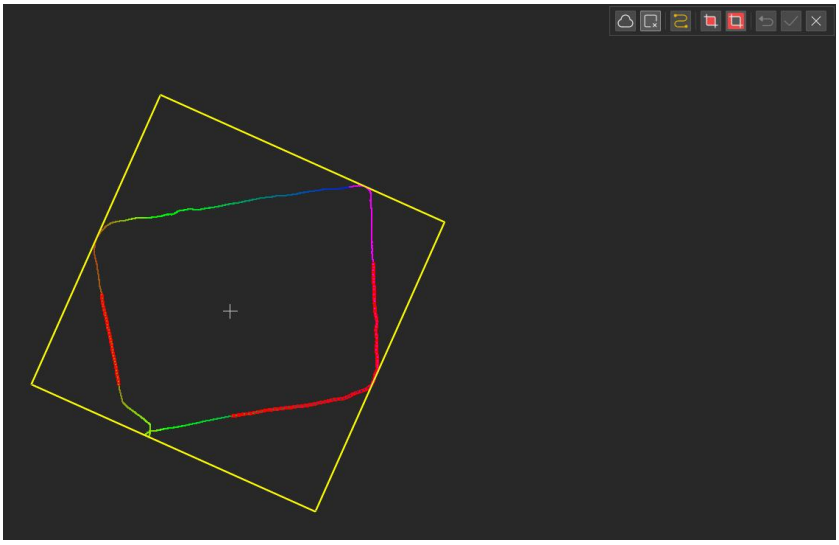


Figure 119. Path selected

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.

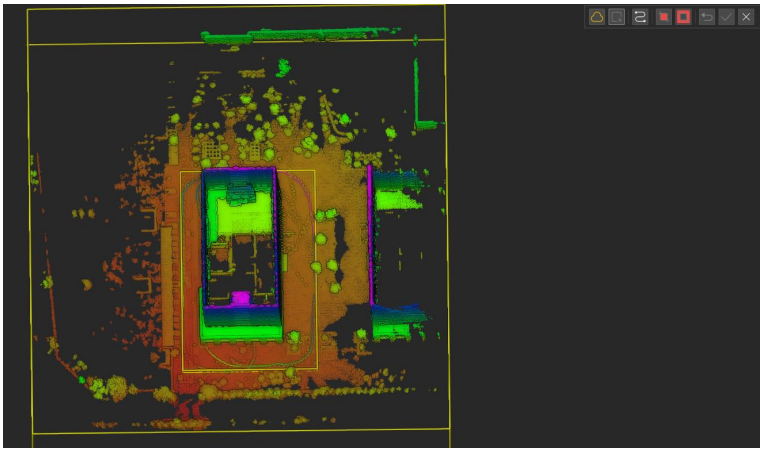


Figure 120. Point cloud before clipping

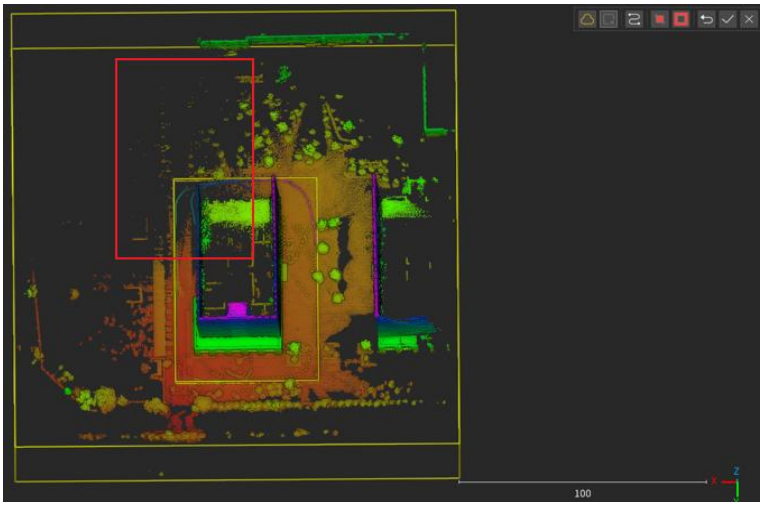


Figure 121. Point cloud after clipping

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.

8.3 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 **Trees** and the **Ground**. The **Indoor** scenario includes **Wall**, **Floor**, and **Ceiling**.

8.3.1 Outdoor

Operation procedure:

1. Select a point cloud file under **Objects** to activate the outdoor classification feature.
2. Click , and the following window appears. The ground types include **Flat**, **Gentle slope**, and **Steep slope**. The extraction accuracy is inversely proportional to the steepness. The steeper the slope, the smaller the extraction accuracy.

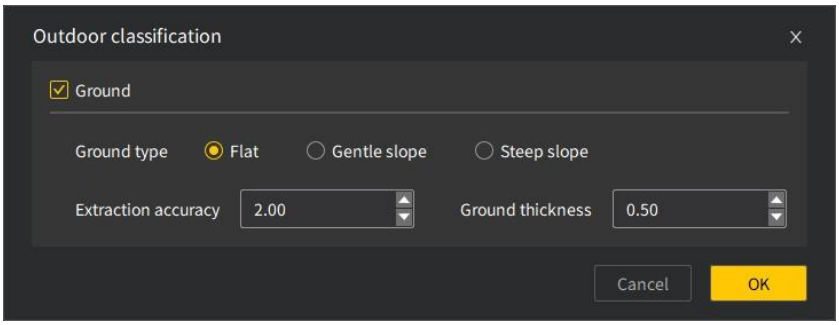


Figure 122. Outdoor classification

3. Select **Flat**, and then click **OK** to extract the ground cloud. The ground and other elements are extracted as below.

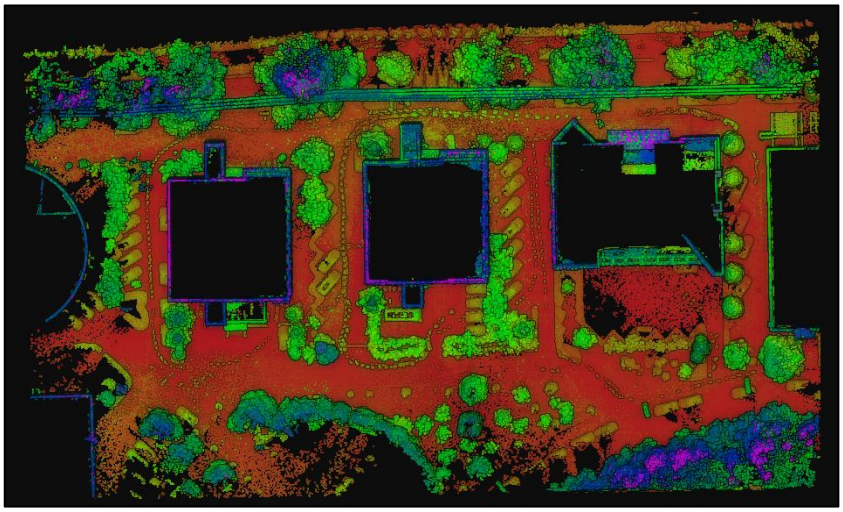


Figure 123. Raw point cloud

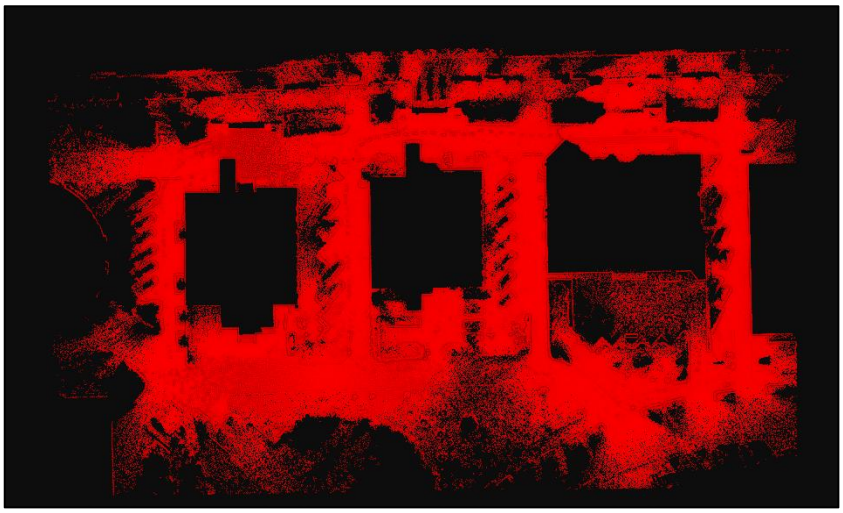


Figure 124. Ground extracted

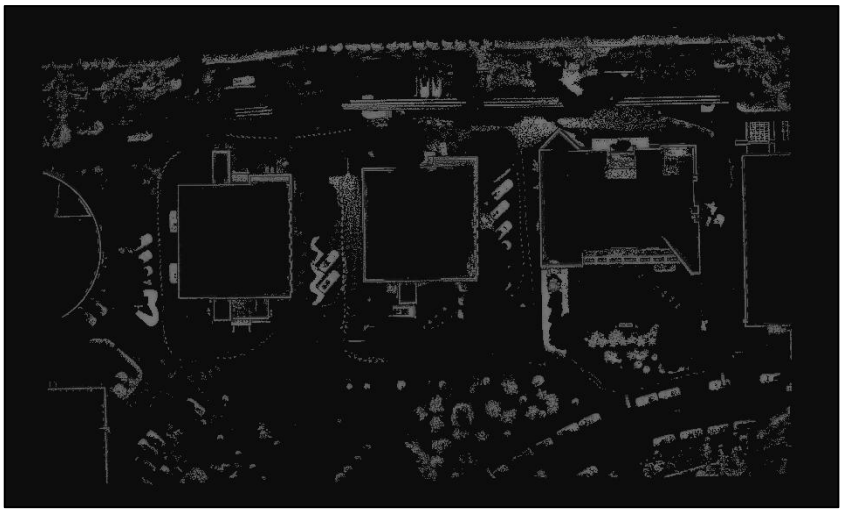


Figure 125. Other elements extracted

8.3.2 Indoor

Operation procedure:

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

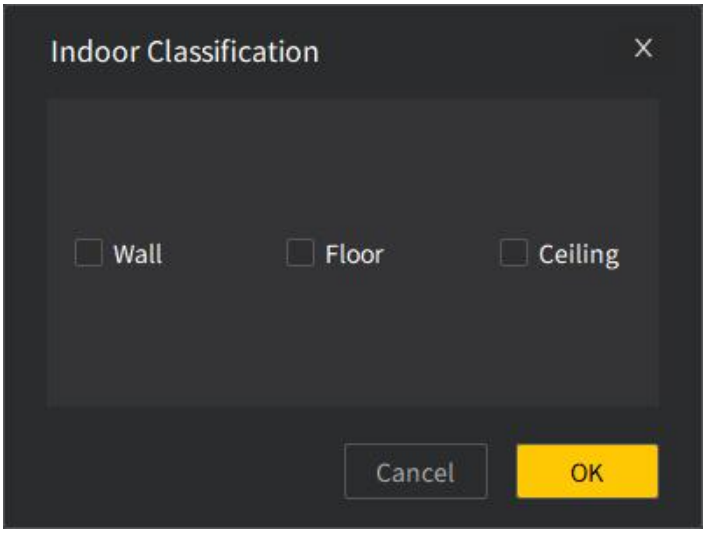
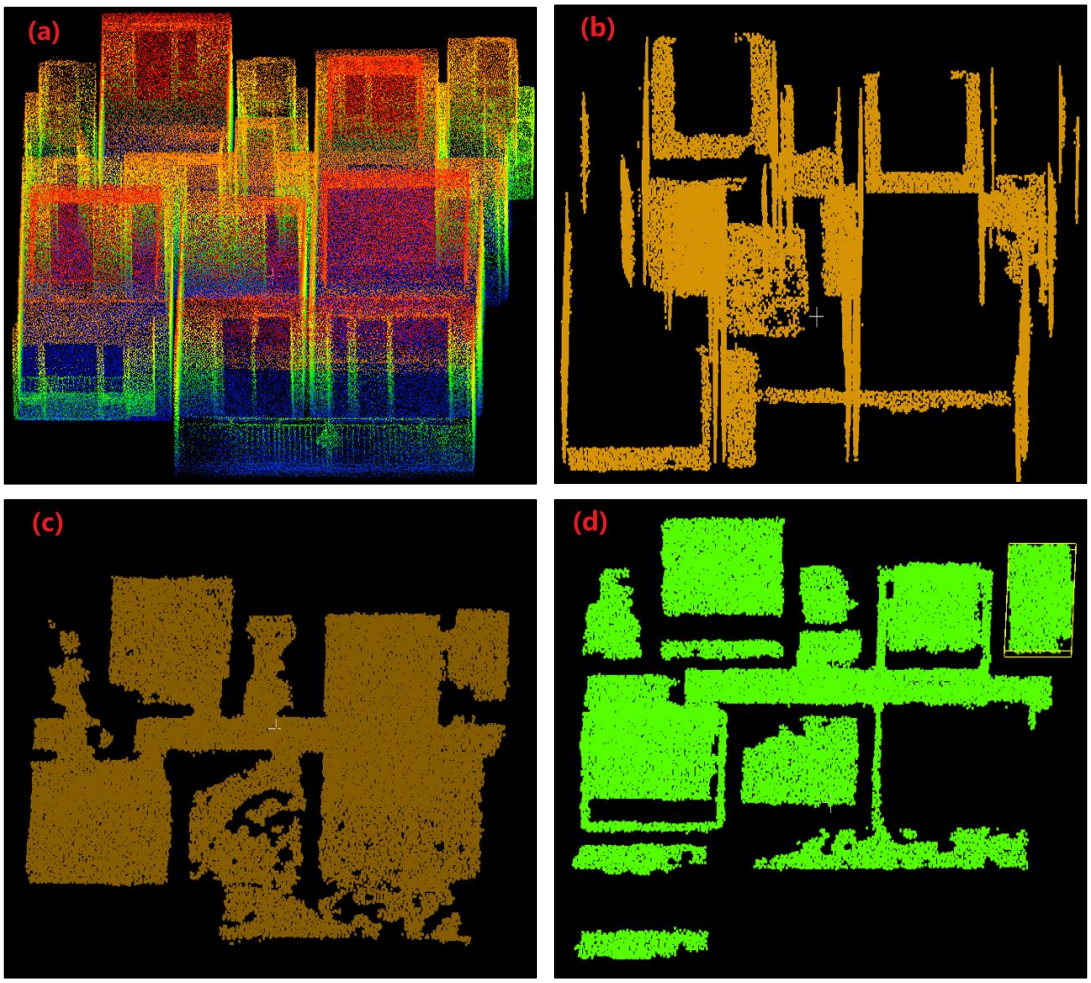


Figure 126. Indoor classification

3. Select one or more of **Wall**, **Floor**, and **Ceiling**, and then click **OK** to extract the clouds of the corresponding types. The walls, floor, and ceiling are extracted as below.



(a) Raw point cloud; (b) walls extracted; (c) floor extracted; (d) ceiling extracted

Figure 127. Indoor classification and extraction

8.3.3 Manual

Description:

Selects portions of a point cloud with different tools and classifies them.

Operation procedure:

1. Select a point cloud file under **Objects**.
2. Click on the **Edit** tab, and the following window appears. The point cloud is rendered by classification.

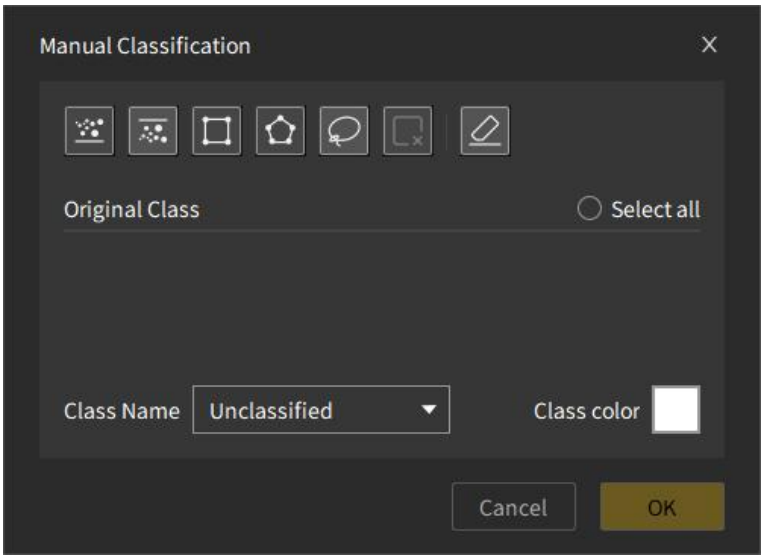


Figure 128. Manual classification (no data selected)

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. The unclassified portion is selected by default. You can also select other classes to change the class settings.

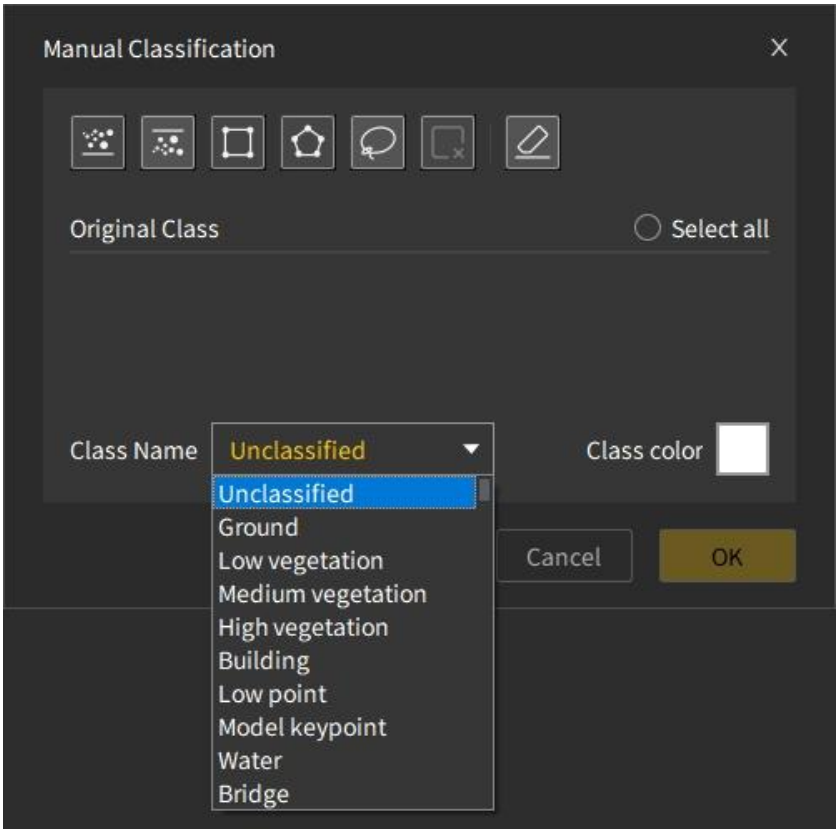


Figure 129. Manual classification (data selected)

6. The class color varies with the class name. Click the class color box to reset the color.
7. Click **OK** to complete the settings. The point cloud becomes unselected and is colored as set in the 3D view area.
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 130. Manual classification result

Note:

- When no portion of the point cloud is selected, the class options are not displayed, **Class Name** is **Unclassified**, the class color is white, and the **OK** button is disabled.
- When the point cloud has multiple classes, classification is to be performed on the unclassified portion by default, and you can select other classes as required.

8.3.4 Extract by Class

Description:

Extract points of a certain class or classes to generate point cloud subsets of different classes.

Operation procedure:

1. Select a point cloud file under **Objects**.
2. Click on the **Edit** tab.
3. The point cloud is rendered by classification. The following window appears.

Figure 131. Extracting the data

4. The above window disappears when the extraction is completed. The extracted point cloud is displayed, and the original point cloud is hidden.
5. The extracted point cloud is named after the class name and is at the same level as the original point cloud on the file list.
   The following figure shows the vegetation and buildings extracted from the point cloud after the manual classification.

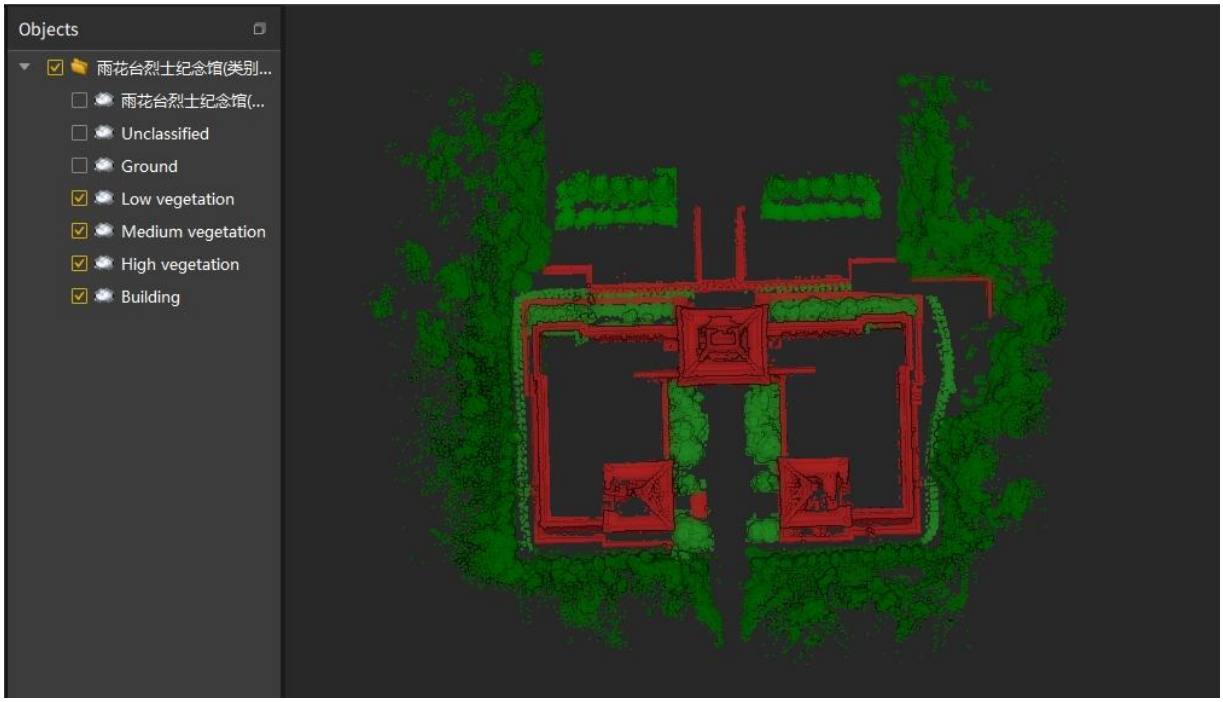


Figure 132. Extraction by class

8.3.5 Floor

Operation procedure:

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

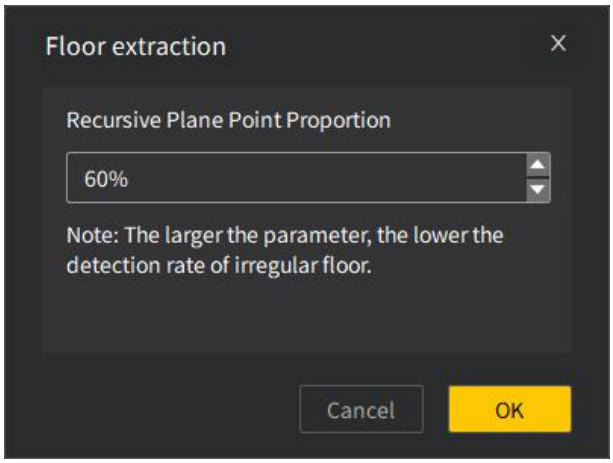


Figure 133. Floor extraction

3. Set the **Recursive Plane Point Proportion** and click **OK** to extract the floor data, as shown below.

   Note: The higher the recursive plane point proportion, the weaker the ability to detect irregular slabs.

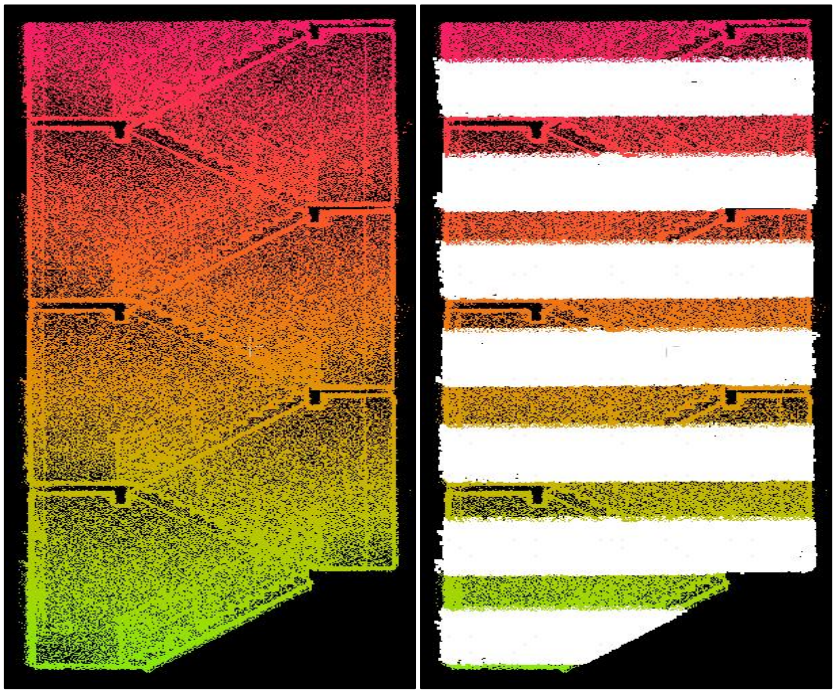


Figure 134. Floor extracted

8.4 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.

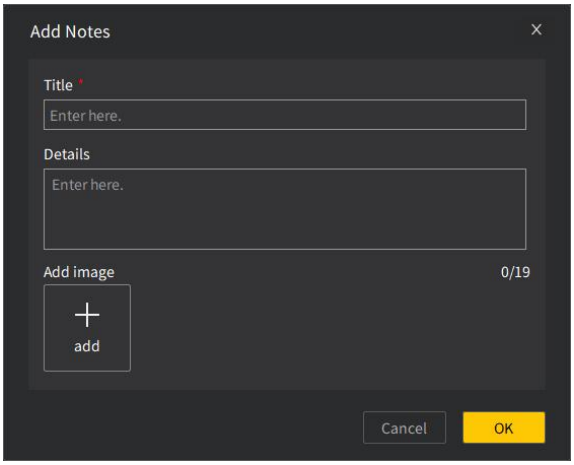


Figure 135. Adding notes

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.

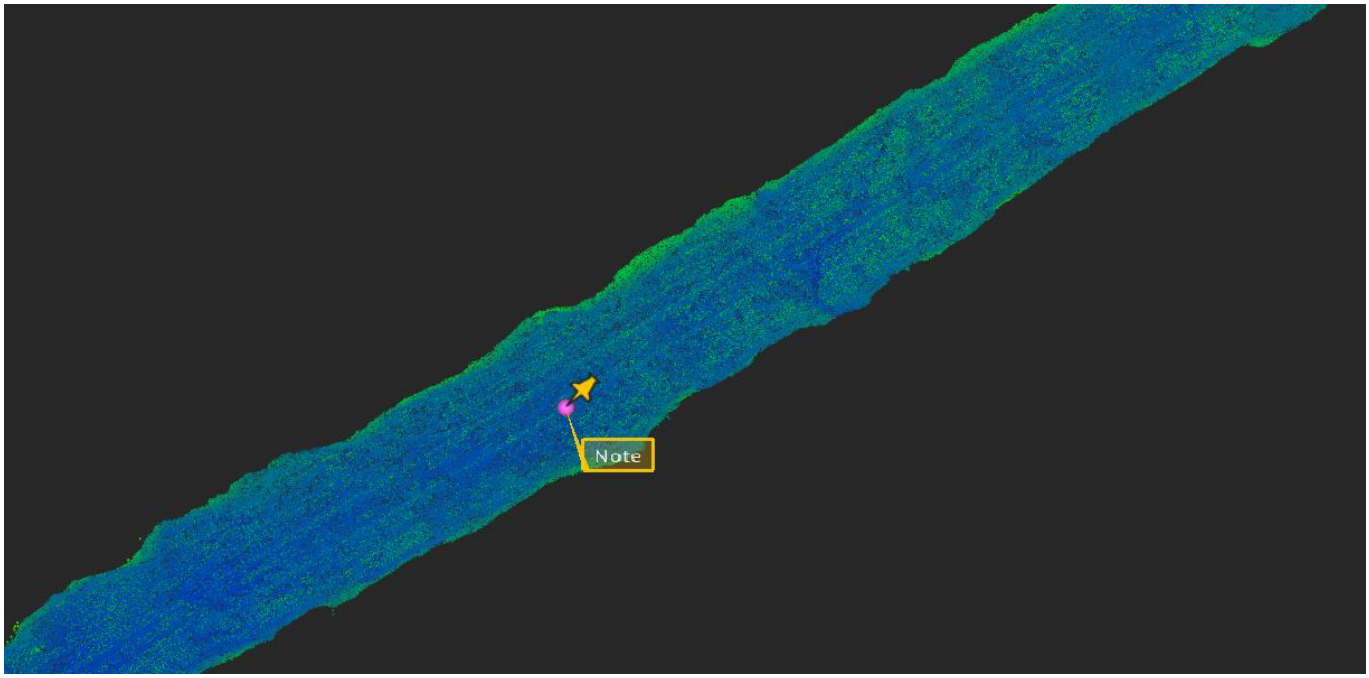


Figure 136. Note label

8.5 Triangular Mesh

8.5.1 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**.

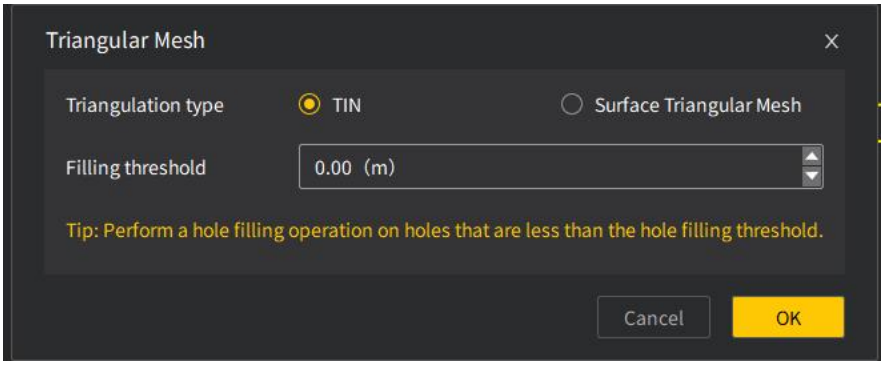


Figure 137. Creating a TIN mesh

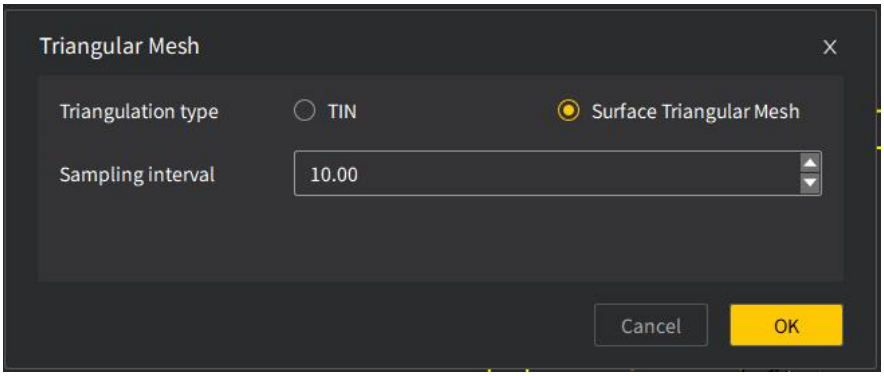


Figure 138. Creating a surface triangular mesh

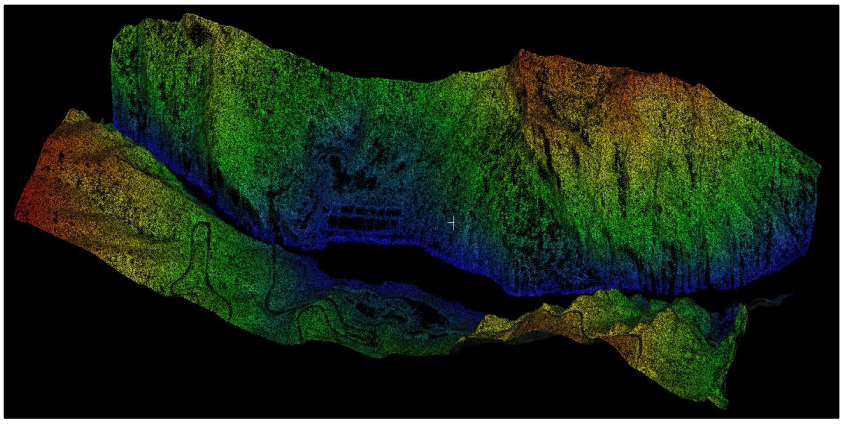


Figure 139. Raw point cloud

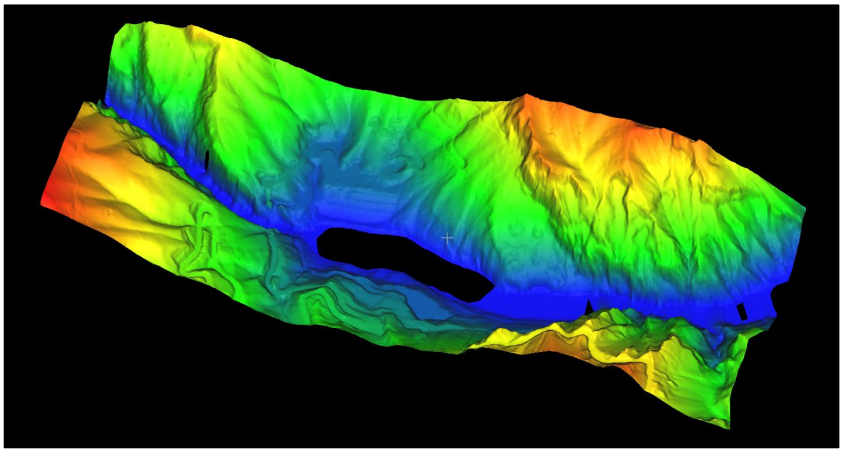


Figure 140. Mesh created

8.5.2 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.

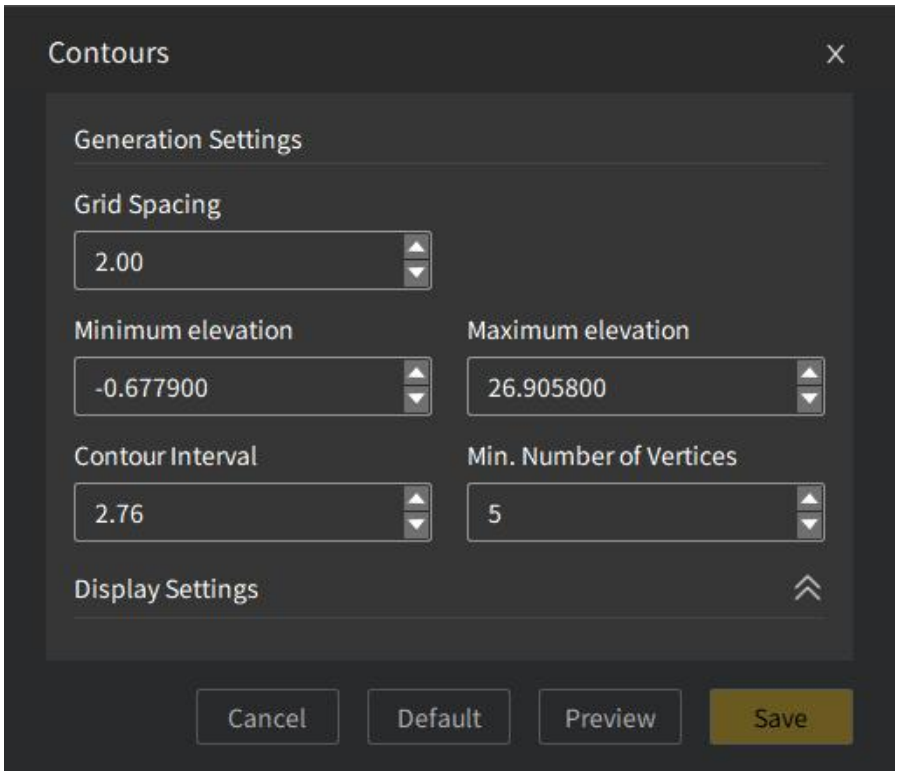


Figure 141. Contour generation settings

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.

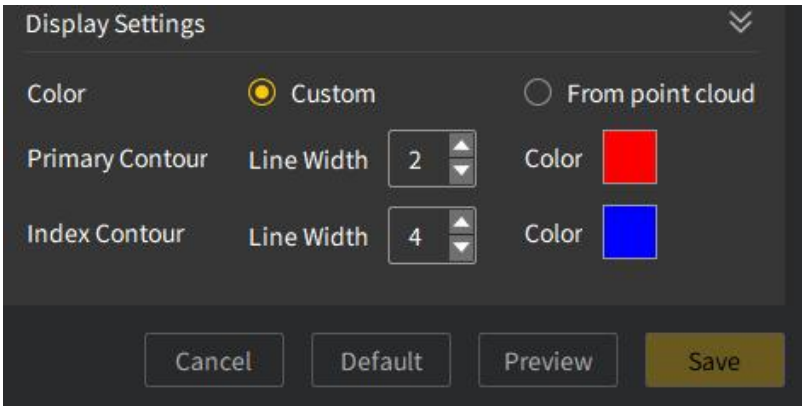


Figure 142. Contour display settings

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

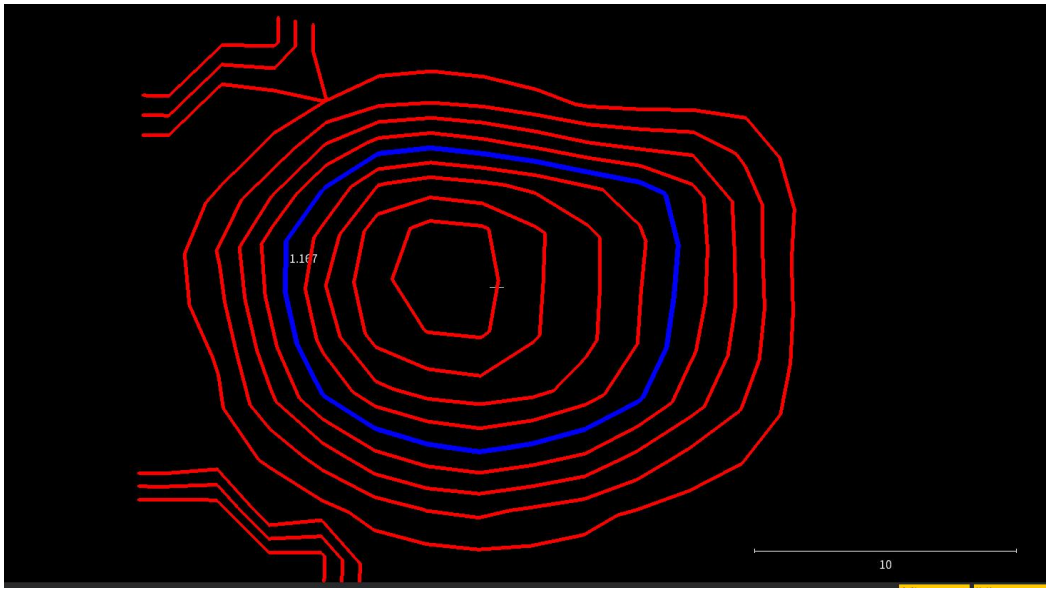


Figure 143. Preview of contour lines in custom colors

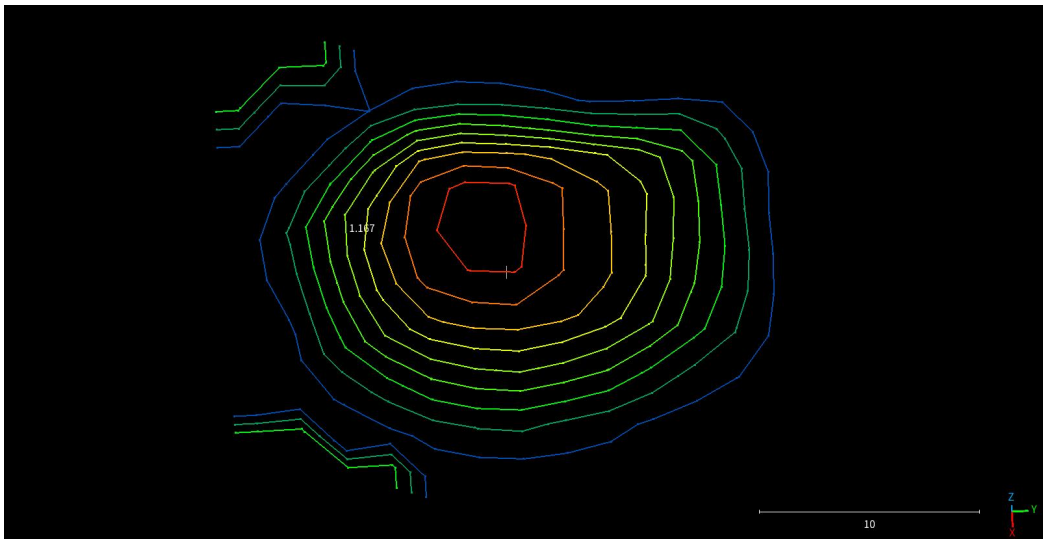


Figure 144. Preview of contour lines in point cloud colors

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.

8.5.3 Fill Holes

Description:

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

Operation procedure:

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

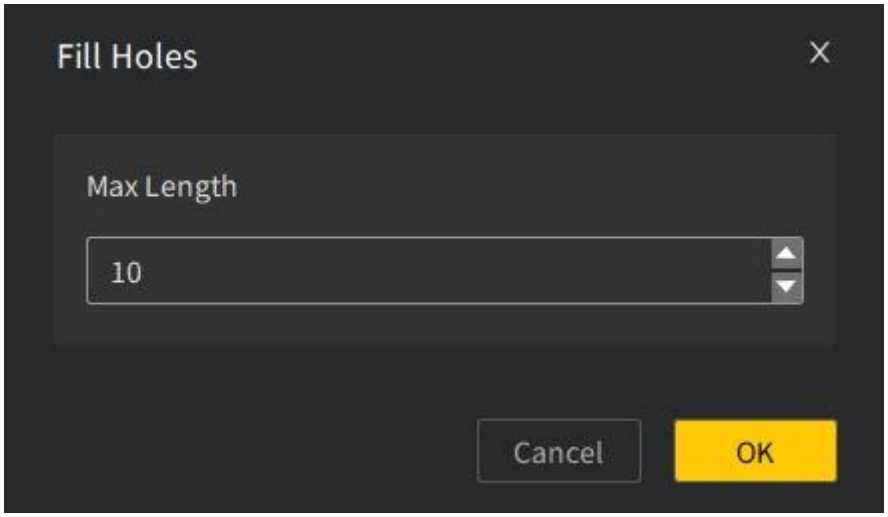


Figure 145. Setting the parameter

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

Figure 146. Filling holes

The following figure shows a triangular mesh with holes filled.

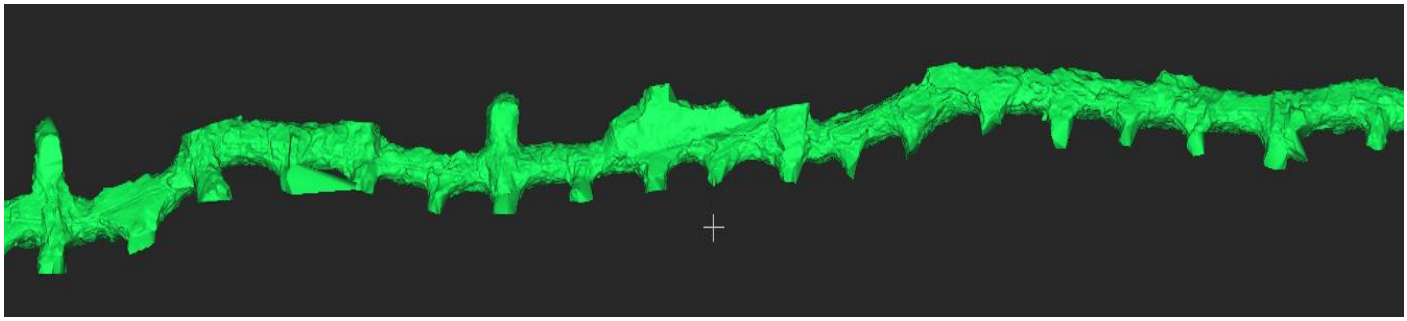


Figure 147. Triangular mesh with holes filled

8.5.4 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 isosurfaces 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.

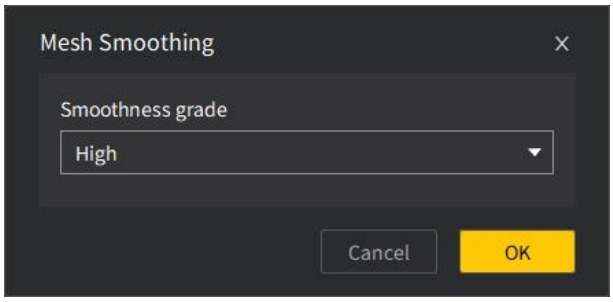


Figure 148. Mesh smoothing

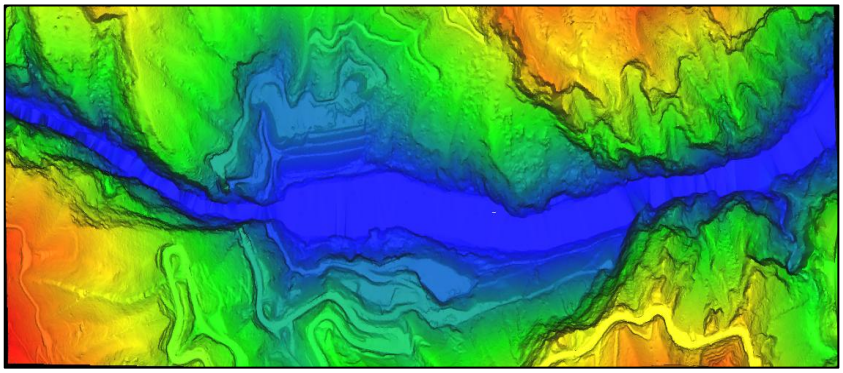


Figure 149. Original mesh

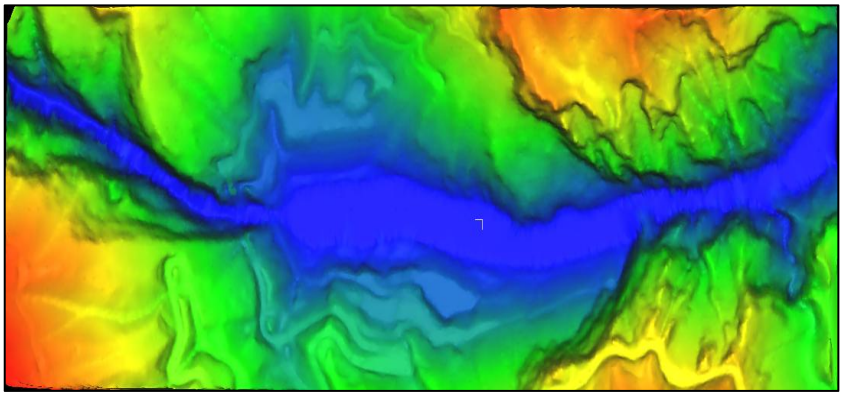


Figure 150. Mesh smoothed

8.5.5 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. Select the sampling mode, set the parameters, and then click **OK**.

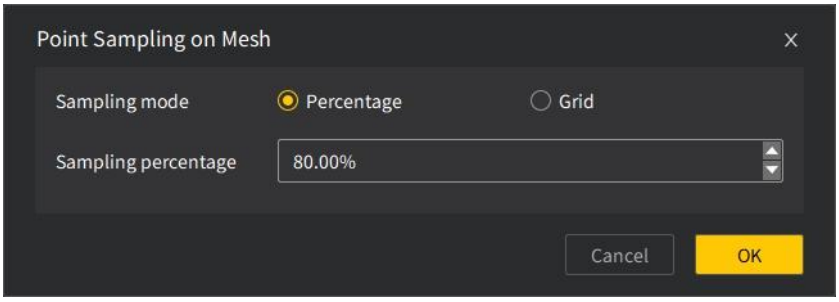
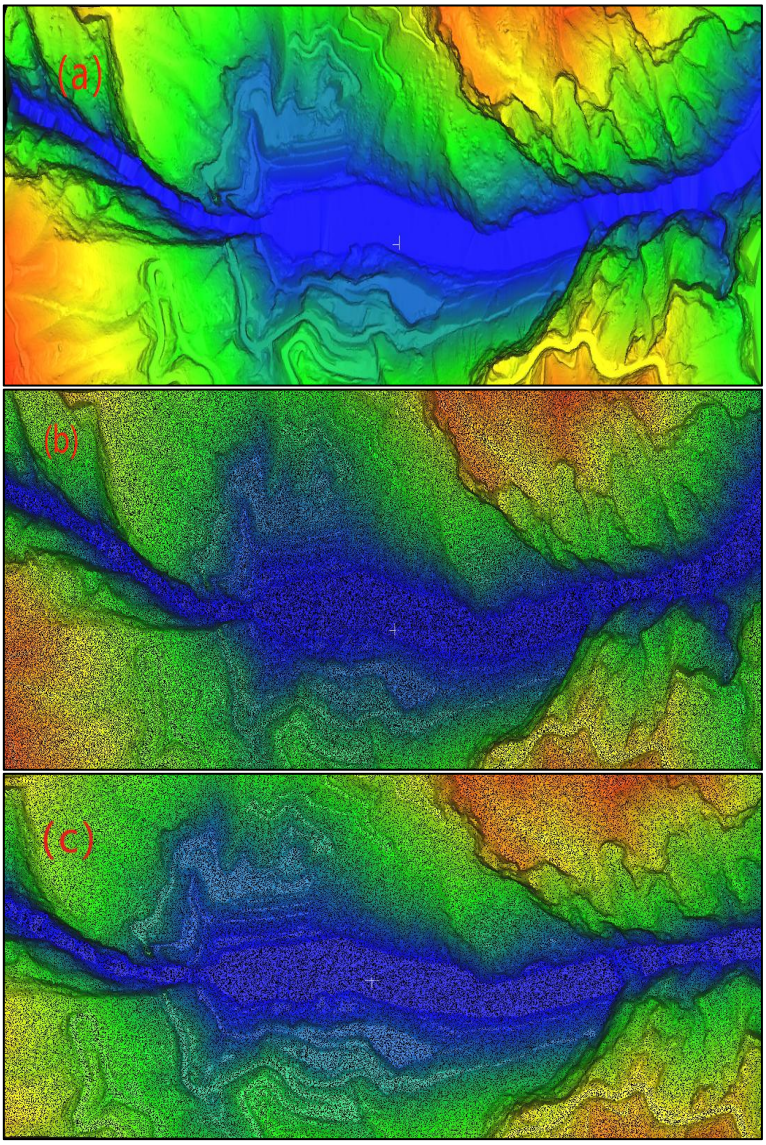


Figure 151. Sampling



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

Figure 152. Sampling results

8.6 Volume Calculation

8.6.1 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.

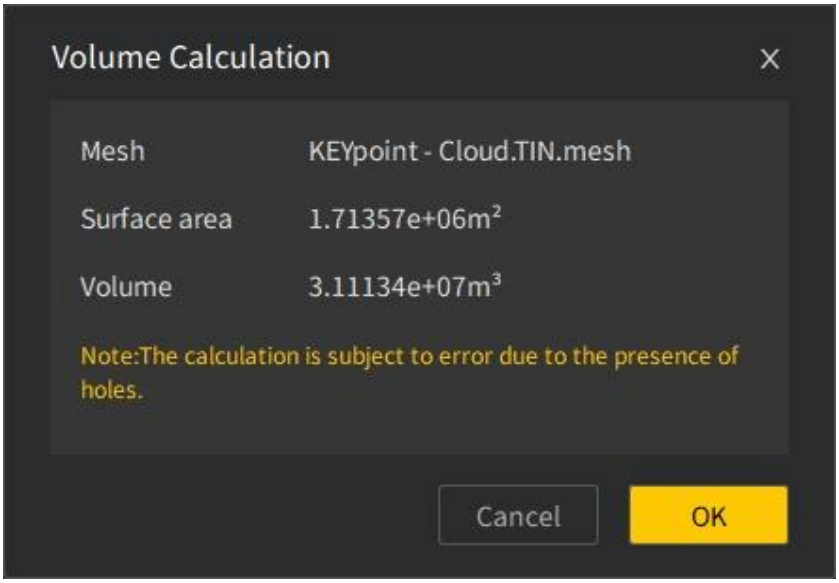


Figure 153. Calculation result (with holes)

Note:

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

8.6.2 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.

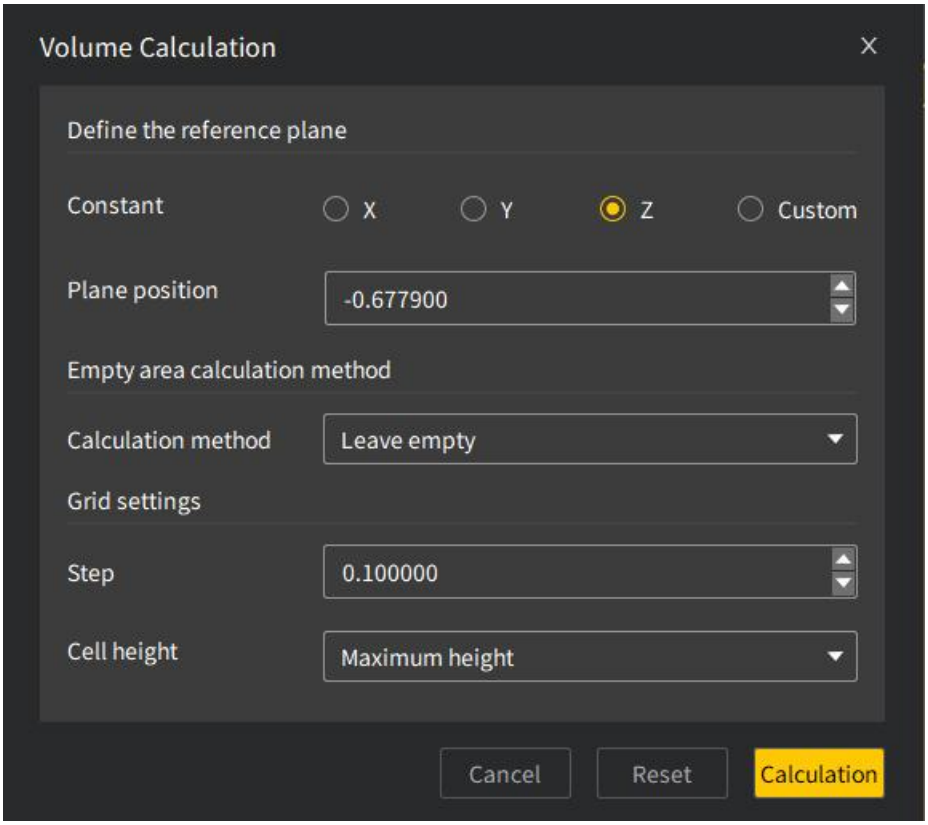


Figure 154. Volume calculation

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.

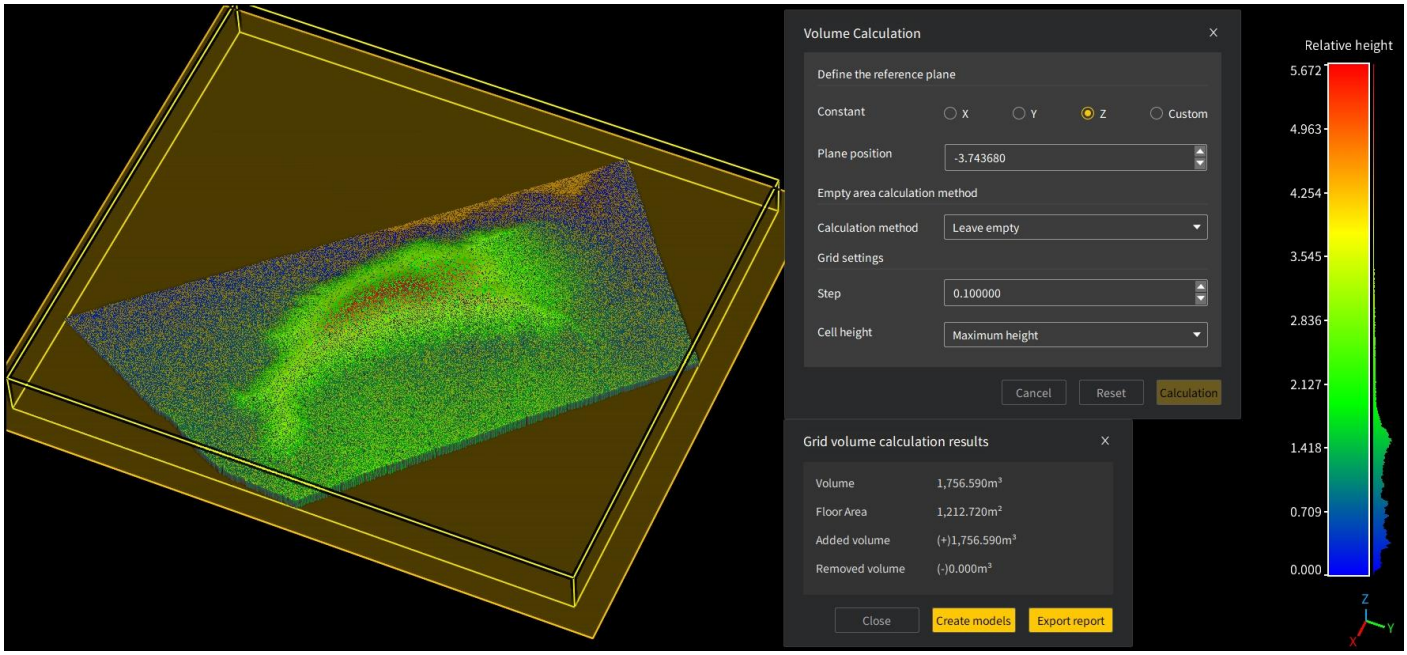


Figure 155. Volume calculation results

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.

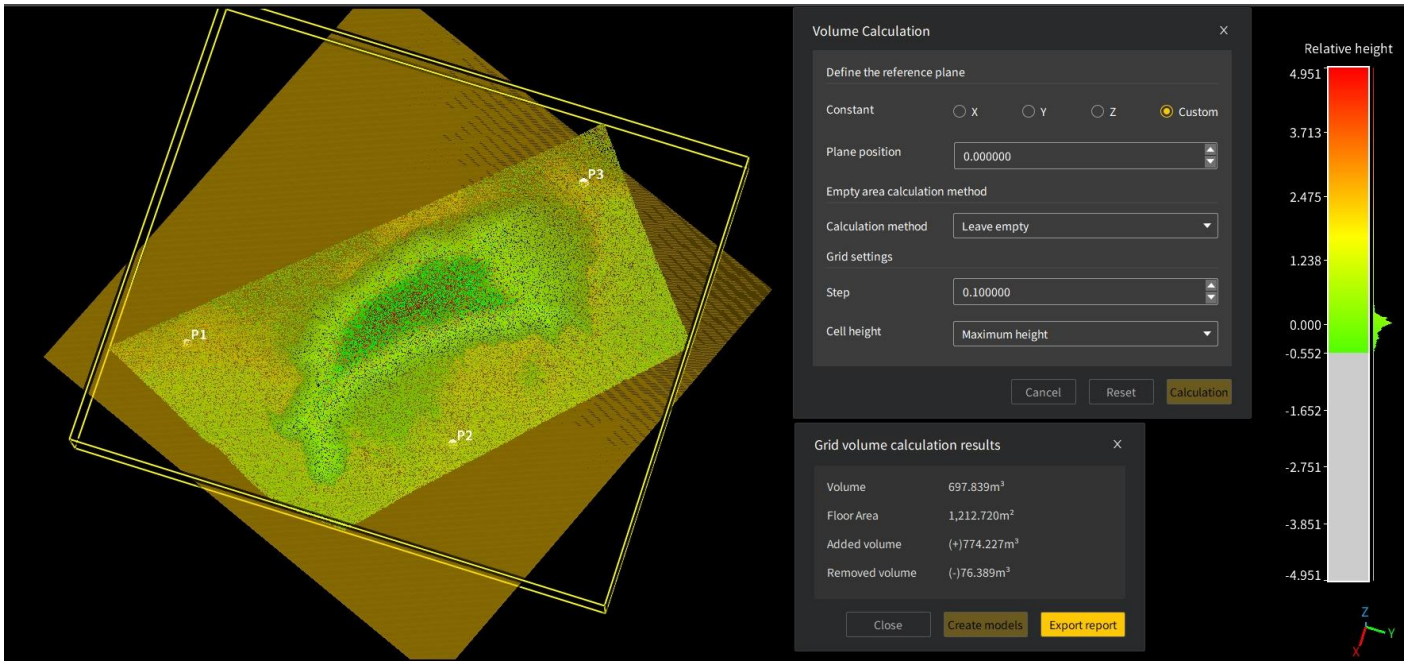


Figure 156. Volume calculation results with a custom constant

8.6.3 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.

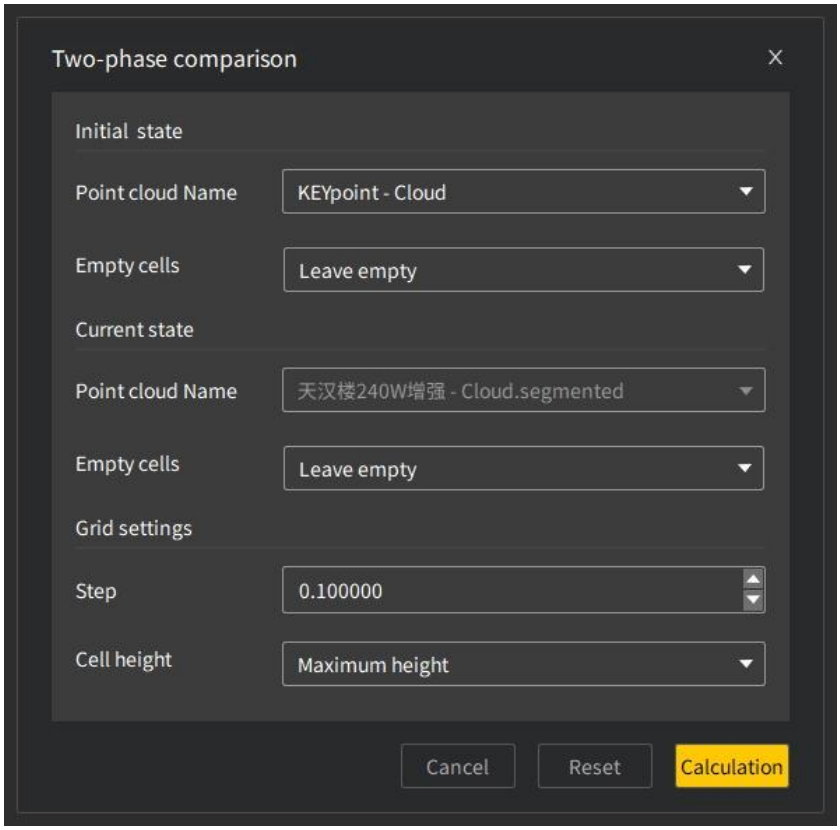


Figure 157. Two-phase comparison settings

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.

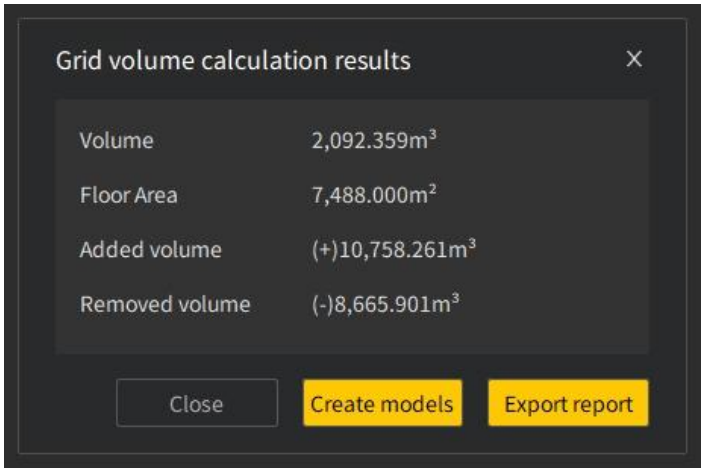


Figure 158. Two-phase comparison results

9 Display

9.1 View

9.1.1 Background

Description:

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

Operation procedure:

1. 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.

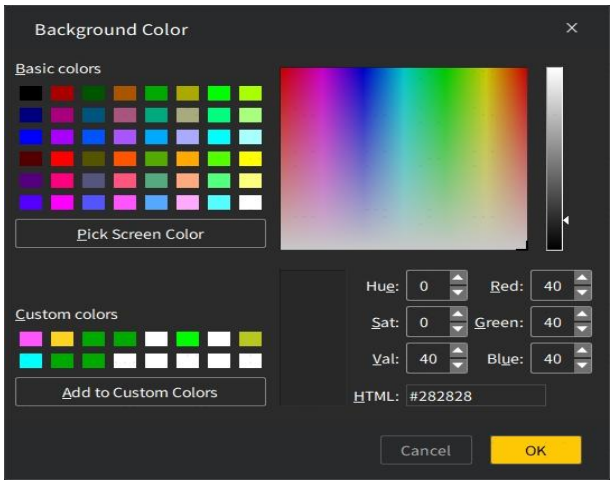


Figure 159. Changing the background color

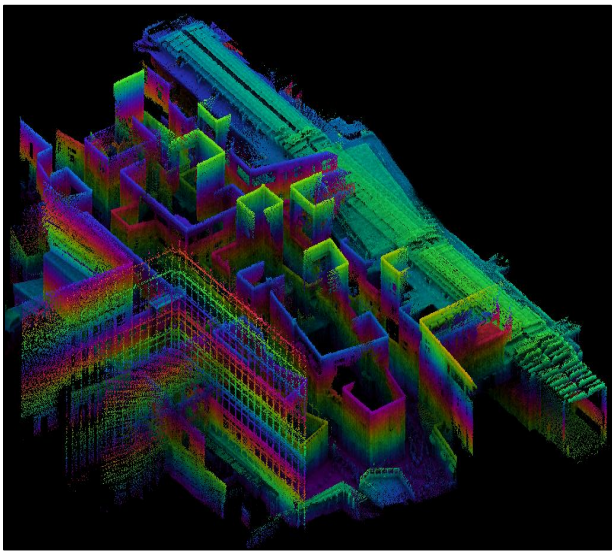


Figure 160. Default background color

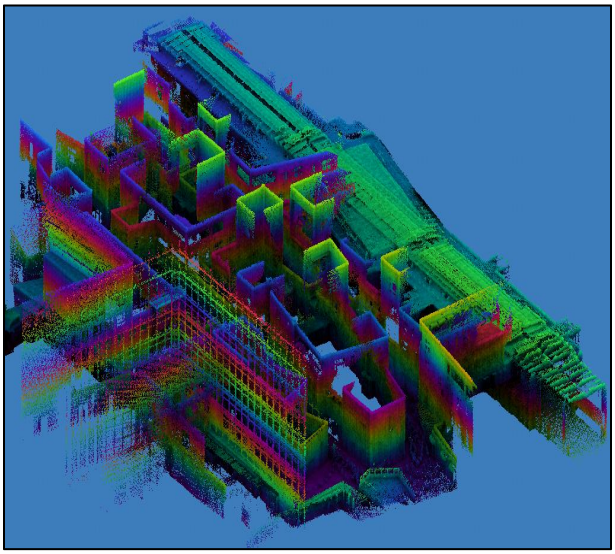


Figure 161. Background color changed

9.1.2 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:

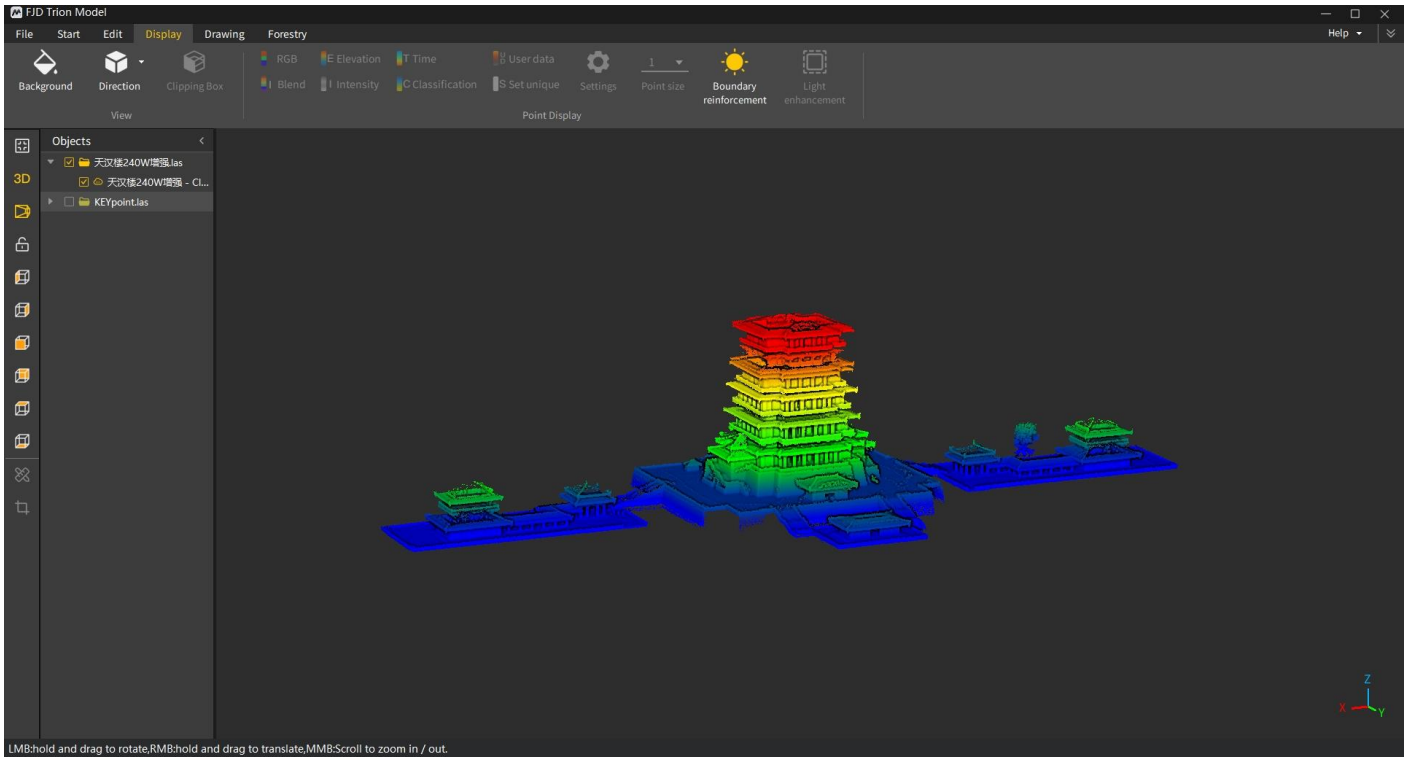


Figure 162. 3D view

- 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:

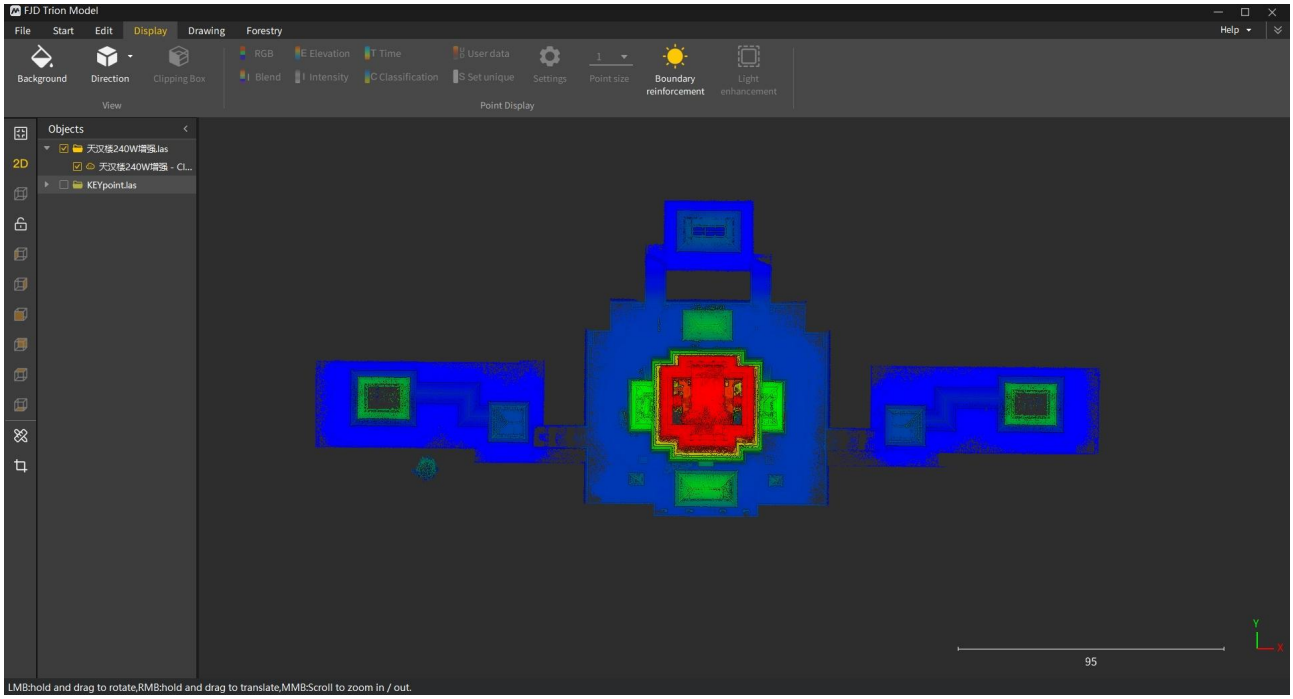


Figure 163. 2D view

- 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:

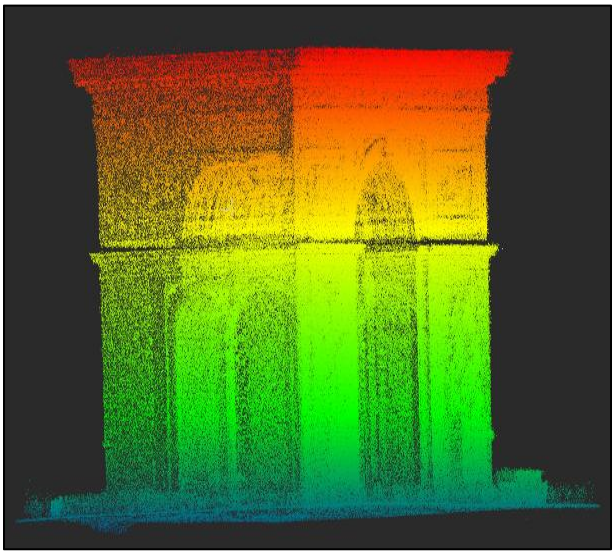


Figure 164. Left side view

- 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:

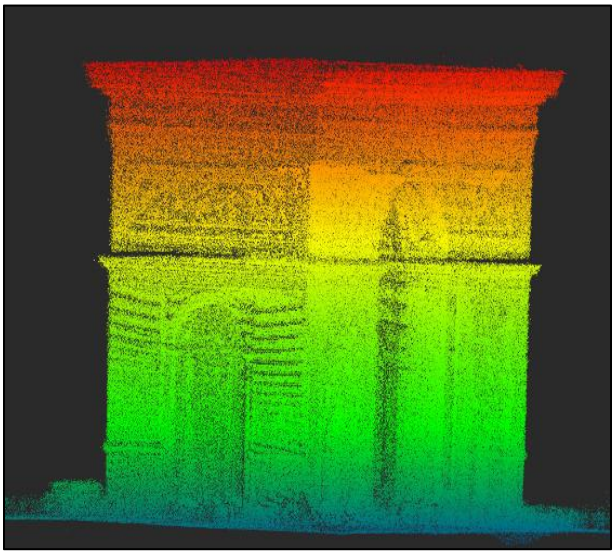


Figure 165. Right side view

- 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:

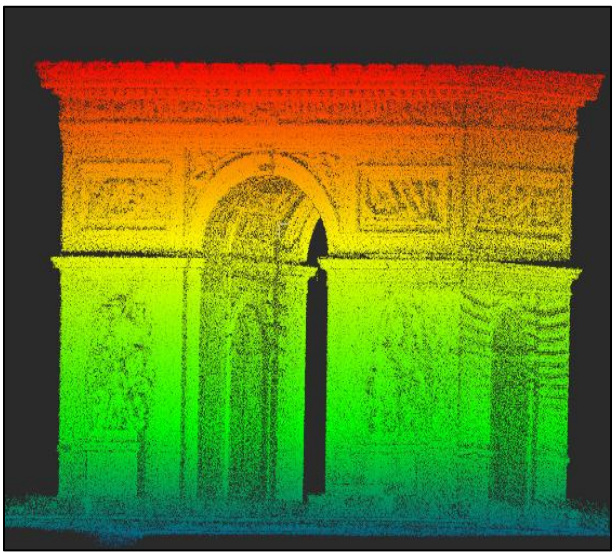


Figure 166. Front view

- 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:

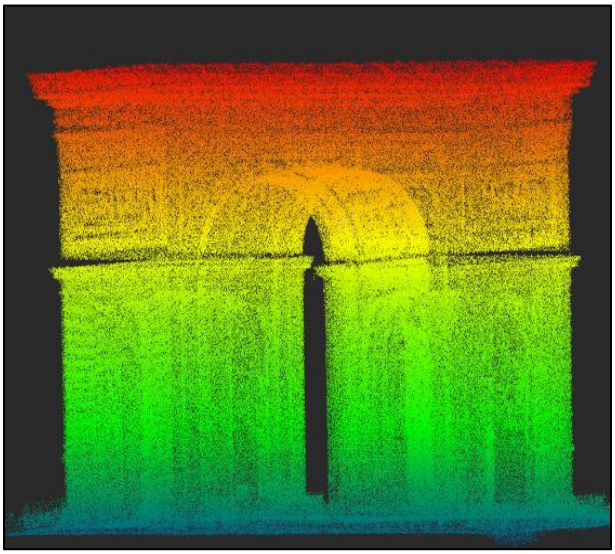


Figure 167. Back view

- 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:

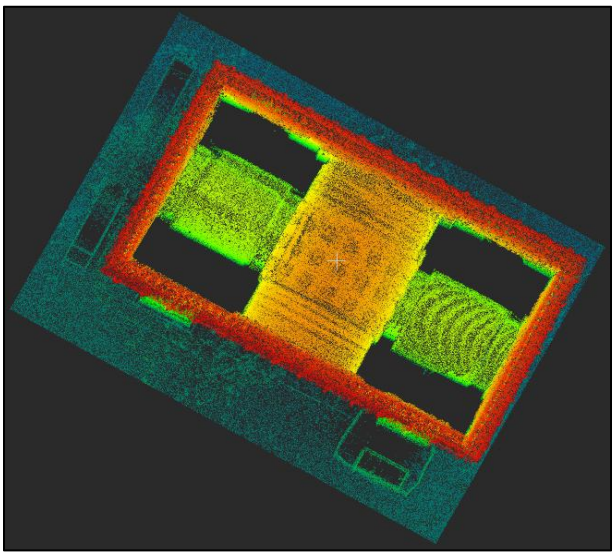


Figure 168. Top view

- 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:

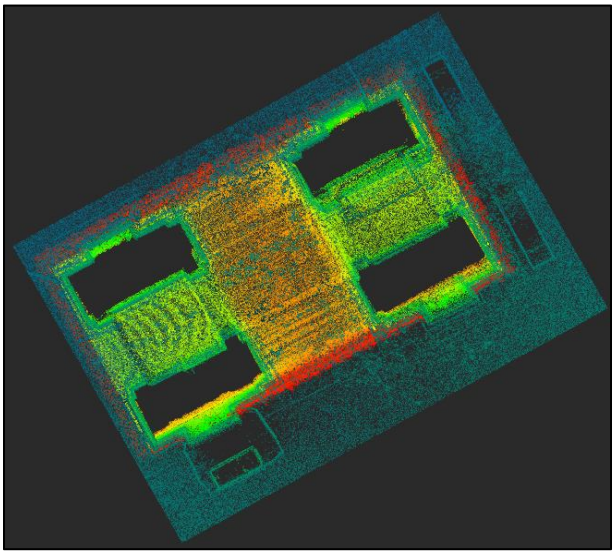


Figure 169. Bottom view

- 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.

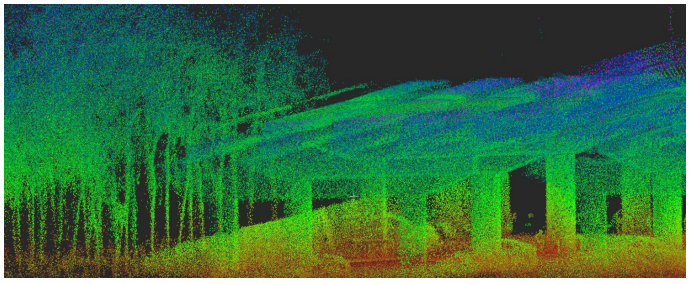


Figure 170. Orthographic projection effect

- 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.

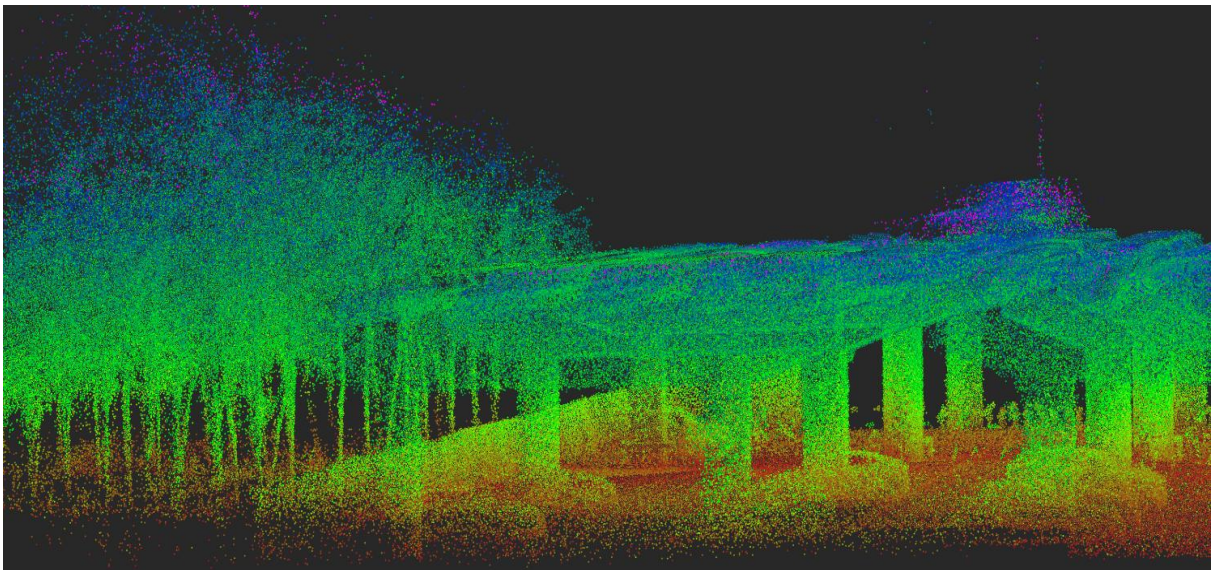


Figure 171. Perspective projection effect

Note:

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

9.1.3 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.

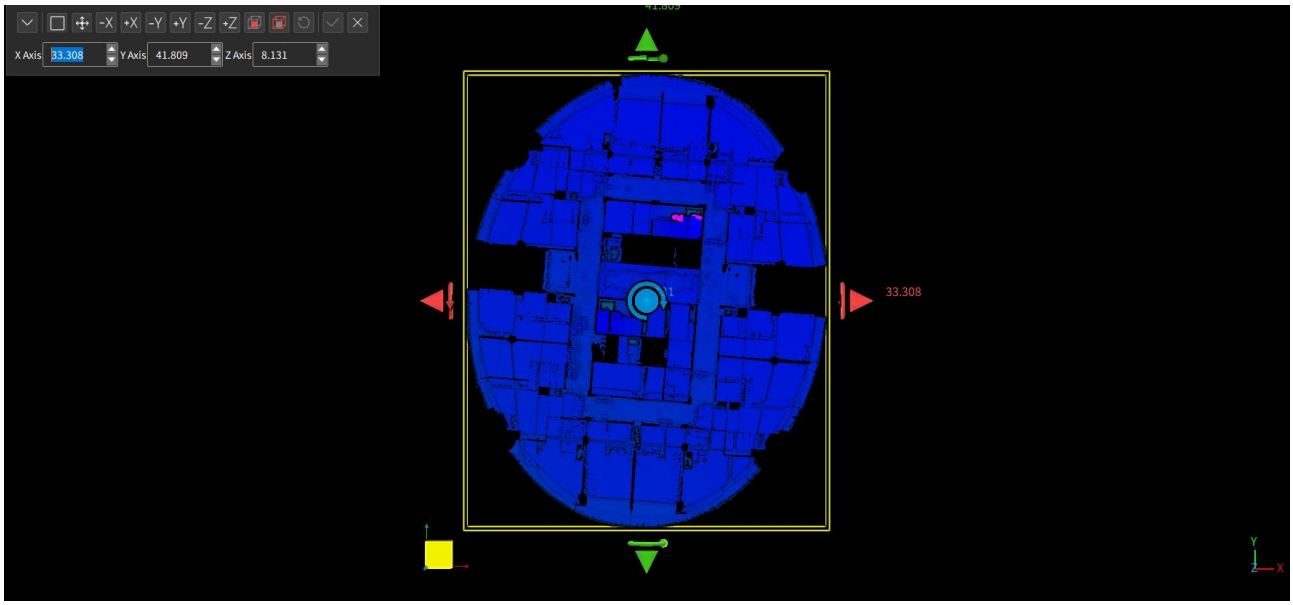


Figure 172. Initial clipping box

3. Click to set the box dimensions. Click -X +X -Y +Y -Z +Z 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.

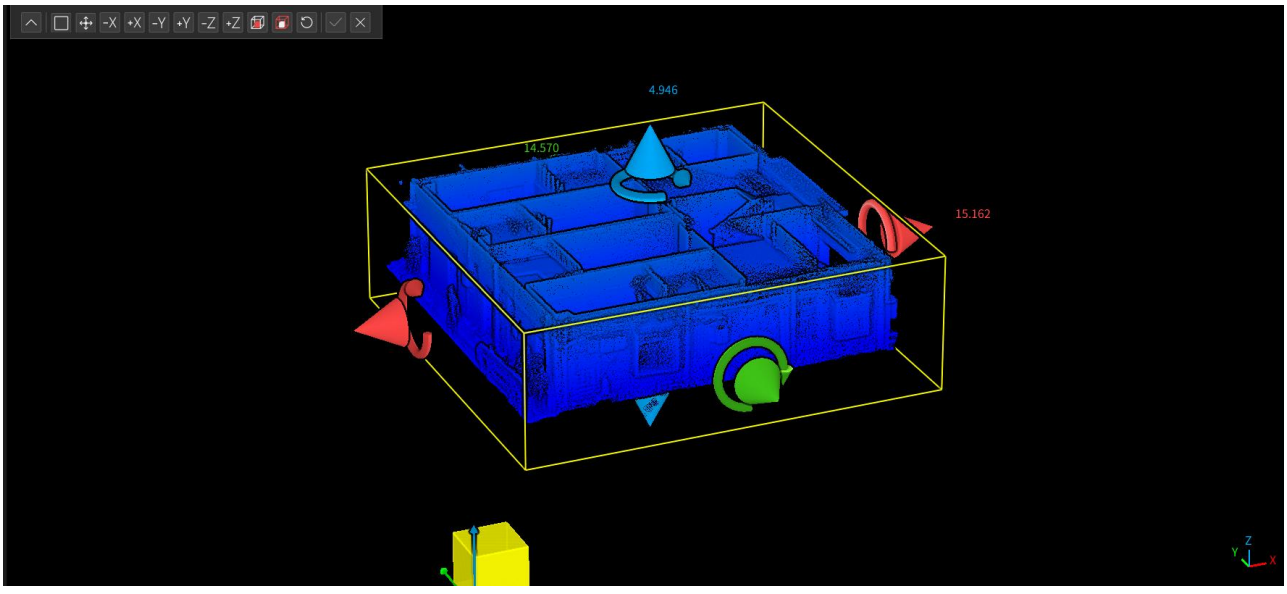


Figure 173. Effect of hiding the top of the point cloud

4. 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.

9.2 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.

9.2.1 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.

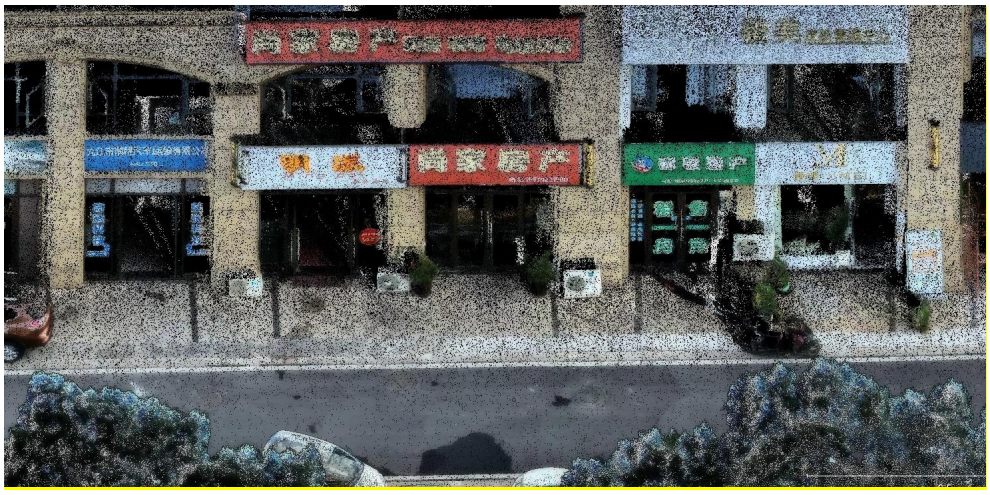


Figure 174. RGB display

9.2.2 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.

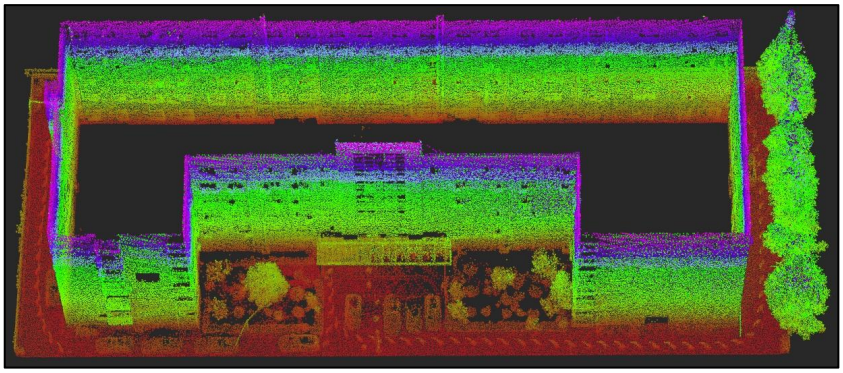


Figure 175. Rendering by elevation

9.2.3 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 176. Rendering by intensity

9.2.4 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.

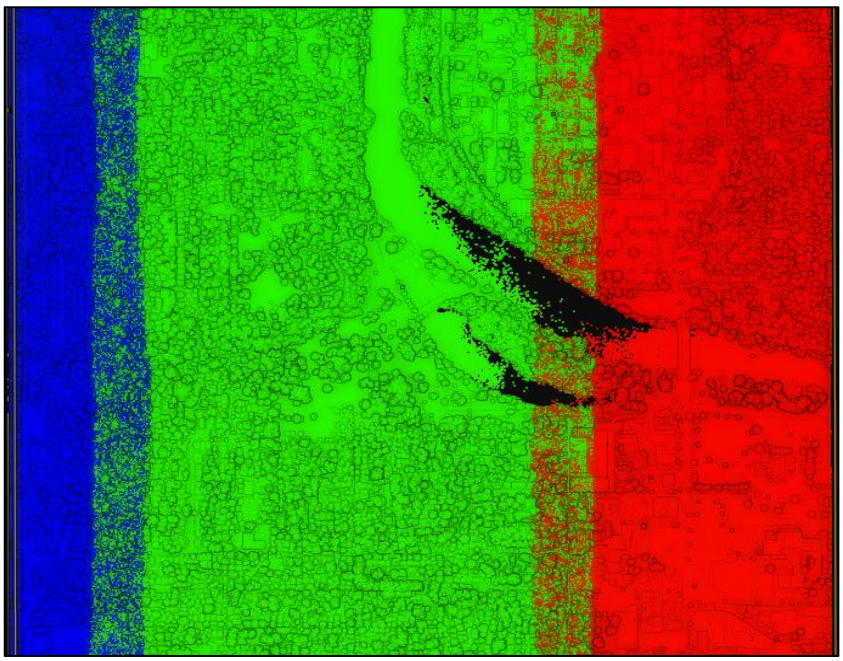


Figure 177. Rendering by time

9.2.5 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.

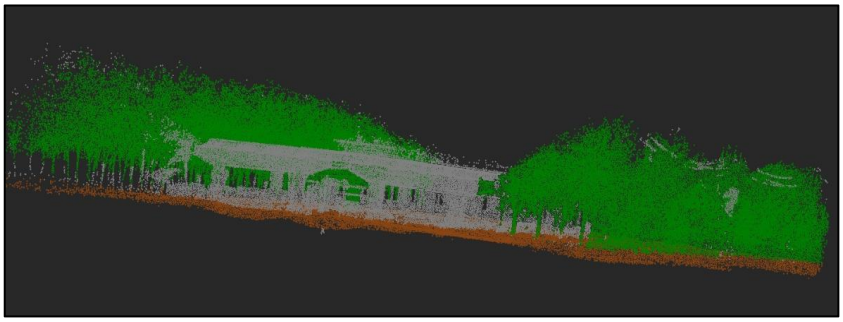


Figure 178. Rendering by classification

9.2.6 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.

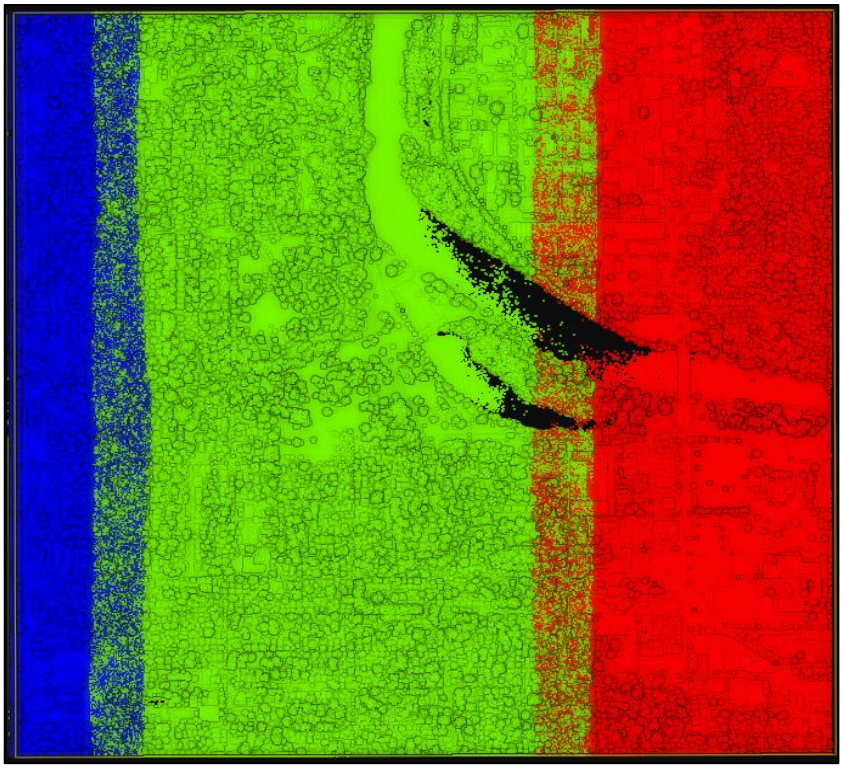


Figure 179. Rendering by user data

9.2.7 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.

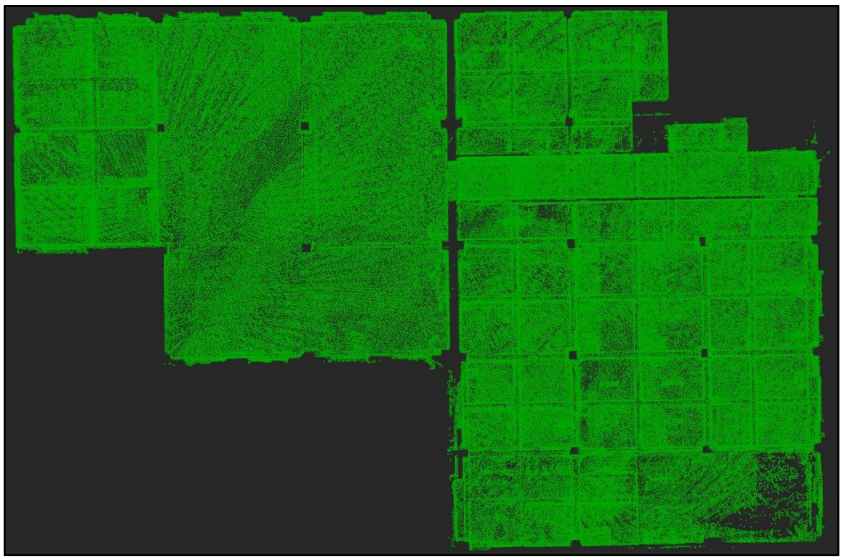


Figure 180. Rendering in a specified color

9.2.8 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.

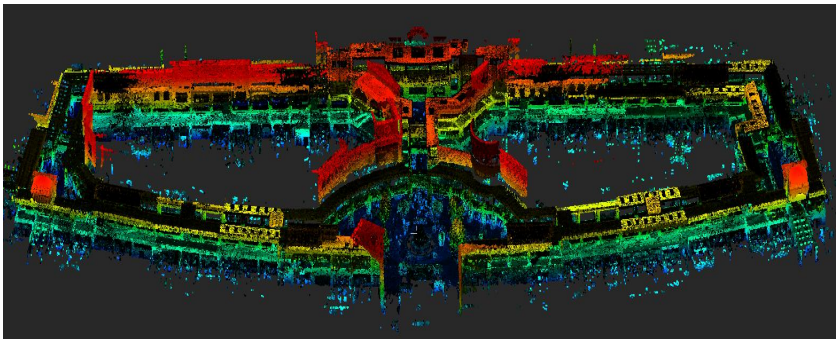


Figure 181. Rendering by elevation and intensity

9.2.9 Point Size

Description:

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

Operation procedure:

1. 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.

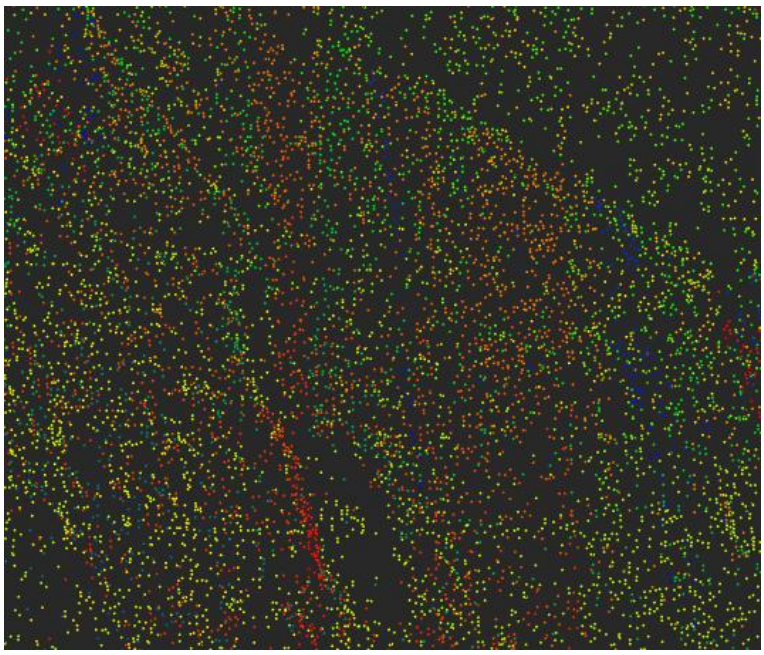


Figure 182. Before changing the point size

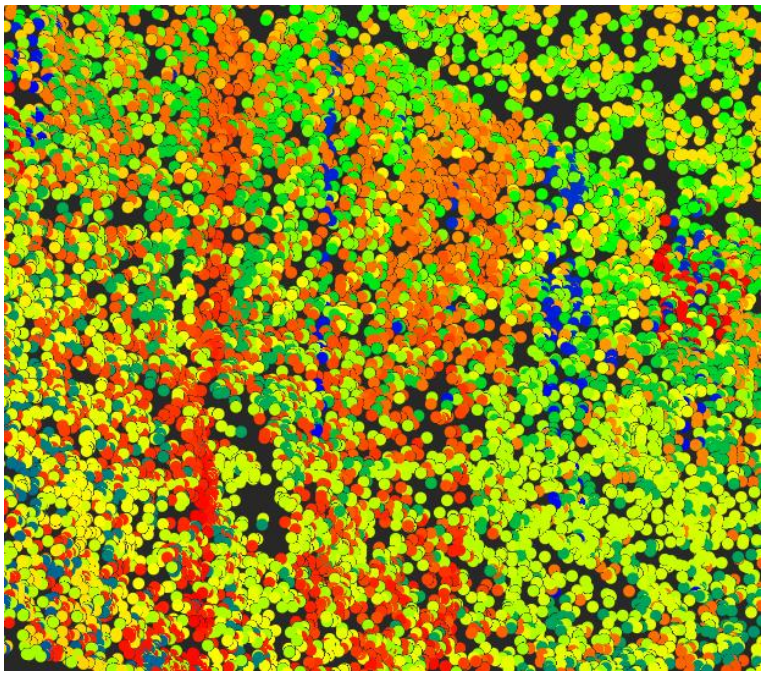


Figure 183. After changing the point size

9.2.10 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.

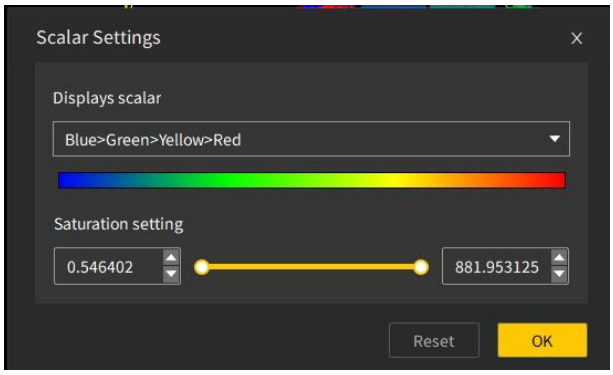


Figure 184. Scalar settings for rendering by time

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.

9.2.11 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.

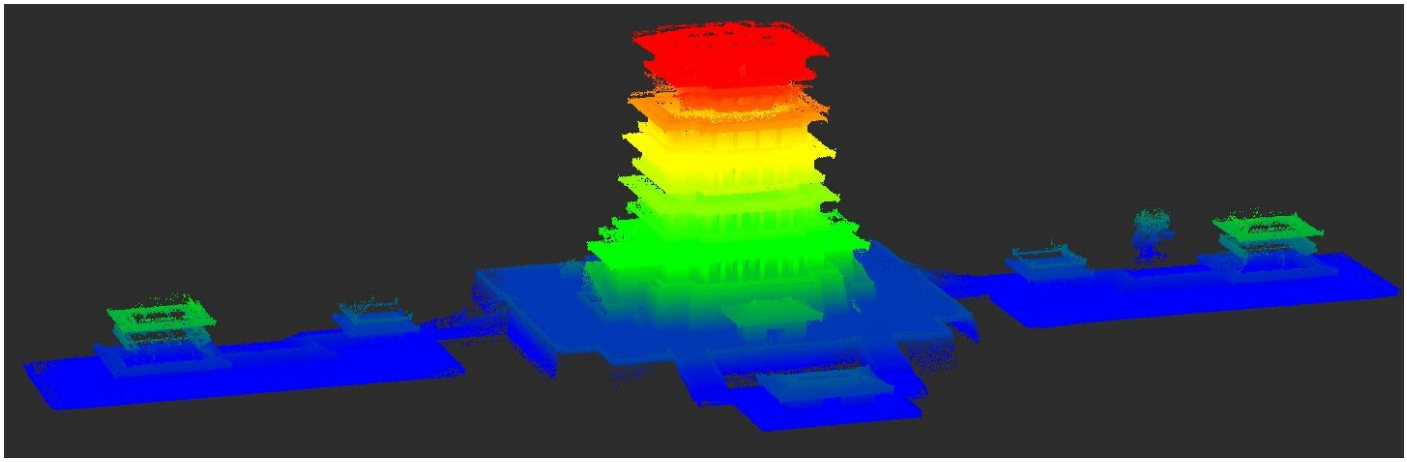


Figure 185. Before boundary reinforcement

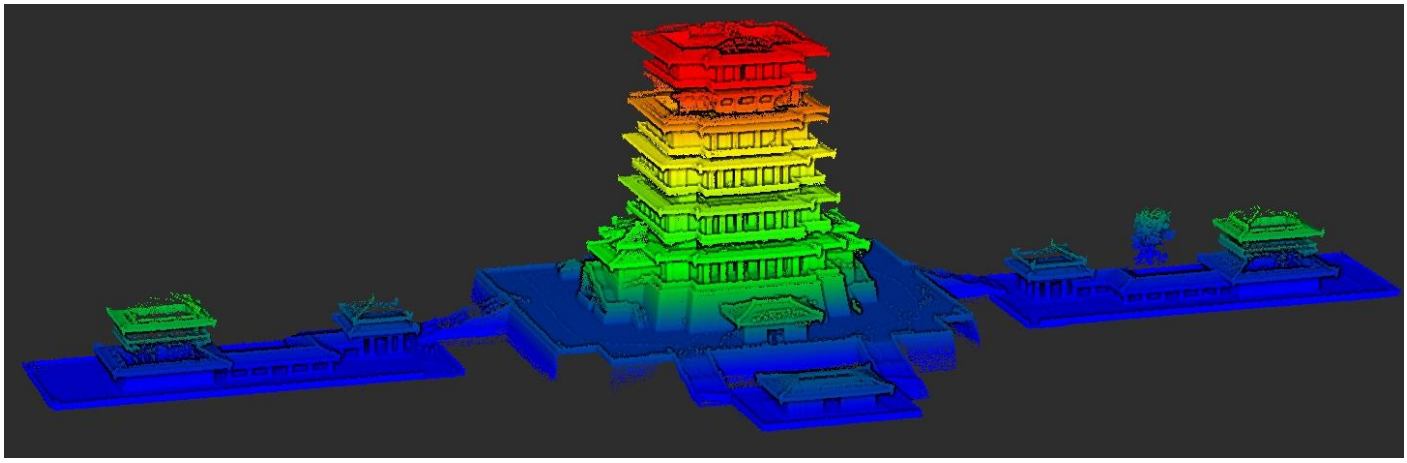


Figure 186. After boundary reinforcement

9.2.12 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.

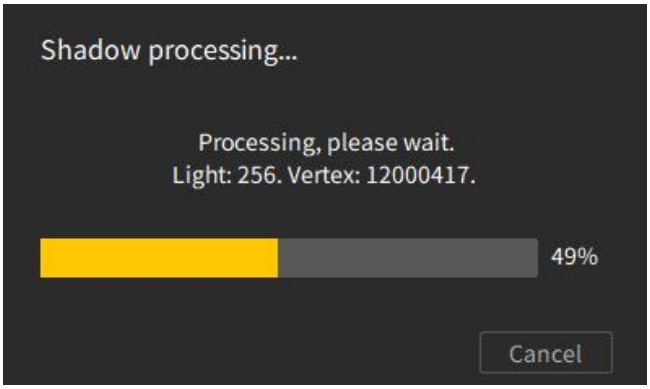


Figure 187. Shadow processing progress window

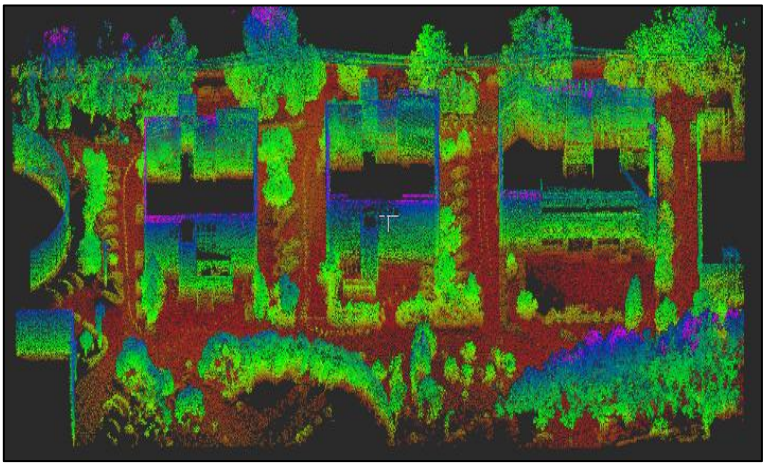


Figure 188. Before light enhancement

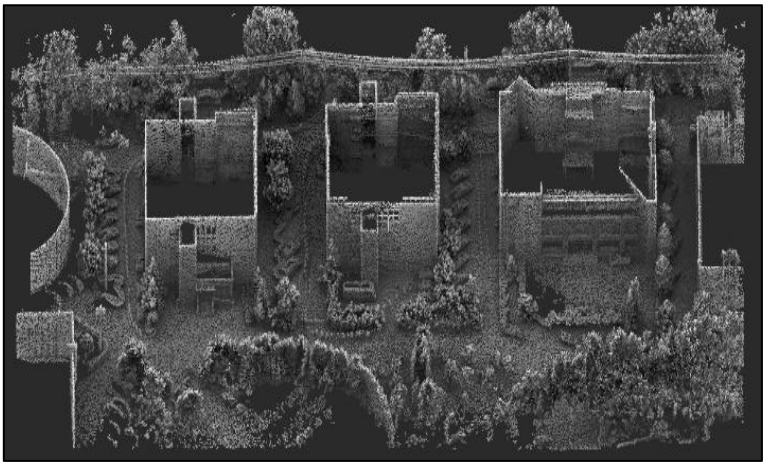


Figure 189. After light enhancement

10 Drawing

10.1 Slice

10.1.1 Slice Plane

Description:

Slices a point cloud along an axis or vertically to the screen to obtain a plane of a certain thickness.

Operation procedure:

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

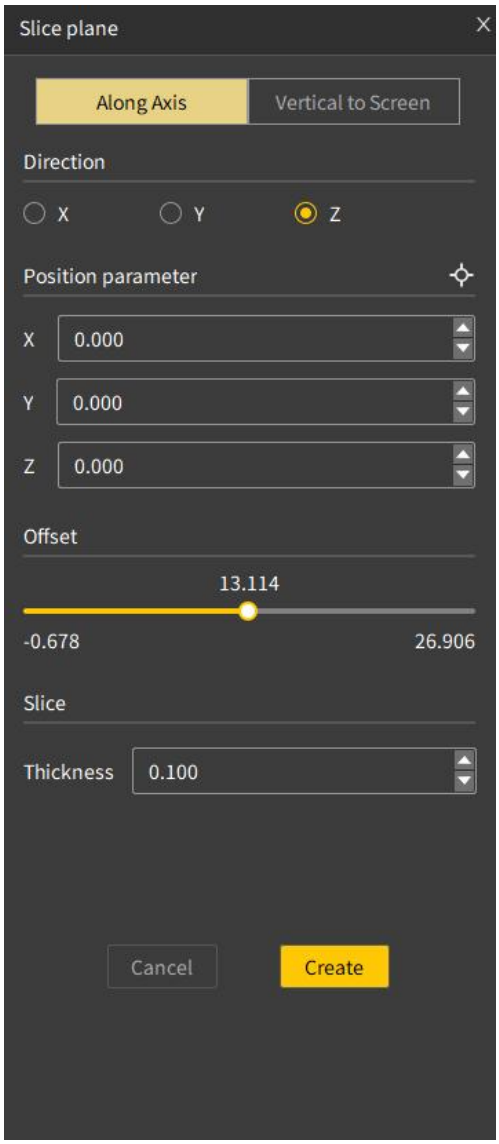


Figure 190. Plane slicing window

3. Click **Along Axis**, select **X**, **Y**, or **Z** as the slicing direction, adjust the plane position by entering values or dragging the slider, set the plane thickness, and click **Create**. The result is shown below. A **Slice plane** file is added to the file list under **Objects**.

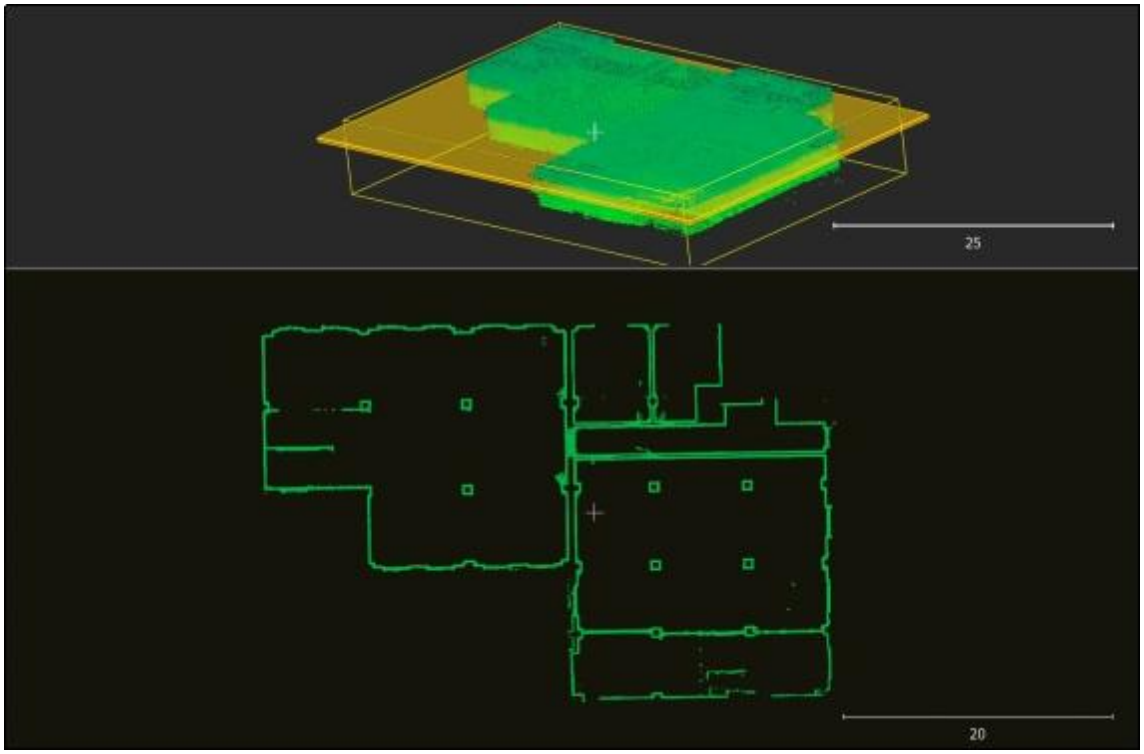


Figure 191. Result of slicing along an axis

4. Select **Vertical to Screen**, and select two points on the point cloud to create a slice. The slice is displayed in the 2D view area. Drag the slider to set the slice offset, and change the thickness value to adjust its thickness. Double-click to modify the corresponding point coordinate in the table as shown below. Click **Create** to obtain the slice, and the file list is updated under **Objects**.

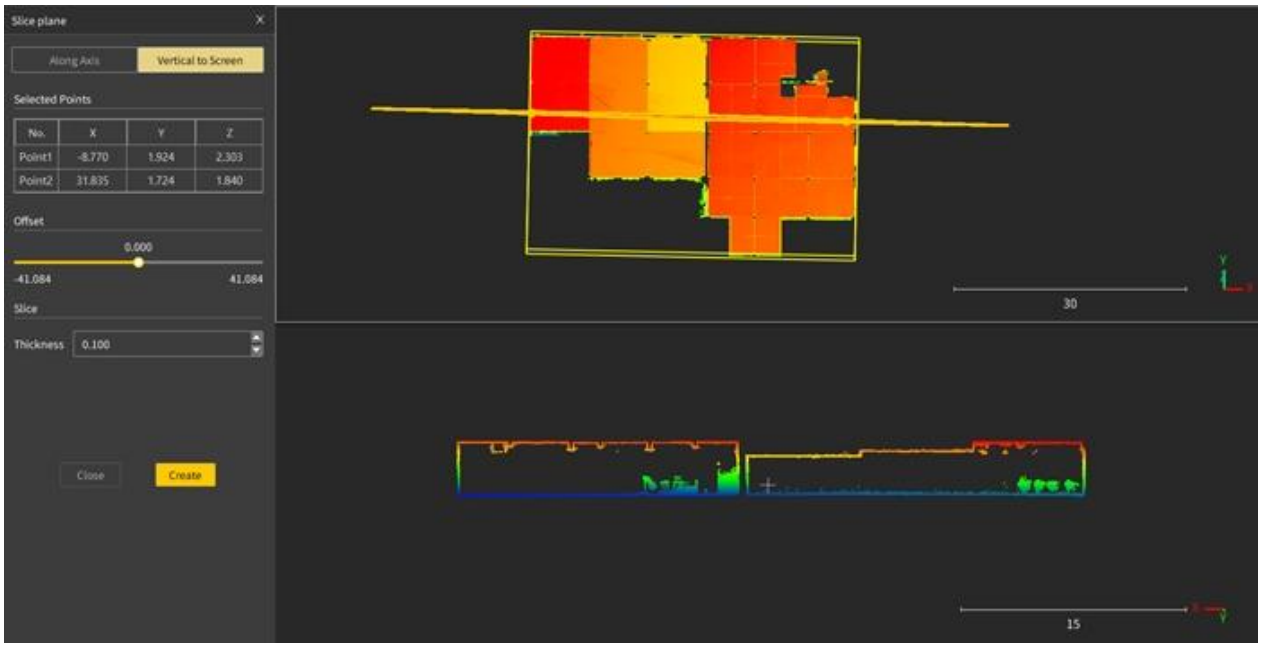


Figure 192. Result of slicing vertically to the screen

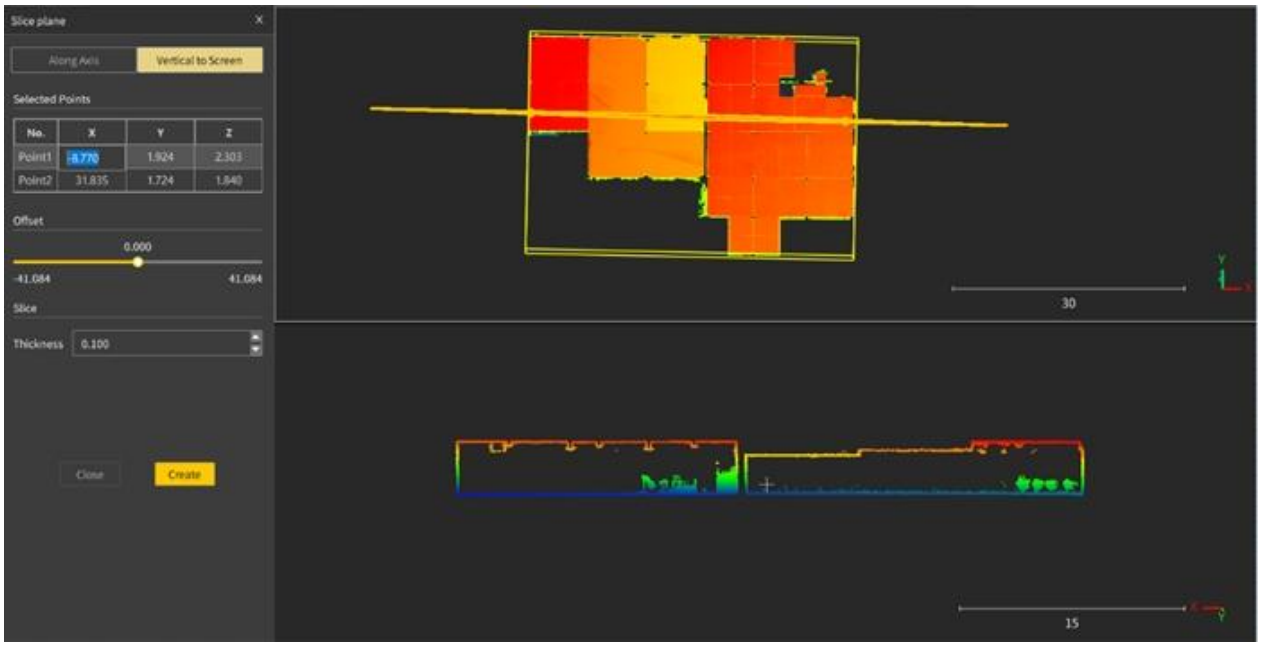


Figure 193. Editing coordinates

10.2 Drawing Management

10.2.1 Draw

Description:

Enables drawing related features to draw 2D vector graphics.

Operation procedure:

1. Select a "Slice plane" or "Vectors" file under **Objects**.
2. Click to enable drawing related features. The slice is projected into a 2D view, where you can draw vector graphics.

10.2.2 Save

Description:

Saves the vector graphics.

10.2.3 Undo

Description:

Undoes the last drawing action. Click it multiple times to undo more than one action.

10.2.4 Redo

Description:

Redoes the last undone action. Click it multiple times to redo more than one action.

10.2.5 Delete

Description:

Deletes the selected vector graphics.

10.2.6 Exit

Description:

Exits the vector graphic drawing mode. Then, the graphic drawn is shown in a 3D view and cannot be edited.

10.3 Fitting

10.3.1 Extract Contours

Description:

Obtains 2D contours of a slice of a point cloud through automatic fitting.

Operation procedure:

1. Select a "Slice plane" file under **Objects**, click the **Drawing** tab, and click **Draw**.
2. Click , and the 3D view is switched to the 2D view. The 2D view is the projection plane perpendicular to the slice.
3. The message "Waiting..." appears during the extraction and disappears when the extraction is completed. Click **Save**, and a file named "Vectors" is added to the file list under **Objects**. The current view is also updated.

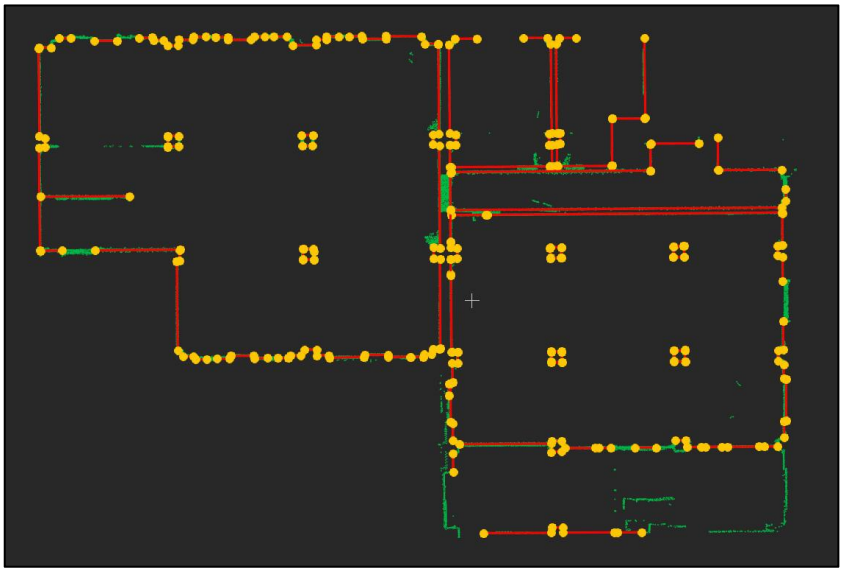


Figure 194. Extracted contours

10.4 Graphic Drawing

10.4.1 Straight Line

Description:

Draws a straight line.

Operation procedure:

1. Click in the **Graphic Drawing** area, and the icon is highlighted.
2. Move the cursor to the view area to select points.
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.

10.4.2 Polyline

Description:

Draws a polyline.

Operation procedure:

1. Click in the **Graphic Drawing** area, and the icon is highlighted.
2. Move the cursor to the view area to select points.
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.

10.4.3 Arc

Description:

Draws an arc.

Operation procedure:

1. Click in the **Graphic Drawing** area, and the icon is highlighted.
2. Move the cursor to the view area to select points.
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. An arc is formed.

4. Click again to start drawing the next arc.

10.4.4 Rectangle

A rectangle can be drawn with two or three points.

Two-point Rectangle

Description:

Draws a rectangle by specifying the diagonal with two 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 is highlighted.
2. Move the cursor to the view area to select points.
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 is highlighted.
2. Move the cursor to the view area to select points. 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.

10.4.5 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 is highlighted.
2. Move the cursor to the view area to select points.
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 is highlighted.
2. Move the cursor to the view area to select points.
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 is highlighted.
2. Move the cursor to the view area to select points.

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.

10.5 Graphic Editing

10.5.1 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 software 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. This operation can be repeated.
4. Right-click or press the Esc key to exit the feature.

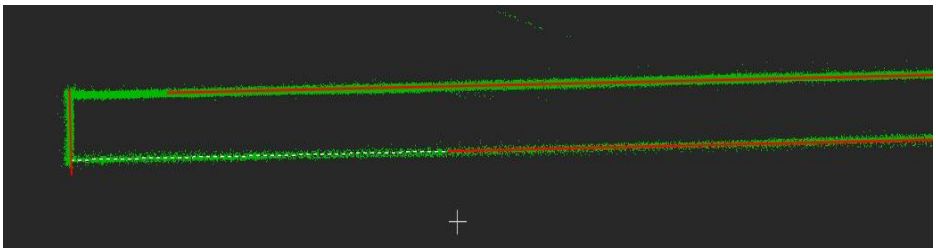


Figure 195. Before extension

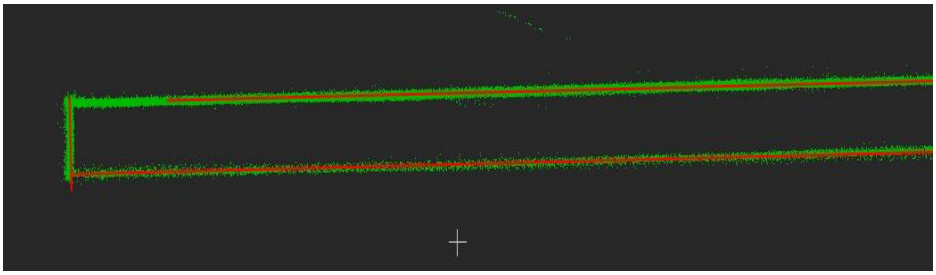


Figure 196. After extension

10.5.2 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 197. Primitive copying result

10.5.3 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 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.

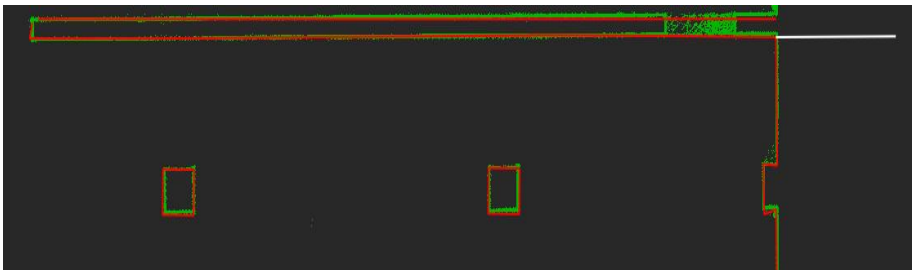


Figure 198. Before trim

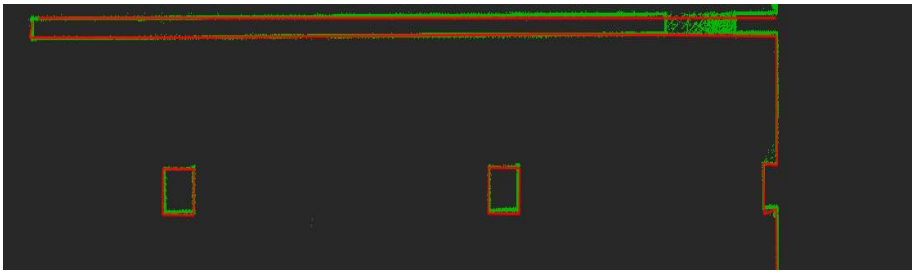


Figure 199. After trim

10.6 Drawing Settings

10.6.1 Object Snap

Description:

Enables or disables the object snap.

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 one end point of the last vector line.
- Any other points on any of the above graphics. The cursor type is different from that for snapping the above elements.

11 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.

11.1 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**.

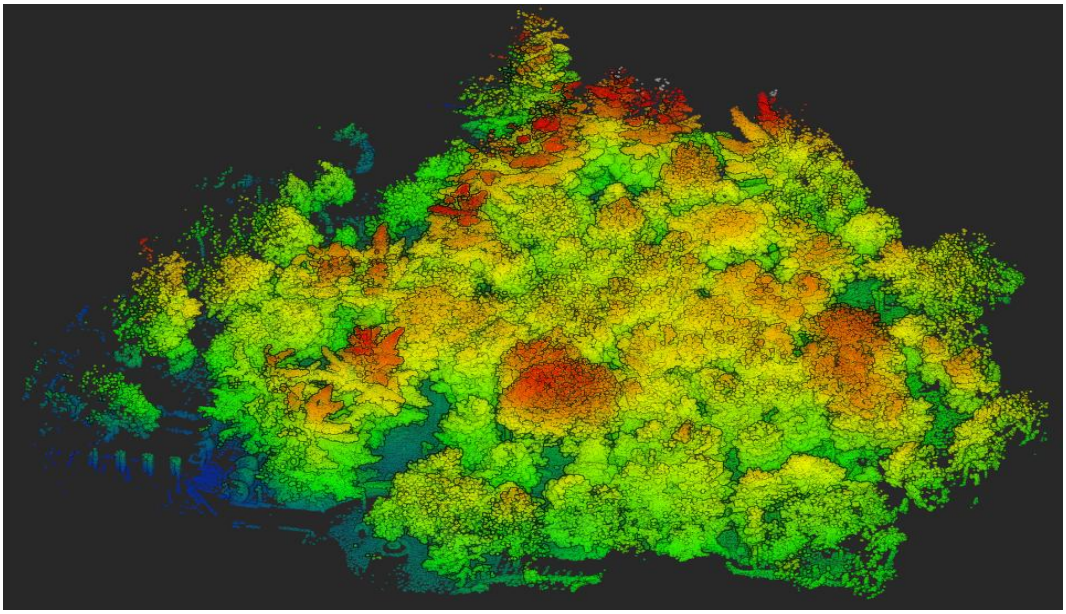


Figure 200. Forestry scenario point cloud

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

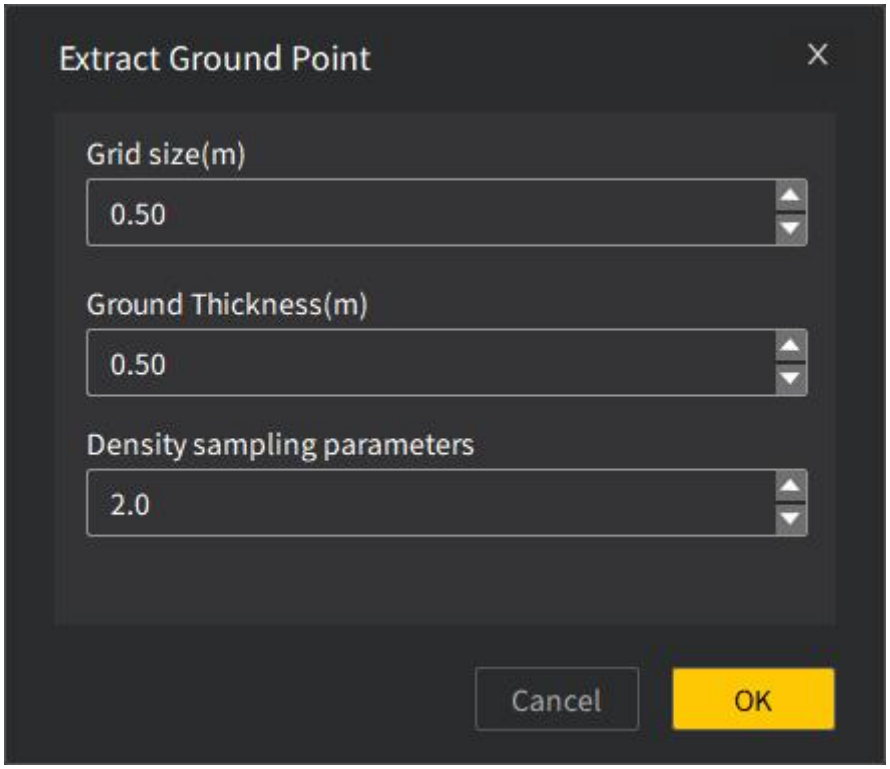


Figure 201. Setting parameters

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

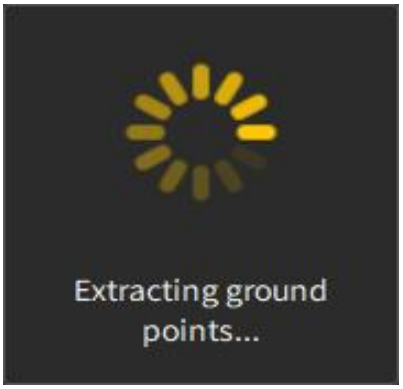


Figure 202. Extracting ground points

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.

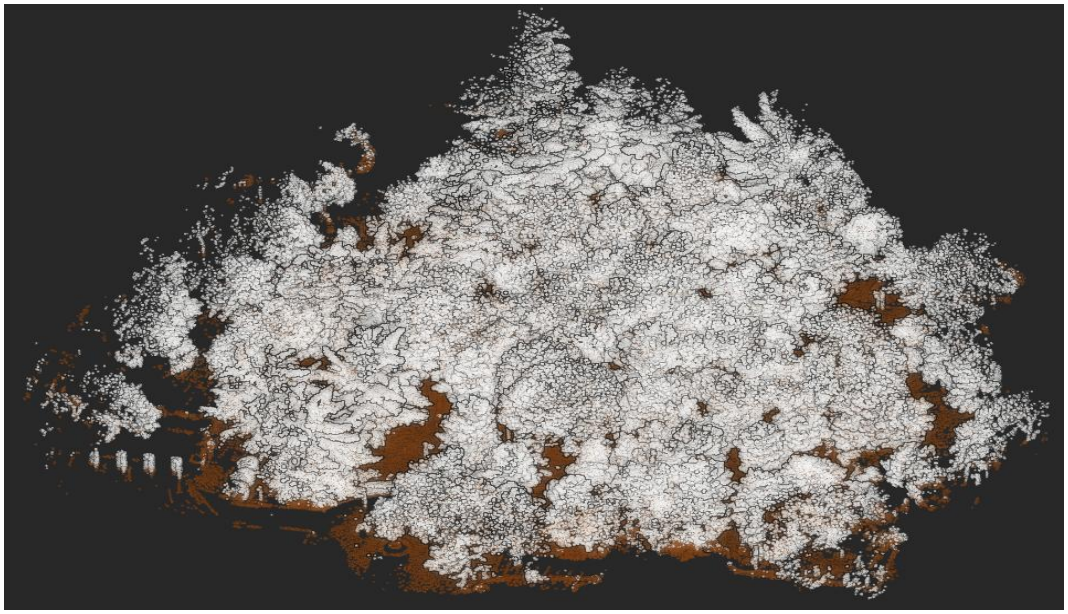


Figure 203. Ground points extracted

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.

11.2 Segment by Tree

Description:

Obtains the tree boundary from the point cloud with ground points extracted to segment the point cloud by tree and extract the trees.

Operation procedure:

1. Select a point cloud file under **Objects**.
2. Click on the **Forestry** tab, and the following window appears.

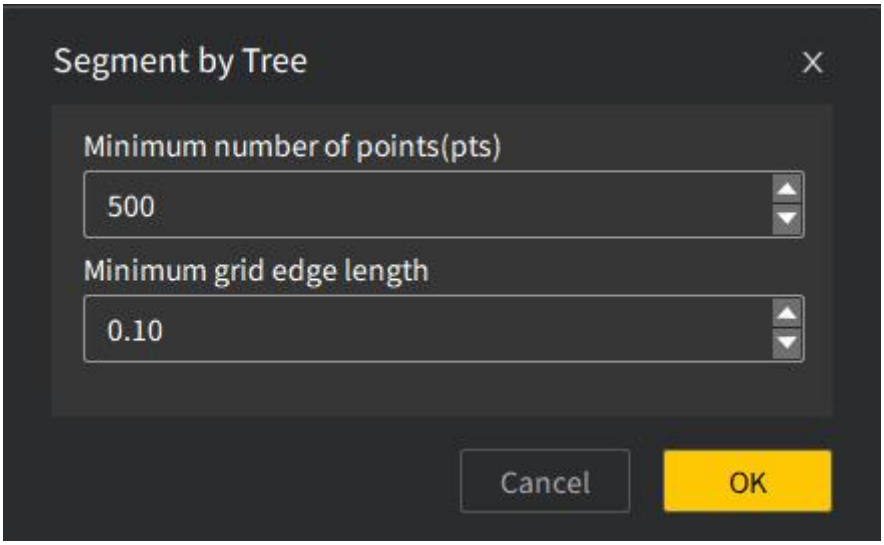


Figure 204. Setting parameters

3. Click **OK** to start the segmentation by tree. The following window appears.

Figure 205. Segmenting by tree

4. The above window disappears when the segmentation is completed. The segmentation result rendered by tree ID is displayed in the 3D view area.

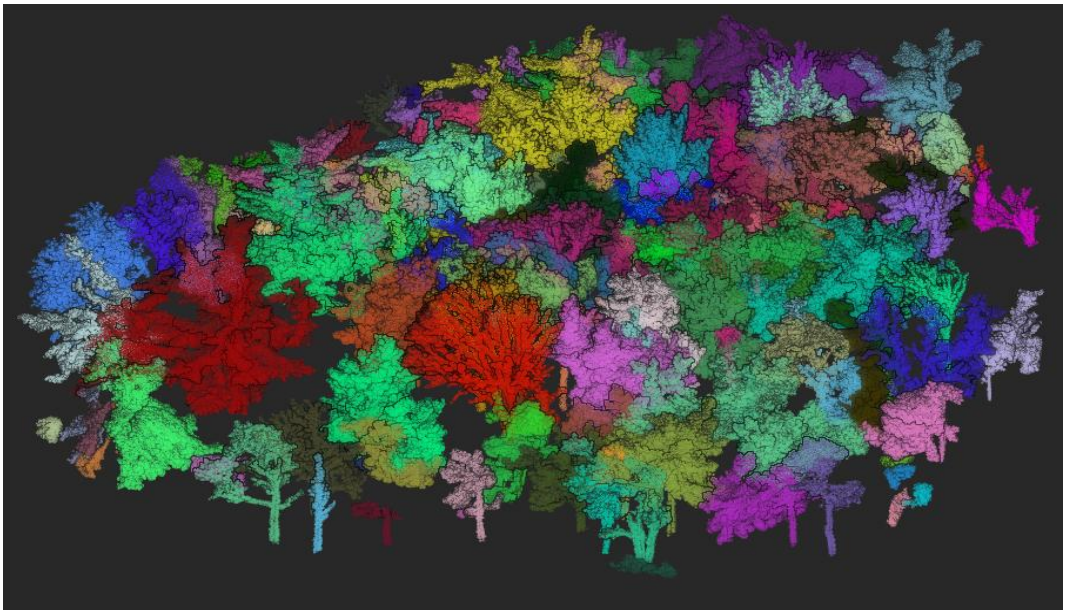


Figure 206. Segmentation result

Parameters:

- Minimum number of points: minimum number of points per tree.
- Minimum grid edge length: It is set based on the point cloud spacing, and is often set to the point cloud spacing.

11.3 Edit

Description:

The tree point clouds differentiated by colors are obtained through segmentation by tree. Oversegmentation or undersegmentation may occur due to the complexity of scanning scenarios. You can edit the segmentation result to make it more realistic.

Operation procedure:

1 Select the folder *original point cloud name*.segment under **Objects**.

2 Click on the **Forestry** tab, and the screen is displayed as below. By default, view 1 is a top view of the point cloud with trees highlighted in different colors, while view 2 is a 3D view of the selected area.

3 Select the area to edit in view 1, and view 2 is updated accordingly.

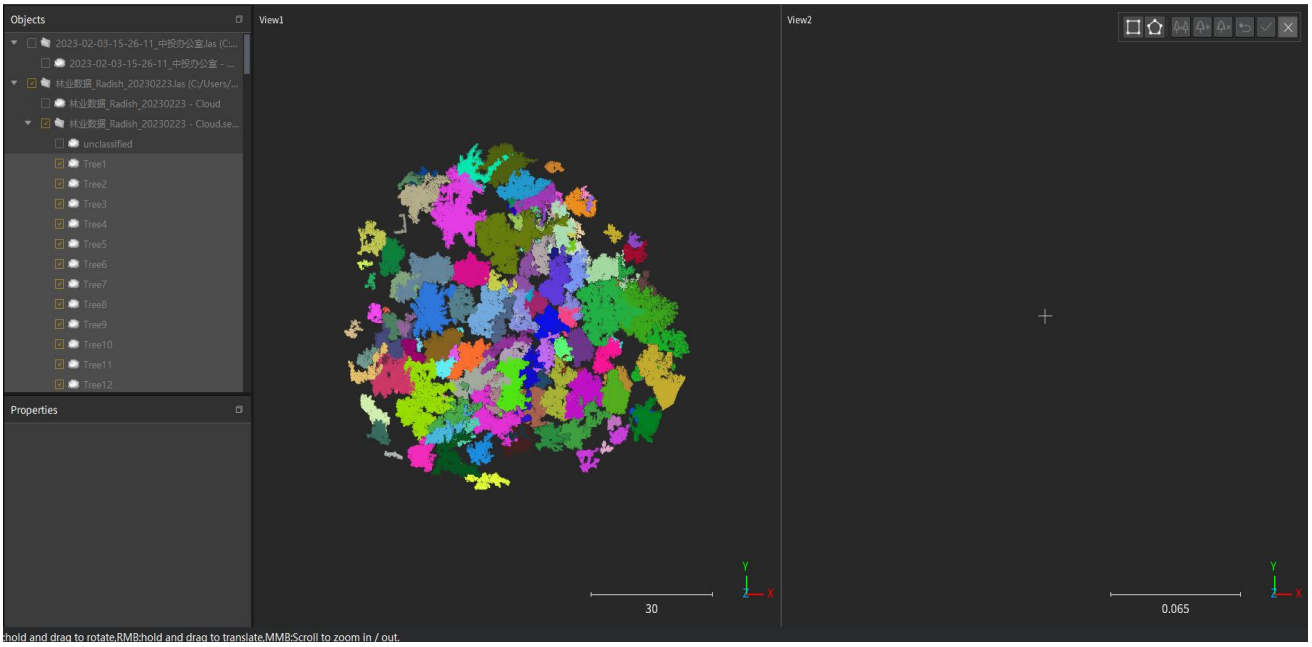


Figure 207. Tree point cloud editing screen

4 If unexpected objects are found in the selected area in view 2, click the ID labels of such objects and then click in view 2 to set them as unclassified data.

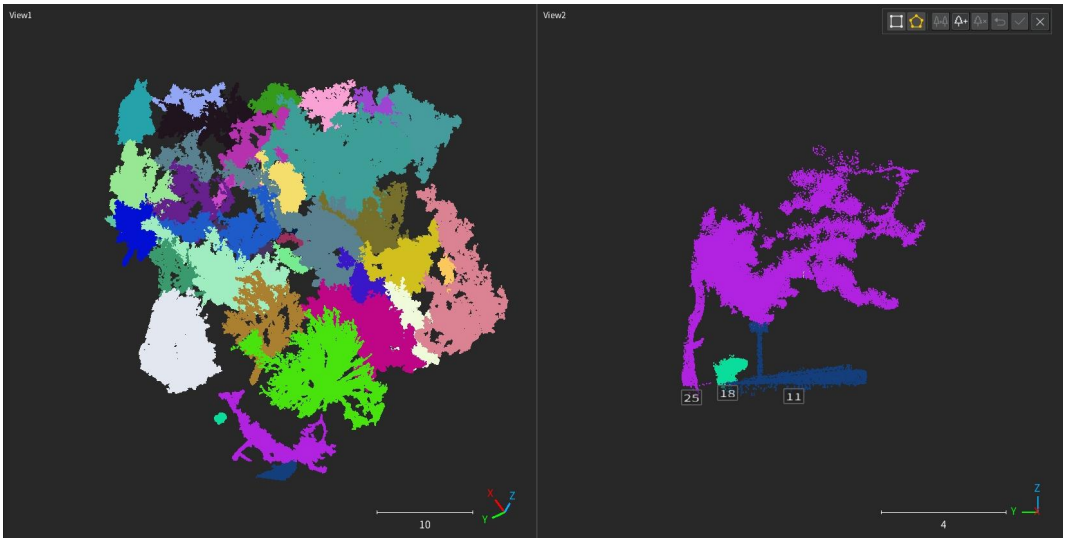


Figure 208. Before deleting the wrong ID labels

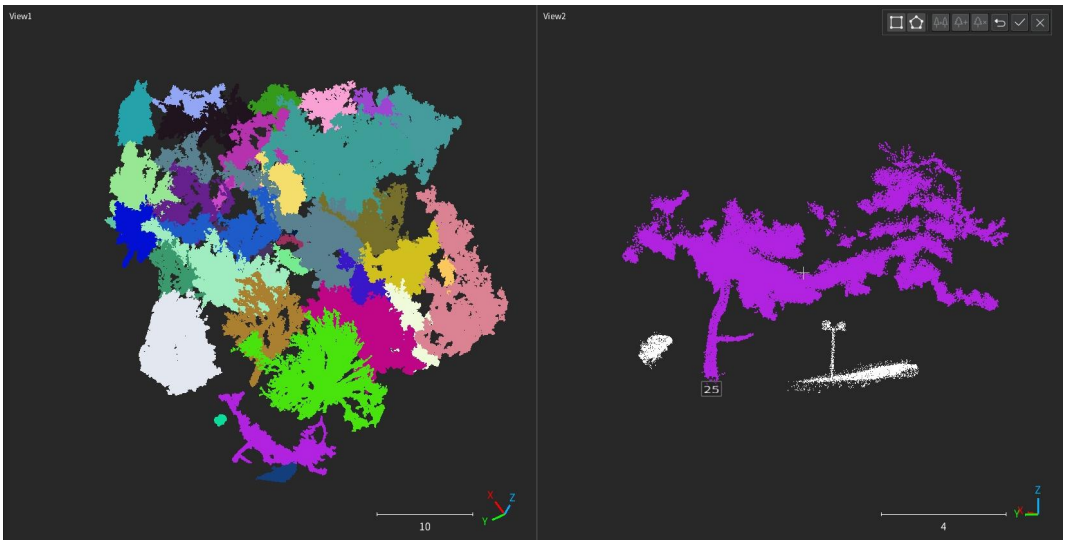


Figure 209. After deleting the wrong ID labels

5 If multiple trees are identified as one tree or trees are not identified, label them manually as individual trees. On the original point cloud shown in view 1, select the portion you want to edit and the object is displayed in view 2, and then click to label it as a tree and assign a new tree ID.

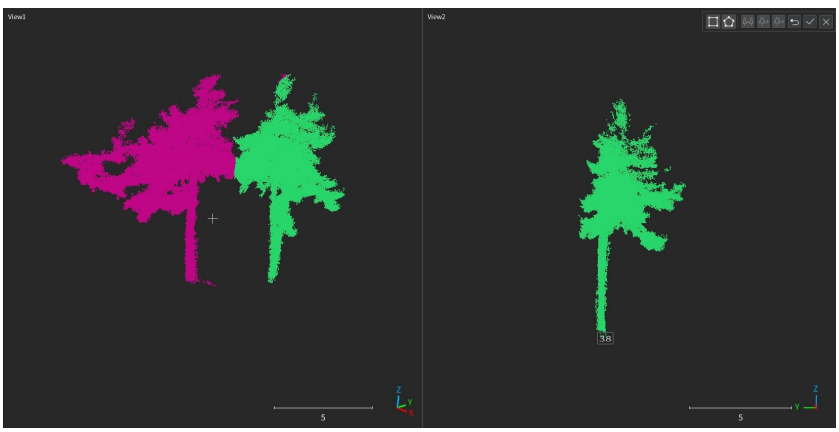


Figure 210. Adding a new tree ID label

6 If one tree is identified as multiple trees, click the corresponding tree ID labels in view 2 while pressing the Ctrl key, and then click to merge them into one. The new tree uses the ID of the first selected tree.

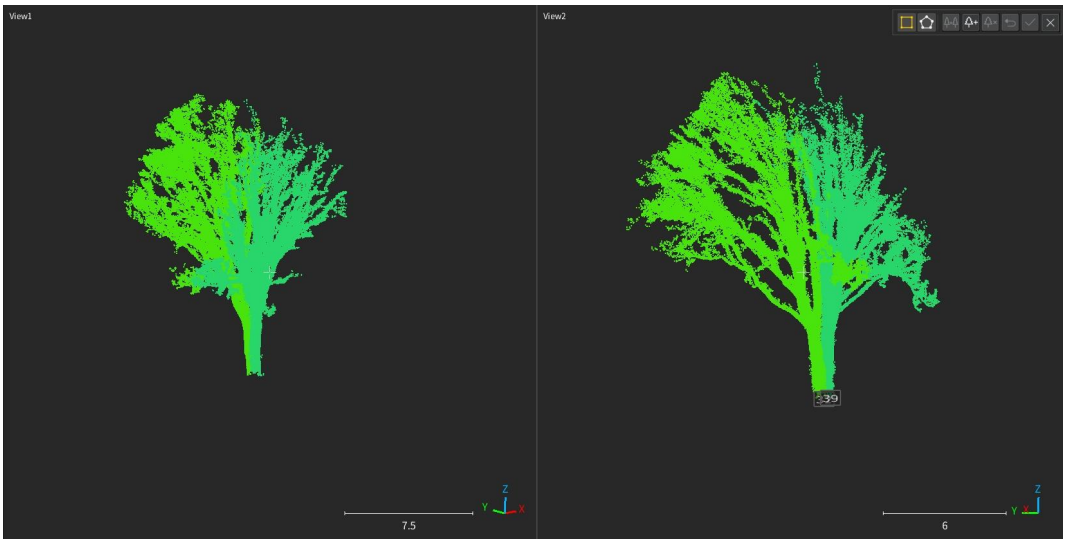


Figure 211. Before merging

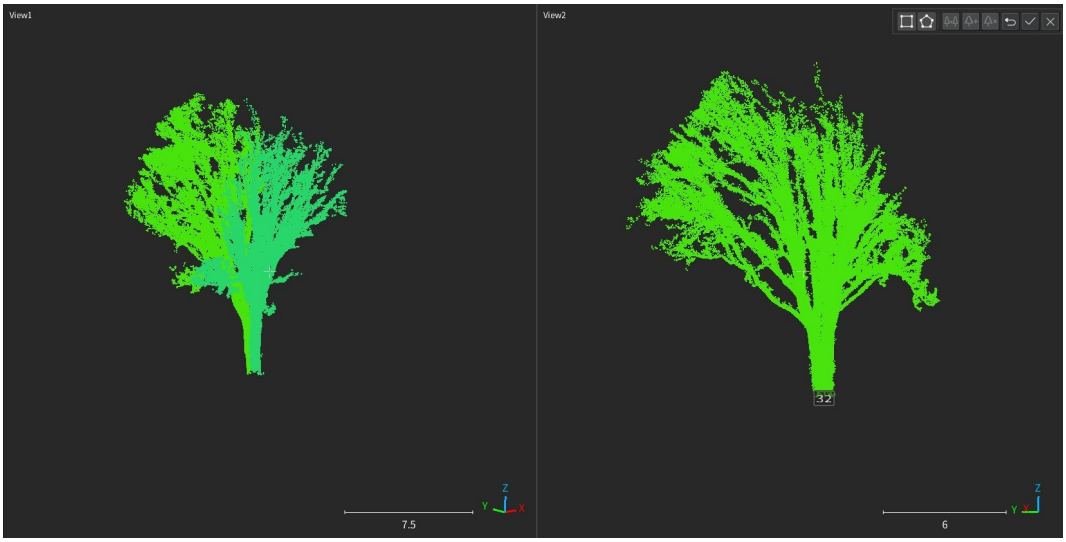


Figure 212. After merging

Note:

- 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.

11.4 Property Calculation

Description:

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

Operation procedure:

1 Click the folder *original point cloud name*.segment, or the required tree point cloud under **Objects**.

2 Click on the **Forestry** tab, and the following window appears. Select parameters as required.

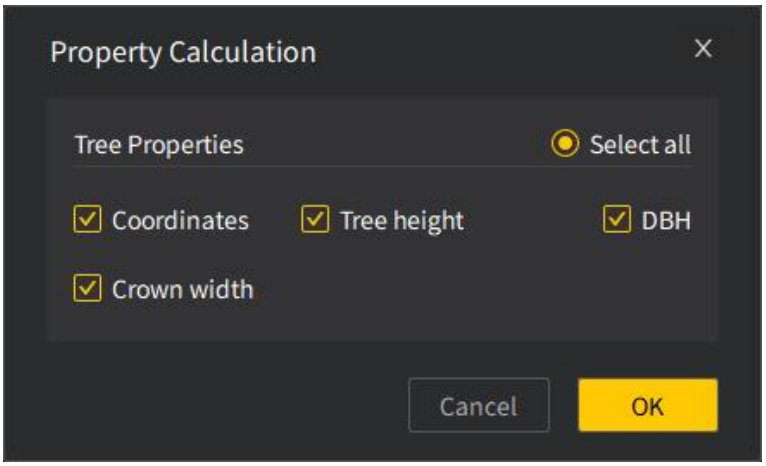


Figure 213. Selecting parameters

3 Click **OK** to start calculation. The following window appears.

Figure 214. Calculating parameters

4 The tree properties are displayed in a table after the calculation is completed. Click any row of data in the table, and a bounding box appears for the corresponding point cloud in the 3D view area.

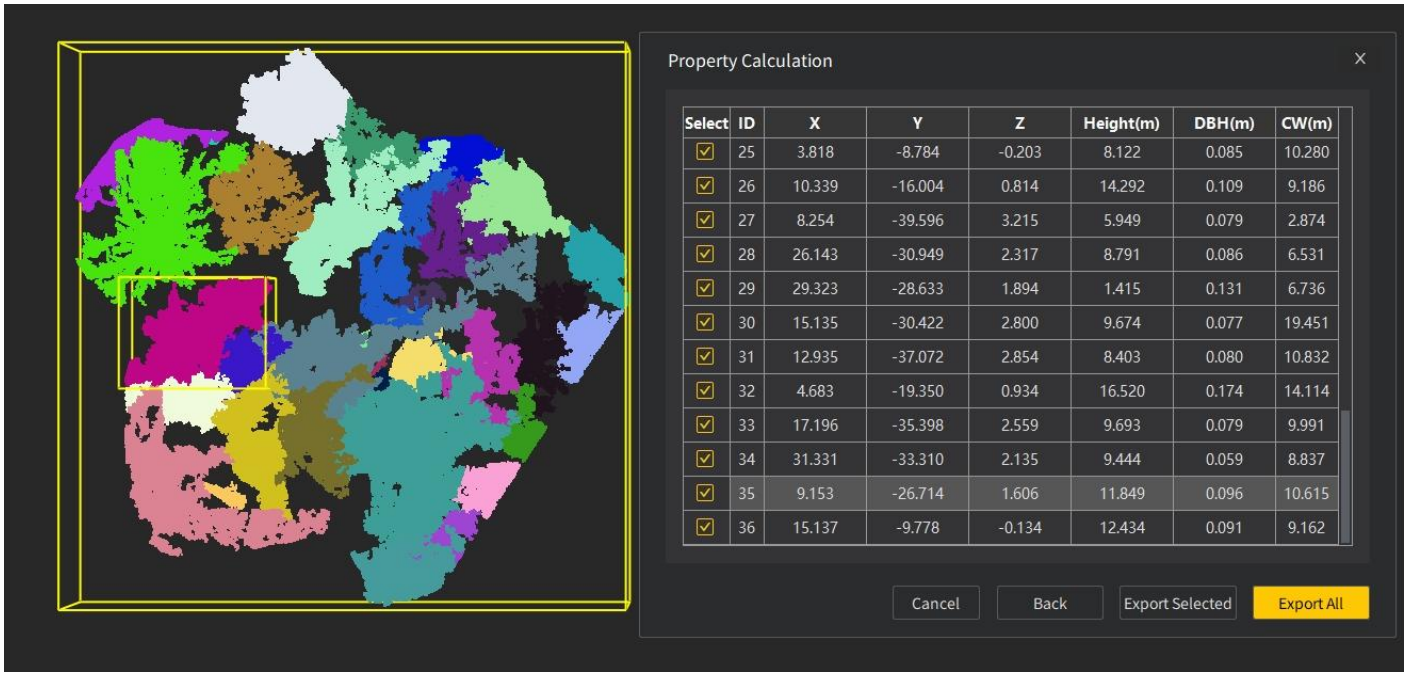


Figure 215. Calculation result preview

5 To export the tree properties, unselect the unwanted IDs and click **Export Selected**, or click **Export All**, select the storage path, and click **Save**. The export file name is the segment file folder name by default. The .xlsx, .csv, and .txt file formats are supported.

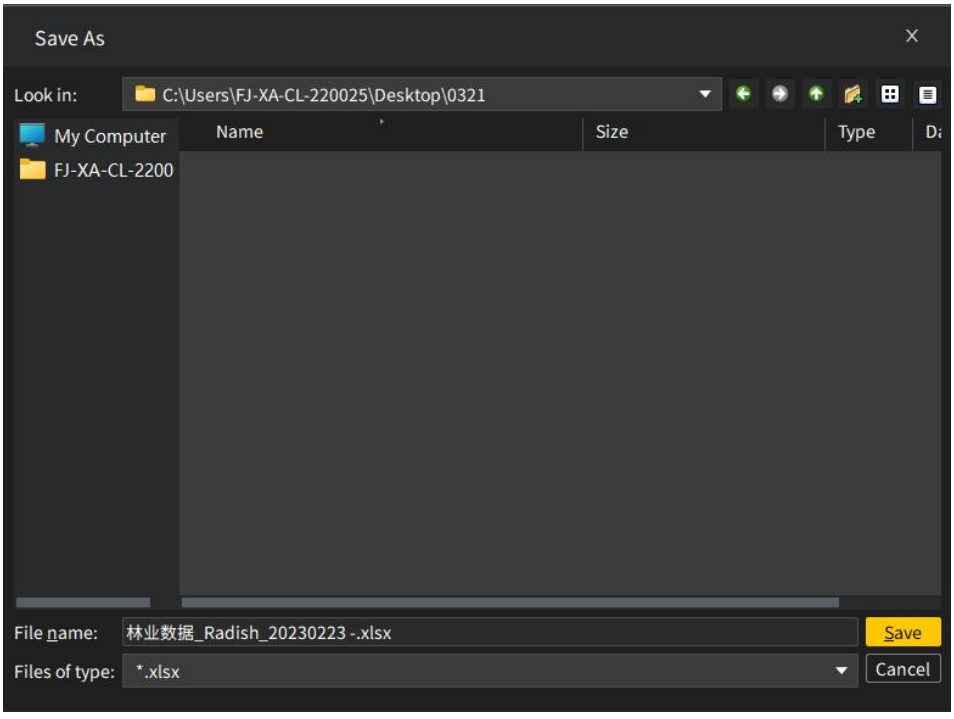


Figure 216. Export settings

12 Appendix

12.1 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.

12.2 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**.

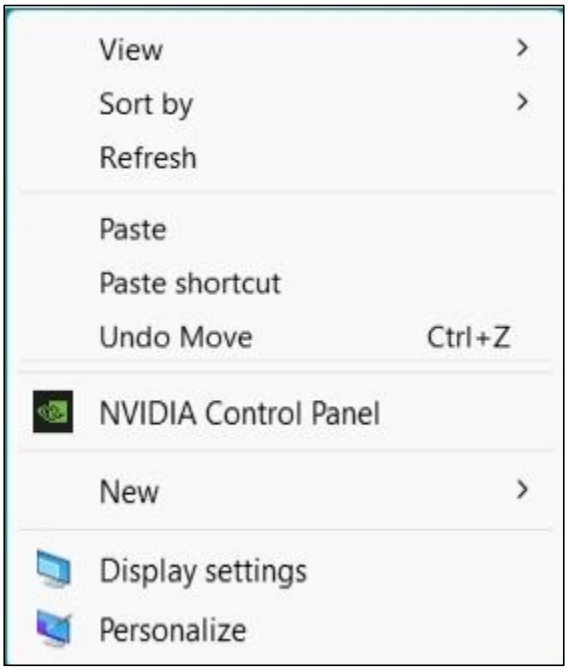


Figure 217. NVIDIA control panel

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

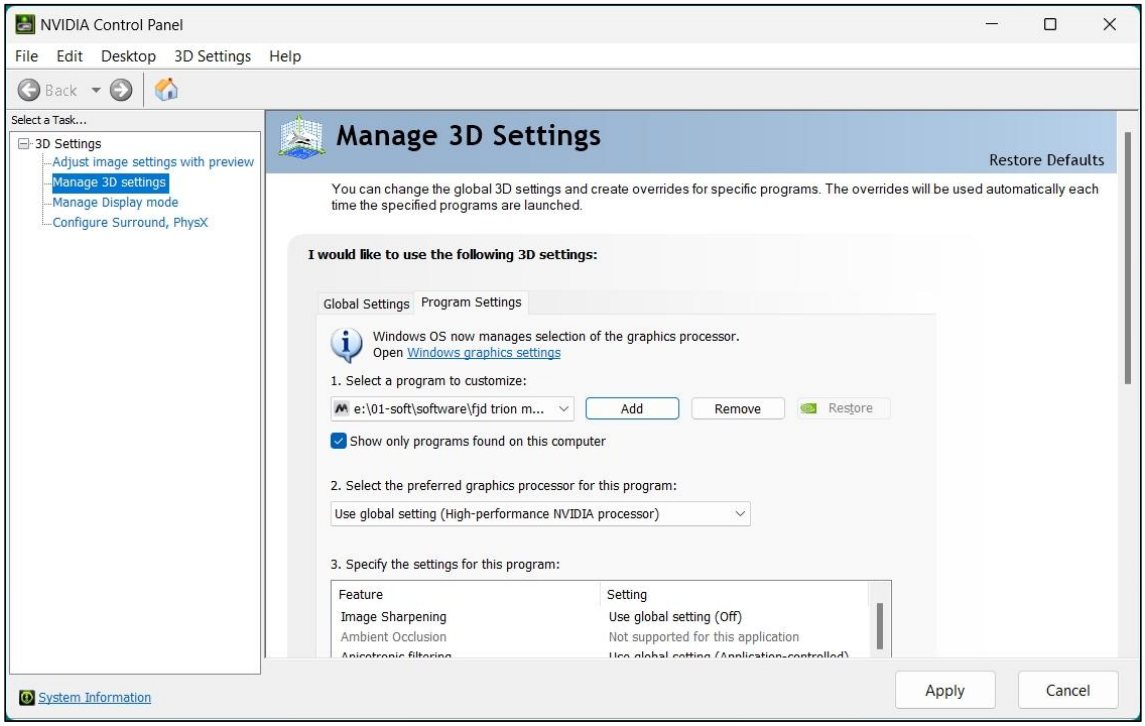


Figure 218. Graphics mode settings

13 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 to get the encryption lock. The encryption lock is shown in the figure below.

Figure 219. Encryption lock