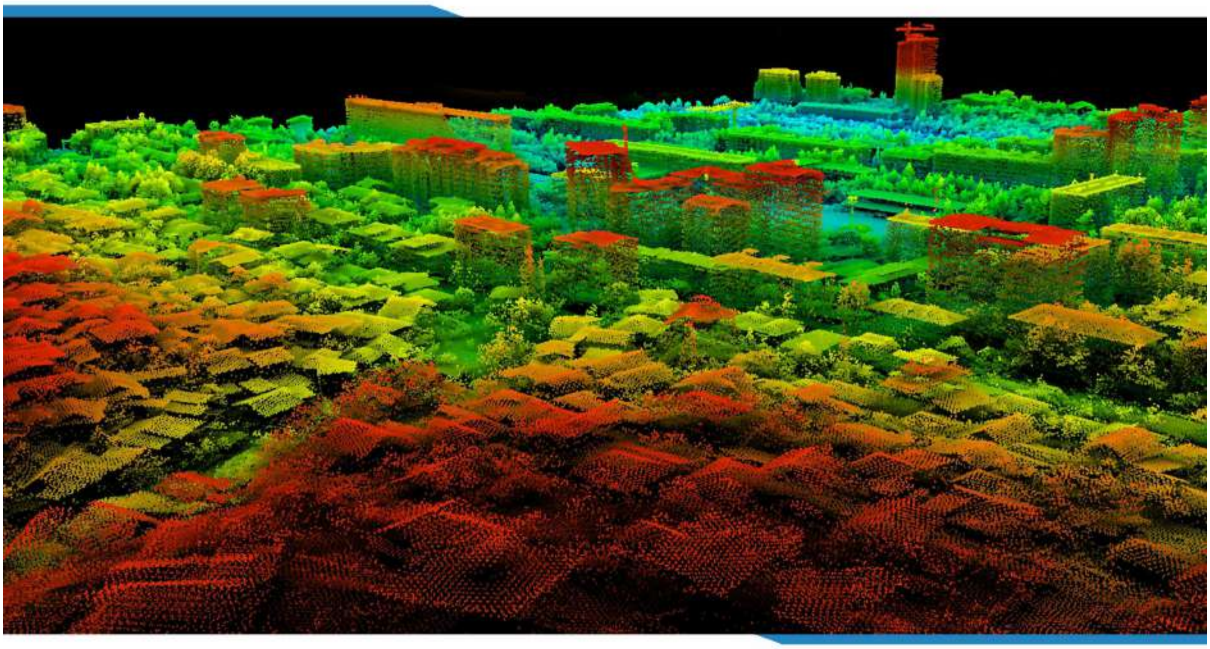
Table of Contents

GreenValley International

LiDAR360 User Guide

LiDAR point cloud processing and analyzing software



www.greenvalleyintl.com

Copyright

GreenValley International

LiDAR360 V8.0

User Guide

Imprint and Version

Document Version 8.0

Published by:

GreenValley International Inc.

Web: https://greenvalleyintl.com/

Dear User,

Thank you for using LiDAR360 software. We are pleased to be of service to you with LiDAR point cloud manipulation solutions. At GreenValley International, we constantly strive to improve our products. We therefore appreciate all comments and suggestions for improvements concerning our software, training, and documentation. Feel free to contact us via info@greenvalleyintl.com, thank you.

Legal Disclaimer

Date of printing: August 15, 2024

Introduction

LiDAR360 is a powerful LiDAR point cloud data processing and analysis platform which can simultaneously process more than 300G point cloud data with more than 10 advanced point cloud data processing algorithms. The platform includes a wealth of editing tools and automatic navigation tape splicing capabilities for the terrain, forestry, mining and power industries (see LiPowerline software).

The Terrain module is a suite of GIS tools used primarily for the generation of industry-standard topographic products. Our point cloud filtering algorithm can precisely extract ground points under complex landscapes and therefore improve the terrain surveying accuracy. The module can also fuse point cloud and image to produce real projective image and other products.

The Forestry module brings important technological innovations to forest inventory and analysis. The parameters of tree height, DBH and crown diameter can be obtained by individual tree segmentation algorithm. A range of regression models for predicting forest structures from LiDAR variables are also provided.

Specifically, LiDAR360 has the following modules:

- Strip Alignment: — Point clouds from overlapped strips can be automatically aligned based on strict geometric model. The software can display aligned strips in real-time and generate aligned point clouds with high accuracy. In addition, the software provides a range of data quality inspection and analysis tools to ensure data accuracy.
- Data Management: — LiDAR360 provides management tools for point cloud and raster data, which include format conversion, point cloud de-noising, normalization, raster band calculation and so on.
- Statistics — Statistical analysis of point clouds based on points, point density, Z value, etc., and evaluate data quality.
- Classification — LiDAR360 provides a variety of classification functions, including ground point classification, model key points classification, selection area ground point classification, machine learning model classification (can efficiently separate buildings, vegetation, street lights and other general categories), deep learning model classification, custom deep learning classification, etc.
- Vector Editing — Vector editing function completes the vectorization part of the digital line drawing process. Relying on the excellent display effect of point cloud, it provides a high-contrast base map, which can clearly distinguish the contouring of ground objects such as houses, vegetation areas, roads, street lamps, water areas and bridges to assist the vectorization of ground objects. At the same time, vector results can be edited, checked and drawn by combining image or model data. A variety of semi-automatic and manual vector chemical tools can be provided to facilitate the acquisition of two-dimensional and three-dimensional vectorization results.
- Terrain — LiDAR360 generates useful terrain and forest information by generating digital elevation models, digital surface models and canopy height models; the provided section analysis tools can generate cross-section products; It can also generate contour line, mountain shadow, slope, slope direction, roughness and other products. At the same time, it also provides the model data editing function.

- ALS Forest — Based on ALS (Airborne Laser Scanning) data, the module can extract a series of forestry parameters (such as height quantile, leaf area index, canopy density, etc.), segment individual trees and extract their parameters (including tree position, tree height, DBH, CBH, crown width, etc.), and use multiple regression analysis functions of the software combining with ground survey data to invert functional parameters such as forest biomass and stock.
- TLS Forest — Based on TLS (Terrestrial Laser Scanning) or backpack lidar point cloud data, it can extract DBH (Diameter at Breast Height) and the number of trees with batch processing, segment individual tree and calculate parameters of them (position, height, DBH, etc.), measure and edit individual tree attributes.
- Mine - Based on LiDAR point cloud data, it can build underground roadway 3D model construction, closed model volume measurement, volume change analysis, and open-pit mine slope line extraction, etc.
- Building Modeling - Providing a set of airborne point cloud data 3D vectorization construction tools. The 3D building model can be automatically constructed by using 2D base map, which preserves the topological structure of the building model, and provides a series of surface editing and edge editing tools. The model is at LOD2 level according to the model level description.
- Geology — Extraction of terrain features, geological structure surface features, etc. based on airborne lidar point cloud data.
- Power Line — A clearance distance analysis report can be obtained based on airborne LiDAR point cloud data, including calibration tower, data classification, and danger point detection.
- Photo — The photo module can recover the detailed 3D geometric structure of objects based on overlapping image data, merge point clouds, and generate a series of standard mapping products. The image module can process nadir, oblique, and multispectral images. Core functions include feature point extraction and matching, block adjustment, self-calibration of cameras, generation of DEM/DSM, intelligent orthoimage mosaicking, orthoimage generation, seamline editing, and more.
- Photo — Supports the construction of photogrammetric models from nadir, oblique true color, infrared, and hyperspectral images. It allows for the addition of control points to generate image projects with real-world coordinates. Features include camera calibration, distortion correction, regional network adjustment, and interactive feature point handling, ensuring users achieve optimal orthophoto results.

LiDAR360 Versions

Modules	Free Version	Full Version
Data Visualization	√	√
Mass Data Support	√	√
Data Management	7-Day Free Trial	√
Strip Alignment	7-Day Free Trial	√
Statistics	7-Day Free Trial	√
Classify	7-Day Free Trial	√

Terrain	7-Day Free Trial	√
Vector Editing	7-Day Free Trial	√
ALS Forest	7-Day Free Trial	√
TLS Forest	7-Day Free Trial	√
Mine	7-Day Free Trial	√
Building	7-Day Free Trial	√
Photo	7-Day Free Trial	√
Geology	7-Day Free Trial	√
Power Line	-	√

Get Started

Please refer to Installation and License to install LiDAR360 software. Please refer to the User Guide to use the software.

Installation

Download the latest version of LiDAR360 Suite from the GreenValley International official website before installation.

System Requirements

- **RAM**: at least 8 GB or more.
- **Central Processing Unit (CPU)**: Intel® Core™ i5/i7; Dual-core processor.
- **Display adapter**: NVIDIA graphics above GTX 970, video memory no less than 4GB.
- **Operating system**: Microsoft Windows 7 (64-bit), Microsoft Windows 8 (64-bit), Microsoft Windows 10 (64-bit), Microsoft Windows 11 (64-bit), Microsoft Windows Server 2012 and higher.

> Note: Please enable high-performance graphics mode for running the software.
>
> Note: For "use shaders to render the point cloud color strategy","Mesh Editor", "Building Model Display", etc. you will need a display adapter that supports at least 4.6 OpenGL and glsl versions and at least 4GB of video memory.

Setup

1. Run the LiDAR360 Suite Setup Wizard.
2. Click "Next" button in the Welcome Interface.
3. Click "I Agree" button to continue if you accept the License Agreement.
4. Choose the installation path (or use default path), then click "Install" button.
5. Click "Finish" button after installation.

License Manager

There are two licensing approaches to activate LiDAR360, by license dongle or license code. A hard lock license provides a USB flash drive, and a soft lock license provides an authorization code. For license dongle, users must not format, delete, or copy the license dongle.

1) License dongle

Properly insert a license dongle to USB port to activate LiDAR360.

2) License code

License code would be generated based on activation information given by LiDAR360 users. After purchasing a license code, please follow the following steps to activate LiDAR360.

1.Run the software LiDAR360.

2.Click File > Activate License, the License Manager window will pop up.

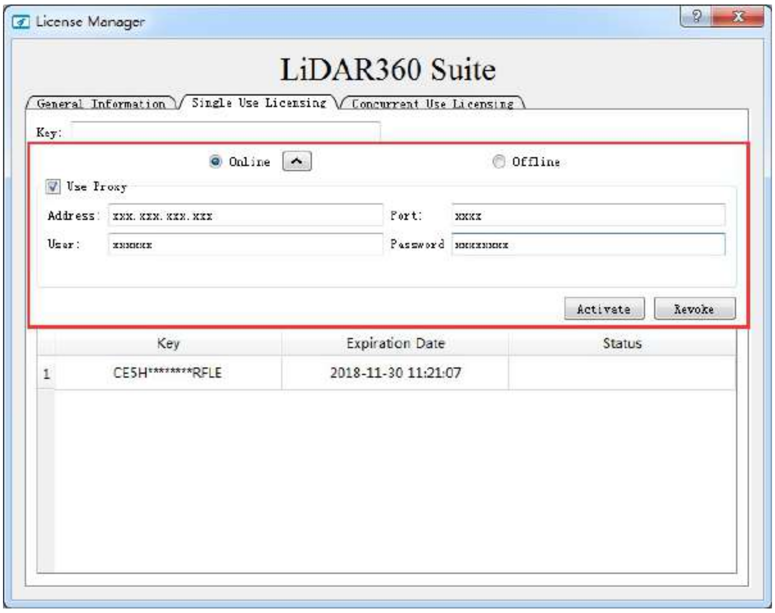
3.Fill in your name and company name, select the modules you want to activate, and then click “Copy”.

4.Email the copied information to info@lidar360.com.

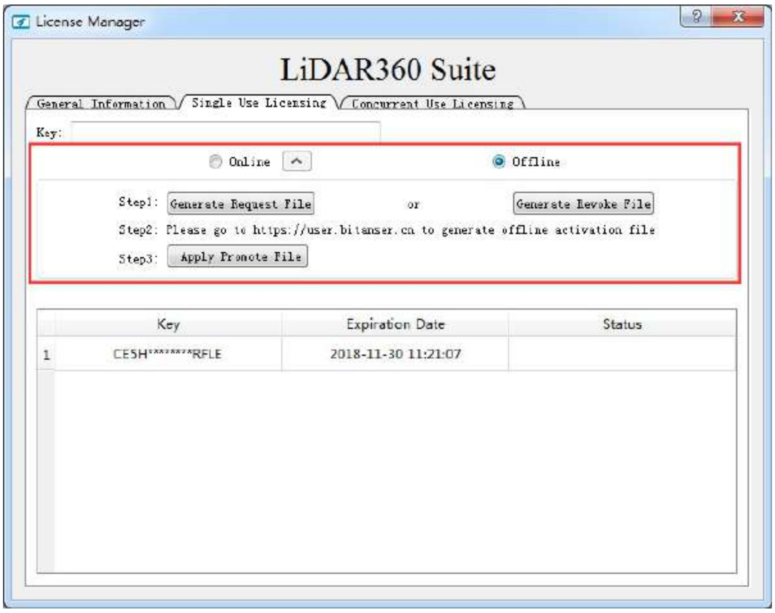
5.There are two licensing modes: single use licensing and concurrent use licensing.

- Single Use Licensing
    - Activation/Update

      Online activation/update: When connecting to the Internet, under the “Single Use Licensing” tab, enter the authorization key, select “Online”, and click “Activate” to activate or update, or you can select the authorization code in the Key list, right-click the right mouse button, and choose “Activate key” from the displayed menu. Under “General Information” page, you may also check the expiration date for each individual module. If you need to set up proxy to connect to the Internet, click to use proxy and to set up the address, port, user name, and password.



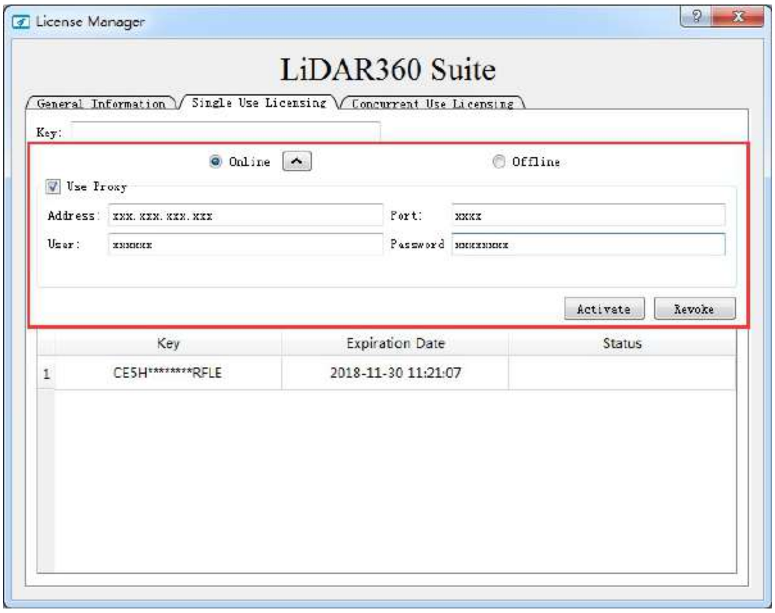
      Offline activation/Update: Enter the authorization code, select Offline, click “Generate Request File” button to generate the request file (.req). Use a computer that can connect to the Internet to browse to https://user.bitanswer.cn, enter the authorization key to log in, click “Offline Update”, upload the request file (.req), and download the generated upgrade file (.upd). On the license management interface, click “Apply License File”.



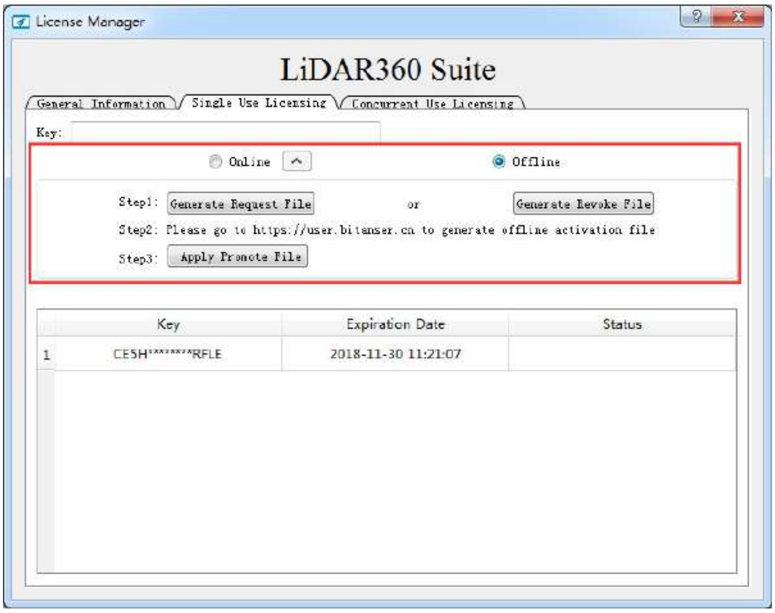
- Revoke

  To unbind an activation key from a computer, user could revoke the activation key online or offline. After the authorization key is revoked, it can be reused on the same computer or a different one.

  Online Revoke: In “Single Use Licensing” tab, enter the authorization key, select “Online”, and click “Revoke” to revoke the key. If you need to set up proxy to connect to the Internet, click to use proxy and to set up the address, port, user name, and password.

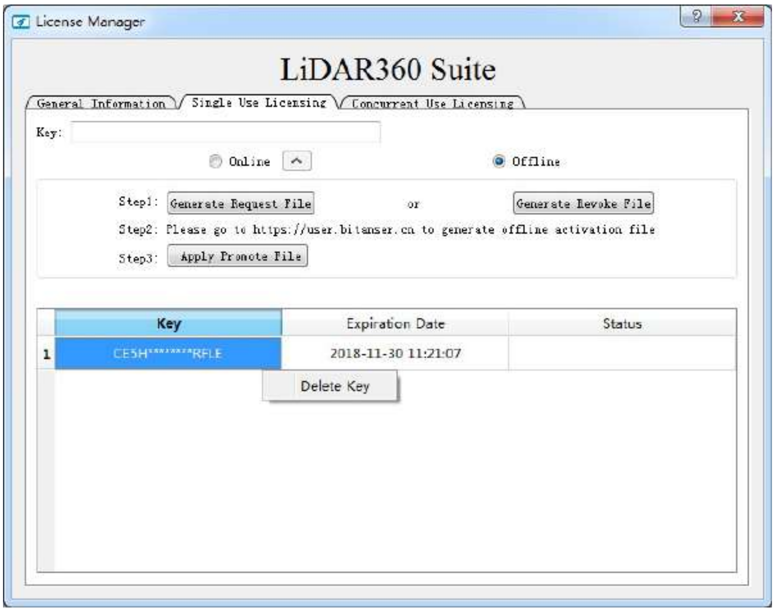


Offline Revoke: Enter the authorization code, select “Offline”, click “Generate Revoke File” button to generate the revoke request file (.req). Use a computer that can connect to the Internet to browse to https: //user.bitanswer.cn, enter the authorization key to log in, click “Update Offline”, upload the request file (.req), and download the generated upgrade file (.upd); On the license management interface, click “Apply License File”.



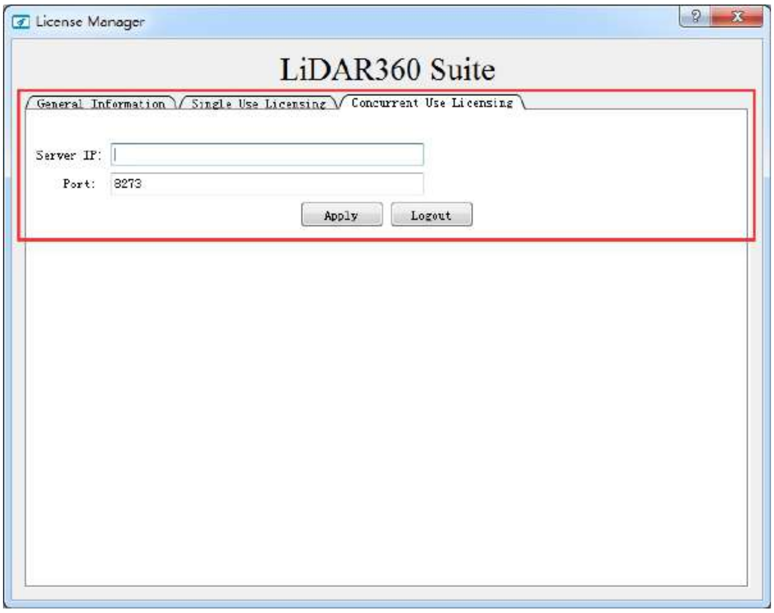
- Delete

  To delete authorization information from the computer, right-click on the authorization key and select “Delete Key”. After being deleted, the same activation key can only be used on the same computer.



- Concurrent Use Licensing

  Install the Group authorization service tool on the group server on the LAN and add the Group service extension module. Activate the authorization code online or offline in the Group authorization management center. Other users on the LAN enter the server IP address on the license activation page. The default port number is 8273. Click Apply.



6.Click the button on the top right corner of License Manager interface to view the license manager user guide.

> Note: If any of the software instance is being opened while updating the license, please restart the software.
>
> Note: If an user would like to move a key from one computer to another, he/she should revoke first, then activate on the other. If the license code has been deleted, the user should reactivate on the same computer first, then revoke and activate this key on the other computer.
>
> Note: Please contact info@lidar360.com to make an inquiry and purchase a license key to activate LiDAR360.

Languages

Currently, the supported languages are English, Chinese, French, and Japanese. The user can change the menu language in the following way:

1. Click Ribbon Toolbar in the upper right and click Display Setting > Language > English/Chinese/French/Japanese* in the menu bar.
2. Click “Yes” in the pop-up to restart the software immediately, or click Cancel and restart later. If “Cancel” is selected, the software does not restart and will be displayed in the selected language the next time the software is started.

Use shaders to render the point cloud color strategy

Currently, the software provides the option of using shaders to render the color strategy of point cloud. Users can choose according to their needs. The selection steps are as follows:

1. Click Ribbon Toolbar in the upper right and click Display Setting > Rendering > Use shaders to render color strategy

2. Click “Yes”, you will be prompted whether to restart the software. You can complete the switch of the color strategy of the rendering point cloud if you restart. If “Cancel” is selected, the software does not restart and will be rendered with the preset color policy the next time the software is started.

3. "Use shaders to render color strategy" refers to that the same point cloud can be displayed in different Windows according to different color strategies, and the rendering adopts the programmable pipeline shader technology.

Tool Reference

This chapter describes the usage of menu bar, toolbar, project management windows and toolbox of LiDAR360 in detail.

- Start Page
- Toolbar
- Quick Toolbar
- Ribbon Toolbar
- Project Manager Window

Ribbon Tab

- File
- Tools
- Preprocessing
- Classification
- Terrain
- ALS Forest
- TLS Forest
- Power Line
- Mine
- 3D Building
- Display
- Profile Editor
- Profile View

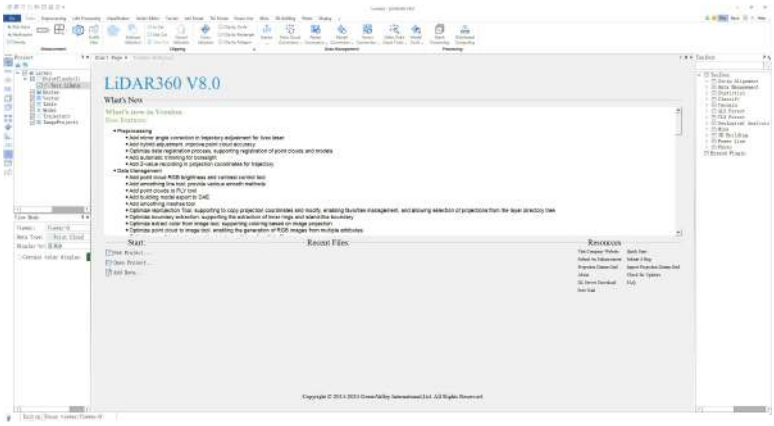
ToolBox

- Strip alignment
- Data Management
- Statistics
- Classification
- Terrain
- Vector editing
- ALS Forest

- TLS Forest
- Mine
- Building modeling
- Power Line
- Geological Analysis

Start Page

Start page is shown as figure below:



Start page can be closed by the close button. Also, users can set Start or Close Start page by checking the box in [Display]->[Show/Hide]->[Start Page].

What's New

What's New shows the new features in the new version.
Start includes:New Project, Open Project, Add Data

Recent Files

Recent files is shown here. Users can left-click and add the file into viewer. If there is no current viewer, a viewer will be created and the data will be added to it.

Resources

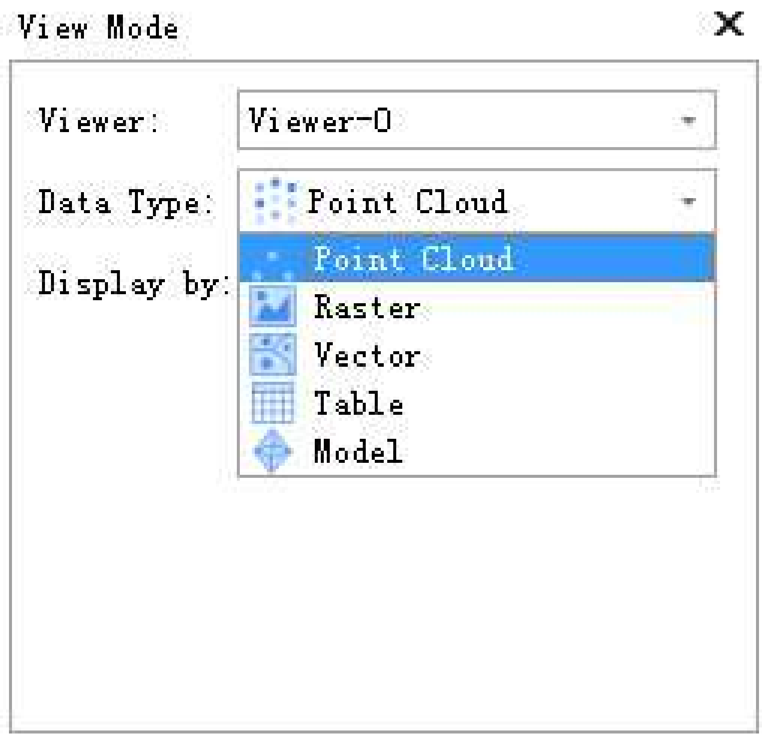
Resources includes：Visit Company Website, Quick Start, Submit An Enhancement, Submit A BUG, Check for Updates, Projection Datum Grid, About.

Toolbar

This chapter provides a detailed introduction to the functions in the toolbar located on the left side of the LiDAR360 software interface.

View Point Cloud Display

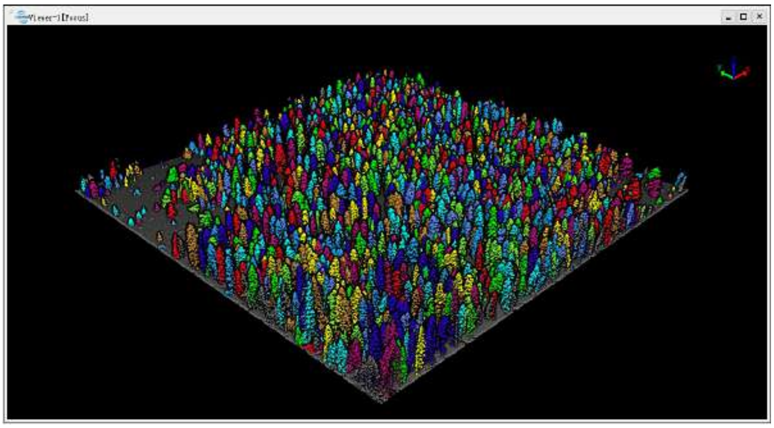
Function Description：Display or hide the point cloud display mode setting interface, as shown in the figure.



Steps

1. Select the window to set the point cloud data display mode.
2. Select the display mode to be set: Currently, there are nearly 20 supported display modes, including display by elevation, display by intensity, display by category, display by RGB, display by GPS time, display by tree ID, and mixed display.
3. Set the parameters according to the selected mode. The selected window will update and display the color of the point cloud data according to the set parameters.

The following image shows the rendering by tree ID:



Display by RGB

1. Select the window to set the display mode.
2. Choose the type of point cloud.
3. Select "Display by RGB" to show the point cloud according to its RGB properties.
4. Click on the color selection button and a pop-up window will appear for selecting colors. Set up your desired color in this window or click on "Pick Screen Color" button, then click on a pixel in the point cloud window that you want to pick (for example ground color). Click "OK" when finished with color selection.
5. Check "Certain color display", which will bring up a buffer setting interface where you can choose to display only points with selected colors (for example, only showing ground points).
6. Modify buffer values to adjust allowed range of displayed colors. The left image below shows display by RGB while right image shows specified color rendering.

Vector Display

Vector data supports four display modes: Height, File, Contours, and Layers.

1. Height: Colors are assigned based on the elevation of the vector data.
2. File: Different vector files are assigned different color schemes.
3. Contour: Displays contours based on elevation data.
4. Layer: Different layers are displayed with different colors, with visibility control for individual layers.

> Note: This mode is only applicable to .shp files generated from point clouds.

Raster Display

Raster data supports Single Band and Hillshade display modes.

1. Single Band: Displays raster data based on the cell values. Supports selection of different color ramps and overlaying of hillshade.
2. Hillshade: Applies hillshade rendering using grayscale imagery.

Table Display

Table data supports Elevation and File display modes. 1.Height: Displays based on the elevation of the coordinates in the table file. 2.File: Different table files are displayed with different colors, which can be customized.

Model Display

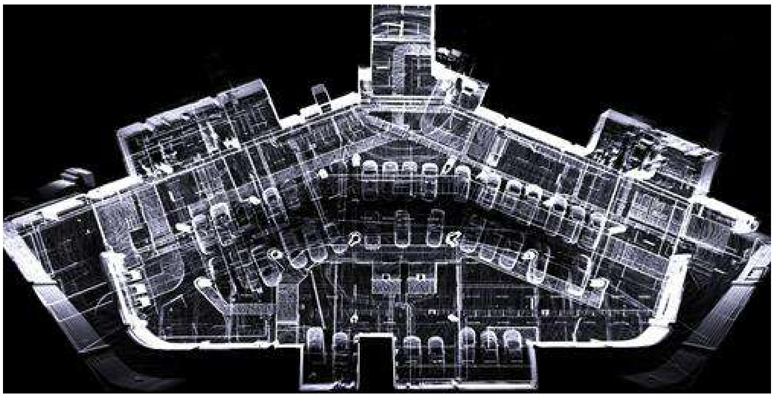
Models support display modes by Height, Class, Texture, Semantic, and Vertex Color.

1. Height: Displays the model based on its elevation.
2. Class: Displays the model in a single color, which can be customized.
3. Texture: For models with textures (e.g., .LiModel, .LiBIM files), display by the texture colors.
4. Semantic: For .LiBIM files, roofs, walls, and ground are displayed in different colors.
5. Vertex Color: For models with vertex colors (e.g., .obj files generated from surface reconstruction), displays as a colored model based on vertex colors.

Glass Shader

Brief: With the glass shader point cloud object appears translucent.

This effect can be enabled/disabled for individual display window. An example is shown as follows.

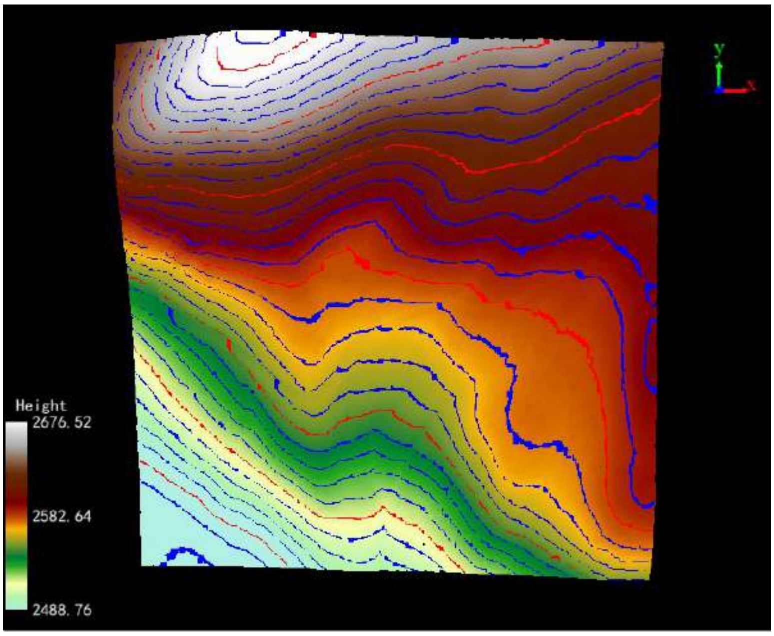


> Note: The Glass Shader is only applicable to point cloud data.

Real-time Contours

Brief：Enables real-time contour effects on point clouds and model data within the window.

After clicking this function, you can enable or disable the real-time contour line effect on the point cloud and model data in the current window. The following figure shows the effect of enabling real-time contour lines:



Contour parameter settings refer to Display Settings > Render Settings > Real time Contour Settings.

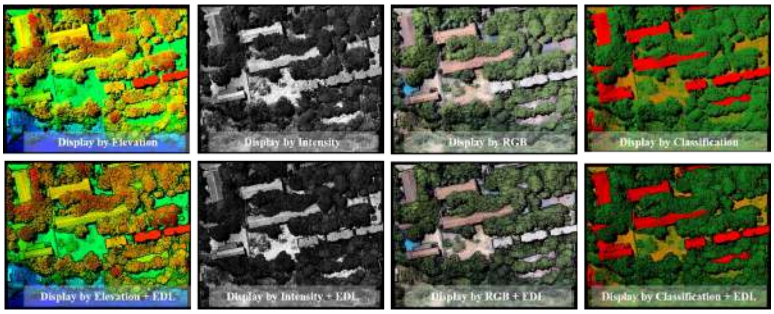
> Note: Real-time contour line effects can be superimposed with other display effects, and are effective for point clouds and model data.

Display by EDL

Description: This tool is used to display the point cloud data and enhance the visual effects of the contour features using the Eye Dome Lighting (EDL) mode. EDL is a shading technique that works with other display mode (e.g. display by height, display by intensity) to improve the depth perception in 3D point cloud visualization.

Usage

1. Click the window that loads point cloud data with the left mouse button and set it as the active window.
2. Click the button on the toolbar. The visual effects of the point cloud data in the scene will be improved with EDL mode. The following picture shows the comparison before and after using EDL display mode.



> Note: This tool only works with point cloud data.

Viewing Tools

Set current active window to some view.

Top View: Set the camera position to view the top view, that is, view from +z to -z direction Three-dimensional data, the plane is the xy plane.

Botton View: Set the camera position to view the bottom view, that is, view from -z to +z direction Three-dimensional data, the plane is the xy plane.

Left View: Set the camera position to view the left view, that is, view from -x to +x direction Three-dimensional data, the plane is the yz plane.

Right View: Set the camera position to the right view, that is, view from +x to -x direction Three-dimensional data, the plane is the yz plane.

Front View: Set the camera position to the front view, that is, view from -y to +y direction Three-dimensional data, the plane is the xz plane.

Back View: Set the camera position to back view, that is, view from +y to -y direction Three-dimensional data, the plane is the xz plane.

Front Isometric View: Set the camera position to 45° front tilt in xy.

Back Isometirc View: Set the camera position to 45° backward in the xy plane.

Set Projection Mode: Change the projection mode of the view, the view supports two projection modes: orthogonal and perspective . View only for 3D display. Shortcut key: F3

> Note: The above function is only for 3D display view; this function does not reset the center position of the viewpoint, if you want to reset to the default viewpoint, please click Global Display .

Full Extent

 **Brief**: This tool is applicable to all data types supported by LiDAR360. It makes all the data cover the entire window in top view and provides full view of all the data.

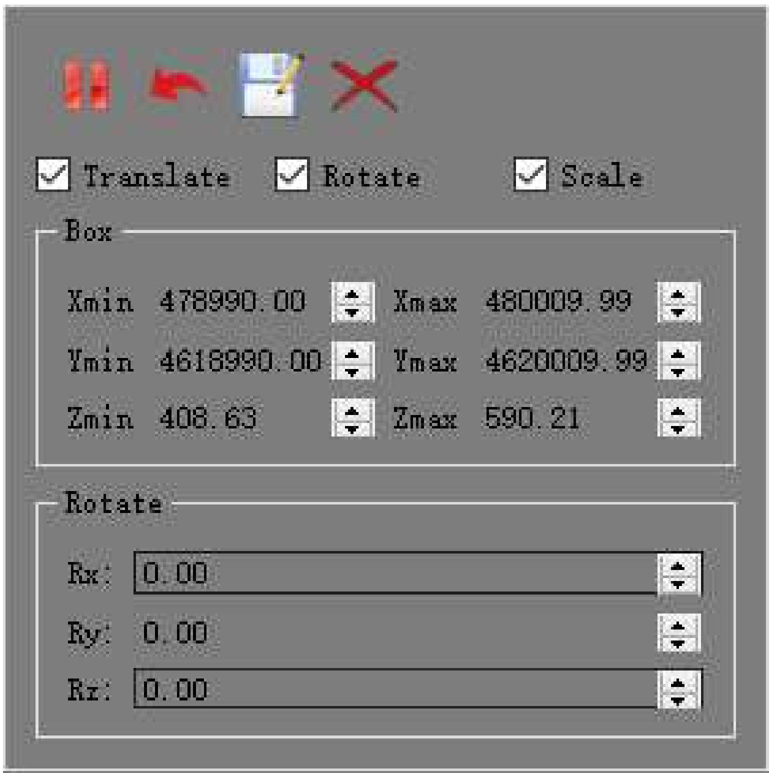
> Note: When there are multiple windows, this tool only works on the active window.

Cross Selection

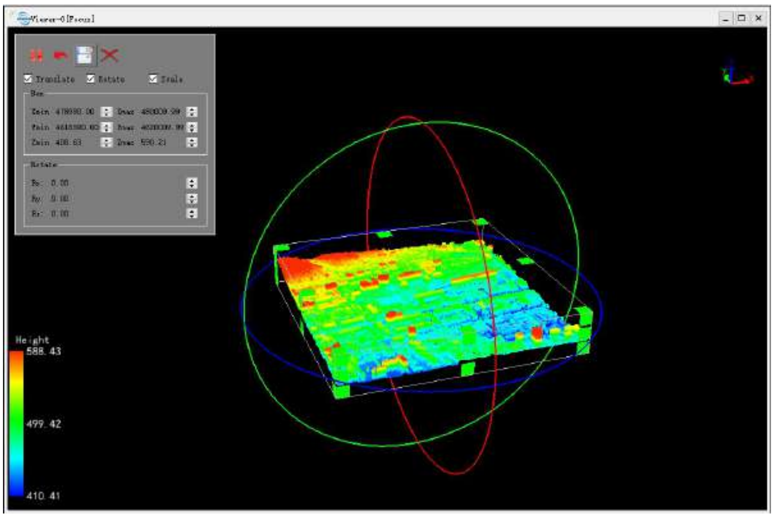
Brief: Cross select partial point cloud using ROI. The boundary and the rotation angles of the ROI can be entered in the user-interface, or interactively changed by using the mouse. This function can be used for flood analysis and other applications.

Steps

1. Click this button to open the dialog shown below:



2. The buttons of translation, rotation, and scale ratio control the translation, rotation, and scale ratio of the ROI.
3. Click button to stop the editing of the ROI and return to the normal interactive user-interface.
4. Click button to reset ROI.
5. Click button to export point cloud inside the ROI to a new LiData File.
6. Click button to exit cross selection.
7. The whole display effect is shown below. Drag any face of the ROI to translate it. And the dragged face will be highlighted. Drag the red, green, or blue track circle to rotate the ROI about the X, Y, or Z axis. The dragged track circle will be highlighted. Drag the green square to rescale the ROI. The dragged square will be highlighted.



Settings

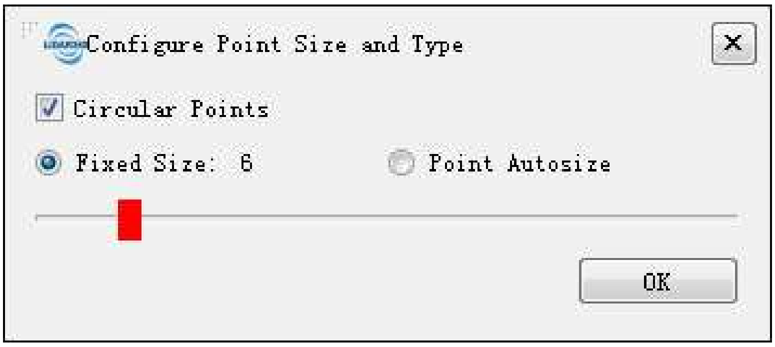
- **Hot Key**: Not applicable.
- **Prerequisite**: Current viewer contains point cloud.
- **Translation**: Switch of using mouse to translate the ROI.
- **Scale**: Switch of using mouse to rescale the ROI.
- **Rotation**: Switch of using mouse to rotate the ROI.
- **Boundary of ROI**: Precisely control the max. and min. value in X, Y, and Z direction of the ROI.
- **Rotation angles of ROI**: Control the rotation angles of the ROI in X, Y, and Z axis.

> Note: This function is for all point clouds in the current viewer.

Configure Point Size and Type

 **Brief**: Configure point size and type.

Steps

1. Click this button to open the dialog shown as below:


2. Configure point size and type.

Settings

- **Circular Points**: Determine point type. If checked, point will be rendered as circle, otherwise, point will be rendered as square.
- **Fixed Size**: If checked, point size would be fixed. User can adjust point size using the slider below (range 0-50 pixels).
- **Point Autosize**: If checked, point size is auto changing based on the depth of the viewer.

> Note: This function is global. If user wants to configure a single point cloud, just right-click it, select Configure Point Size and Type from the context menu.

New Window

Description: This function can add a new empty window to the system.

2D/3D Viewer

2D **Function Description**: The currently active window is displayed in 2D mode. After switching the current window to 2D mode, a rotation angle adjustment slider will appear on the status bar. Dragging the slider allows you to set the window's rotation angle, which ranges from 0° to 360°, with clockwise being positive.

3D **Function Description**: The currently active window is displayed in 3D mode.

> Note: If the view contains raster data, it will default to 2D mode and cannot switch to 3D mode. After deleting the raster data, it can switch to 3D mode.

Profile View

Description: The section editing function allows the user to display the point cloud data of any rectangular area in the main window in the section window, which is convenient for the user to view, measure, and modify the class.

The use of LiDAR data to produce high-precision terrain products requires manual inspection of the results of automatic classification. In this case, profile viewing tools are required. LiDAR360 provides profile view function starting from V5.0 version, which can complete classification inspection more efficiently.

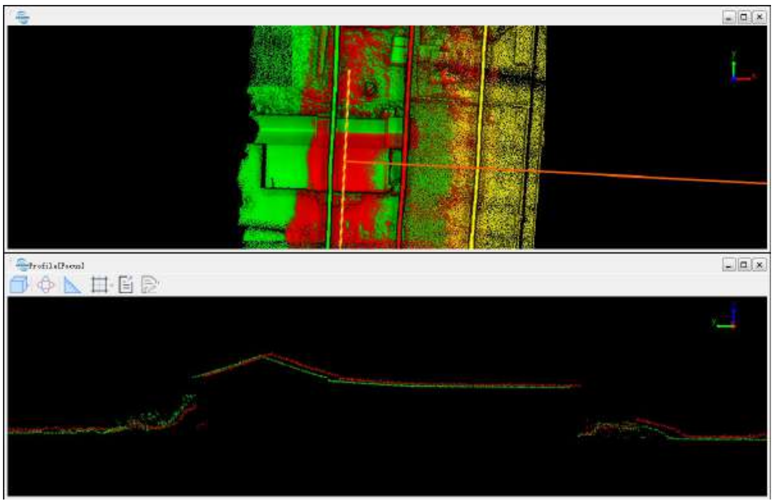
- Draw Profile
- Profile Viewer Tool

Profile and Measurement Tools

When the profile is turned on, the point cloud window does not support the measurement function. The profile window supports single-point selection, multi-point selection, length measurement, angle measurement, slope measurement, and height measurement.

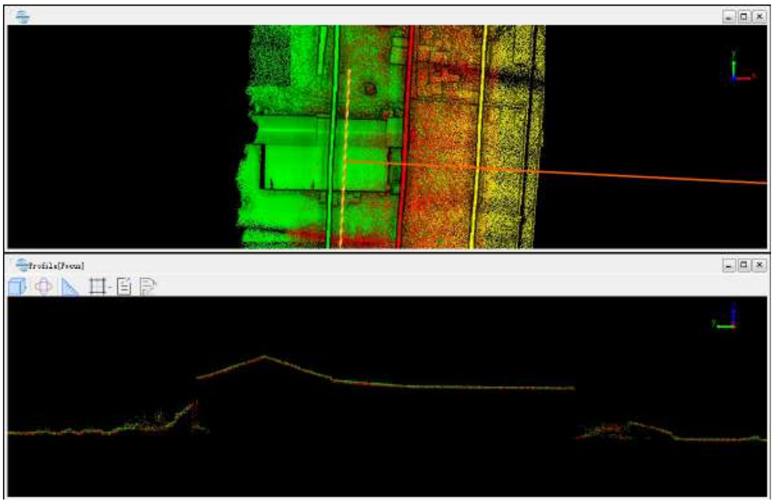
Section Window and Strip Alignment

In the track stitching function, we need to view the data position difference between different flight belts caused by the set-up angle error in the profile window.



The above picture can clearly see the relative position of the point cloud before splicing.

During the splicing process, you can view the splicing effect of different parameters in real time through the section window:



After ensuring that it is correct, the point cloud can be transformed according to the current parameters and written directly to the disk.

Profile View and Strip Alignment

The measurement tools of the profile window can also assist in the error estimation of the pitch, roll and heading angles in Strip Alignment function. Please refer to Strip Alignment for more details.

Profile View and ALS Editor

When the ALS Editor toolbar is opened, the host window will be set to 2D view. The Profile tool can help to check whether the seed points positions are accurate in 3D. Please refer to ALS Editor for more details.

> Note： ALS Editor has its own profile tools.

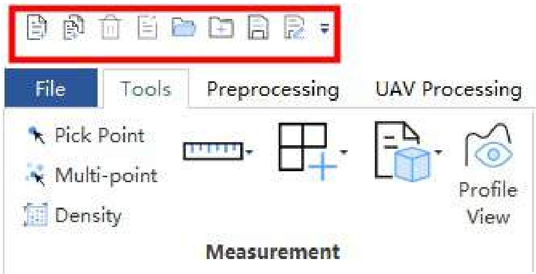
Profile View and TLS Editor

When the TLS Editor toolbar is opened, the host window will be set to 2D view. The Profile tool can help to check whether the seed points positions are accurate in 3D. Please refer to TLS Editor for more details.

> Note： TLS Editor has its own profile tools.

Quick Start Toolbar

As shown in the image below, the quick toolbar is located at the top left corner of the software and includes the following functionalities: Add Data, Delete Data, Export Data, Project Operation.



Add Data

Functional Description: Data types supported by LiDAR360 include point cloud, raster, vector, table, model, trajectory and aerotriangulation project.

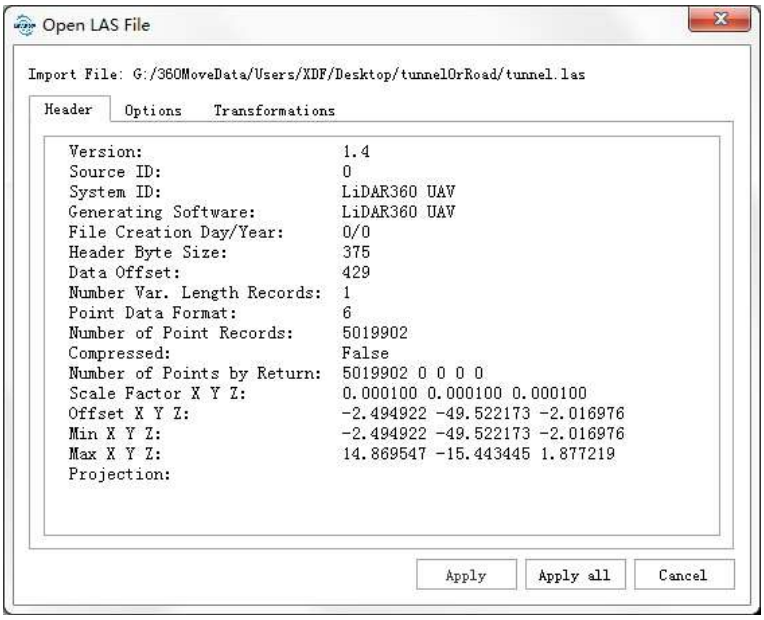
LiDAR360 supports point cloud data formats including LiData (*.LiDataProprietary LiData File), LAS (*.las,*.laz), ASCII (*.txt, *.asc, *.neu, *.xyz, *.pts, *.csv), PLY (*.ply), E57(*.e57), PCD(*.pcd).LiData is a customized point cloud data format. All analysis and processing operations related to point clouds are based on this format. After importing point cloud data in LAS, ASCII, PLY, E57, PCD and other formats into the software, the corresponding LiData format will be automatically generated.

When loading data for the first time, data will be converted into LiData format which can support efficient browsing and processing of massive data according to the pop-up loading interface and different requirements of users.

Click*File->Data->Add Data* (in addition, you can also click) to bring up the Add Data interface.

Add LAS Data

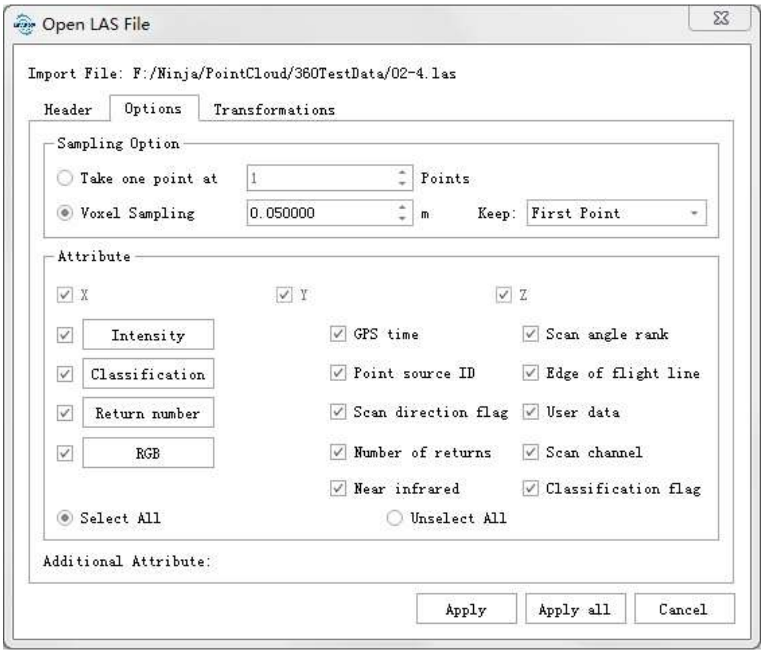
1. Select the LAS file to be loaded, the first load will bring up the screen shown in the following figure. The top of the screen shows the path of the LAS data to be opened. The Header tab describes the information about the LAS header file.Contains the LAS data version number, source ID, system identifier, generating software, file creation date, file header size......From the start of the file to the first point data record the number of bytes of the first field, the number of variable length records, the point data format ID, the number of point data records, whether to compress, the number of points of each echo count, X/Y/Z scale factor, X/Y/Z offset, the minimum X/Y/Z coordinates, the maximum X/Y/Z coordinates and other information.



2. Thinning The option tab allows you to thin the point cloud when it is opened. Thinning supports two modes. Spatial thinning and voxel thinning.
    - **Take one point at n Point (default "1")**: Import one point every n points.
    - **Voxel Sampling**: The voxel thinning method reduces the point cloud density by removing some points that are close to each other or within a grid cell of a given size. Firstly, the mesh is refined based on 3D gird, and then a point in the grid is retained with the corresponding strategy.
    - **Voxel size (default: 0.05)**: Grid cell size for voxel sampling.
    - **Keep (default: "first point")**: Used to determine how the algorithm preserves points entering the grid. See the table below for methods and descriptions.

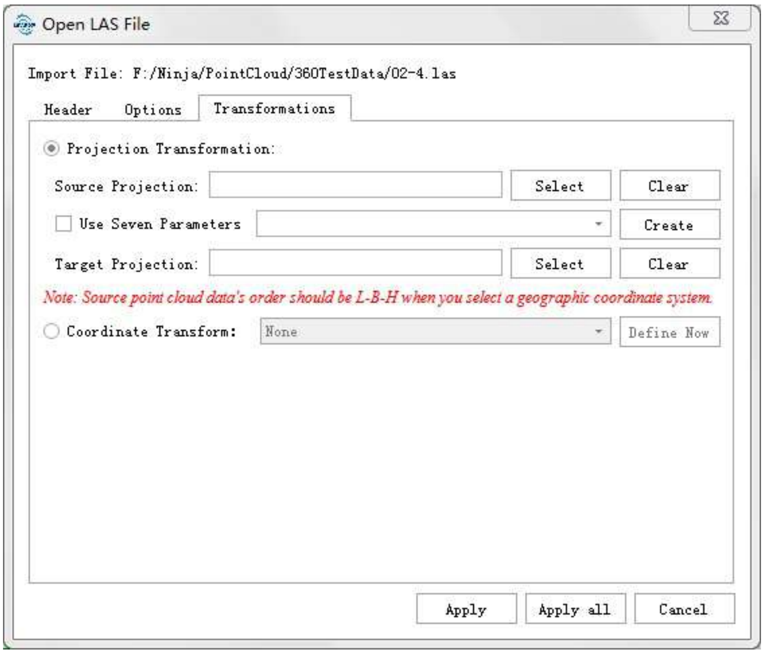
Reserve the point policy	description
First point	The first point to enter the voxel is reserved
Highest point	The highest point in the voxel is reserved
Lowest point	The lowest point in the voxel is reserved
Median Z Point	Points in the grid whose Z value is in the middle position are reserved
Average Point	The average point is the average of all the points in the voxel
Voxel Center point	Keep the center of the voxel, not the point cloud
Highest Intensity	The most reflective points in the voxel will be retained
Lowest Intensity	The least reflective points in the voxel will be retained
Max GPS Time	The latest point in the voxel is reserved
Min GPS time	The earliest point in the voxel is reserved

1. Attribute selection. Select attributes and additional attributes of LAS data. By default, all properties of LAS data are imported.



2. Select a coordinate system or coordinate conversion. You can define coordinate system for point cloud data, or do re-projection or custom coordinate transformation. The corresponding coordinate system can be quickly found by inputing the keywords of coordinate system, or click the drop-down

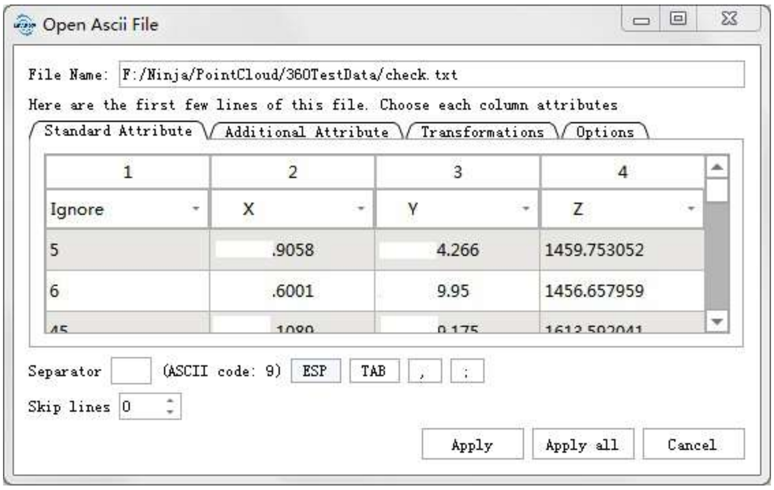
menu of add coordinate system button and import coordinate system from WKT or PRJ. Refer to Reprojection.



3. Projection transformation.
    - Source coordinate system. If the user only selects the "source" coordinate system, the "source coordinate system" will be defined by the imported point cloud and overwrite the existing coordinate system of the Las file. Similar to the function "Define Projection".
    - Target coordinate system. If the user has selected the "source" coordinate system, the "target projection" can also be selected to complete the reprojection operation. When the user selects the target coordinate system, the "source" coordinate system must be selected.
4. Coordinate Conversion.
    - Users can define and apply coordinate conversion relationships such as seven-parameter, four-parameter, 3D affine conversion, XYMultiply, and linear conversion to point clouds. Refer to creating a transformation relationship.
5. Click*Apply*to import the selected LAS data into the software using the current settings, and start to load the point cloud. If you select*Apply All*, this setting will also be adopted for other LAS data that you imported before closing the software, and the dialog box for opening LAS data will not pop up again.

Import TXT File

1. Select the TXT file to be loaded, and the interface as shown in the following figure will pop up:



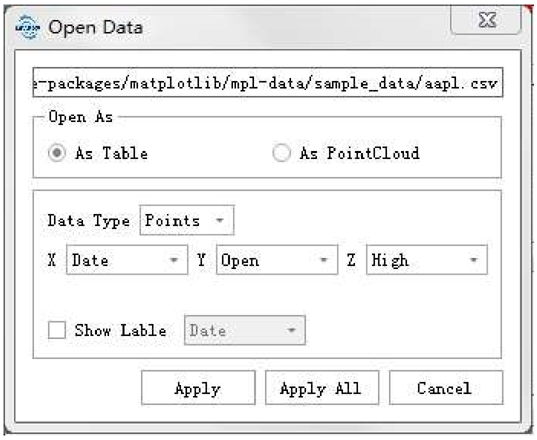
The data path to be opened is displayed at the file name, and if there is a header in the data, the header line is highlighted in red. Open the additional attribute page, check the required additional attribute information, double-click the column name of "Attribute name" to edit the additional attribute name, and select the data type of the additional attribute. Currently, only Integer and real data types are supported for ASCII data.

1. Delimiter. In general, the software can automatically identify the Separator of a file and the user can select the corresponding **Separator** in the separator column.
2. Skip data: File headers and other unwanted data can be ignored by skipping line X on the Skip lines.
3. Data selection: The software will select the column where X, Y and Z are located by default. The user can click the drop-down button above each column to select the corresponding attribute of the data in each column. Select Ignore to ignore the column data, which can be imported as an additional attribute.
4. Click the additional attributes page and select the additional attributes to be imported. The attributes can be renamed. Double-click the corresponding attribute name in the "Attribute name" column for modification. The types of additional attributes support integers and reals.

5. Thinning Refer to **"thinning"** in LAS Data.
6. For coordinate system and coordinate conversion options, see **"Select Coordinate System or Coordinate Conversion"** in Add LAS Data.

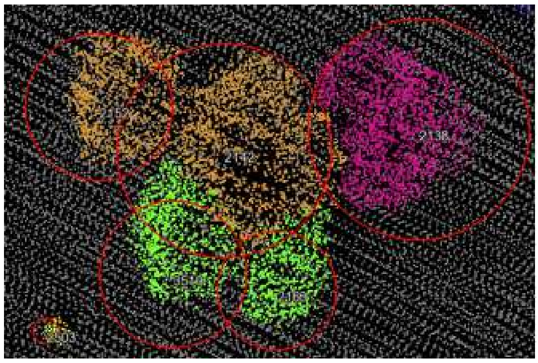
Import CSV File

1. Select CSV data to load and the interface as shown in the following figure will pop up:



   CSV data can be optionally opened as a table or a point cloud.

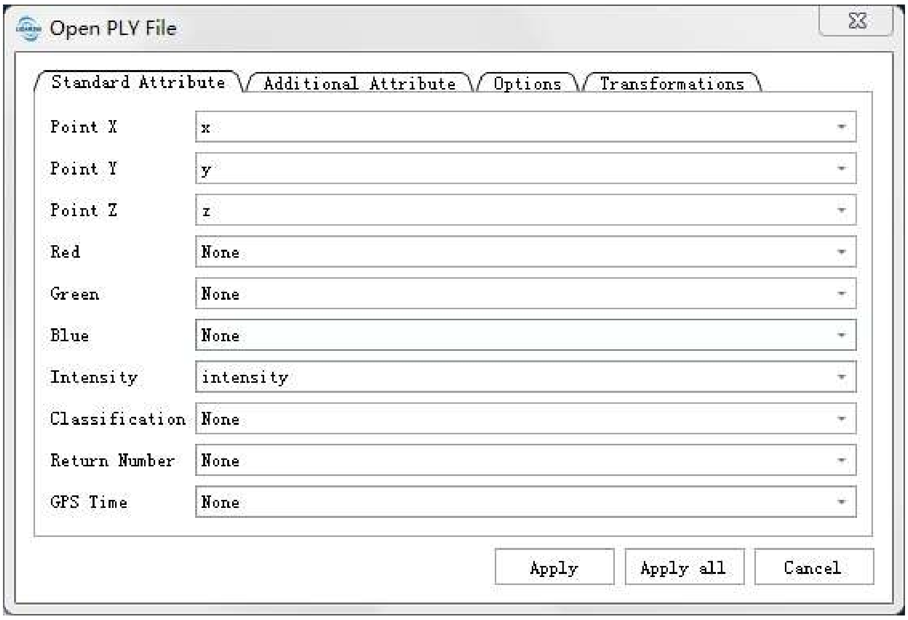
2. In general, if the CSV file is generated by tree segmentation, you are advised to open it as a table. The data type can be points or circles. If you select the point mode, specify the columns corresponding to X, Y, and Z. If you choose to display in a circle, in addition to X, Y, and Z, you also need to specify the columns corresponding to the diameter of the circle:



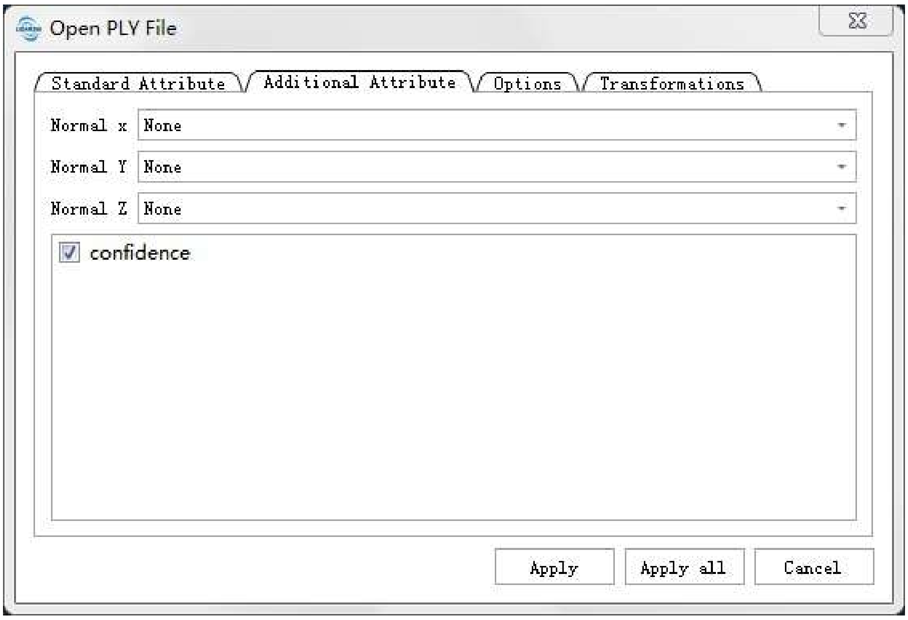
3. The Display label's check box determines whether to display labels of each point or circle in the software. If you select display labels, specify columns corresponding to the labels, for example, ID of the segmented tree. If you select open as a point cloud, it will pop up the same page as that displayed for opening TXT data. For details, see the description of adding TXT data.

Import PLY File

1. Select CSV file to load and the interface as shown in the following figure will pop up:



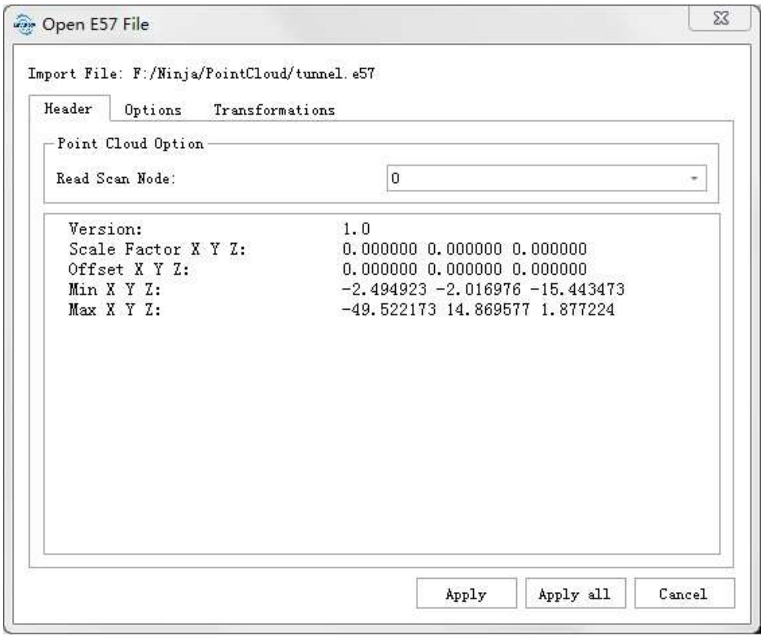
2. Specify the attributes corresponding to the X, Y, and Z coordinates.
3. Select the corresponding fields if there is Intensity information, and select the attributes corresponding to R, G, and B if there is color information. If not, select None.
4. Click the additional attributes page. If the PLY file has Normal X,Normal Y. Then you can import the Normal information as an additional attribute, and other additional attributes are displayed in the list. Choose the selected information to generate related additional attributes.



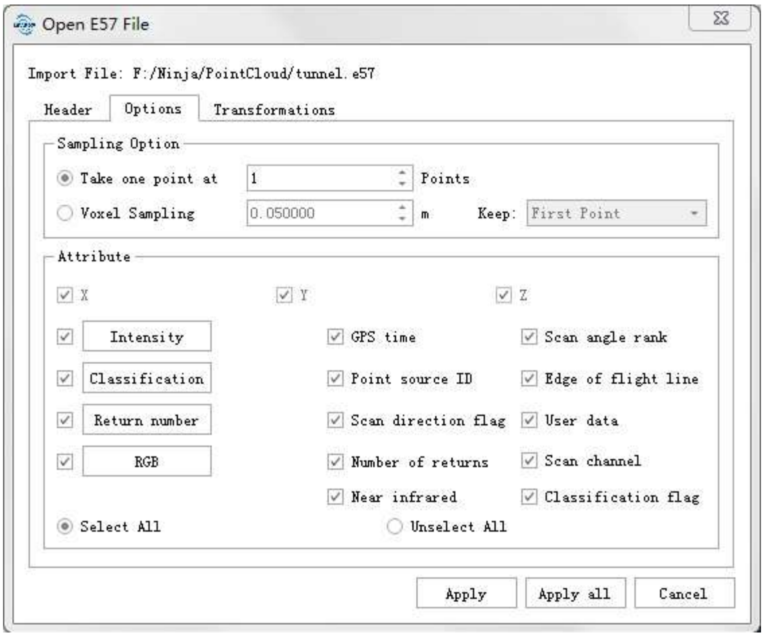
5. For coordinate system and coordinate conversion options, see **"Select Coordinate System or Coordinate Conversion"** in Add LAS Data
6. Thinning Refer to **"Thinning"** in LAS Data.
7. After setting, click*Apply*.

Import E57 File

1. Select the E57 file to be loaded.When loading for the first time, the interface as shown in the following figure will pop up. The top of the interface displays the path of E57 data to be opened. The Header tab describes the E57 header file information, including the name of the scanned data node of E57 data, the version number, the XYZ scaling factor, the offset, and the surrounding box information.



2. In Option tab, you can select to thin the point cloud when it is opened and open all points by default. You can also select the attributes of E57 data and additional attributes and import all attributes of E57 data by default.



3. For coordinate system and coordinate conversion options, see **"Select Coordinate System or Coordinate Conversion"** in Add LAS Data

4. Thinning Refer to **"Thinning"** in LAS Data.

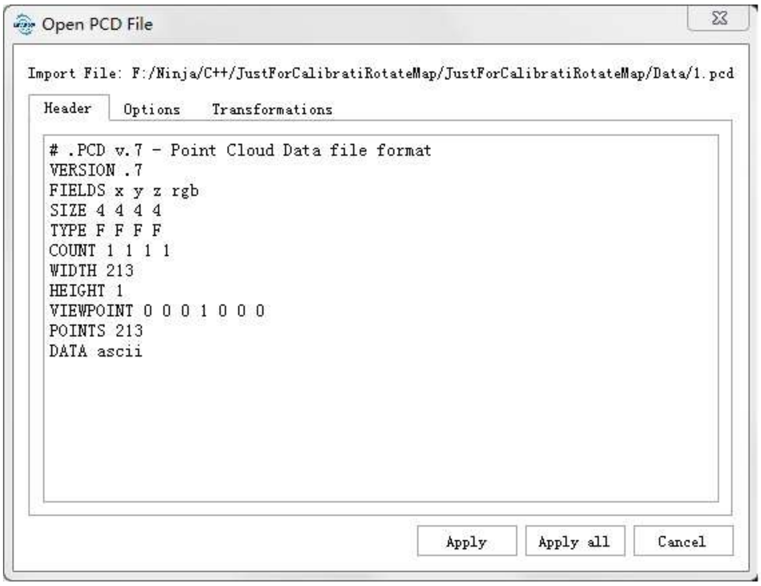
5. Click*Apply*Indicates that the selected E57 data is imported to the software using the current settings, and loading the point cloud . If you choose*Apply All*, this setting is adopted for all data opened this time. The E57 data opening dialog box will not pop up again until E57 data is opened again.

Import PCD File

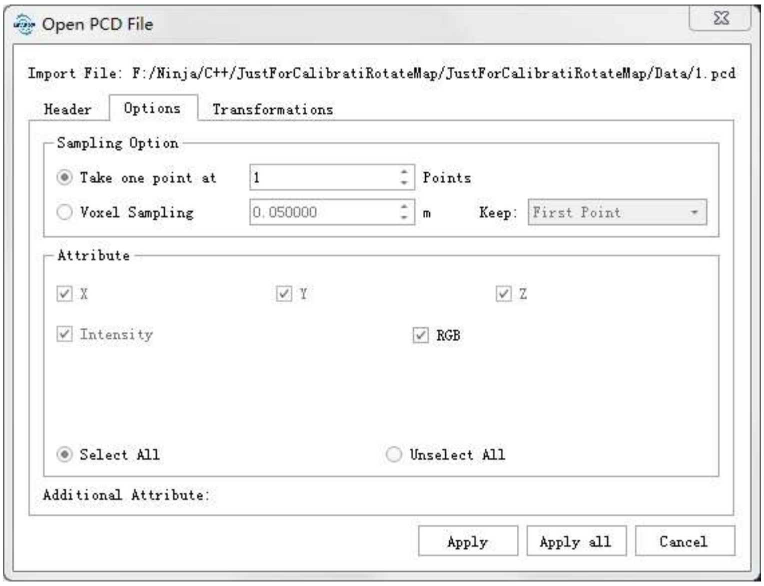
LiDar360 supports the reading of several PCD fixed fields:

The PCD field	corresponds to the attributes read into 360
x/y/z	x/y/z
normal_x	Additional attribute - The x-component of Normals
normal_y	Additional attribute - The y-component of Normals
normal_z	Additional attribute - The z-component of Normals
intensity	Intensity values
rgb	r, g, b components of rgb point cloud

1. Select the PCD file to be loaded. The interface shown in the following figure will pop up when the PCD file is loaded for the first time.



2. In Option tab, you can select to thin the point cloud when it is opened, and open all points by default. You can also select the attributes of PCD data and additional attributes, and import all attributes of PCD data by default.



3. For coordinate system and coordinate conversion options, see **"Select Coordinate System or Coordinate Conversion"** in Add LAS Data

4. Thinning Refer to **"Thinning"** in LAS Data.

Import Raster File

Raster file is a data form that divides the space into regular grids, each grid is called a cell, and assigns corresponding attribute values to each cell to represent the entity. In its simplest form, a raster consists of a matrix of cells (pixels) organized into rows and columns (grid) where each cell contains a value representing information. The position of each cell (pixel) is defined by its row number, and the entity position represented is implicit in the raster row position. Because of this, it is often easy to write efficient code quickly while doing data analysis.

1. Click *File* > *Data* > *Add Data*.

2. Select the raster data to be loaded and click *open*.

Import Vector File

Vector data can be represented in its original resolution and form, graphical output is often more aesthetic (compared to traditional map representation), it doesn't require data conversion, and can maintain accurate geographic location. .shp, .dxf, and.kml formats are supported.

1. Click *File* > *Data* > *Add Data*.

2. Select the vector data to be loaded and*click open*.

Import Model File

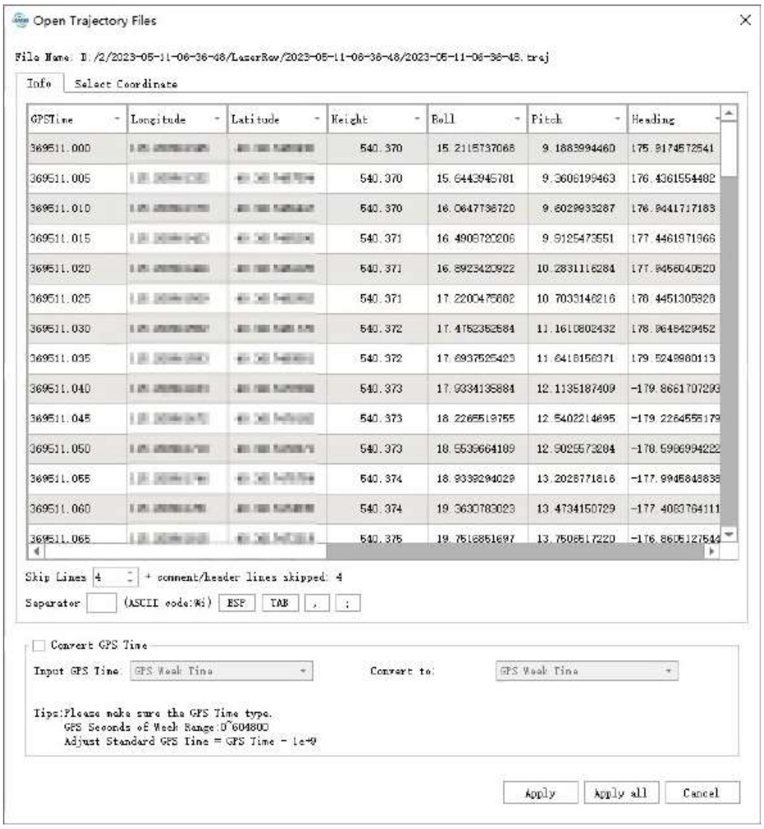
LiDAR360 support Model data formats including Model (*.LiModel,*.LiTin,*.LiBIM,*.LiTree), OSG data(*.osgb,*.ive,*.desc,*.obj), DXF File (*.dxf), among which LiModel file generates regular triangulation model according to DEM or DSM, saves regular grid dot, organizes and stores regular triangulation model in blocks according to quad-tree, and can also superimpose DOM texture information on it. LiTin fileis an irregular 2.5D triangulation model generated according to the point cloud. It is colored according to the elevation, and the light and shadow effects are used to improve the display effect. It can be flat, delete, add vertexes, increase the fracture line and other editing, improve the quality of contour generated according to it. LiBIM is a building model based on the point cloud. LiTree is a tree model filegenerated based on individual tree information.

1. Click *File* > *Data* > *Add Data*.

2. Select the model data to be loaded and*click open*.

> Note: LiDAR360 supports drag and drop to add data. If it doesn't work, clickhereto see more.

Import Trajectory File



Currently, LiDAR360 supports three types of trajectory files（*.traj、*.pos、*.out）。

For traj and pos formats, the fields to be set, including field order, skip lines, and delimiter. In the fields, the required fields include GPS time, longitude and latitude (or GridX, GridY), height, Roll angle, Pitch angle, and Heading angle. For the *.out format, no fields need to be selected.

If the GPS time of the input point cloud data is SOW(seconds of aweek), it needs to be converted to GPS Time. The acquisition date of the input data calculates the GPS week number internally, firstly calculates the GPS standard time, and then subtracts 109 from the GPS Coordinated Universal Time.

When the longitude and latitude fields are set but the GridX and GridY fields are not set, the coordinate system to be converted needs to be set on the coordinate system selection page. The recommended default coordinate system is the corresponding WGS 84 UTM projection coordinate system for the longitude. If set at the same time, the GridX and GridY will be used directly as the projection coordinate values without projection transformation. The out format trajectory file only contains longitude and latitude information, so a coordinate system also needs to be set. The method for setting the coordinate system can refer to the Define Projection page.

Import aerotriangulation Result

The aerial triangulation results in LiDAR360 contain both interior and exterior orientation elements of the images, and some formats also contain tie point information. LiDAR360 currently supports several formats for aerial triangulation results, including internal formats (.*LiAep), LiMapper projects (*.limap), LiGeoreference projects containing images (.*ligeo), BlocksExchange xml format (*.xml), Pix4d projects (.*p4d), and Inpho projects (*.prj). When a format other than the internal image project is selected, it will be converted to the internal image project (*.LiAep) for use in other functions.

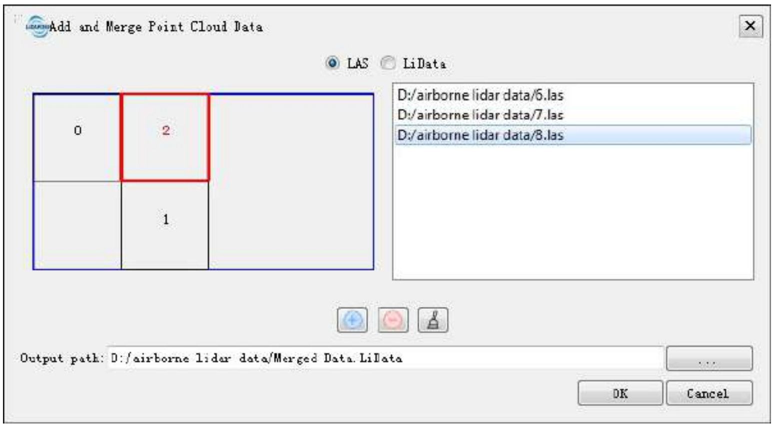
> Note:LiDAR360 supports adding this type of data directly by drag and drop. If dragging and dropping does not work, click here to see more.

Add and Merge Point Cloud Data

Brief: Merge two or more point clouds in LAS/LAZ or LiData format to one single point cloud in LiData format.

Steps

1. Click tool botton to open the Add and Merge Data window.



   Bounding boxes of point clouds are displayed in the left area. Bounding box of the selected point cloud is highlighted in red.

2. Select LAS or LiData as your data type.
3. Click to add LAS/LiData file(s).
4. Click to remove LAS/LiData file(s).
5. Click to remove all file(s).

Delete Data

Function Description: Adding and deleting data are frequent operations during data processing.

Steps

1. Select the data to be deleted from the data directory tree. At this point, the icon in the main menu bar will be activated.
2. Click the icon to remove the selected data.

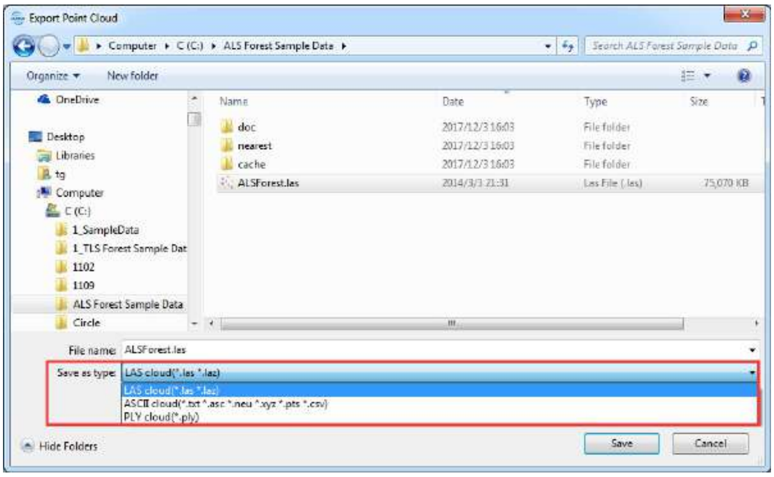
> Note: During certain processing functions, LiDAR360 does not allow data to be removed. Even if data is selected in the directory tree, the icon will not be activated until the function is completed, such as during profile operations.

Exporting Data

Function Description: As described in Add Data, LiDAR360 supports the import of various point cloud data formats and converts them into the LiData point cloud file format. Additionally, it supports exporting LiData data into multiple formats, such as LAS (*.las, *.laz) (versions 1.2-1.4), ASCII (*.txt, *.asc, *.neu, *.xyz, *.pts, *.csv), PLY (*.ply), and E57 (*.e57), among others.

Steps

1. Select the data to be exported from the data directory tree.
2. Right-click to open the menu, choose *Export*, or directly click the icon in the main menu to open the settings menu.
3. Configure the export file type, export path, and file name:



4. Click *Save* to complete the operation.

Project Operations

New Project

 **Function Description**: When starting to process new data, it is best to create a new project. If you do not create one, a new unnamed project will be created by default, and you only need to save the project when closing the software.

1. Click  or select *File* -> *New Project* from the main menu.
2. If you make changes to the current project, ensure you save the changes before closing.
3. Saving a project file only saves the loaded data layers and project settings, not the actual data. When transferring a project file to another storage location, move all files associated with that project (keeping their relative paths).

Open Project

 **Function Description**: Click on *File -> Open Project* in the main menu to open the project dialog, then select the project file you need to load and click *Open*. Opening a project file (*.LiPrj) quickly restores the original settings.

Save Project

 **Function Description**: Save the current project. Clicking this button will bring up the save dialog. Set the storage path and project name before saving.

Save Project As

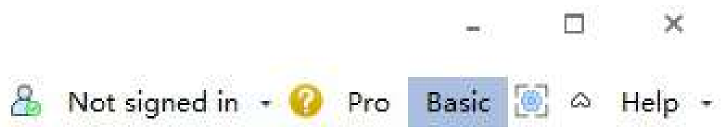
 **Function Description**: Click this button or select *File -> Save Project As* from the main menu to open the "Save As" dialog, set the save path and new project name. Click *Save* to complete the operation. Save the project file (*.LiPrj).

Close Project

After completing your current work, you can click *File* -> *Close Project* in the main menu. This will close the current project and open the next one.

Ribbon Toolbar

Ribbon Toolbar is at the upper-right corner of interface, includes:Batch Processing, Help documents, Advanced mode, Basic mode, Display Optionand Options. Options includes Theme, Title Group Visible,Help, Quick Start, FAQ, Visit Company Website, Submit A BUG, Submit An Enhancement, and Check for Updates.



Advanced and Basic modes are available for Terrain, ALS and TLS modules. Advanced mode provides all related tools regarding to each application. Users can decide which tool to use on their demands, based on different datasets. While Basic mode provides a fast way of analyzing. After setting all parameters and click "Run", data can be batch processed and saved to specified output path.

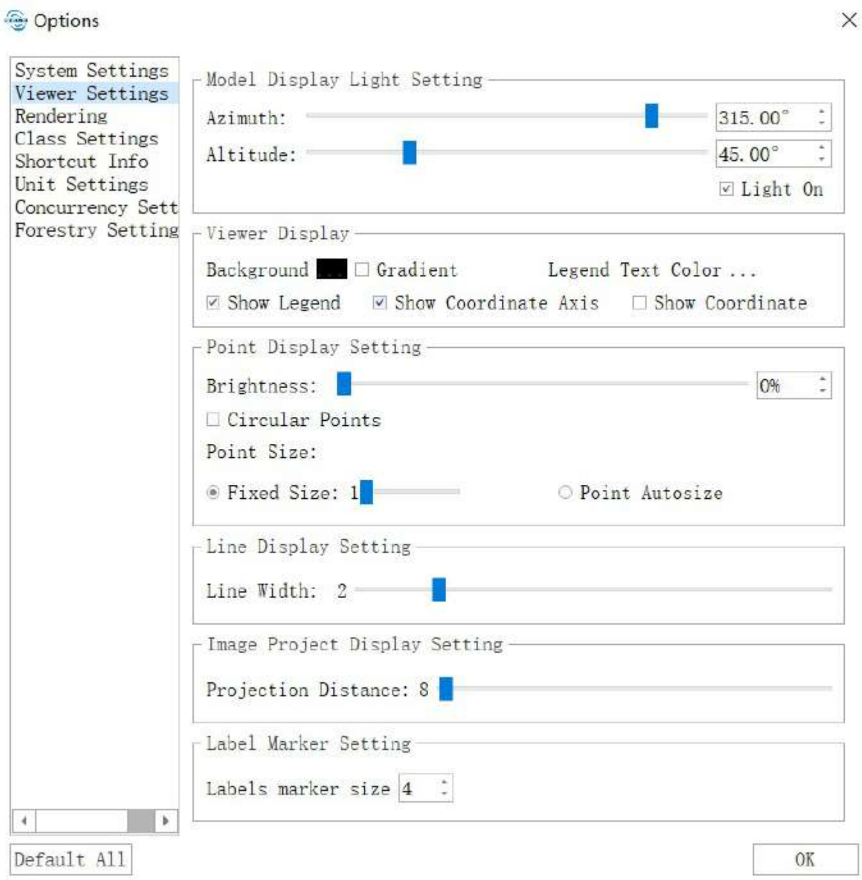
Options

Function Description: This feature includes software system settings, view settings, rendering, category priority settings, shortcuts, and measurement settings. Category priority determines the order of categories in cross-sections, and the order is reflected in the classification interface and category settings interface. Shortcuts allow for quick access to commonly used system functions.

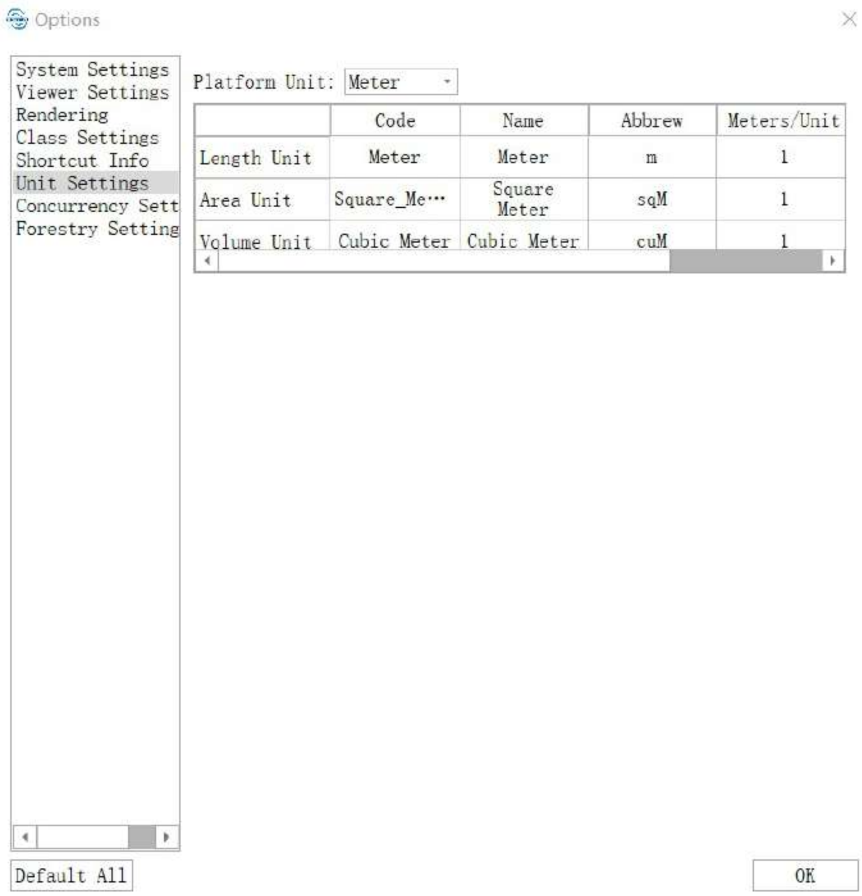
Steps

1. Upon selecting this feature, the following interface will appear. System settings include interface theme and language settings. Users can choose from themes such as Landscape, Blue, White, and Dark Gray. Languages available are English, Chinese, French, Japanese, Korean, and Russian.

2. The "Viewer Settings" option allows users to configure the display of models in all software views, including lighting settings, background color (enabling the "Gradient" option next to the background color allows for a gradient background effect), point display in point clouds, and label properties. Lighting settings apply only to model files (such as LiModel, LiTin, OSGB, etc.) and allow for changes to the light source position.

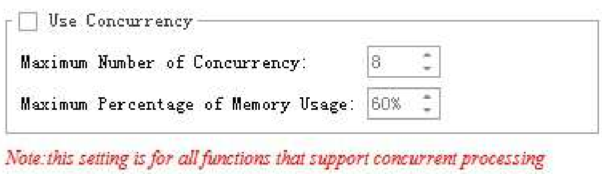


3. The "Rendering" option includes real-time contour settings. Users can configure the contour base, contour interval, line width, primary and index contour colors, etc. The "Use shader rendering color strategy" checkbox determines whether to use shaders for rendering the color strategy of point clouds. The shader rendering color strategy allows the same point cloud to be displayed with different color strategies in different windows, utilizing programmable pipeline shader technology.
4. The "Class Settings" option allows users to define default categories within the software.
5. The "Shortcut Info" option enables users to customize keyboard shortcuts. Refer to Shortcuts for details on how to set them.
6. The "Unit Settings" option changes the interface as shown below.



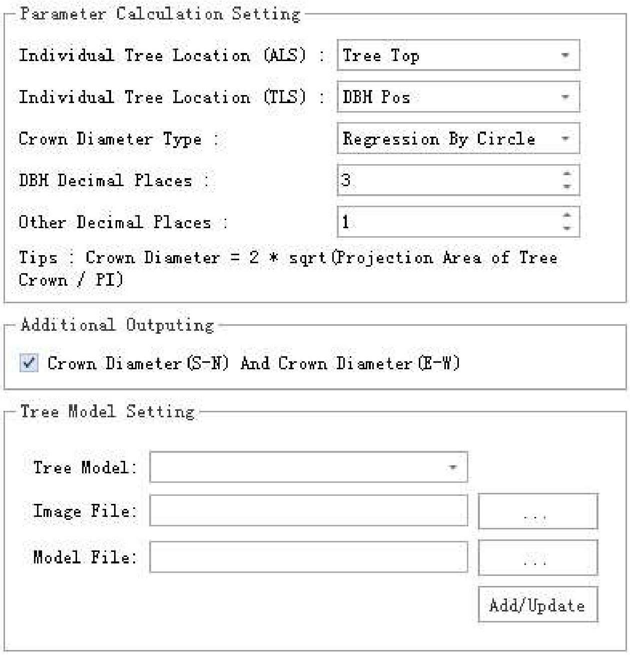
If the source and target units are the same, the measurement results will be displayed in the corresponding units. If the source and target units differ, the results will be multiplied by the scale factor between the units, and the displayed unit will be the actual target unit.

7. Clicking the "Concurrency Settings" option changes the interface as shown below.



Here, users can configure the concurrency parameters used in certain computational processes. The "Use Concurrency" checkbox determines whether concurrency is enabled. Concurrency parameters can be applied to functions such as noise removal, noise filtering, and point cloud smoothing. "Maximum Concurrency Count" refers to the maximum number of threads used during computation, while "Maximum Memory Usage Ratio" specifies the maximum percentage of physical memory that can be used during concurrency.

8. Clicking the "Forestry Settings" option changes the interface as shown below.



This section allows users to configure the settings for individual tree segmentation, attribute statistics output, and tree models within the forestry module.

- **Parameter Calculation Settings**:
  - **Individual Tree Location (Airborne Forestry)**: Used to set the type of tree orientation used in the output results of airborne forestry individual tree segmentation or attribute statistics. Options include "Tree Top" and "Centroid."
  - **Individual Tree Location (Ground-based Forestry)**: Used to set the type of tree orientation used in the output results of ground-based forestry. Options include "Tree Top," "Centroid," and "Diameter at Breast Height (DBH) Position."
  - **Crown Diameter Calculation Method**: Used to set the calculation method for "Crown Diameter" in the results. Options include "Average," "Derived from Fitted Circle," and "Derived from Fitted Ellipse."

```
- The "Average" crown diameter calculation formula is "Crown Diameter = (North-South Crown Diameter + East-West Crown Diameter) / 2."
- The "Derived from Fitted Circle" calculation formula is "Crown Diameter = 2 * sqrt(Canopy Projection Area / π)."
- The "Derived from Fitted Ellipse" calculation formula is "Crown Diameter = sqrt(North-South Crown Diameter * East-West Crown Diameter)."
```

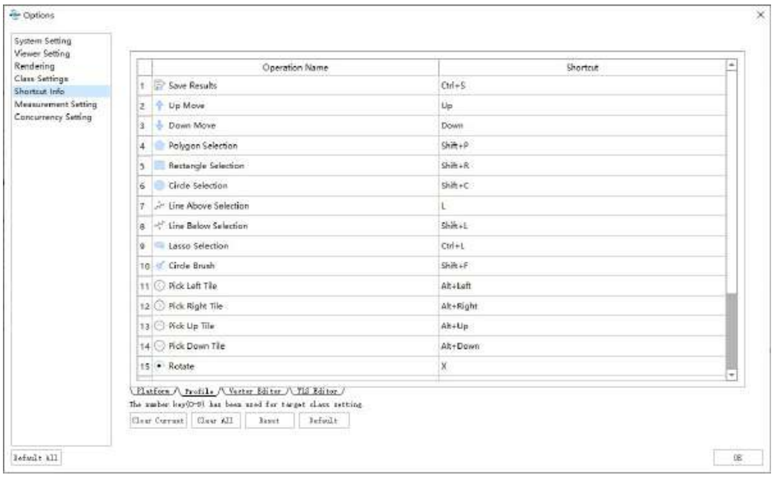
  - **Decimal Places for DBH Output**: Used to set the number of decimal places retained for DBH values in the output results.
  - **Precision of Other Attributes**: Used to set the number of decimal places retained for other attributes in the output results, excluding DBH.
  - **DBH Height Range**: Used to set the height range of point clouds used to calculate DBH. Most countries use 1.3m (1.2~1.4), while Japan typically uses 1.2m.
  - **Maximum Crown Width (Default is 15m)**: The maximum crown width of all standing trees in a stand. This parameter is mainly used for individual tree segmentation and attribute statistics. If set too small, it may lead to over-segmentation; if set too large, it may reduce program efficiency.

    - **DBH Fitting Type (Default is Circular Fitting)**: Supports circular fitting, elliptical fitting, and cylindrical fitting.
  - **Additional Output Items**: Used to determine whether to include "North-South Crown Diameter" and "East-West Crown Diameter" in the output results.
- **Tree Model Settings**:
  - **Tree Model**: Select the tree model to update or delete. Choose "Add Custom Tree Model" to add a new tree model.
  - **Model Name**: Set the name for the new tree model. This option appears when "Add Custom Tree Model" is selected in the tree model dropdown.
  - **Image File**: Set the image file used for distant view display in this model. The tree will be displayed using a crossed pattern of this image file when viewed from a distance.
  - **Model File**: Set the detailed model file used for close-up display. The tree will be displayed using this model when viewed up close. Note: The Y-axis direction of the model corresponds to the tree height direction, and the origin of the model is at the base of the tree.
  - **Remove**: Remove the currently selected tree model.
  - **Add/Update**: Add the currently set tree model. If a model with the same name exists, it will be updated.

9. Click the "Restore Defaults" button to restore system defaults.
10. The "P" keyboard shortcut can be used to adjust the light source position.

Shortcut

Function description: Set the shortcut keys for each function of the software.



STEP

1. Click the lower tab and select the module whose shortcut needs to be modified.
2. Find the function to be modified, double-click the edit box on the right, enter the edit mode and enter the shortcut key.

   > Note: If the set shortcut keys conflict with other shortcut keys, a prompt box will pop up, and the modification will be restored, which needs to be reset. Note: There are some general shortcut keys, such as **undo**, **redo**, etc. The program does not support modifying these shortcut keys. You can use one or more of the three keys Ctrl, Alt, and Shift to form key combinations with other keys.

3. Click the mouse to select a row, and click the **Clear Current** button to clear the current selection shortcut.
4. The **Clear All** button can clear the shortcut keys of all functions on the current page.
5. The **Reset Modification** button can reset the modified shortcut keys of all pages to the unmodified state.
6. The **Restore Default** button can restore the button shortcut keys of all pages to the default state of the software.
7. After the modification is completed, click **OK**, and the modified shortcut key will be saved and take effect.

Project Management Window

The data list in the project is managed by layers and windows separately.

- Layer Management
    - Point Cloud
    - Raster
    - Vector
    - Table
    - Model
    - Trajectory
    - Image Project
- Window Management

> Note: The data loaded into the project can be displayed in single/multiple window(s), or not displayed at all. Please drag data to a certain window for display.

Layer Management

Functional Overview

Layer management manages the data in the software by group, including: point cloud, raster, vector, table, model, among which:

- Point cloud data types include: LiData File (*.LiData Proprietary LiData File), LAS cloud (*.las, *.laz), ASCII Cloud (*.txt, *.asc, *.neu,*.xyz,*.pts,*.csv), E57 Cloud (*.e57), PLY cloud (*.ply);
- Raster data types include: Image (*.tif,*.jpg);
- Vector Data types include: Vector (*.shp,*.dxf,*.kml);
- Table data types include: Table (*.csv);
- Model data types include: Model (*.LiModelProprietary Model File, *.LiTin Proprietary TIN File, *.LiBIM,*.LiTree), OSG Data (*.osgb, *.ive, *.desc, *.obj);

The functions include the whole software system (all Windows) data removal, implicit control and so on. You can control the display and hiding of data in the entire software by selecting the check box of the data node, and drag the data in the data node to different Windows for display. The right-click menu of a data node is mainly used to query, display, collect statistics, export, and remove data. Different data types (including point clouds, rasters, vectors, tables, models, etc.) have different right-click menus.

Click the button (the red box in the figure below) to display the layer list as shown below:

Context Menu

Right-click the data to open the corresponding management menu.

- Layer management - Point Cloud
- Layer management - Raster
- Layer management _ Vector
- Layer management _ Table
- Layer management _ Model
- Trajectory
- Image Project

> Note: The right button menu of layer management tree is effective on all windows with the specified data loaded; while the right-click menu of window management tree is only effective on specified data in the specified window.

Point CLoud Context Menu

Summary

The context menus are used to for data import, removal, query, display, statistics, export, etc.

Data Type Context Menu

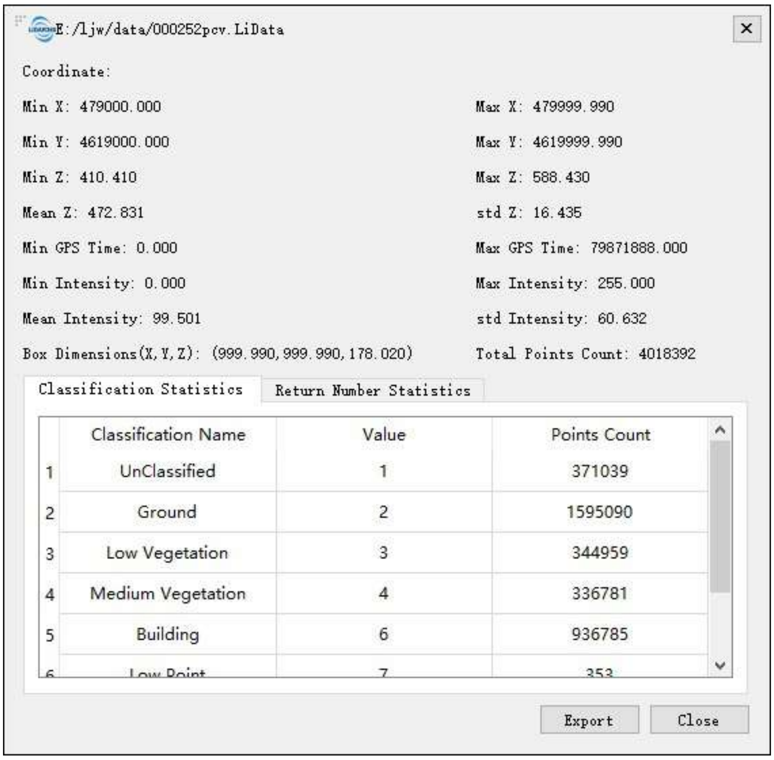
The user can open this menu by right clicking the point cloud data type.

- **Import Data**: The point cloud data formats supported by LiDAR360 include LiData (*.LiData), LAS (*.las, *.laz), ASCII (*.txt, *.asc, *.neu, *.xyz , *.pts, *.csv), PLY (*. ply), and E57 (*.e57). The LiData is the proprietary point cloud data format, on which the point cloud processing are based. Other imported formats of LAS, ASCII and PLY will be converted to LiData for subsequent processing. This function is the same as the tool Add Data.
- **Remove All**: Remove all the point cloud data from the project.

Data Context Menu

The user can open this menu by right clicking a point cloud data object.

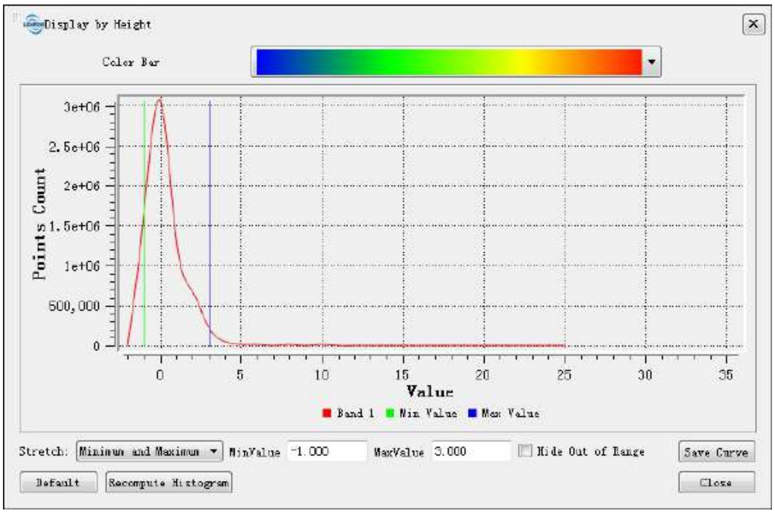
- **Info**: View the basic information of the point cloud, including the path of the data, the coordinate information, the minimum and maximum values of X, Y, and Z coordinates, the average and standard deviation of Z, the minimum and maximum GPS time, the minimum, maximum, and average intensity Value and standard deviation, point cloud bounding box, total points, point cloud category statistics and echo frequency statistics. If additional attribute information is included, the maximum, minimum and type of each additional attribute will be displayed. as the picture shows. Click the "Export" button to export the basic point cloud information as a txt file.



- **Open Containing Folder**: Open the folder.

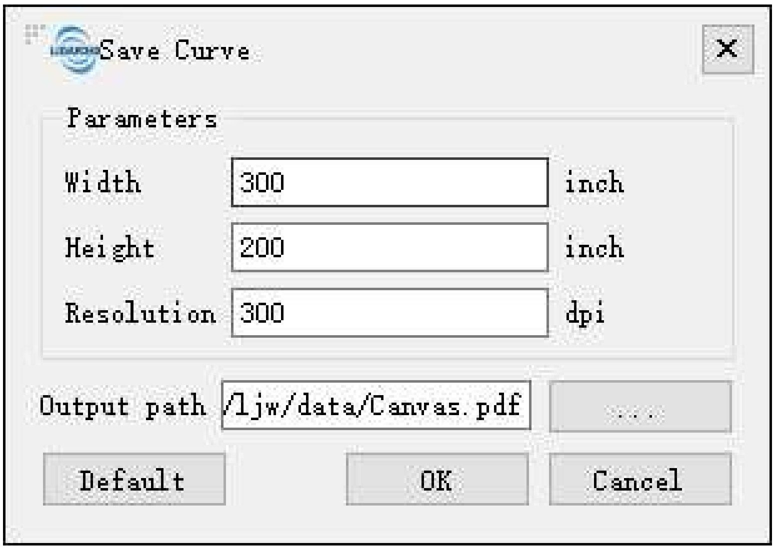
- **Rename**: Rename the file.
- **Display**: Set the display mode of a single point cloud file. LiDAR360 supports up to 20 display modes for a single point cloud file, including display by elevation, display by intensity, display by category, display by RGB, display by back Wave count display, GPS time display, tree ID display, route edge display, echo count display, point source ID display, scan angle display, scan direction display, user data display, scan channel display , Display by near-infrared, display by selected color, display by mixed display, display by combination display mode, display by additional attribute, display additional attribute by RGB, etc. Several display mode setting methods are described in detail below.

  - **Display by Elevation**: The interface as shown in the figure will pop up, and you can stretch by the minimum and maximum values or standard deviation to improve the display effect. When stretching according to the minimum and maximum values, you can set whether to display or hide the data outside the range of the minimum and maximum values.

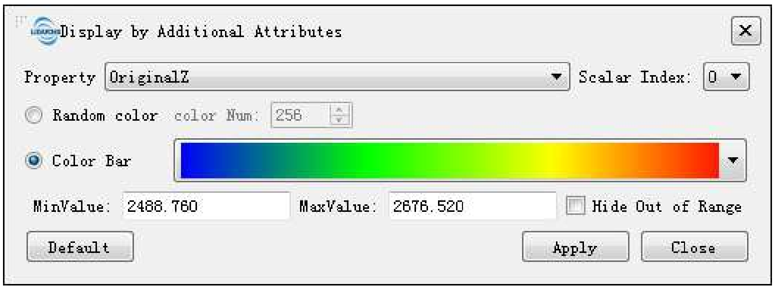


"Recalculate the histogram" is used to count and generate a histogram of all the points' elevation. When opening this function, the histogram of sampled points' elevation is displayed by default, and the histogram may be inaccurate. The recalculate the histogram function can take all the points' elevation into account, and generate an accurate histogram.

The curve can be saved in pdf files. Click the button **"Save Curve"** to pop up the following interface. Set the width, height, resolution, output path of the curve and click **"OK"** button to save the curve to local disk.



- **Display by Additional Attributes**：The following window will popup.



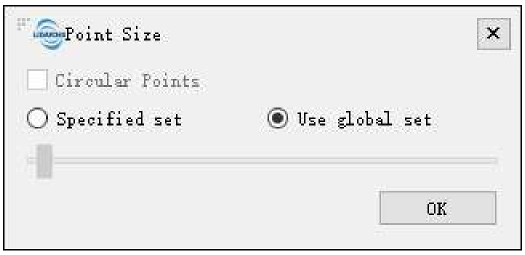

```
1. (Optional) Select the attribute needs to set the color from the attribute drop-down list.
2. (Optional) Select the index of the attribute needs to modify from the scalar index drop-down list box.
3. (Optional) Select a random color and set the number of colors to be configured in the Number of Colors box
.
4. (Optional) Select a color bar from the color bar drop-down list.
5. (Optional) Set the stretching range for the additional attribute component using the minimum and maximum v
alues.
6. (Optional) Check "Hide data outside range" to prevent points outside the minimum and maximum values from b
eing displayed.
7. (Optional) Click "Restore Defaults" to set the minimum and maximum value fields to the default range for t
he corresponding attribute component data.
8. (Optional) Click the "Apply" button to finalize the display effect as shown in the image.
9. Click the "Close" button to exit the settings interface.

 <div align = center><img src="../../ScreenShot/en/ProjectTree/LayerTreePC_DisplayByColorResult.png"alt="Laye
rTreePC_DisplayByAdditionalAttribute.png" width="480" style='margin-left:24%; '/></div>

- **Display Additional Attributes by RGB**: Open the interface as shown in the image.

<div align=center>
```

```
<img src="../../ScreenShot/en/ProjectTree/LayerTreePC_DisplayRGBFromAdditionalAttribute.png" alt="LayerTreePC_DisplayRGBFromAdditionalAttribute.png" style='margin-left:24%;'/>
</div>
1. (Optional) Select the attribute to configure colors from the attribute dropdown list.
2. (Optional) Set the display component indices for the red, green, and blue channels respectively.
3. (Optional) Click "Restore Defaults" to reset the red, green, and blue channel indices to their initial default values.
4. (Optional) Click the "Apply" button to finalize the display effect as shown in the image.
5. Click the "Close" button to exit the settings interface.

<div align=center>
<img src="../../ScreenShot/zh/ProjectTree/LayerTreePC_DisplayByRGBFromAdditionalAttributeResult.png" alt="LayerTreePC_DisplayByRGBFromAdditionalAttributeResult.png" width="480" style='margin-left:24%;'/>
</div>

- **Display by Tree Attributes**: Display point clouds based on tree attributes. This function requires associating the point cloud with the corresponding tree attribute file.
1. Select both the point cloud and the tree attribute file to be associated in the directory tree, right-click and select "Bind Tree Attributes" to associate the tree attribute file with the point cloud.
<div align=center>
<img src="../../ScreenShot/zh/ProjectTree/BindTreeDB.png" alt="BindTreeDB.png" style='margin-left:24%;'/>
</div>
2. Select the point cloud, right-click > Display Mode > Display by Tree Attributes, and the following interface for displaying by tree attributes will pop up. Choose the attributes and color bar to display, and click "Apply" to display the point cloud according to the corresponding tree attributes.
<div align=center>
<img src="../../ScreenShot/zh/ProjectTree/DisplayByTreeDB.png" alt="DisplayByTreeDB.png" style='margin-left:24%;'/>
</div>
```

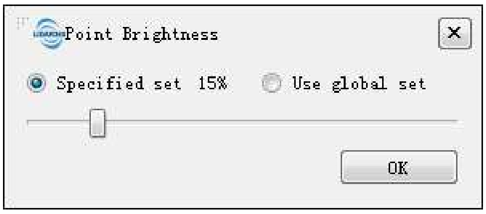
- **Zoom to Layer**: Calculate the bounding box of the current point cloud data. All the windows, in which the data object is loaded, will show full extent of the bounding box.

- **Restatistics**: Recalculate the *Mean Z*, *Std Z*, *Mean Intensity* and *Std Intensity* of point cloud data. This function is used to repair older versions of LiData which may contain incomplete information.

- **Export**: Export the point cloud data to LAS (*.las, *.laz), ASCII (*.txt, *.asc, *.neu, *.xyz, *.pts, *.csv), PLY (*.ply), and E57 (*.e57) format. The supported LAS version for export are 1.2,1.3,1.4, and the default version is 1.4. This function is the same as the tool Export.
- **PCV**: Process the point cloud data with PCV to improve the visual effects.
- **Point Size**: The point size of each point cloud data object can be set separately or set uniformly according to the global settings. The shape of points can be set to circle or square. The interface is shown as follows.



For global point size settings, see the usage of the tool Configure Point Size.
- **Point Brightness**: Set the brightness of a single point cloud data or all the point cloud data, as shown below:



To set the brightness of all point cloud data, users can click the icon in the settings tool bar. For more details, please refer to Display Options.

- **Remove**: Remove the selected point cloud data from the current project. This function is the same as the tool Remove.

Settings

- **Display by Height**:
  - **Color Bar**: The colorbar supports several uniformly varying color intervals for color mapping.
  - **Stretch**: The stretch methods of the histogram.
    - **Minimum and Maximum (Default)**: Apply a linear stretch based on the minimum and maximum pixel values, with the minimum and maximum pixel values as the endpoints for the histogram. For example, the minimum and maximum pixel values of the image could be 2488 and 2656 respectively. The values can be stretched linearly between 0 and 255. Distributing pixel values over the entire histogram range, the brightness and contrast of the image are increased and features in the image are easier to distinguish.
    - **Std deviation**: Apply a linear stretch between the pixel values defined by the std deviation (n) value. For example, the minimum and maximum pixel values of the image could be 2488 and 2656 respectively. If the value of standard deviation (n) is defined as 2, then the values beyond 2 standard deviation become 0 or 255, the remaining values stretch between 0 and 255.
- **Display by Intensity**:
  - **Stretch**: The stretch methods of the histogram.
    - **Minimum and Maximum (Default)**: Apply a linear stretch based on the minimum and maximum pixel values, with the minimum and maximum pixel values as the endpoints for the histogram. For example, the minimum and maximum pixel values of the image could be 2488 and 2656 respectively. The values can be stretched linearly between 0 and 255. Distributing pixel values over the entire histogram range, the brightness and contrast of the image are increased and features in the image are easier to distinguish.
    - **Std deviation**: Apply a linear stretch between the pixel values defined by the std deviation (n) value. For example, the minimum and maximum pixel values of the image could be 2488 and 2656 respectively. If the value of standard deviation (n) is defined as 2, then the values beyond 2 standard deviation become 0 or 255, the remaining values stretch between 0 and 255.
- **Save Curve**:
  - **Width**: The width of the saved curve.
  - **Height**: The height of the saved curve.
  - **Resolution**: The resolution of the saved curve.
  - **Output path**: The output path of the saved curve.
- **Point Size**:
  - **Circular Points (Optional)**: Set the shape of the point to circle or square.

- **Specified set (Optional)**: Set the point size of the specified point cloud data separately.
- **Use global set (Optional)**: Set the point size of the specified point cloud data with global settings.

> Note：Except the import data function, other right-click menu functions work on all viewers loaded with the point cloud.

Raster Context Menu

Summary

The context menus are used to for data import, removal, query, display, etc.

Data Type Context Menu

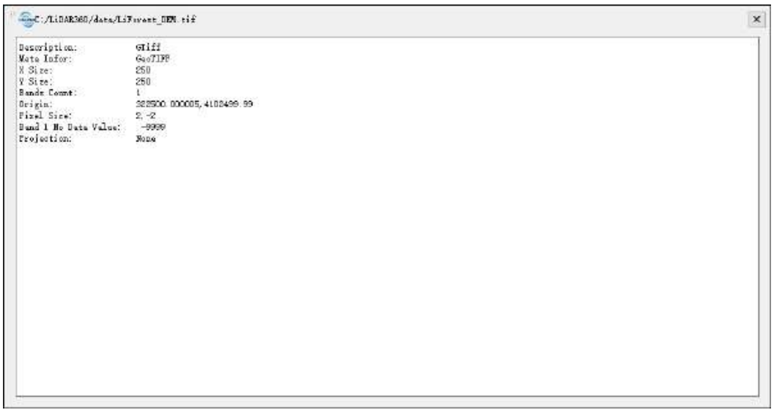
The user can open this menu by right clicking the raster data type.

- **Import Data**: The raster data formats supported by LiDAR360 include TIF and JPG. The function is the same as the tool Add Data.
- **Remove All**: Remove all the raster data from the project.

Data Context Menu

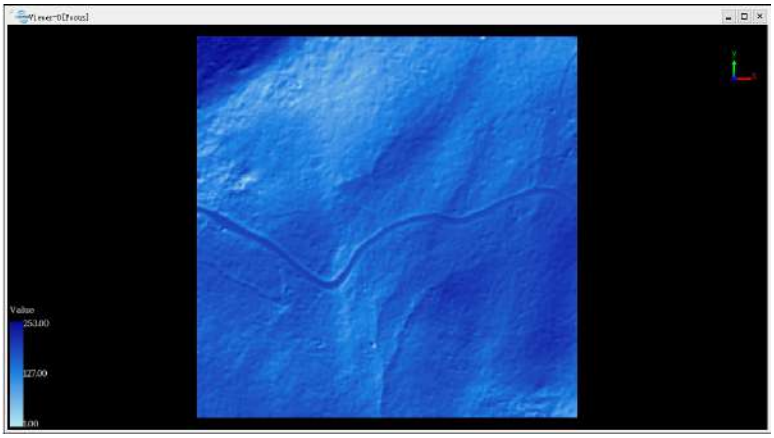
The user can open this menu by right clicking a raster data object.

- **Info**: View the basic information of raster data, including file path, description, X size, Y size, bands count, pixel size, invalid value of each band, projection, etc. The interface is shown below.

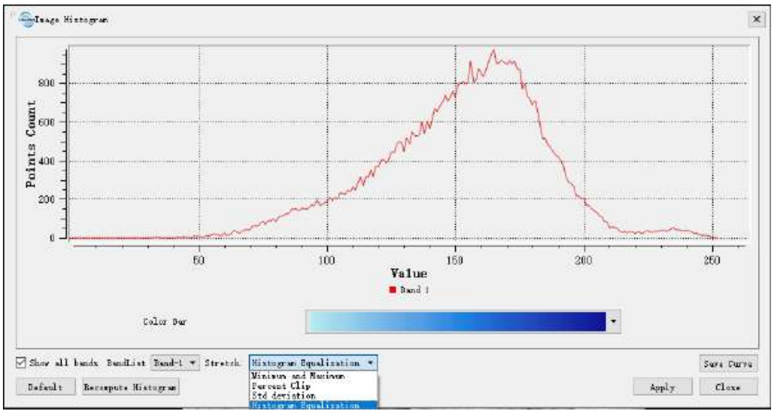


- **Open Containing Folder**: Open the folder.

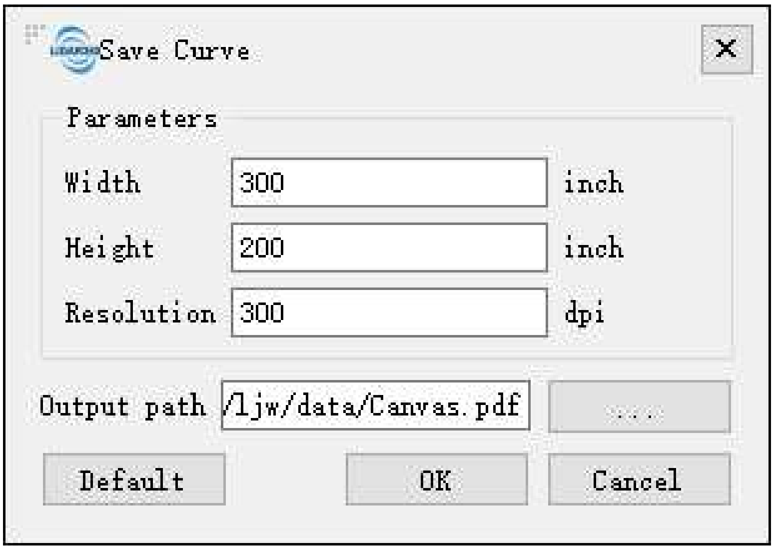
- **Histogram**: View the histogram of the raster data, where the values of each band can be stretched separately. The raster data is displayed in gray color bar by default. Select an appropriate color bar in the combo box. Click the button "Apply", the raster data in the scene will be rendered according to the above settings, as shown below.



Select the stretch method and generate the corresponding histogram, as shown below.



The button "Default" is used to restore the default settings. The button "Recompute Histogram" is used to calculate all the pixel values of the raster data again. The sampling points are 250000 by default. When the pixel size of the raster data exceeds the value, the statistical results may be inaccurate, then this button is needed. The histogram can be saved in *.pdf format. Click the button "Save Histogram" to pop up the following interface. Set the width, height, resolution, output path of the histogram and click "OK" button to save the result to local disk.



- **Zoom to Layer**: Calculate the bounding box of the current raster data. All the windows, in which the data object is loaded, will show full extent of the bounding box.
- **Zoom to Native Resolution (100%)**: Display the raster data in 1:1 ratio according to the resolution of

the raster data.

- **Remove**: Remove the selected raster data from the current project. This function is the same as the tool Remove.

Settings

- **Histogram**:
  - **Stretch**: The stretch methods of the histogram.
    - **Minimum and Maximum (Default)**: Apply a linear stretch based on the minimum and maximum pixel values, with the minimum and maximum pixel values as the endpoints for the histogram. For example, the minimum and maximum pixel values of the image could be 2488 and 2656 respectively. The values can be stretched linearly between 0 and 255. Distributing pixel values over the entire histogram range, the brightness and contrast of the image are increased and features in the image are easier to distinguish.
    - **Percent Clip**: Apply a linear stretch between the pixel values defined by percent clip minimum and percent clip maximum. For example, the minimum and maximum pixel values of the image could be 2488 and 2656 respectively. Percent clip minimum and percent clip maximum values are 0.02 and 0.98, values less than 0.02 mean the values between 2488 and 2492, values more than 0.98 mean the values between 2652 and 2656, values between 2488 and 2492 become 0, values between 2652 and 2656 become 255, the remaining values are between 0 and 255.
    - **Std deviation**: Apply a linear stretch between the pixel values defined by the std deviation (n) value. For example, the minimum and maximum pixel values of the image could be 2488 and 2656 respectively. If the value of standard deviation (n) is defined as 2, then the values beyond 2 standard deviation become 0 or 255, the remaining values stretch between 0 and 255.
    - **Histogram Equalization**: Apply a non-linear stretch between the pixel values, redistribute the pixel values so that the pixel values in a certain range is approximately equal. This method works well when there are a lot of pixel values that are closely grouped together.
  - **Color Bar**: The colorbar supports several uniformly varying color intervals for color mapping.
  - **BandList**: Select a band from the band list to draw the histogram.
  - **Show all bands (Optional)**: Whether to show all bands.
- **Save Curve**:
  - **Width**: The width of the saved curve.
  - **Height**: The height of the saved curve.
  - **Resolution**: The resolution of the saved curve.
  - **Output path**: The output path of the saved curve.

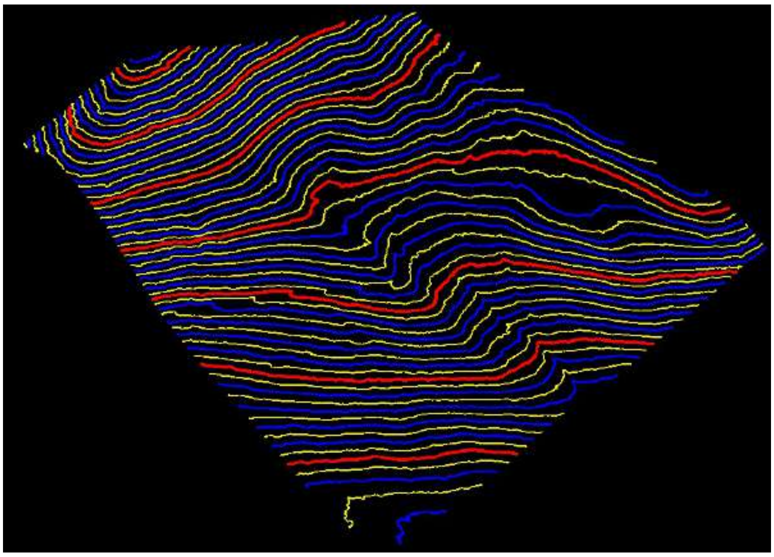
Vector Context Menu

Functional Overview

The right-click menu of vector data in the layer management tree mainly includes import and removal of vector data, as well as information display for a single vector data, opening the attribute sheet, scaling to layers, display by elevation, display by selected color, and remove.

Data Type Context Menu

- **Input Data**: Import data format of vector data should be .gbkg, .shp or .dxf format. This function is basically the same with the function , see Add Data. Import LiDAR360 from contour generated by point cloud as shown in the figure:

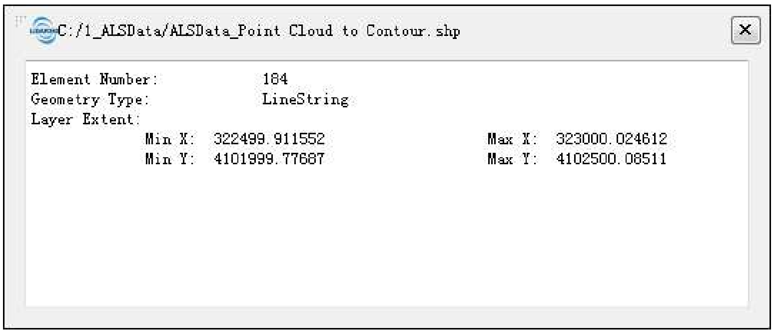


- **Unit Conversion**: When importing DXF format files, unit conversion is supported. The current unit (unit recorded in the file) can be converted to the target unit for import.

- **Remove All**: Remove all vector data from the software.

Data Context Menu

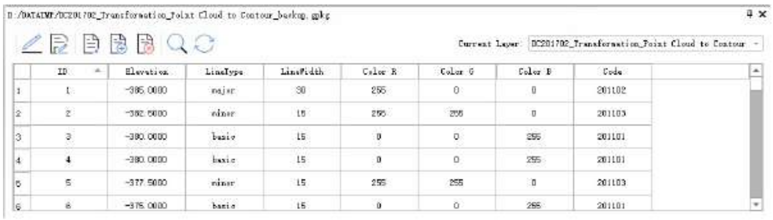
- **Info**: View the basic information of vector file, including file path, elements count and bounding box.



- **Open Containing Folder**：Open the folder.

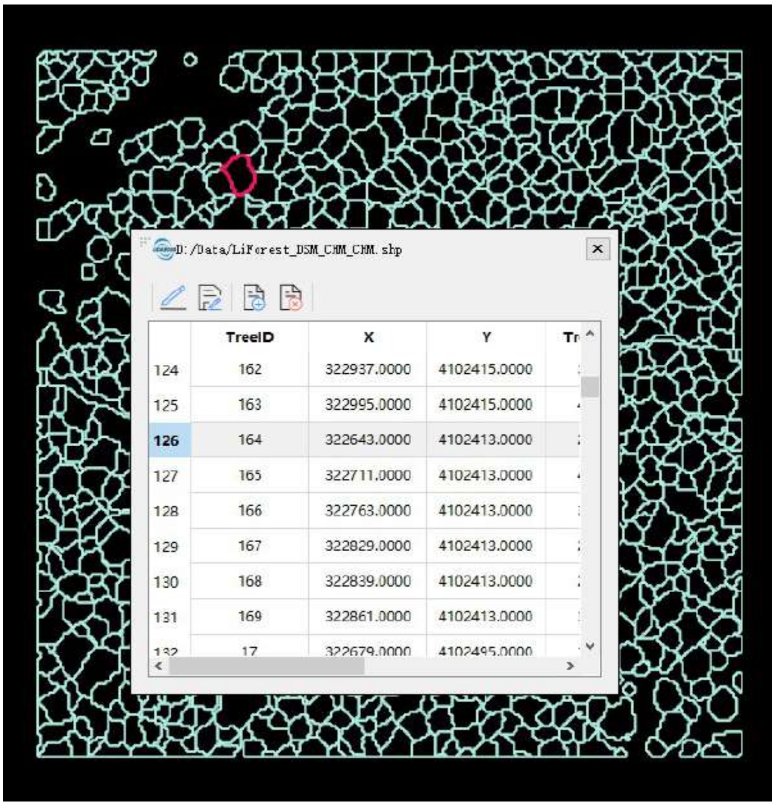
- **Rename**: Rename the file.

- **Open Attribute Table**: This function displays the attribute table information of vector data, and the result is shown as follows:



	ID	Elevation	LineType	LineWidth	Color R	Color G	Color B	Code
1	1	-385.0000	major	30	255	0	0	201102
2	2	-382.5000	minor	15	255	255	0	201103
3	3	-380.0000	basic	15	0	0	255	201101
4	4	-380.0000	basic	15	0	0	255	201101
5	5	-377.5000	minor	15	255	255	0	201103
6	6	-375.0000	basic	15	0	0	255	201101

  - Single-click the row or the cell in the attribute table, the corresponding data will be highlighted as shown in the following picture.

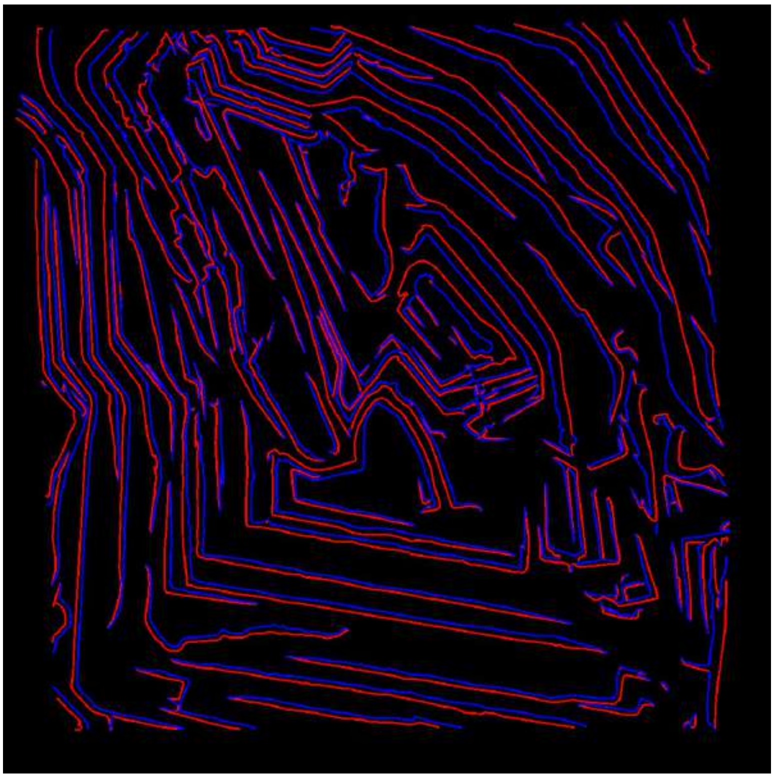


	TreeID	X	Y
124	162	322937.0000	4102415.0000
125	163	322995.0000	4102415.0000
126	164	322643.0000	4102413.0000
127	165	322711.0000	4102413.0000
128	166	322763.0000	4102413.0000
129	167	322829.0000	4102413.0000
130	168	322839.0000	4102413.0000
131	169	322861.0000	4102413.0000
132	17	322679.0000	4102495.0000

  - Double-click the header of a row in the attribute table, and the view will locate the location of the double-clicked data.

- **Zoom to Layer**: Calculates the bounding box of current vector data and displays all windows opened for this data globally within this bounding box range.

- **Line Width Setting**: The line width of an individual vector file can be set separately or using global line width settings as shown below.



- **Display by Elevation**: Maps elevation attributes of vector data to uniformly changing color intervals.
- **Display by Selected Color**: Displays vector data with fixed colors.
- **Remove**: Removes the selected vector data from the software.
- **Save**: Provides a save function for files in vector editing.

> Note: Except "Import Data” and “Remove”, other functions in context menu work on all windows that contains this vector file.

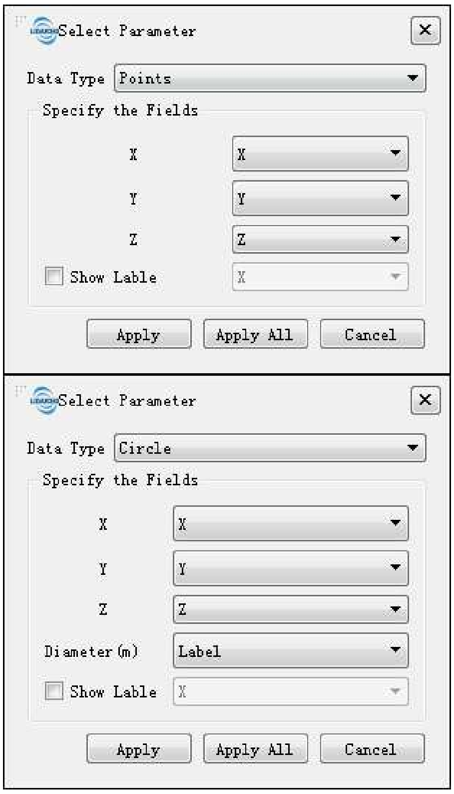
Tables Context Menu

Functional Overview

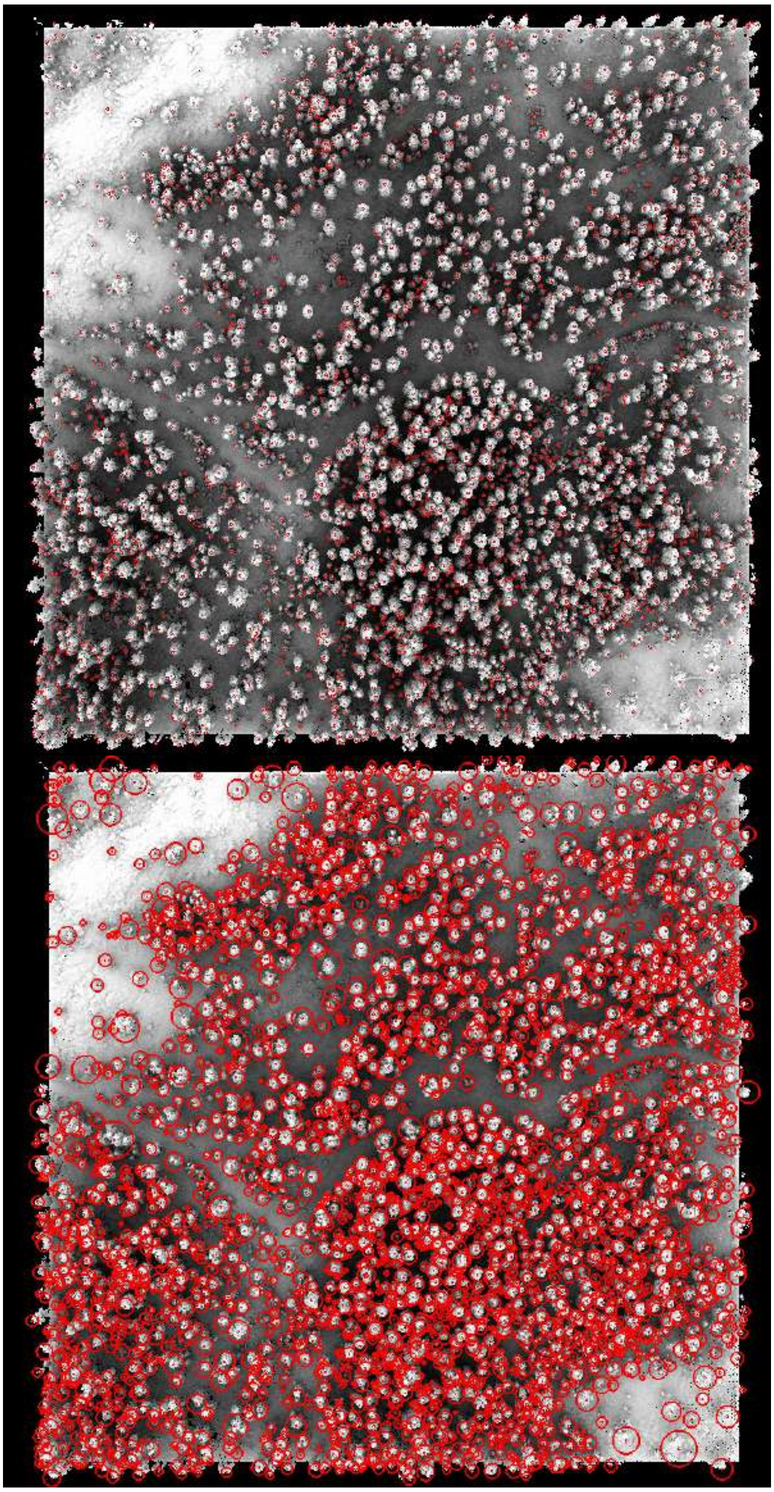
The right button menu of table data in the layer management tree mainly includes import and export table data, it also contains display information, display settings, zoom to layer, display by selected color, display by height, recalculate the statistics, export, and other operations for single model data.

Data Type Context Menu

- **Import Data**: The import data format is CSV（* .csv）. If click this function, pop-up the dialog, select data type as "Point" or "Circle", and set the specified field X, Y, Z, Diameter (if the data type is circle), and check whether to show labels.



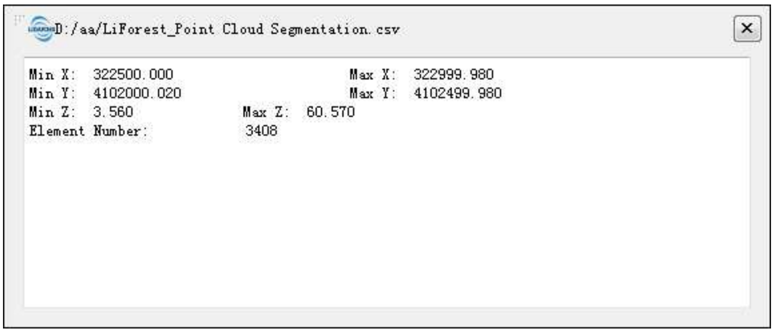
Select data type as Point/Circle, the table file displayed as below:



- **Remove All**: Remove all table files from LiDAR360.

Data Context Menu

- **Info**: View the table's basic information, including path, the number of elements, the minimum and the maximum of X, Y, Z values.

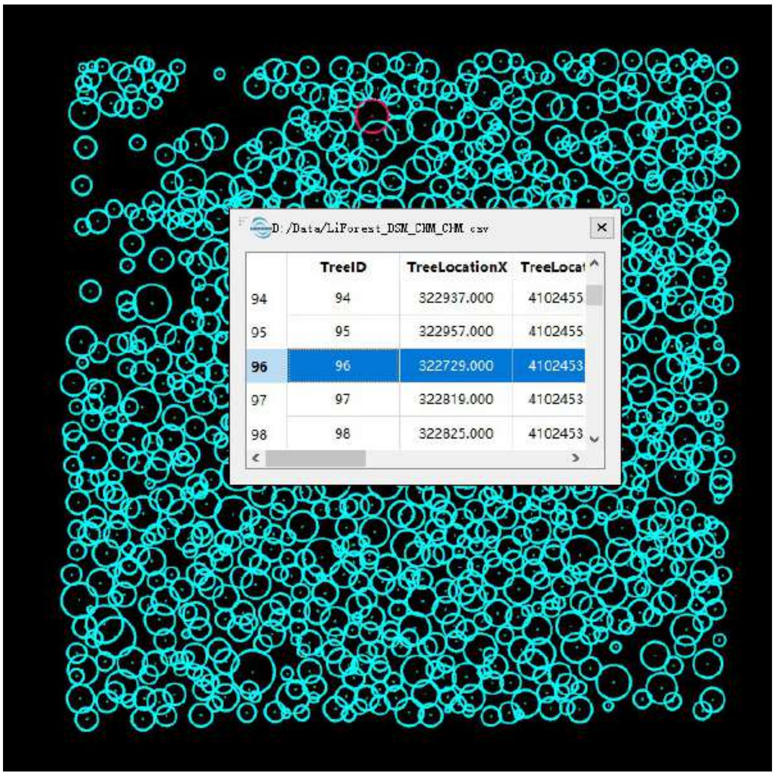


- **Open Containing Folder**: Open the folder.
- **Rename**: Rename the file.
- **Attribute Table**: Display the contents of the table. As shown in the following figure, double click specific row, the window will go to the position.

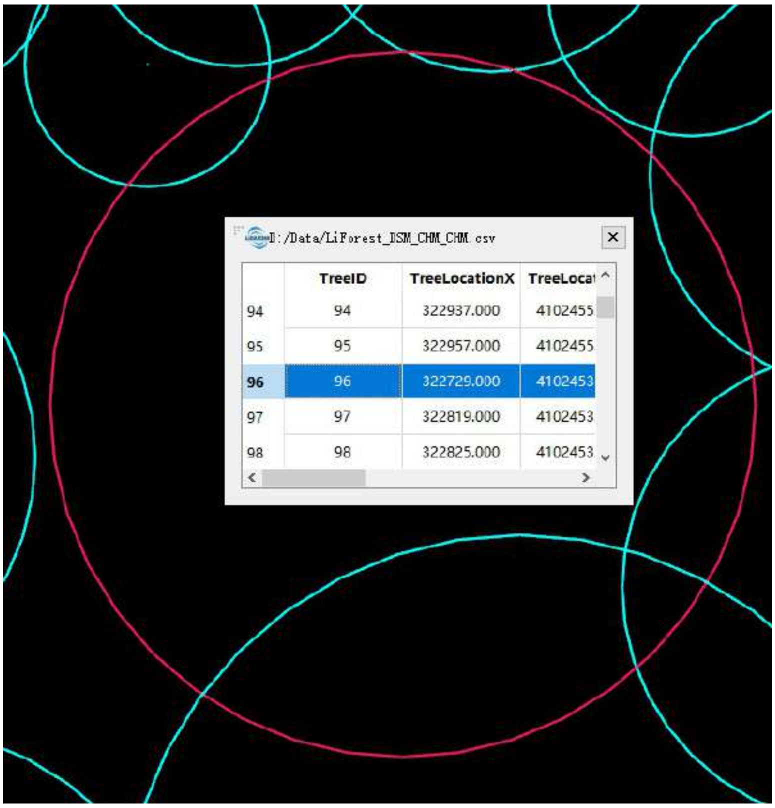

D:/aa/LiForest_Point Cloud Segmentation.csv

	TreeID	TreeLocationX	TreeLocationY	TreeHeight	CrownDiameter	CrownArea	CrownVolu
1	1	322511.520	4102089.780	60.570	10.913	93.534	3348.00
2	2	322511.810	4102015.140	57.100	9.670	73.449	2548.50
3	3	322537.430	4102062.510	55.720	11.769	108.782	3763.62
4	4	322529.420	4102073.100	53.850	10.929	93.803	3236.31
5	5	322525.070	4102101.070	53.740	12.906	130.820	3520.82
6	6	322501.160	4102109.680	53.510	7.978	49.989	1399.01
7	7	322522.530	4102143.800	52.390	7.669	46.192	1309.62
8	8	322520.650	4102152.530	52.070	9.612	72.559	1960.91
9	9	322514.280	4102001.290	51.430	11.659	106.756	3493.82
10	10	322533.990	4102053.190	51.070	8.075	51.216	1411.41
11	11	322523.570	4102070.020	50.950	6.626	34.479	929.889
12	12	322619.080	4102017.240	50.010	12.047	113.983	3273.20
13	13	322522.240	4102076.870	49.740	7.524	44.456	1332.91
14	14	322526.700	4102079.520	49.570	3.536	9.820	229.217
15	15	322537.320	4102163.130	49.220	8.669	59.022	1814.00

  - Single-click the row or the cell in the attribute table, the corresponding data will be highlighted as shown in the following picture:

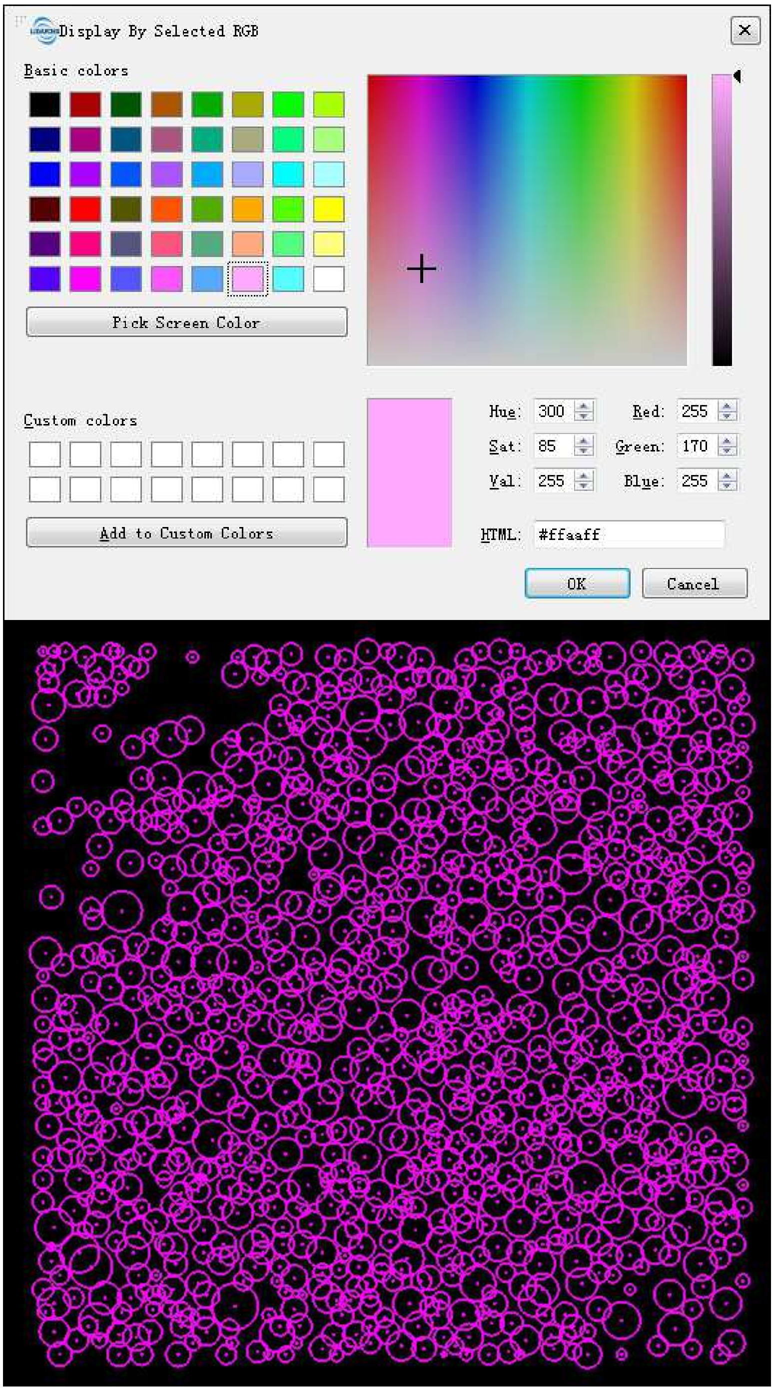


- Double-click the button in front of the row in the attribute table, it will zoom to the corresponding data, as shown in the following picture:

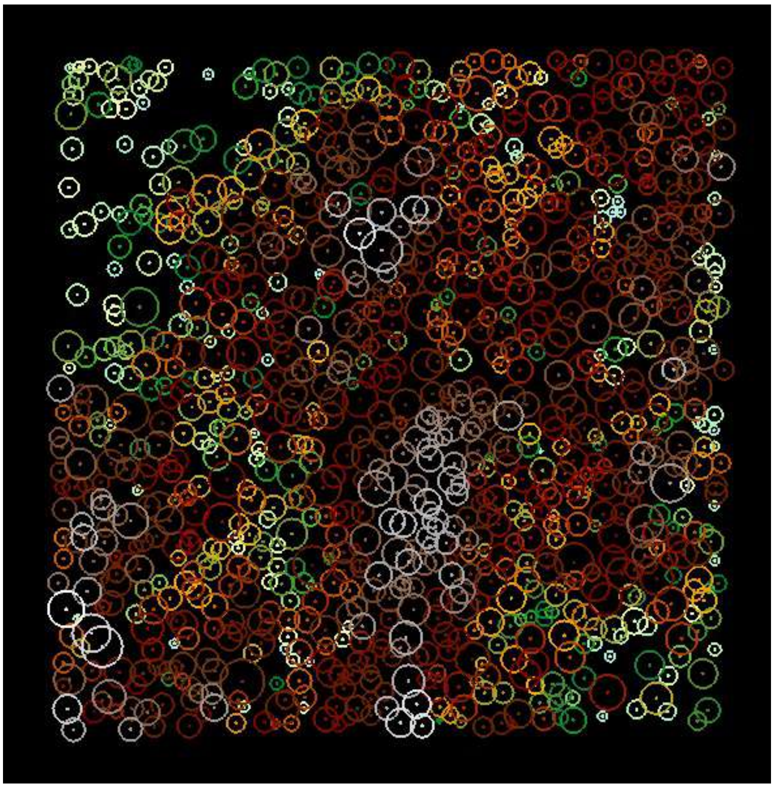


- **Zoom to Layer**: Calculates the bounding box of the current table data, and all windows that contains this file will display globally in this bounding box range.
- **View Mode**: Configure the display mode for the current table data, including setting it to display as points or circles, whether to show labels, and the fields to be displayed on the labels.

- **Point Size**: Set the size and shape (circular or square) of the display points for the current table data.
- **Label Size**: Set the size of the display labels for the current table data.
- **Display by Selected**: Display the table data by selected color, as shown in the following picture:



- **Display by Height**: Correspond the elevation information of the table data to a uniformly changing color band. And display the table data on the screen as shown below:



- **Remove**: Remove the selected file from LiDAR360.

> Note: Except "Import Data", the other function in context menu work on all windows that contains this table file.

Model Context Menu

Functional Overview

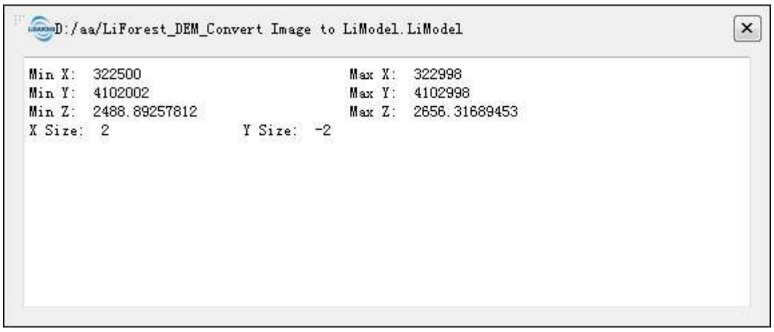
The right-click menu of model data in the layer management tree mainly includes import and export table data, it also contains information, display setting, zoom to layer, restatistics, export for single model data.

Data Type Context Menu

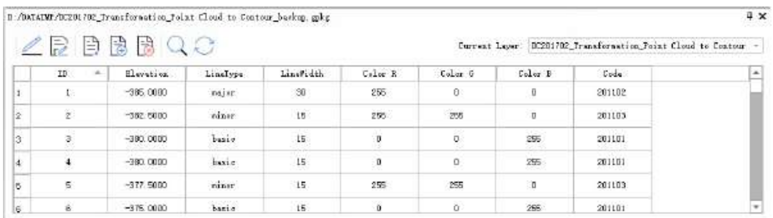
- **Import Data**: The model formats that LiDAR360 supported include: (*.LiTin, *.LiModel, *.LiBIM, *.LiTree), OSG data (*.osgb, *.ive, *.desc, *.obj), among which the LiTIN format and LiModel format are LiDAR360 defined model format. This function is the same with the function , see Add Data.
- **Remove All**: Remove All: Remove all model files from LiDAR360.

Data Context Menu

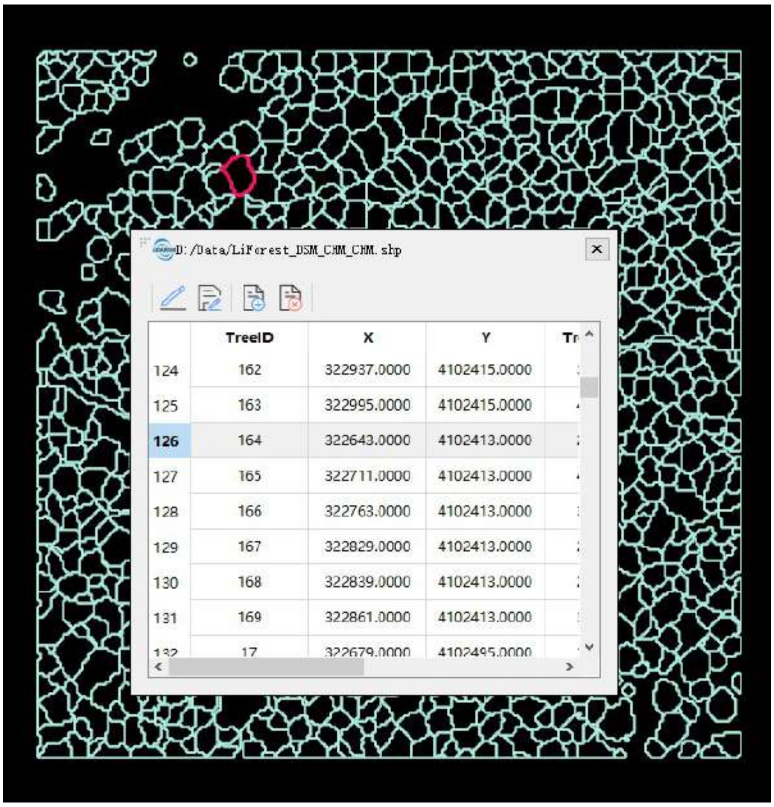
- **Info**: View the basic information of model file, including the path, resolution, the minimum and maximum of X, Y, Z. This function just applies to LiTin and LiModel file.



- **Table Attribute**: This function is only applicable to LiBIM files, displaying the attribute information of LiBIM model data. It supports adding, deleting, viewing, querying by attribute fields and modifying attribute values.

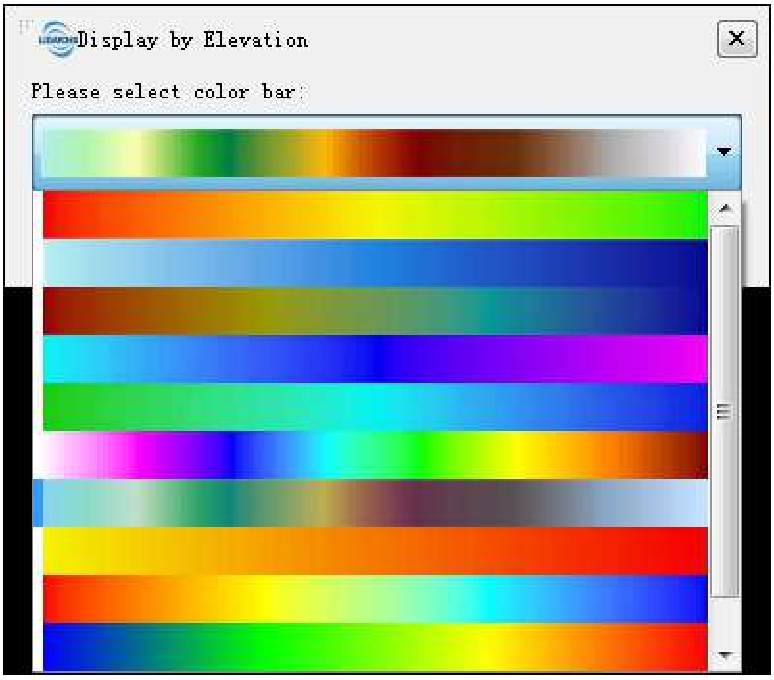


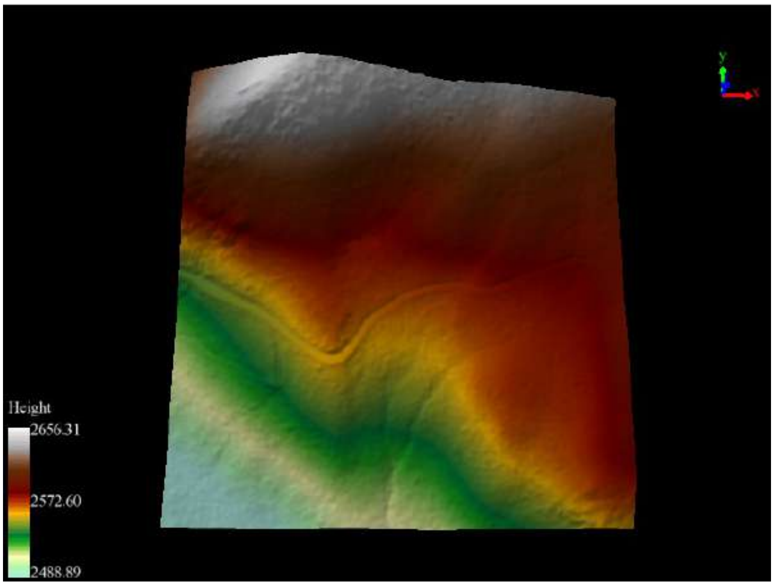
When clicking on a row or cell in the attribute table, the corresponding data in the view will be highlighted as shown in the picture.



Double-clicking on the header of a row in the attribute table will locate the position of the double-clicked data in the view.

- **Open Containing Folder**：Open the folder.
- **Rename**: Rename the file.
- **View Mode**: Set the model file's display mode, including display by elevation, display by texture, display by light. If "Display by Elevation" is selected, the color bar selection dialog will be popped up, and the display effect is as below:





- **Zoom to Layer**: Calculate the bounding box of the current model data, and all windows that contains this file will display globally in this bounding box range.
- **ReStatistics**: Recalculate basic information such as Min X, Y, Z and Max X, Y, Z. In general, if you edit the LiModel data, such as smooth height or repair height, you can use the Restatistics function.
- **Export**: This function only applies to LiModel file, and the export format is TIF. In General, after 3D visualization editing of LiModel and LiTIN generated by DEM, it is necessary to convert the editing files to TIF format file.
- **Remove**: Remove the selected model data from viewer window or project.

> Note: Except "Import Data", other functions in context menu work on all windows that contains this model file.

Trajectory

Functional Overview

The right-click menu for trajectory data in the layer management tree mainly includes importing and removing trajectory data, as well as operations such as displaying information, renaming, zooming to the layer, opening the containing folder, and removing individual trajectory data.

Right-Click Menu for Data Types

- **Import Data**: LiDAR360 supports trajectory data formats including *.out, *.pos, *.traj etc., where *.pos and *.traj are software-defined custom trajectory data formats.
- **Remove All**: Remove all trajectory data from the software.

Right-Click Menu for Individual Trajectory Data

- **Information**: View basic information about a trajectory including start time, end time, maximum/minimum longitude and latitude values and maximum/minimum height values.
- **Open Containing Folder**: This function opens the directory where the file is located.
- **Rename**: Rename selected files.
- **Zoom to Layer**: Calculate bounding box of current track data and display all windows that open this dataset globally within this bounding box range.
- **Remove**: Remove selected track data from software.

AEP Project

Functional Overview

The right-click menu for airborne LiDAR project data in the layer management tree mainly includes importing and removing airborne LiDAR project data, as well as operations such as displaying information, opening the containing folder, and removing individual airborne LiDAR project data.

Right-Click Menu for Data Types

- **Import Data**: The supported formats of airborne LiDAR project data in LiDAR360 include *.liaep, *.ligeo, *.limap, *.p4d, *.prj. Among them,*.liaep,*.ligeo,* .limap are software-defined formats.
- **Remove All**: Remove all airborne LiDAR project data from the software.

Right-Click Menu for Individual Airborne LiDAR Project Data

- **Information**: View basic information about the airborne LiDAR project including image count, connection point count and coordinate system information.
- **Open Containing Folder**: This function opens the directory where the file is located.
- **Zoom to Layer**: Calculates the bounding box of the current data and scales all open windows displaying that data to fit within this bounding box.
- **Show Camera**: Determines whether to draw the camera's viewing frustum in the 3D window.
- **Show Thumbnails**: Determines whether to display the thumbnail of the image in the 3D window.
- **View Mode**: Sets the display mode for the current data, including elevation-based display, color-coded connection points, or additional attributes.
- **Camera Projection Distance**: Sets the size of the camera's viewing frustum.
- **Save**: Saves the image project after making modifications.
- **Export Camera**: Exports the image project to a third-party file. For specific functionality, see the Export Camera page.
- **Remove**: Removes the selected aerial triangulation project data from the software.

Camera Group Right-Click Menu

- **Info**: View basic information about the specified camera group in the aerial triangulation project, including the number of images in the group, principal points, focal length, internal orientation elements, etc.

Image Right-Click Menu

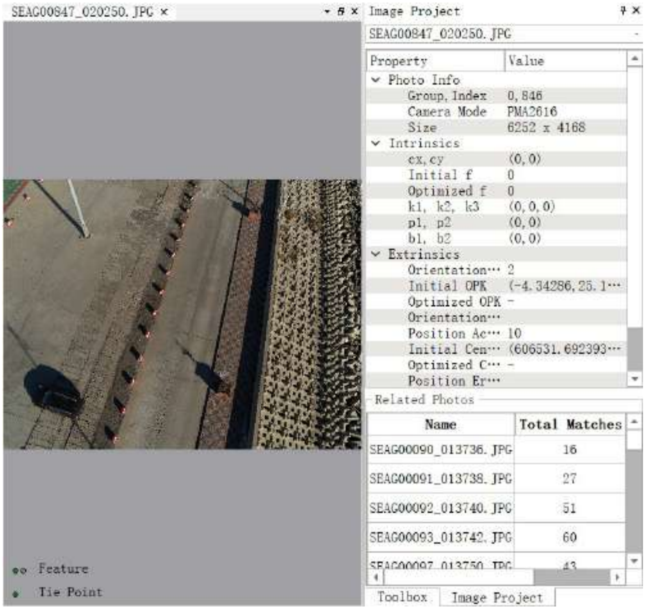
- **Remove**: Removes the selected image from the image project.

Point cloud set information viewing is supported.

- **Info**: View basic information about all points in the point cloud set.
- **Export**: Export the point cloud set to another format.

Image Double-Click View

- Double-clicking a single image will open the image information window, displaying all details about that image, as shown below.

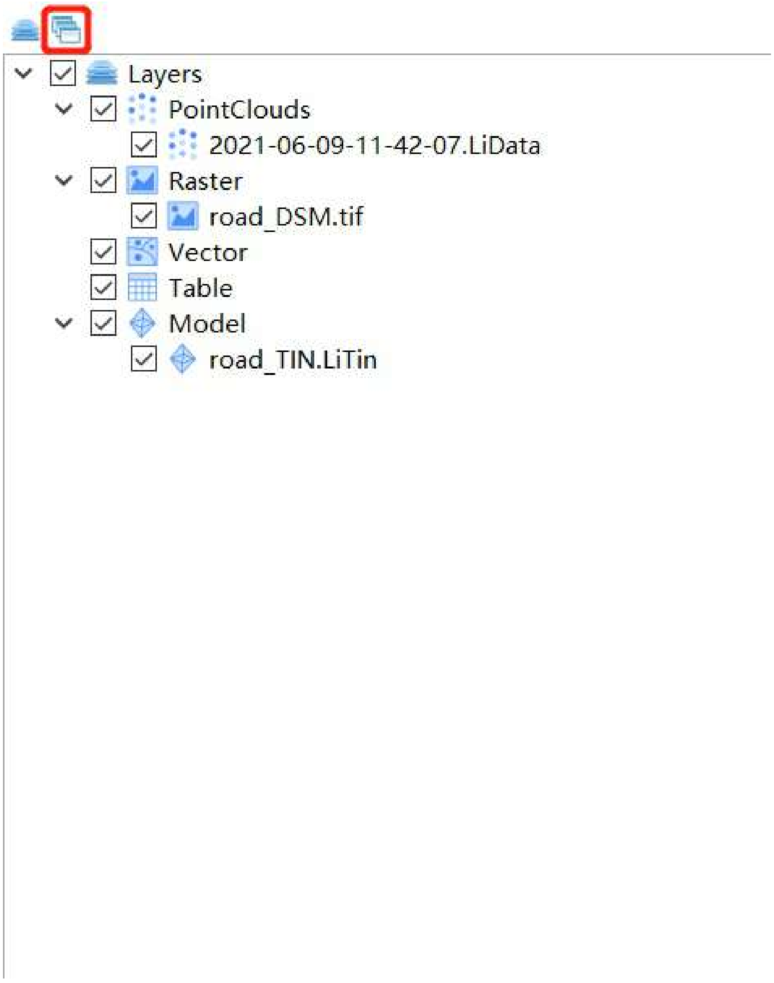


Window Management

Summary

The **project windows** manage all windows(i.e. viewers) and data in windows. The user can remove data from window, edit display order by dragging data node, show/hide data in specified window by checking/unchecking the box before window/data node. The context menu(i.e. right-click menu) of data node, which differs depending on data types, is mainly used for data query, display, statistics, export, and removal, etc. These functions are effective on specified window.

Click the button to show the **project windows** as follows:



Context Menu

Open the context menu by right clicking window or data.

Window Context Menu

- **Remove All**: Remove all data from the selected window.

Data Context Menu

- Same as LayerManagement.md.

> Note: The context menu of window management is only effective on specified window, while the context menu of layer management is effective on all windows.

Ribbon Page

Ribbon Page is the quick toolbar at the top of the software, featuring tabs for various modules. Each page contains commonly used functions for that module and can display key business processes in simple mode, providing one-click full-process operations. This helps users quickly get started and locate main functions. The available tabs include:

- Tools
- Preprocessing
- Classification
- Vector Editor
- Terrain
- ALS Forestry
- TLS Forestry
- Powerline
- Mine
- 3D Building
- Photo
- Display

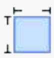
Tools

This module includes Measurement,Clipping and Data Management.

- Measurement
- Clipping
- DataManagement
- Process

Measure Tools

The measurement tools are used to measure geometric information of point cloud data, including single point selection, multiple point selection, point density measurement, length measurement, area measurement, angle measurement, slope measurement, height measurement, volume measurement, etc.

Icon	Function	Function Description
	Pick Point	The Pick Point tool can be applied to point cloud, raster, and model data. When applied to point cloud data, it allows interactive querying of attributes of individual points, including their position, intensity, return number, classification, GPS time, etc. When applied to raster data, it enables interactive querying of attributes of individual points, including their position, stretched RGB values, pixel values, etc.
	Multi-point	The Multi-point tool can be used for both point cloud and raster data. It allows interactive querying of attributes of multiple points by clicking with the mouse, and supports exporting selected point sets to various formats such as txt, asc, neu, xyz, pts, csv, etc. Attributes that can be queried for point cloud data include point ID, position, classification, return number, GPS time, intensity, etc. For raster data, attributes that can be queried include point ID, position, band values, etc.
	Density	Point density is one of the indicators used to measure the quality of point cloud data. The point density measurement tool is used to measure the density of point clouds, calculating the average number of points per square meter.
	Length Measurement	The length measurement tool can be used for point cloud, raster, and model data. This tool allows interactive querying of distance information between multiple points by clicking with the mouse. For 2D data, the measurement results indicate planar distance. For 3D data, such as point cloud and model data, the measurement results indicate the Euclidean distance between points in three-dimensional space.
	Area Measurement	The area measurement tool can be used for all data types supported by LiDAR360. By clicking with the mouse, the tool allows interactive drawing of polygons and calculates the projected area within the polygon. For 3D data area measurement, the current window will automatically switch to orthographic projection mode.
	Angle Measurement	The angle measurement tool can be applied to point cloud, raster, and model data. By clicking with the mouse, the tool allows interactive selection of measurement points. In a 3D view, it queries the angle between three points, while in a 2D view, it queries the projection angle of the three-point line on the horizontal plane.
	Height Measurement	The height measurement tool can be applied to point cloud and model data. By clicking with the mouse, the tool allows interactive selection of measurement points and queries the relative height between two points.
	Gradient Measurement	The gradient measurement tool can be applied to point cloud, raster, and model data. By clicking with the mouse, the tool allows interactive selection of measurement points and queries the gradient between two points, which is the angle between the line connecting the start and end points and the horizontal plane.

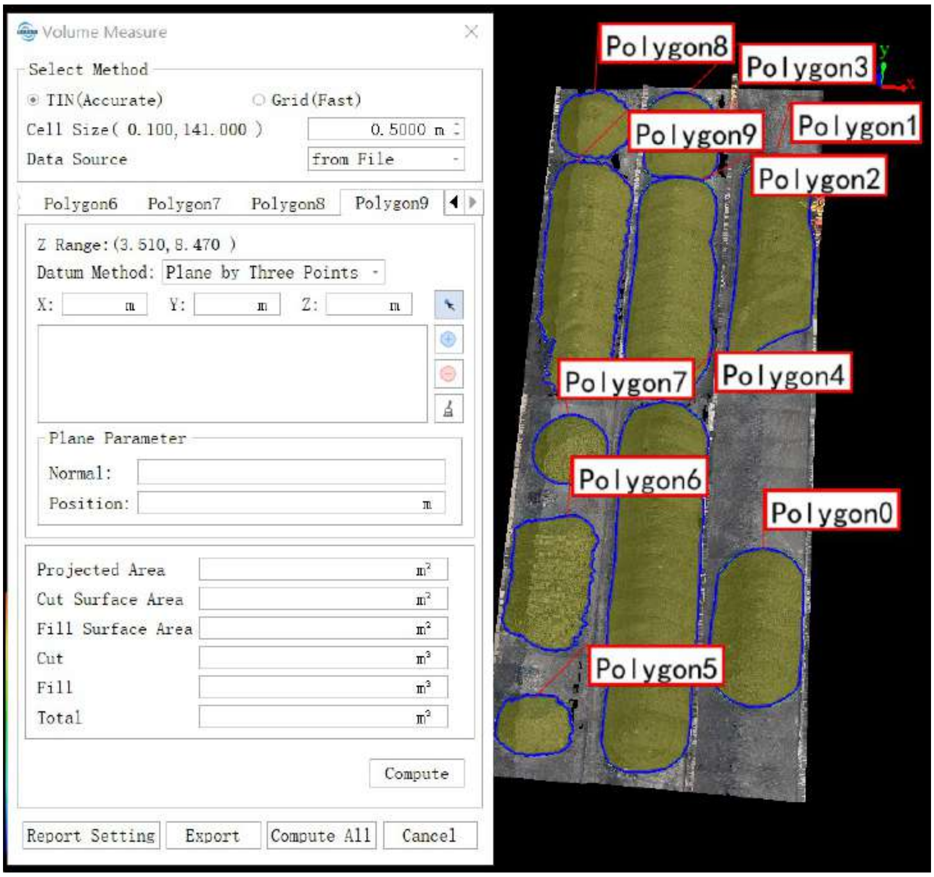
Volume Measurement

Function Description:The volume measurement tool can be applied to point cloud and model data. This tool allows interactive selection of a reference plane for measurement by selecting data from a file or by clicking with the mouse. It calculates the volumes of fill, cut, and combined cut-and-fill relative to a certain height, commonly used for applications such as measuring the volume of coal piles or ship hulls.

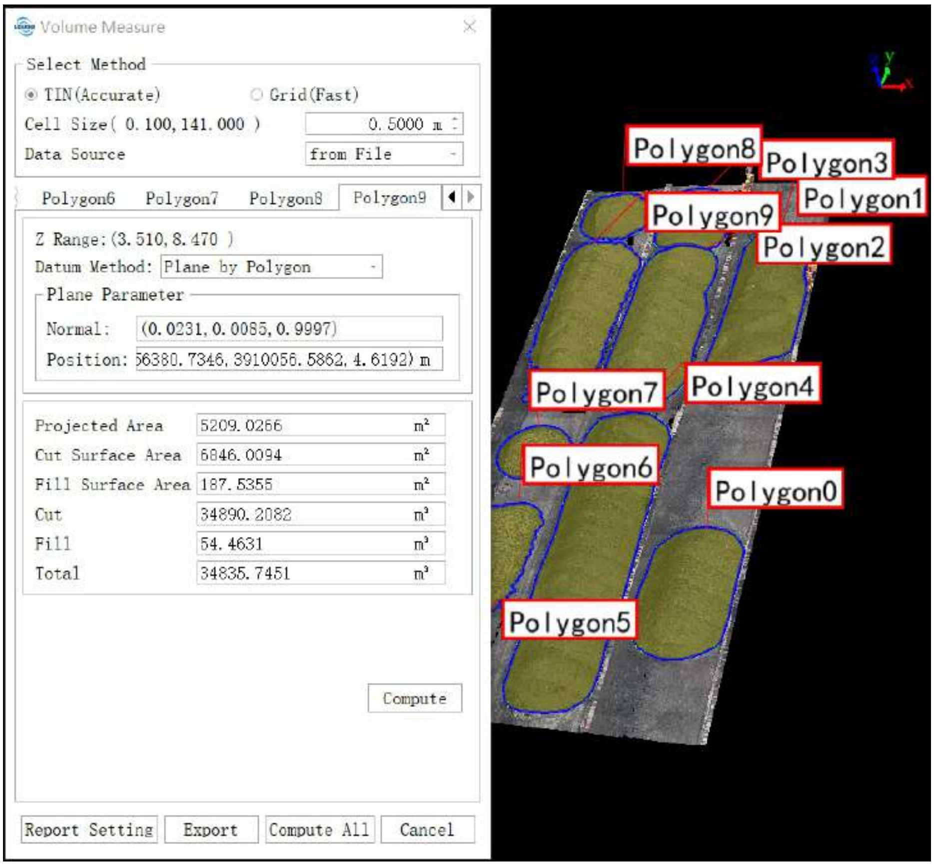
Steps

Click *Tools* > *Measurement* > *Volume Measurement / Volume Measurement From File*

1. After clicking this function, a volume measurement options dialog will appear. If you choose volume measurement, set the mouse to interactively select the measurement reference plane by clicking. If you choose to select a file for volume measurement, select the file from the folder to delineate the measurement boundary. It is generally recommended to switch the view to top view before performing volume measurement.
2. (Optional) Use the mouse to interactively select the measurement points. In the window with the loaded point cloud data, continuously click the area of interest with the left mouse button (to generate the reference plane for volume measurement, select at least three points). Double-click to finish the selection. The selected area will be outlined with a red solid line in the scene, and a dialog box as shown will appear.



3. (Optional) Select .shp, .dxf, or .gpkg files from the folder (currently only line and polygon files are supported).
4. Set the cell size.
5. Set the reference plane for volume measurement. The calculation methods for the reference plane include minimum value, polygon plane, custom, three-point plane, and point cloud fitted plane. Once the reference plane is determined, its plane parameters will be displayed.
6. Set the data source for volume measurement. The data source types include loaded points and all points.
7. Click the calculate button to generate results for projected area, surface area, cut volume, fill volume, and net volume (cut and fill). The scene will display the space occupied by the current measurement data, shown as several columns, as illustrated.



8. Click the export button to export the volume measurement results in PDF format.

Parameter Settings

- **Select Method**: Currently supports two volume measurement methods: TIN method and grid method. The TIN method provides higher accuracy in calculation but is less efficient compared to the grid method.
- **Cell Size**: The grid size affects the size of the displayed columns and the accuracy of the grid method, but does not affect the accuracy of the TIN method. This parameter defines the minimum calculation unit size for volume measurement using the grid method. A smaller value provides more precise calculations.
- **Reference Plane**: This parameter defines the reference plane for cut-and-fill volume measurement.
  - **Minimum(Default)**: Uses the minimum Z value within the selected point range as the height

value of the plane.
    - **Polygon Plane**: Fits the best plane based on boundary points within the selected area.
    - **Custom**: Allows the user to input or click to specify a height as the reference plane for volume measurement.
    - **Three-Point Plane**: Determines a plane using three selected points.
    - **Point Cloud Fitted Plane**: Determines the best plane based on selected points and a fitted region radius.
- **Data Source**: This parameter defines the type of data source for volume measurement.
    - **Loaded Points (Default)**: Uses points loaded within the specified range in the scene. This method is relatively faster but may be affected by changes in loaded data in the scene.
    - **All Points**: Uses points from the specified point cloud file within the specified range. This method is slower but provides more stable calculation results.

> Note: This function is specifically for volume measurement of point cloud data and can only be used in the 3D view window.

Volume Measurement on Solid Model

Calculate the volume of a closed solid model and display the result on the interface. If the model is not closed, calculation is not possible and a prompt dialog will appear. This function requires that the connection order of all triangles in the model is either all counterclockwise or all clockwise; otherwise, the volume calculation will not be accurate.

Profile View

Function Description：The profile editing feature allows users to display designated rectangular areas of point cloud data in the profile window within the main window, facilitating operations such as viewing, measurement, and classification modification.

To produce high-precision terrain products using LiDAR data, manual inspection of automatically classified results is necessary. This requires the use of profile viewing tools. Starting from version 5.0, LiDAR360 provides profile viewing functionality, which enables more efficient classification checks.

Snap Options

Snap options are supported only for length measurement, area measurement, angle measurement, height measurement, and slope measurement. When the corresponding capture switch is enabled, measurement information is primarily obtained from the corresponding data format. If capture is not enabled, coordinate information will be obtained from the depth map in the scene.

- **Point Cloud Snap**

  Function Description: When enabled, measurement information is prioritized from the point cloud.

- **Lowest Point Snap**

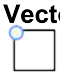
  Function Description: When enabled, measurement information is prioritized from the lowest point in the point cloud near the mouse cursor.

- **Highest Point Snap**

  Function Description: When enabled, measurement information is prioritized from the highest point in the point cloud near the mouse cursor.

- **Vector Endpoint Snap**

  Function Description: When enabled, measurement information is prioritized from the endpoint of the vector.

- **Vector Midpoint Snap**

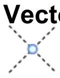
  Function Description: When enabled, measurement information is prioritized from the midpoint of the vector.

- **Vector Nearest Point Snap**

  Function Description: When enabled, measurement information is prioritized from the nearest point of the vector.

- **Vector Intersection Snap**

  Function Description: When enabled, measurement information is prioritized from the intersection points of vectors.

- **Vector Center Snap**

  Function Description: When enabled, measurement information is prioritized from the center point of the vector.

- **Model Snap**
  

  Function Description: When enabled, measurement information is prioritized from the model data.

- **Table Snap**
  

  Function Description: When enabled, measurement information is prioritized from the table data.

- **Trajectory Snap**
  

  Function Description: When enabled, measurement information is prioritized from the trajectory data.

- **Tie Points Snap**
  

  Function Description: When enabled, measurement information is prioritized from the image data.

Selection Toolbar

Select and save point clouds in interest area.

- Select and Clip
- Cross Selection
- Clip by Circle
- Clip by Rectangle
- Clip by Polygon

Select and Clip

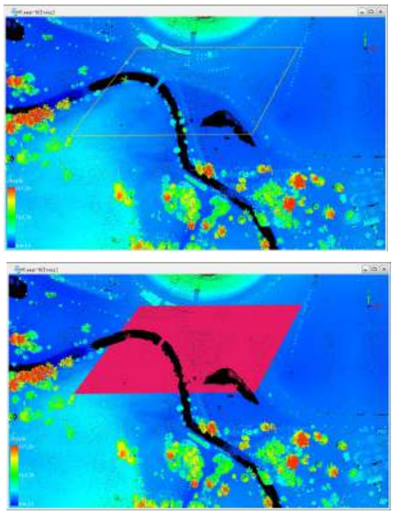
Icon	Function Name	Description	Shortcut
	Polygon Selection	Draws a polygon to select points within the defined area.	Shift+P
	Rectangle Selection	Draws a rectangle to select points within the defined area.	Shift+R
	Sphere Selection	Draws a sphere to select points within the defined area.	Shift+S
	Circle Selection	Draws a circle to select points within the defined area.	Shift+C
	Lasso Selection	Uses a lasso to select points.	Shift+L
	Above Line Selection	Selects points above the polyline using a line.	None
	Below Line Selection	Selects points below the polyline using a line.	None
	Plane Selection	Selects points on a plane defined by three points.	None
	Above Plane Selection	Draws a polygonal plane to select points above the plane.	None
	Below Plane Selection	Draws a polygonal plane to select points below the plane.	None
	Plane Inside Selection	Draws a polygonal plane with thickness to select points inside the plane.	None
	Subtract Selection	Increases or decreases the selection area based on previous selections.	Shift+D
	Cancel Selection	Cancels all selection and clipping operations.	None
	In Cut	Clips the point cloud to keep only the selected points.	Shift+X
	Out Cut	Clips the point cloud to keep only the points not selected.	Shift+Ctrl+X

Selection Tool Steps

1. Click the tool button to activate the selection tool.
2. For polygon tools, click the left mouse button to define the vertices of the polygon in the area of interest. The program will automatically form a closed polygon. Double-click to finish. For other selection tools, click the first point, drag the mouse to cover the area of interest, and double-click to complete the selection.
3. If vertices are misplaced in the polygon selection, right-click to cancel the last selected vertex. This

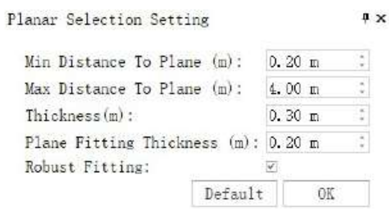
operation can be performed multiple times.
4. Double-click the left mouse button on the last vertex to end vertex selection. The points within the selected polygon area will be highlighted (in red).
5. After completing one selection, you can perform another selection based on the first. Cancelling the result of the last clipping will also cancel the clipping of the corresponding selection area.
6. Each selection area will either add to or subtract from the existing selection area based on the inverse selection state.

Selection results:



Selection Settings

Selection settings. Configure parameters related to selection, and open the plane selection settings interface.



Parameter Settings

- **Min Distance to Plane**: Sets the minimum distance to the plane.
- **Max Distance to Plane**: Sets the maximum distance to the plane.
- **Thickness**: Sets the thickness within the plane.
- **Plane Fitting Thickness**: Thickness used for plane fitting.
- **Robust Fitting**: Provides better fitting results but is slower.
- **Default**: Click this button to restore all parameters to their default values.

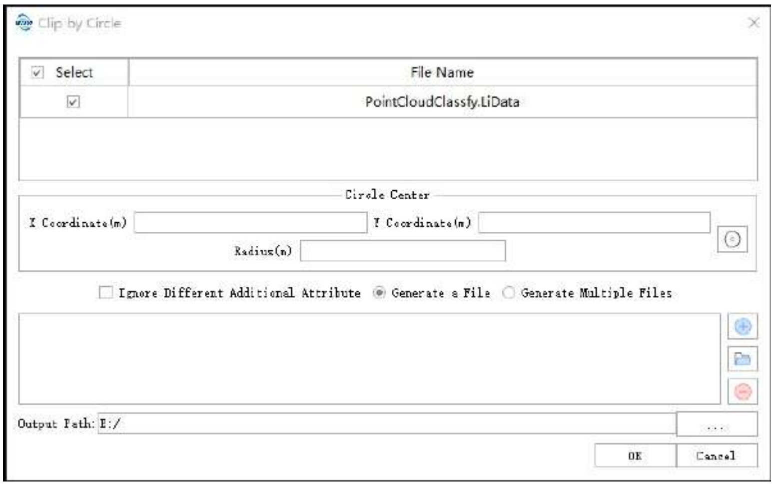
Clip by Circle

Summary

Clip by circle tool extracts the point cloud data within user-defined circle(s), and the extracted points can be saved in one or multiple files.

Usage

Click *Data Management* > *Clip* > *Clip by Circle*



Settings

- **Input Data**: Input one or more point cloud data files. File Format: *.LiData.
- **X Coordinate (m)**: Enter the X coordinate of the circle center.
- **Y Coordinate (m)**: Enter the Y coordinate of the circle center.
- **Radius (m)**: Enter the radius of the circle.
- **Ignore Different Additional Attributes**: When inputs are several point cloud files, and this option is checked, only the same additional attributes of the point cloud files will be merged. If this option is unchecked, and there are some different attributes in the additional attribute tables, the mergence of different point cloud files cannot be done.
- **Generate a File**: Extract all the point cloud data within 2D circle(s) to one file.
- **Generate Multiple Files**: Extract the point cloud data within each circle and save them to one individual file. The file is named by its center and radius by default.
- : After clicking this button, users will be able to draw circle in the current window. Single-click to choose the center of the circle, and drag the mouse to change the size of the circle. Double-click to end the selection. The coordiantes of the center of circle and the radius of the circle will be displayed in the window.
- : Click this button to add the coordinates and radius of a circle to the processing list. Perform the same operation to add multiple clipping circles.
- : Click this button to load external data files. The file format refers to the Clip Range File Format in the appendix.

- : Click this button to remove the selected row(s) in the processing list.
- **Output path**: The generated file will be saved in this output path.

Clip by Rectangle

Functional Overview

Clip by rectangle tool extracts the point cloud data within user-defined rectangular(s), and the extracted points can be saved in one or multiple files.

Usage

Click *Data Management* > *Clip* > *Clip by Rectangle*

Parameters Settings

- **Input Point cloud data**: Input one or more point cloud data files. File format: *.LiData.
- **Center Point**: Determine the rectangular box according to a selected point and area width.
- **Two Points**: Determine a rectangular box according to two selected points.
-  **Select Region**: After clicking this button, users will be able to draw rectangle in the current window. If a single point is selected, click to select a position in the view with the mouse, and the rectangle box information can be determined and displayed according to the area width parameter. If two points are selected, after clicking to select a position in the view with the mouse, drag the mouse to display the real-time rectangle box. Double click the mouse to end the rectangle selection, and the position information of the rectangle box will be displayed in the function interface.
- **Y Maximum (m)**: Enter the maximum Y coordinate of the rectangle.
- **Y Minimum (m)**: Enter the minimum Y coordinate of the rectangle.
- **X Minimum (m)**: Enter the minimum X coordinate of the rectangle.
- **X Maximum (m)**: Enter the maximum X coordinate of the rectangle.
- **Ignore Different Additional Attributes**: When inputs are several point cloud files, and this option is checked, only the same additional attribute tables can be merged, and the mergence of different point cloud files cannot be done.
- **Generate a File**: Extract all the point cloud data within the rectangle to a file.
- **Generate Multiple Files**: Extract the point cloud data within each rectangle and save them to one file.

The file is named by the rectangle's lower left corner coordinate and the rectangle's width and height by default.

- : Click this button to add the entered rectangle range to the processing list. Perform the same operation to add multiple clipping regions.
- : Click this button to load external data files. The file format refers to the Clip Range File Format in the appendix.
- : Click this button to remove the selected row(s) in the processing list.
- **Output path**: The path of the clipped output file after the function is executed.

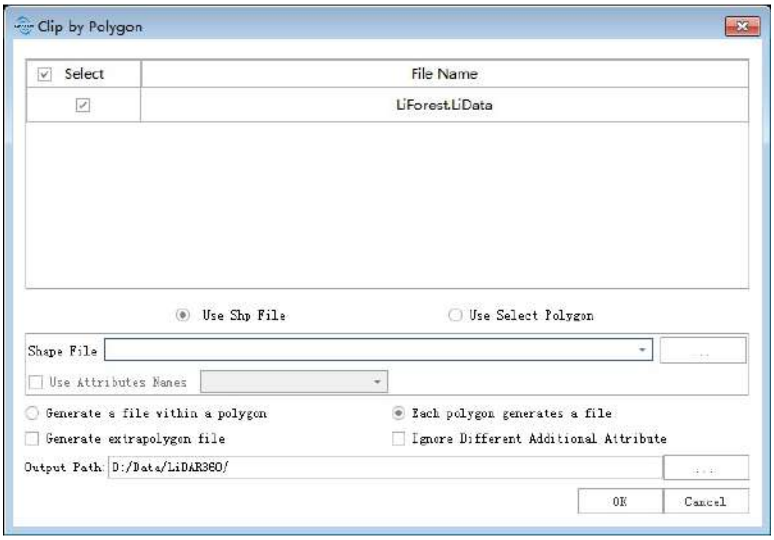
Clip by Polygon

Summary

Clip by polygon tool extracts the point cloud data within a user-defined polygon vector file, and the extracted points will be saved in one or several files.

Usage

Click *Data Management* > *Clip* > *Clip by Polygon*

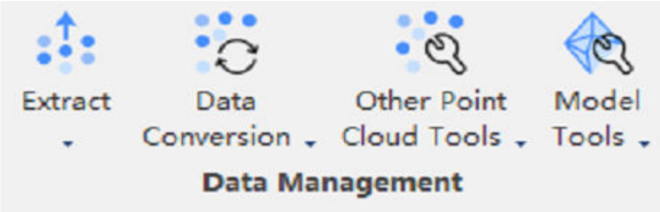


Settings

- **Input Data**: Select one or more point cloud data files. File Format: *.LiData.
- **Interactive Polygon**: Click to start to create the polygon. Click in the viewer to select the nodes and double-click to end selecting. After this, the polygon will be added to the polygon list. Click to delete the polygon in the list.
- **Use Vector File**: User can select the Vector File loaded into LiDAR360 software from the drop-down menu, or select button to load an external vector data file.
- **Ignore Different Additional Attributes**: When inputs are several point cloud files, and this option is checked, only the same additional attributes of the point cloud files will be merged. If this option is unchecked, and there are some different attributes in the additional attribute tables, the mergence of different point cloud files cannot be done.
- **Generate Single File**: When users choose to export a single file, the point cloud will be clipped by all polygons in the vector data. All results will be saved in one file.
- **Generate Multiple Files**: When users choose to export single file, the point cloud will be clipped by all polygons in the vector data. The point cloud in each polygon unit will be saved as one file.
- **Using Shp File Attribute Naming**: Available when **Generate Multiple Files** is selected and the vector data contains attribute tables. The user can select the corresponding attribute value as the file name. When the attribute value does not exist, the files are automatically named in numerical order.
- **Output Path**: The result output folder path.

Data Management

Data management includes Point Cloud Conversion, Raster Conversion, Model Conversion, Vector Conversion and Other Point Cloud Tools.



Extract

Extract includes:

Extract by Class, Extract by Elevation, Extract by Intensity, Extract by Return, Extract by Time, Extract by Segment.

Point Cloud Conversion

Point Cloud Conversion includes Convert LiData to LiData, Convert to ASCII, Convert to TIFF, Convert to Shp, Convert to DXF,

Convert to Las, Convert to E57, Convert LAS to LiData.

Raster Conversion

Raster Conversion includes: Convert TIFF to LiModel, Convert to Texture LiModel, Convert TIFF to LiData, Convert TIFF to ASCII.

Model Conversion

Model Conversion includes: Convert LiModel to TIFF, Convert LiTIN to DXF, Convert LiModel to TIFF, Convert LiBIM to OBJ, Convert LiBIM to FBX, Convert LiBIM to CityJson,

Vector Conversion

Vector Conversion includes: Convert Shape to KML, Convert KML to Shape.

Other Point Cloud Tools

Other Point Cloud Tools include: Denormalization, Remove Outliers, Resampling, Tile by Range, Tile by Point Number, Merge, Extract Point Cloud Boundary,

Extract Color from Image, Noise Filter, PCV, Subdivision, Transform GPS Time, Define Projection, Reprojection, Smooth Points.

Model Tools

Model Tools currently include LiBIM merge LiBIM Merge.

Processing tools

Processing tools include batch processing and distributed processing tools.

- Batch Processing
- Distributed Computing

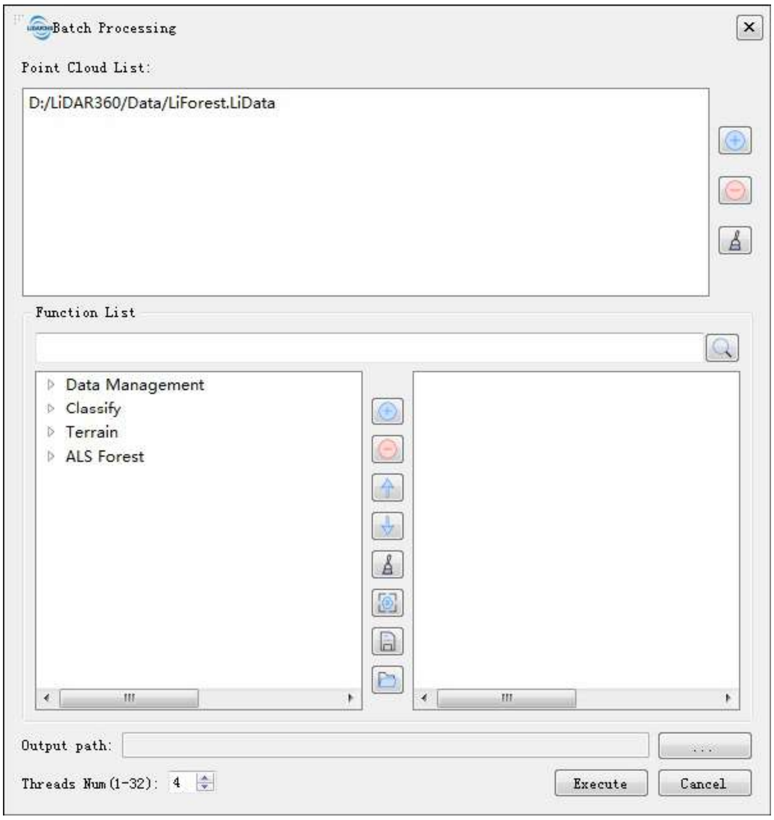
Batch Processing

Functional Description

In view of point cloud data to realize multi-data, multi-function, multi-thread streamlining batch processing operations, support the las and LiData type data, and provides two types of calling methods, these are dialog box and command line call batch processing function, the dialog calling batch processing function provides the operating sequence and parameters are saved in the function,in order to facilitate the use of other data. The following steps are described the calling method:

Structured Tool

1. After clicking the following interface pops up. The file list lists all the point cloud data in the system. The functions that support point cloud batch processing operations are listed in the left list of functions, and the list on the right shows the function sequence of the batch processing operations.



2. Click the on the right side of file list to select point clouds to be batch processing; Click the button to clear the file list; Click the button to remove selected point clouds (.las or .LiData type).

3. Double click the function in the left list (or select this function and click the button) and the function Parameters Settings interface will pop up. After Parameters Settings are completed, it will be displayed in the right list.
4. (Optional) Double click the function in the right list (or select this function and click the button) to modify the parameters.
5. (Optional) Select a function in the execution sequence, then click the or button to adjust the

order of execution.

6. (Optional) Click the button to clear the execution sequence.
7. (Optional) Click the button to save the execution sequence and corresponding parameters to file (.LiProcessList).
8. (Optional) Click the button to load execution sequence and corresponding parameters from file(.LiProcessList).
9. Set the number of threads for multi-thread batch processing in the thread number box (default 4). If the number of threads is set to 1, a single thread processing will be performed.
10. Set output path and click execute button, and the batch processing operations are processed in the order of the function list, and all intermediate results from this process are saved to the output path.

> Note: This tool is only applicable to point cloud data (.las or .LiData type). Some functions without parameters(e.g., PCV, Normalization by Ground Class) can be added directly and; A function that requires input raster data as an input parameter, and ensure the sequence of function operation (generating raster data first). If DEM is required for normalization function, DEM function should be put before normalized function.

11. You can enter a function name in the searching for quick search. Please note the software language version.

Invoke batch processing by CMD

1. Open the cmd.exe command line window, drag LiBatch.exe from the installation directory of LiDAR360 software into the command line window, or enter the installation directory layer by layer to invoke LiBatch.exe, and click Enter, software information, common command line commands and a list of command-line callable batch functions (in English and Chinese) will appear.

2. The command line supports the invocation of json files, the data and functions recorded in a json file can be executed in order by inputting -jsonFile plus the file name of json, and the final result is saved in the output folder json files can be generated using the interface batch processing or modified manually. But users must keep in strict accordance with the format of relevant parameters to avoid any parsing errors. For the list of Plugin ID and Action ID used to perform specific functions in the json file, refer to the appendixID List of Json Callable Functions for Batch Processing.

```
>> -jsonFile BatchProcessList.LiProcessList
```

3. In addition, according to the information prompt, you can enter the specific data file after -i as the input data (full path required). When inputting multiple files, users can enter -i plus multiple files separated by spaces, or -ifolder plus data type for filtering (.las or .LiData), default is LiData. It is worth noticing that users must ensure there is no space in the input file path, otherwise a parsing error will be raised. In fact, all parameter commands and specific parameters must be separated by spaces.

```
>> Outlier_Rmovel -ifolder ..\data\ las
```

4. Users can use -o command to set a folder path, instead of a specific file path. If users do not set this parameter, the output file will be saved in the same directory as the input file by default. Users can use -threadNum command to set the number of threads. Before calling a function, users can enter -h,

-H, -help or -or any command to view the parameter instructions of the specific function. Command format: command name (case sensitive) -----parameter introduction. To run a function without setting any parameters, users can use the command -default/-DEFAULT, or leave it blank.

> Outlier_Rmovel -h

5. Only one function can be called at each time. Taking outliers removal as an example, the valid input will be Outlier_Removal. The exact function name must appear in the parameter list. Input -i plus file name and hit "enter" button to run outliers removal to the input file, using default parameters, and save the output in the same folder as the input.

```
>> Outlier_Rmovel -i ..\data\*.LiData
```

6. In the classification module, see classify ground points as an example: Input classify ground points -h, window appears classify ground points function related command line help. For classification function, - fc is starting category, according to category list behind input corresponding to the category of digital by commas, if you do not input this order, it is starting category for all classes,- tc refers to target classification, we can input the corresponding category number.

```
>> Classify_ Ground_ Points -h
```

7. The meaning of the following command line is: Run the ground point classification function with 8 threads. The input data is ..\input*.LiData, and the output files are saved in ..\output\ (for classification functions, no new output files are generated, i.e., the class attributes are modified in the source data). The initial categories 1, 2, and 3 correspond to unclassified, ground points, and low vegetation points, respectively. The target category is ground points, with an iteration angle of 25 degrees and an iteration distance of 1.2 meters. Other parameters that are not set will use their default values.

```
>> Classify_ Ground_ Points -threadNum 8 -i ..\input\*.LiData -o ..\output\ -fc 1,2,3 -tc 2 -ia 25 -id 1.2
```

Batch Processing - Volume Measurement

Description

This function calculates the fill and excavation volumes for one or more point clouds within specified boundaries and outputs the results. Currently, only the TIN method is supported, which involves constructing a TIN from the point cloud and then calculating the fill and excavation volumes between the TIN and a specified plane.

Example Parameter Json File

```
{
  "FunctionNumbers": "1",
  "IsBatchProcessing": "0",
  "0": {
    "PluginID": "11",
    "enAction ID": "0",
    "Parameters": {
      "planeMmethod": "1",
                         "planeDir": {
        "0": "0",
        "1": "0",
                                 "2": "1"
      },
      "pointOnPlane": {
        "0": "0",
        "1": "0",
                                 "2": "-336.6666"
      },
      "polygonObjectName": {
        "0": "C:\/360MoveData\/Users\/XDF\/Desktop\/Test.shp"
      },
                         "attributeName" : ""
    },
    "enAction NAME": "Volume Measurement",
    "PluginVersion": "1.1"
  },
  "Output Path": "C:\/360MoveData\/Users\/XDF\/Desktop\/",
  "FileList": {
    "0": "C:\/360MoveData\/Users\/XDF\/Desktop\/Test.LiData"
  }
}
```


Parameter Description:

1. "Output Path": Must be specified as a folder path, not a file name. The correct format is "E:/Test". Results will be output as CSV files in this folder, with each LiData producing one CSV file. Each line in the CSV file represents the result for one polygonal area.
2. "FileList": Specifies one or more point clouds, separated by commas in JSON format. Example: "0":"1.LiData","1":"2.LiData".
3. "planeMethod": Method for defining the reference plane. Can be set to "0" or "1". "0" means the user specifies the reference plane, which can be any plane not perpendicular to the Z direction, generally a horizontal plane. "1" means the reference plane is automatically calculated based on the input polygon. When set to "0", parameters "planeDir" and "pointOnPlane" must be specified, defining the plane's direction and a point on the plane.
4. "planeDir": Plane direction in the plane-point form equation. Default is the Z-axis direction, i.e., "0","0","1". (The numbers before the colon "0","1","2" are indexes and cannot be changed.) The plane direction cannot be perpendicular to the Z-axis, otherwise, the calculation cannot be performed.
5. "pointOnPlane": A point on the plane in the plane-point form equation. Together with "planeDir", this parameter defines the plane.
6. "polygonObjectName": Specifies a vector file. In the resulting CSV file, each polygon in the vector file will generate a line of information. The results are shown in the figure below:
7. "attributeName": Optional, default is empty. If not empty and the shapefile contains this attribute field,

a column will be added to the CSV file with the attribute values from the shapefile object.

	A	B	C	D	E	F	G
1	Name	Projected Area	Cut Surface Area	Fill Surface Area	Cut(Cube Meters)	Fill(Cube Meters)	Total(Cube Meters)
2	0	12284.5193	25119.6841	0	543296.9741	0	543296.9741
3	1	11948.788	22008.3516	26537.6413	156.1381	430694.1247	430537.9867
4							

Distributed Processing

Description

Distributed processing supports multi-machine distributed operations for handling large volumes of point cloud data files.

Usage

Assuming you have four computers: one acting as the master node and the other three as computing nodes.

1.Set Up and Start Computing Nodes

On the three computers designated as computing nodes, click "Start Computing Node" on the toolbar in " Tools" Page. In the pop-up settings dialog,enter the IP address and port number for each computer. The port number can generally be left at its default value. After clicking the OK button, a computing node monitoring dialog will appear on each of the computing nodes.

2.Set Up and Start Master Node

Once the three computing nodes are running, click the "Distributed Computing" button on the Lidar360 toolbar to open the master node settings dialog. Use this dialog to configure distributed tasks, which involves three main aspects:

(1) Configure IP and Port for Each Computing Node

Ensure that the IP addresses and port numbers configured here match those entered in the computing node settings dialog mentioned earlier.

(2) Configure Computational Task

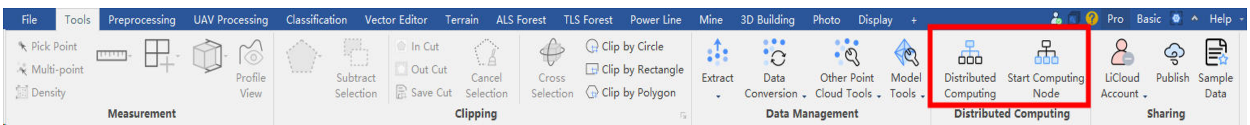
For example, you can configure the tasks to "Classify ground points first, then perform resampling."

(3) Set Source and Result Point Cloud Directories

The source point cloud directory contains the point cloud data to be processed, which can include multiple large data files. The results of the distributed processing will be output to the result point cloud directory.

Start Distributed Computing Process

Click the Execute button to open the master node monitoring dialog and start the distributed computing process.



Monitoring Distributed Tasks

In the master node monitoring interface, the top list displays real-time information about each source point cloud file chunk, including the IP of the computing node it's assigned to, the current stage of the chunk, the status of the chunk, and the time consumed for processing the chunk. The middle list shows real-time information for each computing node, including connection status, point cloud transfer speed, CPU usage, memory usage, the number of current Libatch instances running on the node, and the maximum number of Libatch instances run historically. The lower section displays log output and overall progress information for the distributed computation.

Preprocessing

Preprocessing page includes: Calibration, Adjustment, Classify Ground Points, Control Point Report, 3D Control Point Report、 Quality Inspection, Projections and Transformations.

Quality Inspection

Quality Inspection includes: Trajectory Quality Analysis, Elevation Difference Inspection, Strip Overlap Analysis, Density Quality Inspection

Projection and Transformations

Projection includes： Define Projection、 ReProjection

Transformation includes： Transformation、 Convert Projected Surface、 Elevation Adjustment、 Transformations Calculation、 Fitting Geoid Model

Alignment

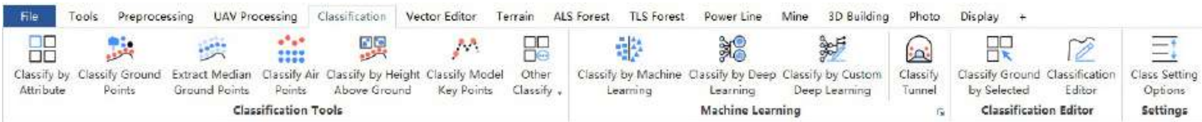
Alignment includes： Data Registration（ICP Registration、 Rectify、 Manual Rotate and Move）

Online Reconstruction

Online Reconstruction includes：DJI L1/L2 Reconstruction

Classification

Classification module is shown as figure below:



Classificaiton Tools includes： Classify by Attribute, Classify Ground Points, Extract Median Ground Points, Classify Air Points, Classify by Height Above Ground, Classify Model Key Points.

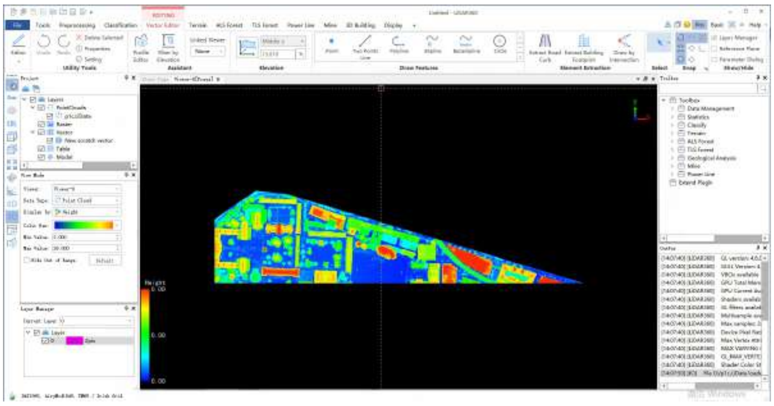
Other Classify includes： Classify Low Points, Classify Below Surface Points, Classify Isolated Points, Classify Noise Points, Classify by Min Elevation Difference, Classify Closeby Points, Classify Buildings.

Machine Learning： Classify by Machine Learning、 Classify by Deep Learning、 Classify by Custom Deep Learning、 Define Classification Tools.

Classification Editor includes: Classify Ground by Selected, Classify by Interactive Editing。

Vector Editor

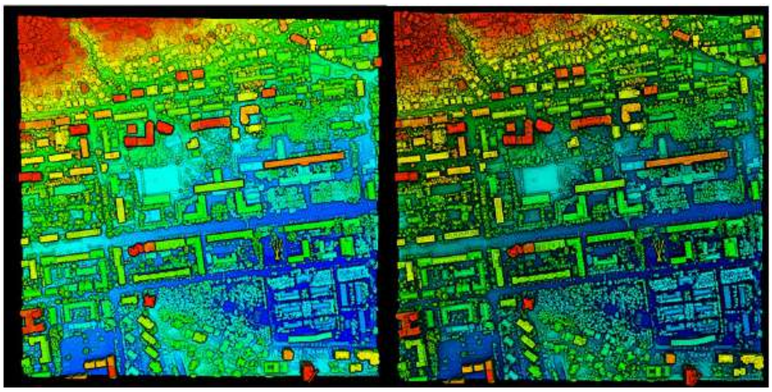
The vector editing function realizes the editing, modification and storage of vector objects, completing the vectorization part of the digital line drawing process. Based on point cloud, model and image data, it provides excellent display effect based on high contrast provided by point cloud, which can clearly distinguish buildings, vegetation areas, roads, street lamps, water bodies, bridges and other features. It generates vectorized results in formats such as SHP and DXF and supports integration with third-party software such as ArcGIS and AutoCAD.



By providing basic entity elements such as points, lines, polylines, rectangles, and polygons, the main key terrain vectorization is completed. The above-mentioned terrain entity elements are managed using a layer-based approach.

The vector object export function is provided to integrate the vectorization results into a DXF file for input into other mapping software. This can be symbolized and edited to output surveying and mapping results that comply with various standards.

Note: In order to make the terrain categories clearer and more distinguishable, PCV processing can be performed on point cloud data before vectorization. The left image in the figure below shows point cloud elevation + EDL display while the right image shows mixed display after PCV processing.



Usage

Click *vector editor* toolbar, which includes the following functions:

- Editor

- Entity Vectorization
- Element Extraction
- Basic Function
- Profile
- ElevationFilter
- Z Mode
- Linked Viewer
- Entity Modify
- Entity Selection
- Entity Snapper
- Layer Management
- Reference Plane
- Show Vertices

Editor

Control the start and end of vector editing, supporting the editing of one vector object at a time. Editable vector objects include: selecting a vector file for editing, creating a temporary vector file, and creating and editing a new file.

Select Vector File

Click Editor > Select Vector File

If there is vector object data in the editing window, a dialog box for selecting vector object data will pop up, listing the vector object data already loaded into the editing window. Select the data you want to edit.

If there is no vector object data in the editing window, a dialog box for selecting vector object data will pop up to load data from the disk for editing.

Once editing is enabled, other functions on the vector editing toolbar will be activated.

During editing, the vector data being edited cannot be removed from the editing window.

Create Temporary Scratch Vector

Click Editor > New Sketch Editing

A temporary editing canvas is constructed in the editing window, allowing for vector editing. The edited vector data is loaded into the directory tree as a temporary object in the window.

Once editing is enabled, other functions on the vector editing toolbar will be activated.

During editing, the vector data being edited cannot be removed from the editing window.

New Vector File Edit

Click Editor > New Vector File Edit

A file selection dialog box will pop up, generating the corresponding vector data file based on the user's selected path (supporting DXF and SHP formats, KML format is temporarily not supported; KML files can be converted to SHP format for editing using a conversion tool) and loading the vector data file into the editing window to enable editing.

Once editing is enabled, other functions on the vector editing toolbar will be activated.

During editing, the vector data being edited cannot be removed from the editing window.

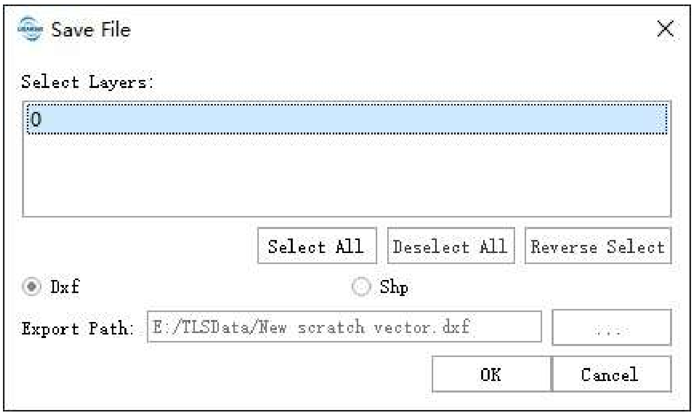
Export LabelMe

Click Editor > Export LabelMe

Save the vector results automatically extracted from the image as a LabelMe format file in the same folder as the image for custom deep learning model training.

Save

Description: Click *Editor > Save*. If it is a sketch and has not been saved before, the file save dialog box will pop up. Otherwise, it will be saved to the original file.



Select the layer, file type, and save path, and click "OK" to save the layer to the specified path.

> Note:
>
> 1. If the saved file has a .shp suffix, a corresponding directory named directoryname_layername_points.shp or directoryname_layername_lines.shp will be created.

End Editing

Click Editor > End Edit

Ending the current editing session will prompt a dialog box to save the editing results. The user can choose as needed, exit the editing function, and the vector editing module will become unavailable.

Basic Function

Vector Editor provides a variety of basic functions, such as Vertex Editing, Attribute Query, Undo, Redo, Delete entity, Zooming, and Setting.

Undo

Undo the last edit.

Redo

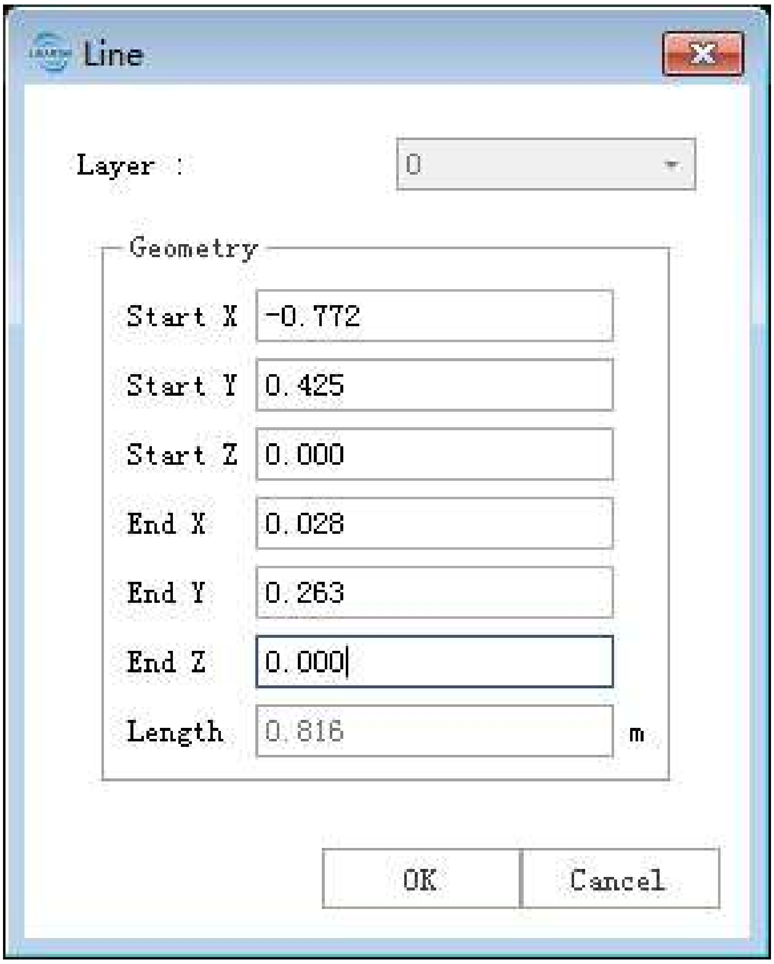
Redo the last edit.

Delete Entity

You need to first select the entities which need to be deleted, and then perform the delete operation.

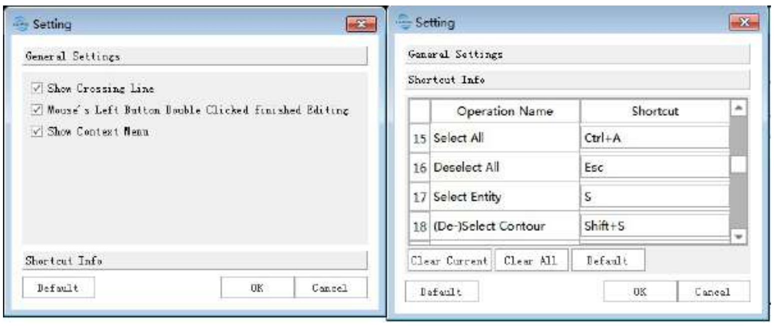
Attribute Query

You can query entity geometry attribute. Users can use the left mouse button to select the entity, which make the geometry attribute of the entity displayed in the pop-up dialog, as shown below.



Setting

Configure various parameters for vector measurement.



- **Show Crossing Line**: This parameter defines whether to display the cross wire in the screen center or not.
  - **Yes(default)**: Show.
  - **No**: Hide.
- **Double Click Mouse's Left Button to Finish Editing**：This parameter defines whether the current editing can be ended when double click the left mouse button.
  - **Yes**: Double click Mouse's left button to finish editing.
  - **No(default)**: Single click Mouse's right button to finish editing.

- **Show Context Menu**: This parameter determines whether to show context menu using right click.
  - **Yes(default)**: Show.
  - **No**: Hide.
- **Capture Distance (15px)**: Capture the distance.
- **Shortcut**：Shortcut settings.

Profile Editing

The vector editing module allows for vector editing with the assistance of the cross-section window.

Click

<p style="position:relative;top:-40px;left:36px;width:92%'> to open the cross-section window and create a new cross-section window for the vector editing module. For related operations, refer to Cross-Section Editing.

> Note:
>
> 1. When drawing lines, rectangles, or polygons in the cross-section window, the drawn vector elements are automatically restricted to the cutting plane.
> 2. When using the selection tool in the cross-section window to modify vertices or move vector objects, only the height values of the vertices or vector objects are modified.

Height Filtering

Parameter Settings

- **Position**: The midpoint of the Z-values of the point cloud to be displayed.
- **Buffer Zone**: The extent of the Z-value range, extending in both positive and negative directions from the center position.

Associated Window

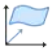
Function Overview

The associated window is an auxiliary function for vector editing, allowing users to load other types of data, such as TIN data and raster data, from the same area into the associated window for assisted editing. Users can perform operations such as drawing lines, placing points, and extracting building boundaries in the associated window.

Parameter Settings

- **Associated Window**: The window list contains all currently open windows in the software. Select an associated window for assisted editing.

Elevation

The elevation mode is an auxiliary function in vector editing used to select between scene elevation mode, fixed elevation mode, or contour elevation mode.

Function Overview

Real Z: During vector drawing and editing, uses the elevation captured from the results.

Constant Z: The elevation captured from points during vector drawing and editing will ultimately use the elevation of the transparent surface. The elevation of the transparent surface can be set by selecting points in the scene or manually inputting them.

Contour Z: During node addition and shaping processes, uses the starting point elevation of the selected vector to ensure consistency in elevation during contour editing.

> Note: During contour editing, it is essential to switch to contour elevation mode.

Parameter Settings

- **Height Source**: The reference plane is the horizontal plane, effective only in fixed elevation mode, supporting various sources for height:

  User Defined: Allows users to set a custom height value. You can manually input the height in the plane height editing box or click the height selection button on the right. Right-click in the scene to select a reference height point, and left-click to exit height selection.

  Min z: Uses the minimum Z value of the point cloud as the height value.

  Middle z: Uses the intermediate Z value of the point cloud as the height value.

  Max z: Uses the maximum Z value of the point cloud as the height value.

  Filter Center: Uses the center position value set in the ElevationFilter as the reference plane's height value.

- **Plane Height**: Displays the current set plane height.

Entity Vectorization

Entity vectorization, including vectorization of point, line and polygon entities, also provides specific editing functions for polylines and polygons respectively.

> Note: Point layers can only add point objects, not line, circle, polyline and polygon objects. Line layers cannot add point entities, only line, polyline, and polygon objects.

Point

Draw the entity point by clicking the left mouse button. The operation steps are as follows.

1. Click the left mouse button to select the point entity position.
2. Repeat step 1 to draw the next point entity.
3. Click the right mouse button to end drawing the point entity.

Two-point polyline

Draw a line segment entity, click the left mouse button to draw the start point of the line segment entity. The operation steps are as follows.

1. Click the left mouse button to select the starting point of the line segment entity, or right-click to end the drawing of the line segment entity.
2. Click the left mouse button to select the end point of the line segment entity, or right-click to go to Step 1.
3. Click the right mouse button to end drawing the line segment entity.

Polylines

Draw polylines interactively with the mouse.

1. Click the left mouse button to select a point, or click the right mouse button to exit drawing a polyline.
2. Repeat step 1 to add a point, or click the right mouse button to end the current polyline drawing. If you want to close, click the close button, and repeat step 1 to start a new polyline drawing.
3. Click the right mouse button to exit drawing the polyline.

B-Style Curves

B Draw B-style curves interactively with the mouse.

1. Click the left mouse button to select the point, or click the right mouse button to quit drawing the B-style curve.
2. Repeat step 1 to add points, or right-click to end the drawing of the current B-style curve, if you want

to close, click the close button, and repeat step 1 to start the drawing of a new B-style curve.
3. Click the right mouse button to exit drawing the B-style curve.

Bezier Curve

Draw a Bezier curve interactively with the mouse.

1. Click the left mouse button to select the point, or click the right mouse button to quit drawing the Bezier curve.
2. Repeat step 1 to add points, or click the right mouse button to end the drawing of the current Bezier curve, if you want to close, click the close button, and repeat step 1 to start the drawing of a new Bezier curve.
3. Click the right mouse button to exit drawing Bezier curves.

Circle


Draw a circle interactively with the mouse.

1. Click the left mouse button to select the center of the circle, or click the right mouse button to exit the circle drawing.
2. Double-click the left mouse button to select a point in the circle, or right-click to go to Step 1.
3. Click the right mouse button to exit drawing the circle.

Two-point circle


Draw a circle interactively with the mouse.

1. Click the left mouse button to select an endpoint on the diameter of the circle, or click the right mouse button to exit the two-point circle.
2. Double-click the left mouse button to select another end point on the circle diameter, or right-click to go back to Step 1.
3. Click the right mouse button to exit the two-point circle.

Three-point circle


Draw a circle interactively with the mouse.

1. Click the left mouse button to select a point on the circle, or click the right mouse button to exit the three-point circle.
2. Click the left mouse button to select the second point in the circle, or right-click to go to Step 1.
3. Double-click the left mouse button to select the third point in the circle, or right-click to go to Step 2.
4. Click the right mouse button to exit the three-point circle.

Three-point circular arc

Draw the arc interactively with the mouse.

1. Click the left mouse button to select the first point on the arc, or click the right mouse button to exit the three-point circular arc.
2. Click the left mouse button to select the second point on the arc, or click the left mouse button to go to Step 1.
3. Double-click the left mouse button to select the third point on the arc, or click to go to Step 2.
4. Click the right mouse button to exit the three-point circle arc.

Three-point rectangle

Draw a rectangle interactively with the mouse.

1. Click the left mouse button to select a point, or click the right mouse button to exit drawing the rectangle.
2. Repeat step 1 to add points, click three points to end the current drawing, repeat step 1 to start a new rectangle drawing.

Draw polygon

Provide a way to draw polygons and provide editing functions for adding and deleting nodes.

1. Click the left mouse button to select a point, or click the right mouse button to exit drawing polygons.
2. Repeat step 1 to add points, or click the right mouse button to end the current polygon drawing, repeat step 1 to start a new polygon drawing.
3. Click the right mouse button to exit drawing polygons.

Text

Add text annotation.

1. Click the text button to open the text setting parameters panel and set the parameters.
2. Left-click with the mouse to confirm the position of the text annotation, or right-click with the mouse to exit drawing the text annotation.

Parameters

- **Texts**: Set the content of the text annotation to be added.
- **Rotation(angle)**: The rotation angle for drawing the text, in a counterclockwise direction.
- **Height**: The height for drawing the text.

Add node

Interactively edit the polygon with the mouse and add nodes.

Interactively edit the polygon with the mouse and add nodes.

1. Click the left mouse button to select the polyline or polygon entity to be edited, or click the right mouse button to exit edit polygon.
2. Left-click a polyline or polygon entity location to add a node at that location, or right-click to end the current edit and repeat the steps to start a new edit.
3. Click the right mouse button to exit adding nodes.

Append node

Interactively edit polylines, B-style curves or Bezier curves with the mouse, adding nodes at the beginning and end.

1. Click the left mouse button to select the polyline entity to be edited, or click the right mouse button to exit editing. 2 Click the left mouse button to select the point, and judge the insertion position of the point according to the distance. If it is closer to the starting point, add a point before the starting point, otherwise add a point after the ending point, or click the right mouse button to end the polyline editing, and repeat the steps begin a new polyline edit.
2. In the attributes panel on the right, select the type of line segment to be added (polyline, B-style curve, Bezier curve). The default is polyline, and you can switch between the three by left-clicking the mouse.
3. Click the right mouse button to exit the append node.

Delete node

Edit the polygon interactively with the mouse and delete nodes.

1. Click the left mouse button to select the multi-line or polygon entity to be edited, or click the right mouse button to exit editing.
2. Click the left mouse button on the polygon entity node to delete the node, or click the right mouse button to end the current edit, and repeat the steps to start a new edit.
3. Click the right mouse button to exit the delete point.

Break in line

Interactively edit polygons with the mouse, breaking lines.

1. Click the left mouse button to select the polyline, B-style curve or Bezier curve to be edited, or click the right mouse button to exit editing.
2. Click the left mouse button to select the first intersection point on the polyline, or click the right mouse button to end the current edit, and repeat step one to start a new edit.
3. Click the left mouse button to select the second intersection point on the polyline to complete the interruption, or click the right mouse button to go back and select the first intersection point.

4. Click the right mouse button to exit the break line.

Break at Point

Edit polygons interactively with the mouse, breaking at points.

1. Click the left mouse button to select the polyline, B-style curve or Bezier curve to be edited, or click the right mouse button to exit editing.
2. Click the left mouse button to select an intersection point on the polyline, or click the right mouse button to end the current edit, and repeat step one to start a new edit.
3. Click the right mouse button to exit the break at point.

Merge

Interactively edit polygons with mouse, merge.

1. Click the left button of the mouse to continuously select multiple non-closed vector lines (polyline, B-style curve or Bezier curve), and click the merge function to merge the selected vector lines.
2. After clicking the merge function, click the left mouse button to continuously select the vector lines to be merged, and click the right mouse button to merge the selected multiple vector lines.
3. Click the left mouse button to select a vector line, click the right mouse button to select another vector line, and click the left mouse button to select another vector line to complete the merging.
4. Click the right mouse button to exit the merge.

Join Polylines

Edit polygons interactively with the mouse and prune it.

1. Click the left mouse button to select a polyline, rectangle or closed polyline as the baseline, or click the right mouse button to exit editing.
2. With a baseline selected, left-click to continue selecting baselines, or right-click to select a vector line to prune.
3. After entering the state of selecting the vector line to be pruned, click the left mouse button to select the vector line to be pruned, and the selected vector line will be deleted according to the intersection of the selected vector line and the reference line.
4. Right-click to go back to reselect the baseline, right-click again to exit pruning.

> Note: The intersection point of the pruning judgment is the intersection point on the xy plane and has nothing to do with the elevation of the vector.

Intersect Polylines

To intersect multiple lines through mouse interaction.

1. Click the left mouse button to select the first vector line for intersection, or click the right mouse button to exit editing.
2. Click the left mouse button to select the second vector line for intersection, or click the right mouse button to go back to step 1.
3. Select the desired intersection point to complete the operation.

Trim Segments

Editing and trimming polygons through mouse interaction.

1. Click the left mouse button to select a polyline, rectangle or closed polyline as the reference line, or click the right mouse button to exit editing.
2. After selecting a reference line, click the left mouse button to continue selecting reference lines, or click the right mouse button to prepare for selecting vector lines to trim.
3. After entering the state of selecting vector lines to trim, click the left mouse button to select the vector line that needs trimming. The selected vector line will be deleted based on its intersection with the reference line.
4. Clicking the right mouse button allows you to go back and reselect a reference line; clicking it again exits trimming.

> Note: The intersection point used for trimming judgment is on xy plane and has nothing to do with elevation of vectors.

Reshaping

By constructing sketch shaping lines or surfaces on selected elements. The element takes the shape of the sketch connected from the first intersection of the sketch and the element to the last intersection.

1. Click the left mouse button to select the multi-line or polygon entity to be edited, or click the right mouse button to exit editing.
2. Pick up points in the scene.
3. Select the line segment type of shaping (multi-segment line, B-style curve, Bezier curve) in the right attribute panel. The default is multi-segment line, and you can switch between the three by clicking the left mouse button.
4. Double-click on the polygon or it’s edge to end the current edit.
5. The non-closed polyline directly replaces the part between the first intersection point and the last intersection point with the newly drawn line segment. The closed polyline will retain the new two closed polylines.You need to click the left mouse button to select a closed polyline to be deleted.

Cut Polygons

Split the selected faces according to the drawn line.

1. Click the left mouse button to select the closed polyline or polygon entity to be edited, or click the right

mouse button to exit editing.
2. Pick points within a polyline or polygon area
3. Double-click on the polyline or polygon boundary to end the current edit.

Line smoothing simplification

 Smooth or simplify the polyline.

1. Click the Line Smoothing Simplification button, the parameter interface will pop up, and set the parameters.
2. Select the vector line object and click Execute.

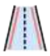
Parameters Settings

- **Enable smoothing**: Whether to use smoothing.
- **Smoothing Distance**: This parameter is valid when smoothing is enabled. Specifies the average distance between two adjacent points after smoothing.
- **Enable Simplification**: Whether to enable Simplification.
- **Simplified Distance**: Specifies the maximum distance deviation from points to lines to keep. As the distance gets smaller, the more points are kept, and the fewer points are removed from the line.

Feature Extraction

The feature extraction module is used for vectorizing elements in the scene. This module provides functionalities for extracting road curbs, intersection lines, building contours, contours from models, toe lines, intersection drawing, and vector extraction using SAM. The objects of vectorization in this module are buildings or roads. When any function of this module is activated, if there is no relevant layer, the software automatically adds a corresponding layer.

Extract Road Curb

 Activate the semi-automatic curb edge vectorization function

1. Click the left mouse button for the first time to determine the search box position, ideally near the middle of the curb face.
2. Click the left mouse button for the second time to determine the search direction, specifying the forward direction of the curb, and start the operation.
3. The software automatically extracts the upper and lower edges of the curb and adds them to the current layer.
4. (Optional) Click the toolbar's undo button to undo the last extracted edge line.
5. (Optional) Click the toolbar's redo button to re-add the last undone edge line.

Parameter Settings

- **Length (meters)**: The length of the search rectangle box, default is 0.5 meters. The larger the value, the faster the search speed, but too large a value may cross over two disconnected road sections.
- **Width (meters)**: The width of the search rectangle box, default is 0.2 meters. It should cover the curb area but not be too large.
- **Height (meters)**: The height of the search rectangle box, set according to the curb height to be extracted. Slightly larger than the curb, adjust using the height indicator line.
- **Curve Angle (degrees)**: The degree of curvature in the direction of the curb. For curved roads, the default is 30°.

Extract Building Footprint

 Activate the semi-automatic building contour vectorization function

1. Click the left mouse button to select the area where the contour is to be extracted.
2. Double-click to end the area selection and automatically extract the contour line of the selected area.
3. The contour line is vectorized into polylines and added to the building layer.
4. (Optional) Click the toolbar's undo button to undo the last extracted contour line.
5. (Optional) Click the toolbar's redo button to re-add the last undone contour line.

Parameter Settings

- **Point Source**: This parameter defines the type of volumetric measurement data source.
    - **from File (default)**: Uses points within the specified range from the point cloud file. The speed is

  - relatively slow, but the calculation result is stable.
  - **from Loaded**: Uses points within the specified range loaded into the scene. The speed is relatively fast, but changes in the scene's data loading may affect the calculation result.
- **Dividing Roof Layer (default off)**: When enabled, roof layers are clustered and roof boundaries are extracted for each height level within the selected area. When disabled, only the overall boundary is extracted.
- **Min Area (square meters)**: The minimum area for the extraction region. Boundaries smaller than this area will be discarded.
- **Inner (default off)**: Whether to extract internal boundaries. If enabled, the internal void boundaries of roofs with holes will be retained.
- **Outer (default on)**: Whether to extract the outer boundary. If enabled, the outer boundary of the roof will be retained.
- **Regularization (default on)**: Whether to regularize the extracted boundary. When enabled, the boundary is simplified and organized.
- **Angle Tolerance (degrees)**: Effective when regularization is enabled. The angle threshold for regularization is typically set between 15°-30°. A larger value will reduce the number of sharp angles on the boundary.

Extract Building Footprint from Mesh

 Activate the building contour extraction from model function

This function extracts building contours from the model. A parameter dialog box will pop up:

- **Min Area (square meters)**: The minimum area for the extraction region. Boundaries smaller than this area will be discarded.

- **Regularization (default on)**: Whether to regularize the extracted boundary. When enabled, the boundary is simplified and organized.

- **Angle Tolerance (degrees)**: Effective when regularization is enabled. The angle threshold for regularization is typically set between 15°-30°. A larger value will reduce the number of sharp angles on the boundary.

Click the left mouse button to confirm the level cutting plane position and execute contour extraction.

Trace Toe Tool

Refer to Toe Line Extraction Tool

Draw by Intersection

 Activate the intersection line drawing method for vectorizing buildings

1. Click the left mouse button to sequentially select two points on one edge of the building.
2. Click the left mouse button to sequentially select two points on another edge of the building.

3. The selected two edge lines will automatically extend to intersect at a point.
4. (Optional) Right-click to bring up the context menu and choose to close the polyline, undo a point, or end the current operation.
5. (Optional) Click the toolbar's undo button to undo a selected vertex.
6. (Optional) Click the toolbar's close button to automatically close the current polyline and end the intersection line drawing.

Extract Feature by SAM

 Activate the function to automatically extract vectors from images

Click on the image with the mouse to automatically recognize vectors from the image.

Parameter Settings

- **Selection Tool**: Specifies the interaction mode for automatic vector extraction.
  - **Point Selection**: Left-click with the mouse to automatically extract the vector. Hold down the shift key while left-clicking to add a positive point. Hold down the alt key while left-clicking to add a negative point.
  - **Rectangle Selection**: Left-click with the mouse to determine the starting point of the rectangle, move the mouse, and left-click to determine the endpoint of the rectangle. The vector will be automatically extracted.
- **Simplification**: Simplifies the extracted vector.
  - **Maximum Pixel Distance**: Specifies the maximum distance deviation to retain points to the line. As the distance decreases, more points are retained, and fewer are deleted from the line.
- **Smoothing**: Smooths the extracted vector.
  - **Mean Smoothing (3\5\7 neighborhood)**: Uses the 3\5\7 neighborhood for mean smoothing for each point on the vector line.
  - **Bezier Smoothing (Line Control)**: Smooths using piecewise Bezier curves, generating a vector line that does not pass through control points.
  - **Bezier Smoothing (Point Control)**: Smooths using piecewise Bezier curves, generating a vector line that passes through control points.
  - **B-Spline Curve (Point Control)**: Smooths using cubic B-spline curves, generating a vector line that passes through some control points.
  - **B-Spline Curve Fitting**: Fits using cubic B-spline curves, generating a vector line that does not pass through control points.
- **Regularization**: Regularizes the automatically extracted vector.
  - **Angle Threshold**: Merges line segments with angles smaller than this threshold into a straight line.
  - **Maximum Pixel Distance**: Specifies the maximum distance deviation to retain points to the line. As the distance decreases, more points are retained, and fewer are deleted from the line.
- **Grounding**: Adheres the automatically extracted vector to the point cloud or model, automatically modifying the elevation of the extracted vector line's nodes.
  - **Low Point**: Uses the 3\5\7 neighborhood for mean smoothing for each point on the vector line.
  - **Mid Point**: Smooths using piecewise Bezier curves, generating a vector line that does not pass through control points.

    - **High Point**: Smooths using piecewise Bezier curves, generating a vector line that does not pass through control points.
- **Output Result**: Specifies the type of automatically extracted vector.
    - **Polygon**: Specifies that the automatically extracted result is a polygon.
    - **Rectangle**: Specifies that the automatically extracted result is a rectangle.

Toe Extractor

Function Overview

This tool can automatically create a polygonal toe line, and the resulting vector can be used for volume calculations.

Usage

Start the vector editing module, select or create a new vector file, and in the feature extraction toolbar, click , to bring up the parameter setting box:

LiDAR360 Terrain

Set the parameters, then click on the pile in the point cloud as the "seed point" to start the extraction.

Parameter Settings

- **Maximum Size**: The range of the point cloud for extracting the toe line. This parameter represents the side length of the square. The point cloud search is conducted centered on the "seed point".
- **Show Rect (default on)**: Whether to display the search box centered on the current mouse position.
- **Grid Size**: This value can be set by sampling the data using the selection tool on the right. By clicking on a position in the point cloud, the local point spacing is estimated. Generally, it should be more than twice the point spacing. If the grid size is too small, there is a risk of extraction failure; if too large, the extracted toe line edges may not be precise enough.

- **Minimum Vertical Change**: The vertical change amount at the grid size step. The toe algorithm grows from the seed point in all directions. If the vertical change at the step distance between adjacent cells meets the setting, it continues to extend. A minimum change amount that is too small will result in a larger than actual range, and too large may result in an incomplete toe line.
- **Missing Data**: Whether the pile data within the extraction range is partially missing. Enabling this option sacrifices some efficiency but extracts a more complete toe line.
- **Smooth**: Whether to smooth the extracted toe line. Enabling this function will provide a smoother result, but there may be some misalignment with the original point cloud.

The extracted toe line result is shown as follows:

LiDAR360 Terrain

Entity Selection

The entity selection module allows for selecting vectorized entities, including options to select, select all, deselect all, select entities, window selection, select intersecting entities, select by layer, and invert selection.

Select

 Default selection, allowing for selecting and modifying entities. The following methods can be used to select and modify entities:

- **Single-click to select an entity**: Left-click to select an entity and clear the previous selection. Hold down the left Shift key to add to the selection.
- **Rectangle window selection**: Left-click to select the top-left and bottom-right corners of a rectangle window. Entities fully contained within this rectangle will be selected. Hold down the left Ctrl key to deselect entities within the rectangle. The previous selection will be cleared by default; hold down the left Shift key to add to the selection.
- **Rectangle window selection**: Left-click to select the bottom-right and top-left corners of a rectangle window. Entities partially or fully contained within this rectangle will be selected. Hold down the left Ctrl key to deselect entities within the rectangle. The previous selection will be cleared by default; hold down the left Shift key to add to the selection.
- **Modify entity vertices**: Click on a selected entity's vertex and drag to move and modify its position. In the main vector editing window and associated window, only the X and Y coordinates of the vertex will be modified; in the profile window, only the Z coordinate will be modified.
- **Modify entities**: Press the M key, left-click to select a reference point, and drag to move and modify the position of all selected entities. In the main vector editing window and associated window, only the X and Y coordinates of the entity will be modified; in the profile window, only the Z coordinate will be modified.

Select All

 Select all entities within the scene.

Deselect All

 Deselect all entities within the scene.

Select Intersecting Entities

 Left-click and draw a line. Entities intersecting with this line will be selected. Hold down the left Shift key to deselect entities.

Select Layer

 Left-click to select an entity, and all entities within the same layer will be selected.

Invert Selection

 Currently selected entities will be deselected, and currently unselected entities will be selected.

Entity Snap

Vector Editor provides a variety of snapper operations, The function can control the point location clicked by the mouse, which can improve accuracy of vectorization.

Snap on Endpoint

Snap startpoints and endpoints, such as starting point and ending point of one line.

Snap on Entity

Select the closed point of entity, snapping can move along the entity.

Snap Center

Snap the center of the entity, such as the center of a circle.

Snap Middle

Snap middle points of an entity, such as the midpoint of the line segment.

Snap Intersection

Snap intersecting points of multiple entities, such as the intersection of two lines.

Restrict Orthogonal

After selecting a current point, the subsequent selected points can only move in the orthogonal direction (vertical or horizontal direction) of the current point.

Snap Point Cloud

Snap points on the point cloud, this snap is suitable for drawing corners of house etc.

Snap Model

Snap the closest point of model.

Show/Hide

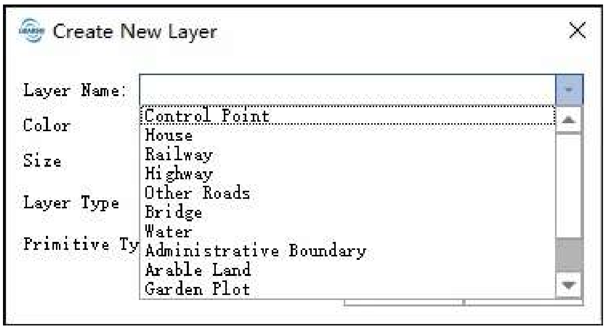
Layer Management

Description: Vector editing is managed with layers, which means a file is treated as a layer. Clicking the triangle button of the "File" will show the drop-down menu, including the "New File", "Open File", "Save File", "Remove File" and "Export File" functions.

Add Layer

Click *Layer>Right-Click Menu>Add Layer*, and the "Create New Layer" dialog box will pop up.

1. Select the layer name from the drop-down menu, including: control points, buildings, railways, roads, other roads, bridges, water bodies, administrative boundaries, cultivated land, gardens, forests, grasslands and other vegetation. Alternatively, you can directly enter the layer name as shown in the following figure:



2. Select layer type (oint or line). If creating point layer, you need to set color and point size; if creating line layer, you need to set line color, size and type.

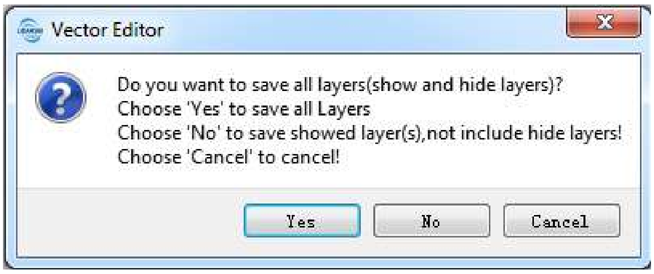
3. Click the layer color to select the layer color.

4. Size setting is between 1-10 pixel.

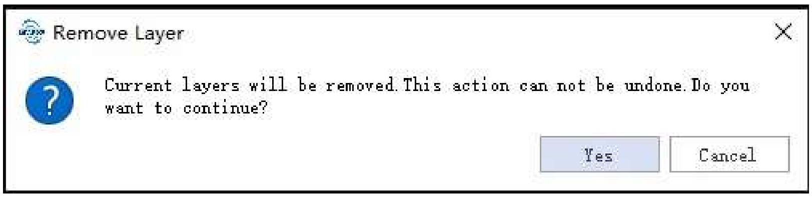
> Note:One vector layer can only store vector entities of current layer. If the created layer is a line layer, only lines, polylines, Bezier curves and other types of lines can be drawn in this layer. If the tool currently being used does not match the type of the active layer, the layer will turn to that the tool corresponded. The default layers for point tools are "point", polyline tools are "polyline", polygon tools correspond to "polygon" and text tools correspond to "text". When activating a tool, if the current layer type does not match and there is no corresponding default layer available, a new default layer will be automatically created.

Save All Layers

Click the "Remove All/Current Layer" function to pop up a dialog box:



Select the file type and path to save, click OK, and save the layer to the corresponding path. Note: If the saved file extension is shp, a directory named after the layer name with _points.shp or _lines.shp will be created in the directory. #### Remove All/Current Layers #### Right click *Layer>Remove All Layers*:



- Click "Yes" to save all/current layers and remove all/current layers;
- Click "No" to not save all/current layers and remove layers;
- Click "Cancel" to exit the current remove all/current layer operation.

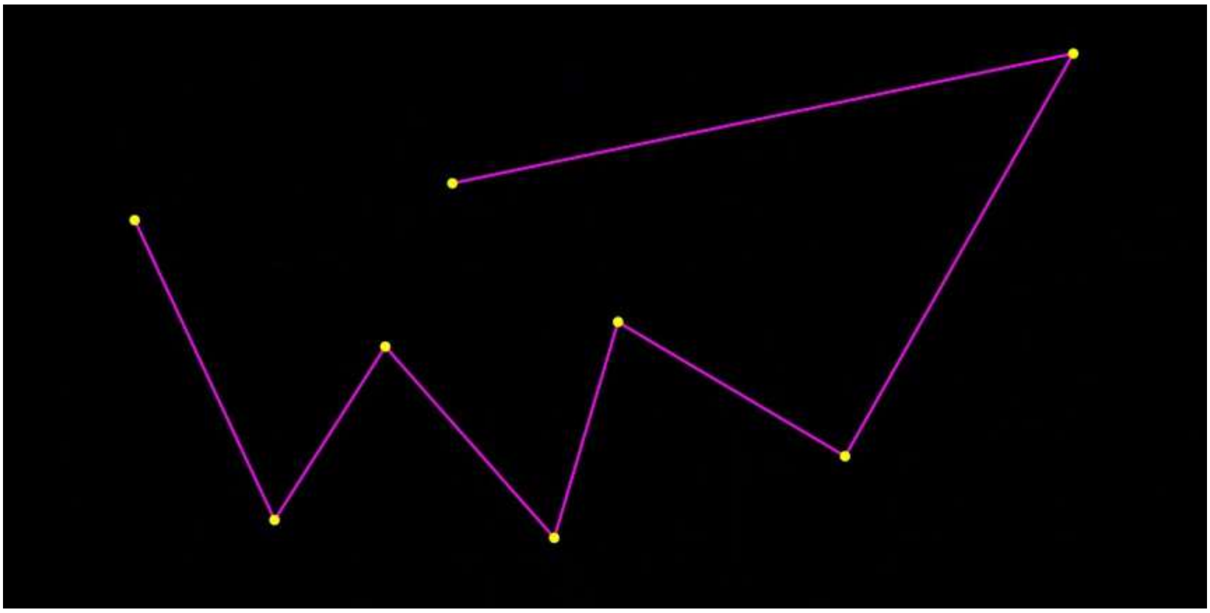
show/hide

Shows or hides the elements drawn by the current layer.

Show Vertices

Feature Overview

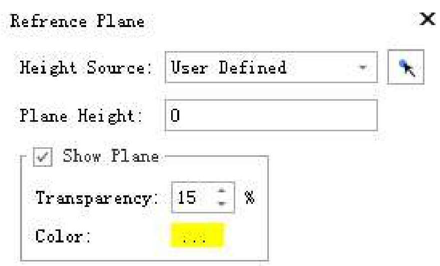
The display vertices toggle allows users to choose whether to display the endpoints of polylines, making it easier to edit entities.



Reference Plane

Feature Overview

The reference plane is a drawing plane set in the main window of the vector editing module. When drawing lines, rectangles, and polygons, the reference plane settings interface will automatically activate. During these drawing operations in the main window and associated windows, the drawn vector objects will be restricted to this reference plane.



Parameter Settings

- **Show Plane**: Toggle to display the drawing reference plane.
- **Transparency**: Sets the transparency of the reference plane, with a range of 0-99.
- **Color**: Sets the color of the reference plane. Click the button to open the color selection dialog box.

Terrain

Terrain module has two modes: Basic and Advanced. Click the button at upper-right corner to select mode.Basic mode provides one-click processing for terrain products generation.

Advanced

Tools of Advanced mode is shown as figure below:

Preprocessing includes： Subsampling, Remove Outliers, Classify Groud Points, Smooth Points

Based on Point Cloud includes： DEM, DSM, Point Cloud to Contour, Generate TIN

Based on Raster includes： Hillshade, Slope, Roughness, Aspect

Based on TIN includes： TIN to Contour, TIN to DEM

Terrain Conversion includes： Convert TIFF to LiModel, Convert to Texture LiModel, Convert LiModel to TIFF

Analysis Tools includes: Deviation Analysis, Change Detection Construction Progress Analysis

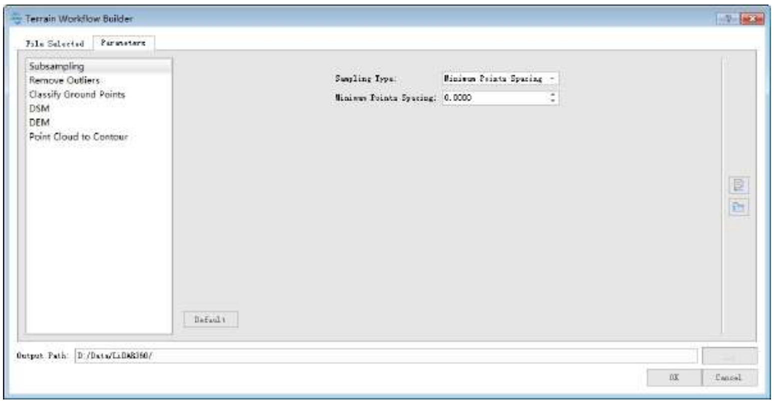
Model Editor includes： LiModel Editor, LiTin Editor

Other tools include: Section Analysis, Query Dip and Strike, Profile Editor

Basic

Tools of Basic mode is shown as figure below:

Click "Run" and a dialog will pop up. Click "File Selected" and select the point cloud file to be processed. Then, go to Parameter tab and set parameters for functions. After that, click "OK" and batch process the data, results will be saved to the specified Output Path.



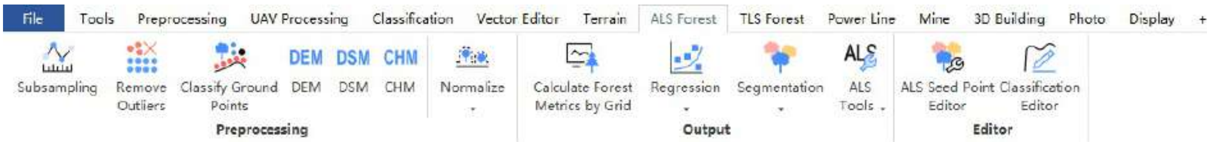
Other tools include: Section Analysis, Vector Editor, Query Dip and Strike, Profile Editor.

ALSForest

ALS Forest module provides two modes:Basic and Advanced.

Advanced

Advanced mode is showed as figure below:



ALS Preprocessing includes： Subsampling, Remove Outliers, Classify Ground Points, DEM, DSM, CHM, Normalize(Normalize by DEM, Normalize by Ground Points)

ALS Tools includes: Calculate Forest Metrics by Grid

Regression Analysis:Linear Regression, Support Vector Machine, Fast Artificial Neural Network

Random Forest Regression, Run Existing Regression Model

Segmentation:Point Cloud Segmentation, Generate Seeds by Layer Stacking, Generate Seeds by CHM

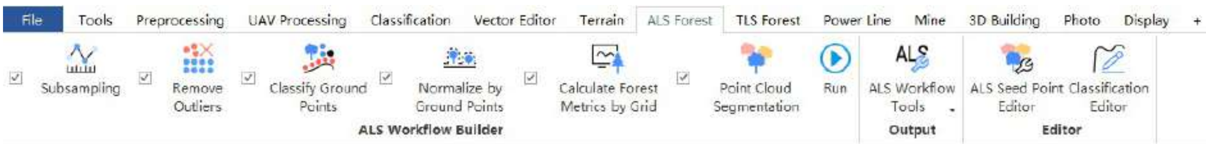
Point Cloud Segmentation from Seed Points, CHM Segmentation

ALS Tools: Clear Tree ID, Extract by Tree ID, Statistic Individual Tree Attributes Denormalization

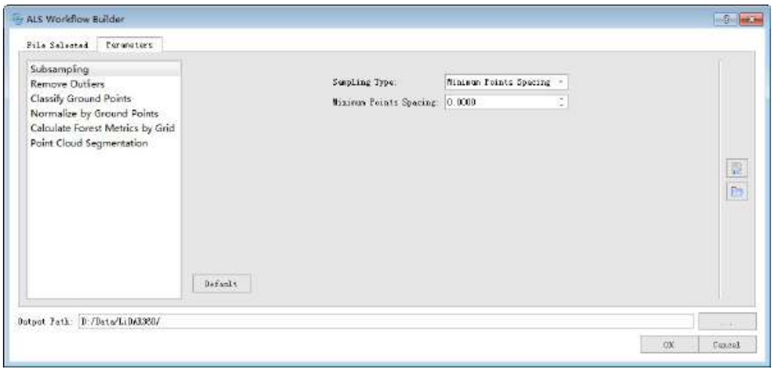
ALS Forest Editor includes:ALS Seed Point Editor, Profile Editor

Basic

Basic mode is showed as figure below:



Click"Run", and a window pop up as figure below. Click "File selected" tab and select the point cloud file.Then, click "Parameter" tab and set parameters for each function. Click "OK" and the functions will batch process. Result data will be saved under "Output Path".



ALS Tools includes:Clear Tree ID, Extract by TreeID, Statistic Individual Tree Attributes), Denormalization

ALS Forest Editor includes:ALS Seed Point Editor, Profile Editor

TLSForest

TLS Forest module has two modes: Basic and Advanced. Click the button at upper-right corner to select mode.Basic mode provides one-click processing for TLS analysis.

Advanced

Tools of Advanced mode is shown as figure below:

Preprocessing includes： Subsampling, Removal Outliers, Classify Ground Points, Normalization(Normalization, Normalization by Ground Points)

TLS Tools include: Leaf Area Index

Segmentation: Point Cloud Segmentation， Point Cloud Segmentation from Seed Points

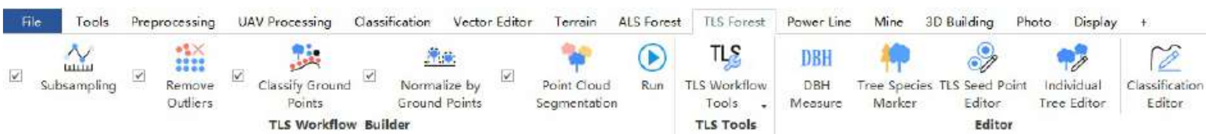
Tree Attribute: Statistic Individual Tree Attributes， Increase Individual Tree Attributes

Other ALS Tools: Clear Tree ID， Extract by Tree ID， Denormalization

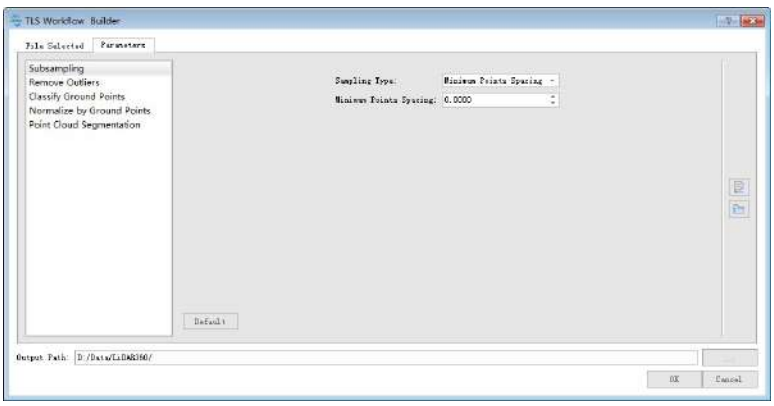
TLS Forest Editor includes： DBH Measure， Tree Species Identification TLS Seed Point Editor， Individual Tree Editor， Profile Editor

Basic

Tools of Basic mode is shown as figure below:



Click "Run" and a dialog will pop up. Click "File Selected" and select the point cloud file to be processed. Then, go to Parameter tab and set parameters for functions. After that, click "OK" and batch process the data, results will be saved to the specified Output Path.



TLS Tools include: Clear Tree ID， Extract by TreeID

TLS Forest Editor includes: DBH Measure， Tree Species Identification TLS Seed Point Editor， Individual Tree Editor， Profile Editor

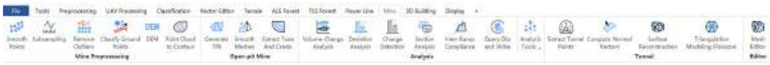
Power Line

Powerline page is shown as figure below:

Tools include: Setting, Mark Tower, Classify, Danger Points, Clear User Data, Profile Editor

Mine

Mine module provides tools related to mines.



Preprocessing

Preprocessing functions include:

Smooth Points, Resampling, Remove Outliers, Classify Ground Points, DEM, Point Cloud to Contour,

Open-pit Mine

Open-pit Mine includes the following functions:

Generate TIN, Smooth Meshes, Slope Line Extraction.

Analysis

Analysis includes the following functions:

Volume change analysis, Deviation Analysis, Change Detection, Section Analysis,

Tunnel

Tunnel includes the following functions:

Extract tunnel points, Compute Normal Vectors, Triangulation modeling, Poisson modeling.

basic mode

Functions of basic mode are shown as figure below:

Click “Run", the dialog will pop up as shown in the picture below. Click "File Selected" and select the point cloud file to be processed; go to Parameter tab and set parameters for functions. After that, click "OK" and batch process the data, results will be saved to the specified Output Path. For corresponding parameters and Settings, refer to the corresponding function documents.

Buildings

The building module includes following functions:

Preprocessing includes:

- Subsampling
- Remove Outliers
- Classify Ground Points
- Classify by DeepLearning

Color Processing includes:

- DSM
- DOM

Model Processing includes:

- Extract Building Footprint from Image
- Auto Create Buildings
- Project Textures fromPhotos
- Attributes Calculation

Model Editing includes:

- Building Editor
- Matched Photo Editor
- Materials Editor

Display

This chapter explains usages of different modules and functions on Display page, includes Display and Record, Viewers and Operations.

Display and Record

Display menu of the software.

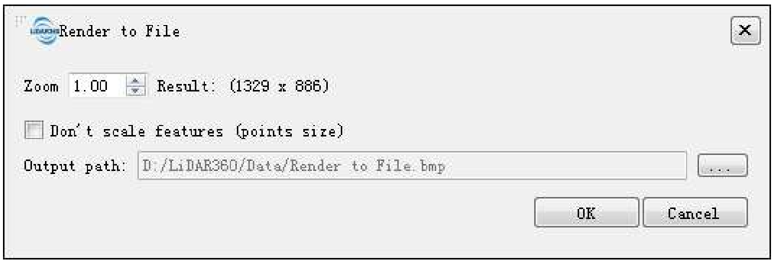
- Render to Image
- Camera Roam
- Save to Video
- Background Grid
- Camera Setting
- Display Solid Model
- Display Wireframe
- Display Vertices

Render to Image

Functional Description: This tool can render the current view as an image file (*.bmp format is supported). The way of scaling to equal proportions based on the window size and from width to height is supported to generate larger than the current window size of the image. Settings to select whether to scale features is supported, and if window size scaling ratio is greater than 1, and no feature is scaled, a more detailed picture than the window display effect can be obtained.

Steps

1. Adjust the 3D view to get the scene you want to render.
2. Click *Menu* > *Display* > *Render to Image*Use this function, the pop-up interface is as follows.



3. (Optional) Set the zoom factor.
4. (Optional) According to demand to choose check or won't check the "Don't scale features (points size)".
5. (Optional) Set the output path.
6. Click "OK" to get the rendered image.

Parameters Settings

- **Shortcut Keys**: None
- **Input Function**: Current active window.
- **Zoom**: This parameter defines how many times the rendered image size can be enlarged based on the original window size.
- **Size**: This parameter does not support the setting. The size of the output image is calculated in real time based on the zoom factor, and the size is displayed in parentheses. When the actual size is too large to be supported by the running environment, the output picture will fail and a prompt will be given in the output window.
- **Don't scale features (points size)**: By default unchecked. At this time, the output picture quality is the same as the original window, and no more detailed content will be obtained. When the scaling ratio is 1, this parameter has no effect. When the zoom ratio is greater than 1 and this function is selected, the pixel size occupied by each point in the point cloud is the same as the pixel size occupied by the window. When the picture size is larger than the window, the details of the point cloud seen in the picture are clearer, and the picture with finer display effect than the window is obtained.
- **Output path**: Set the image output path. Note: When EDL is on, Render to File tool supports adding the EDL effects to the results.

Capture Image

 **Functional Description**: Save the view in current window as a JPG file.

Steps

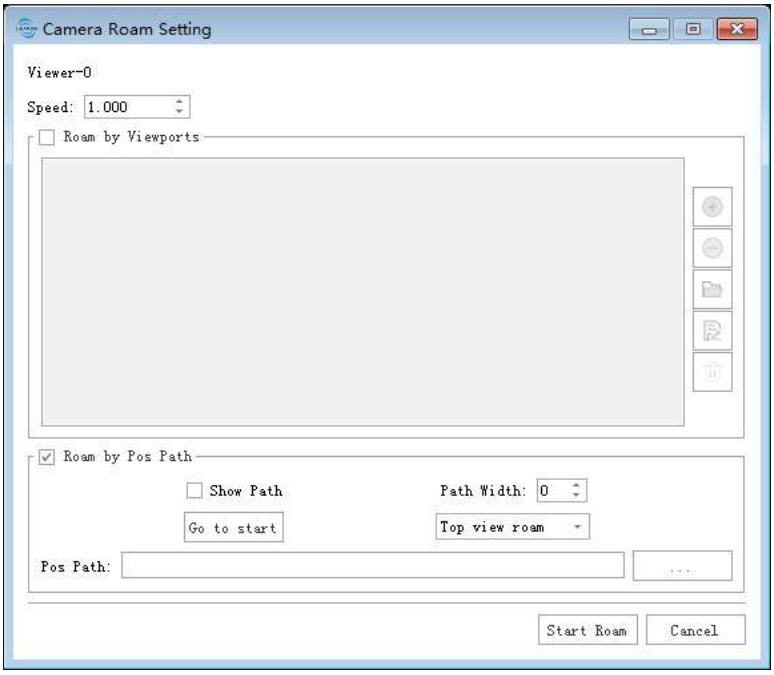
1. Clicking this button, and click "OK" in the pop-up to save the file with the default file name the same as the window name.

Camera Roam

Brief: This function can control the scene camera to navigate through a roam path. Two types of roam path are supported including roam path based on viewport and roam path based on POS file. This function can also generate immersive video if cooperated with Save to Video function.

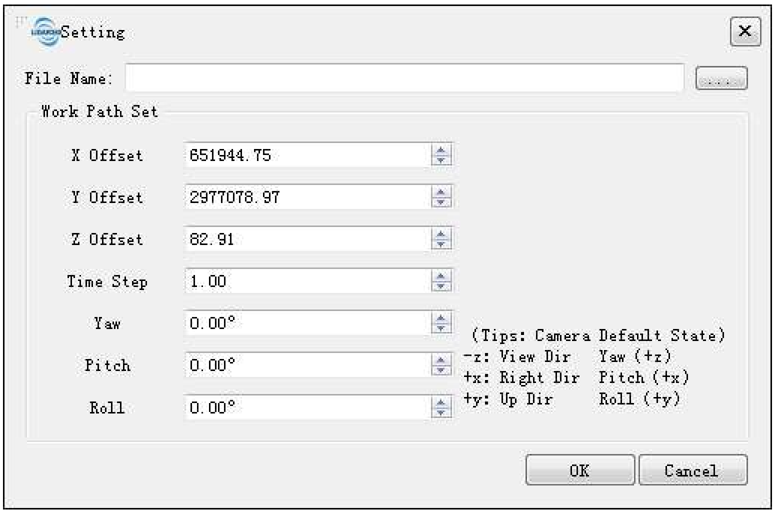
Steps

1. Click *Display* > *Camera Roam* to open Camera Roam Set dialog as bellow.



2. Choose mode as desired(Select "Roam by Viewports" or "Roam by Pos Path").

   - 2.1 Roam by Viewports
     - Speed: Set the speed of Roam by Viewports.
     - Add Key Frame： Click this button to add current viewport as a key frame.
     - Delete Key Frame: Click this button to delete a key frame.
     - Load Key Frames from File: Click this button to import a key frame file.
     - Save Key Frames to File: Click this button to export all key frames to file.
     - Delete All Key Frames: Click this button to delete all key frames.
     - Select a key frame in the list: Scene camera will be adjusted to this key frame.
   - 2.2 Roam by POS File
     - Check or uncheck "Show Path" as desired.
     - Set the Path Width, which will be activated when "Show Path" is checked.
     - Click Settings to open the following dialog.



- Select a POS File: txt format only, each line of the file represents a key frame. The first three columns are assumed to be XYZ. Column delimiters include comma, semicolon and space.
- Set parameters: Including offset, time step, yaw, pitch, roll.
- Click "Ok" to end the roam path setting.

3. Click "Start Roam" to begin roam through current path. After finished, the following dialog will show.

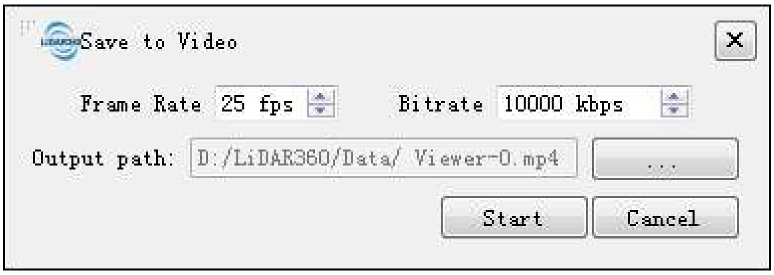
4. (Optional) Click "Pause/Continue" to control the pause and resume of roaming.
5. Click "Stop" to exit this roam tour.
6. Click "Cancel" to exit this function.

Save To Video

Brief: Record the current window screen and save as video (mp4 and avi formats are supported).

Steps

1. Click *Display* > *Save To Video*, an interface will show as follows.



2. Set the frame rate(default is 25 frame per second).
3. Set the bit rate.
4. Set the output path of video.
5. Click "OK". The following interface is shown on the left-top corner of the current window.

6. Change the camera to get the scene which you want to record.
7. Click "Stop" or "Start"
   - Stop：Pause the current record.
   - Start：Recover the record.
8. Click "End" to finish recording. The video will be saved.

Settings

- **Input**: the scene of the window for record.
- **Frame Rate**: The frequency (rate) at which consecutive images called frames appear on a display.
- **Bit Rate**: Refers to the number of bits used per unit of playback time to represent the video. Lower bit rate may result in lower resolution of the picture and has mosaics on the picture, and however the higher bit rate may lead a larger video file. The default bit rate is automatically adjusted according to the system resolution.
- **Output path**: The file path to which the video will be saved.

  > Note: When EDL is on, Save to Video tool supports adding the EDL effects to the results.

Background Grid

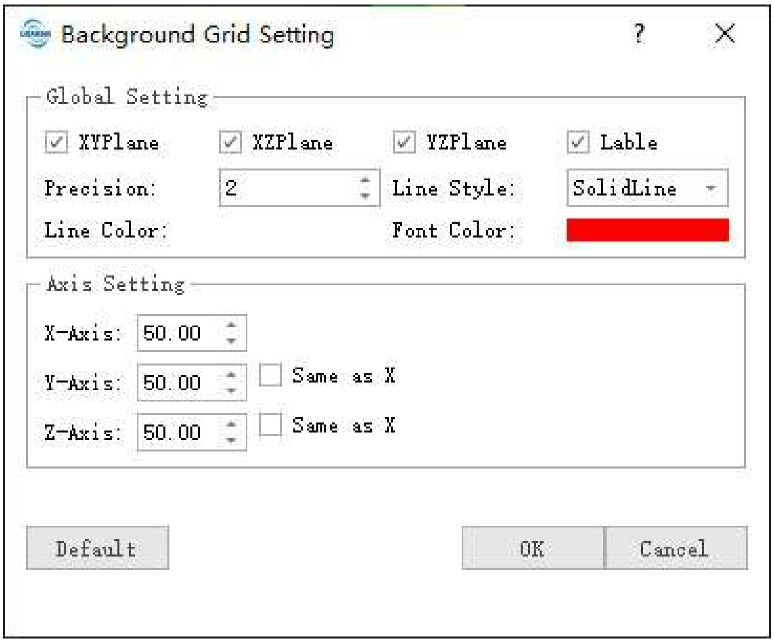
Summary: This tool can be used to display the 3D grid in the viewer window. When there are multiple point clouds in the window and the distance between the point clouds is short, the grid range of all point clouds is calculated and displayed. When there are multiple grids in the window and the point cloud is far away, the grid range of the point cloud within the display range of the current window is calculated and displayed.

Steps

1. Click *Display* > *Background Grid* to use this function.

Settings

Function Summary: Set the display or hide of grid plane (XY plane, XZ plane, YZ plane), display or hide of grid label, grid line style, line color, and font color.



- **Grid Plane (XY Plane, XZ Plane, YZ Plane) Display or Hide (Default is Display All)**: Check or uncheck the checkboxes in front of the three planes to change the display status of the corresponding planes.
- **Grid Label Display or Hide (Default is Display)**: Check or uncheck the checkbox before the text to change the display status of the label.
- **Line Style (Default is Solid Line)**: Click , select the grid line style (solid or dot).
- **Line Color (Default is White)**: Click , a color selection interface will pop up, and one of these colors can be selected as the color of the line.
- **Font Color (Default is Red)**: Click , a color selection interface will pop up, and one of these colors can be selected as the color of the font.
- **Axis Setting (X-Axis, Y-Axis, Z-Axis) Cell Size (Default is "50")**: Click , the value will increase or decrease in steps of 10. User can also directly enter a specific value. The cell size of the

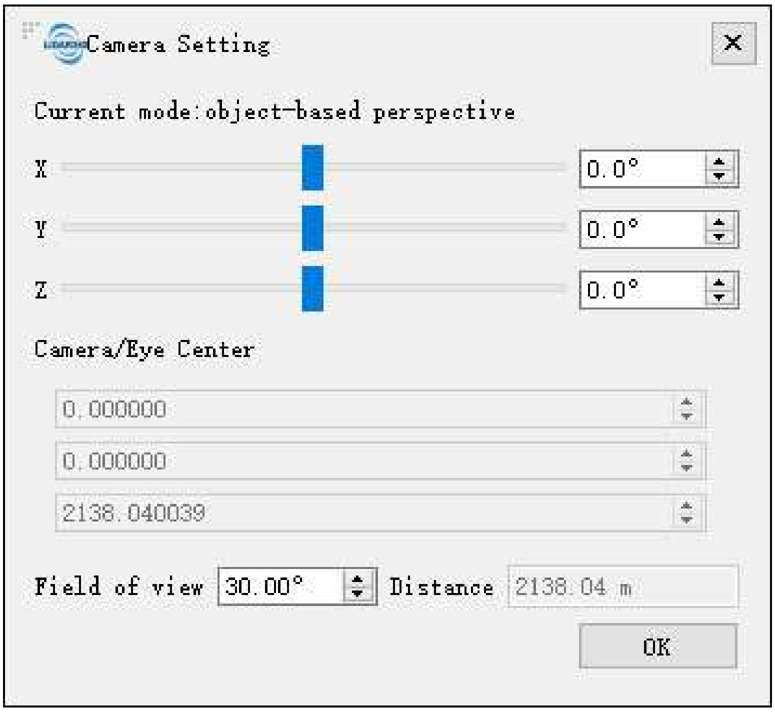
Y-axis and Z-axis can be set to be the same as the X-axis, so that there is no need to set the cell size of the Y-axis and Z-axis separately.

Camera Setting

Function Description: Allows viewing and changing the camera settings of the currently active 3D view.

Steps

1. After clicking this function, the following interface will pop up:



2. Use the corresponding sliders to set the camera's angle position in the X, Y, and Z directions.

Parameter Settings

- **Function Input**: The current active view is displayed in 3D mode.
- **Current Mode**: Displays the projection method of the current view, including perspective projection and orthographic projection.
- **X**: The camera's angle position in the X direction.
- **Y**: The camera's angle position in the Y direction.
- **Z**: The camera's angle position in the Z direction.
- **Camera/Eye Center**: This parameter cannot be manually set; it shows the real-time viewpoint center of the camera in the view.
  - **First Parameter**: The camera's position in the X direction.
  - **Second Parameter**: The camera's position in the Y direction.
  - **Third Parameter**: The camera's position in the Z direction.
- **Field of View**: The field of view angle.
- **Distance**: The distance to the viewpoint, which cannot be manually set and shows the real-time distance to the viewpoint.

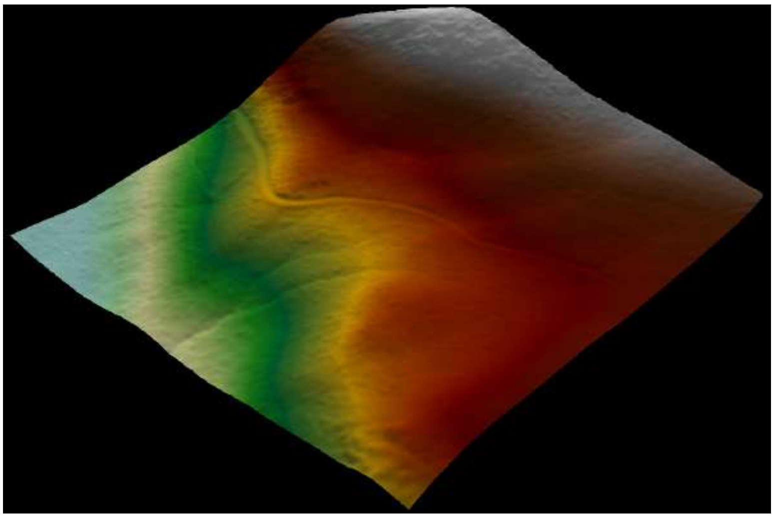
> Note: This function is only for the 3D mode of the view and cannot be set in 2D mode.

Display Solid Model

Brief：Set model files in the current window to display in model type.

Steps

1. Click this function, the models in the current active window will display in model, as shown in the figure below:



Settings

- **Shortcut key**：Press the "W" key to switch the display mode between model, triangle and point.

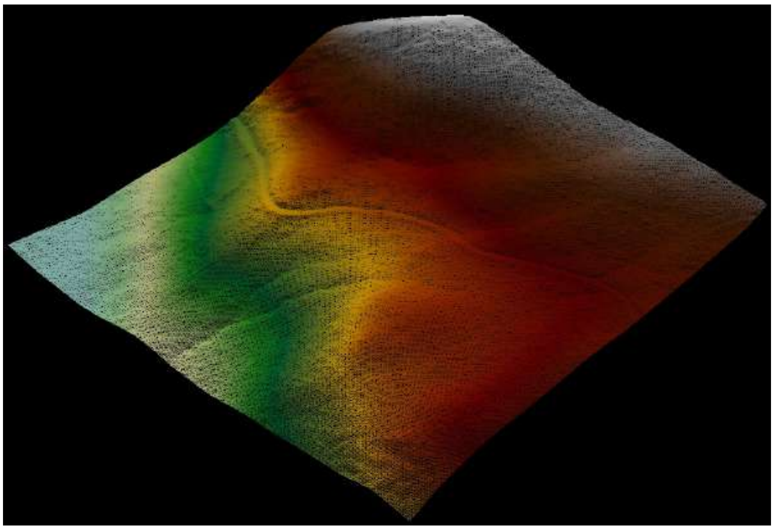
> Note: The model types for this function includes LiTIN, LiModel, IVE and other model types.

Display Wireframe

Brief: Switch display mode of models in current window to triangle mesh mode.

Steps

1. Click this button, the models in current active window will display as triangle mesh, as shown in the figure below:



Settings

- **Shortcut key**：Press the "W" key to switch the display mode between model, triangle and point.

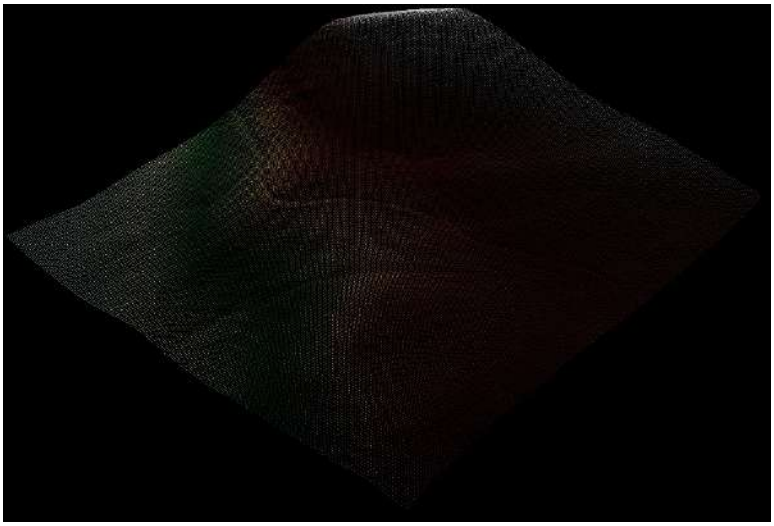
> Note: The model types for this function includes LiTIN, LiModel, IVE and other model types.

Display Vertices

Brief: Switch display mode of models in current window to point mode.

Steps

1. Click this button, the models in current active window will display as points, as shown in the figure below:



Settings

- **Shortcut key**：Press the "W" key to switch the display mode between model, triangle and point.

> Note: The model types for this function includes LiTIN, LiModel, IVE and other model types.

Viewers

This menu provides operations to create, close and arrange windows.

- New Viewer
- Close Active Viewer
- Close All Viewers
- Viewers Layout
    - Tile Viewers
    - Tab Viewers

New Window

Brief：Create new window in the current project.

Steps

1. Click the menu *Viewers* > *Add Window*. The result is as follows.

> **Shortcut Key**：Ctrl+F3

Close Active Viewer

Brief: Close the active window in the current project.

Steps

1. Click the menu *Viewers* > *Close*. The active window will be closed.

> **Shortcut Key**：Ctrl+F4

Close All

Brief: Close all windows in the current project.

Steps

1. Click the menu *Viewers* > *Close All*. All windows are closed.

Viewer Layout

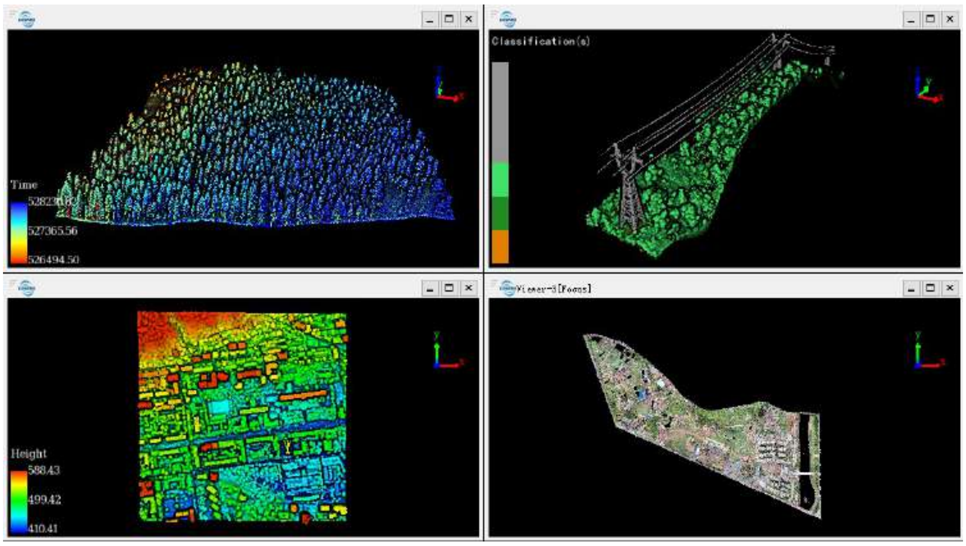
- Tile Windows
- Tab Viewers

Tile Windows

Brief: Rearrange all the viewers in tiled fashion.

Steps

1. Click the menu *Viewers* > *Viewers Layout*> *Tile Windows*. The result is as follows.



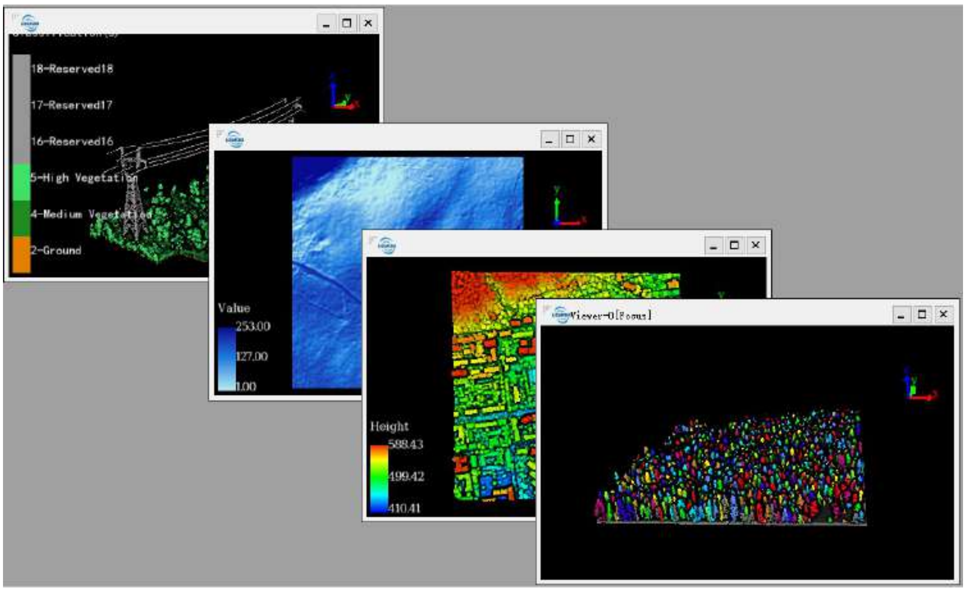
> Note: This function needs at least one window in the current project.

Tab Viewers

Brief: Rearrange all the viewers in tab fashion.

Steps

1. Click the menu *Viewers* > *Viewers Layout*> *Tab Viewers*. The result is as follows.



> Note: This function needs at least one window in the current project.

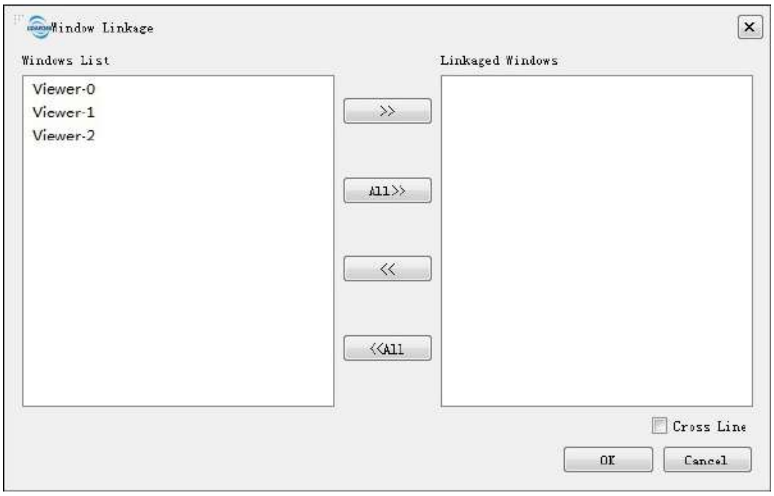
Operations

Window Linkage

 **Brief**: Enable windows linkage of multiple viewers.

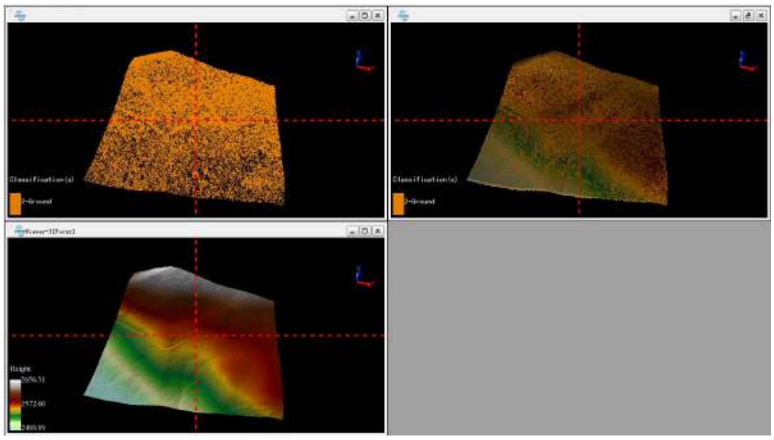
Steps

1. Click this button to open the dialog shown below:



2. The windows list contains all windows in LiDAR360, double-click a window that needed to linkage, or select the window then click button >> to add the window. Also, user can click button All>> to add all windows to linkage windows list. The linkage windows list all windows under linkage, double-click a window to remove, or select the windows and then click button << to remove. Also, user can click button <<All to remove all linkage windows.

3. If "Cross Line" is checked, cross line will be displayed in the linkage windows. The following figure shows if the cross line is checked:



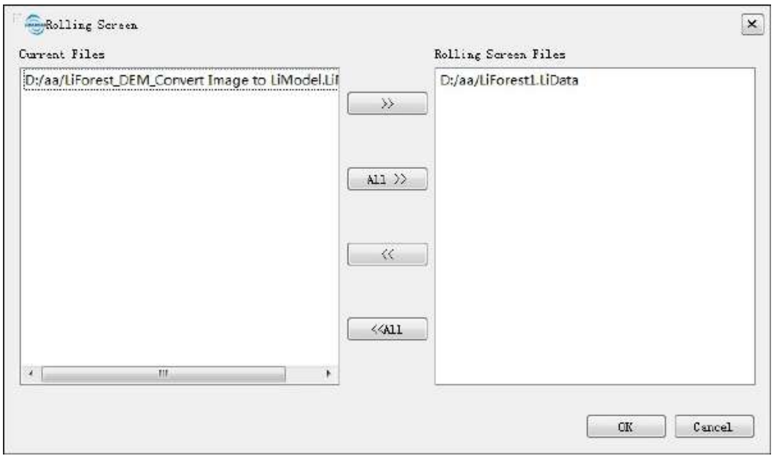
> Note: If the current window contains raster data, it will be displayed in 2D. If the linkage windows contain 3D window and 2D window, the windows will be displayed in 2D mode.

Rolling Screen

 **Brief**: Enable rolling screen for current viewer.

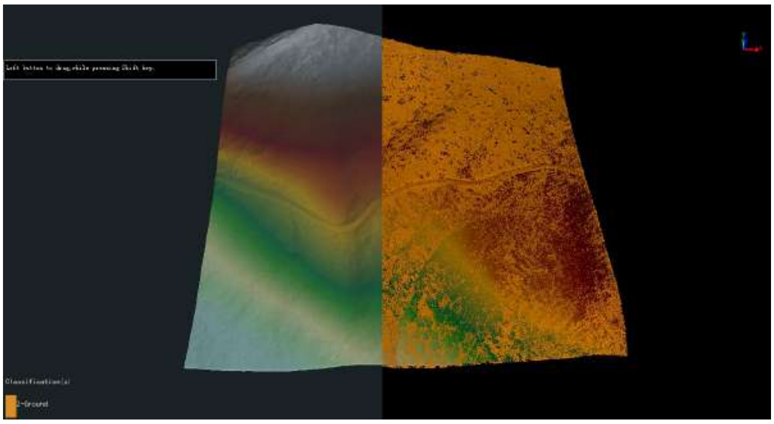
Steps

1. Click this button to open the dialog shown below:



2. Current files list contains files in current viewer. Double-click a file that needs to be added to rolling screen, or select files and then click >>, or click button All>> to add all files to rolling screen files. The rolling screen files list contains files in rolling screen, double-click a file to remove, or select files and then click button << to remove, or click button <<All to remove all files.

3. Click "OK". In the viewer, press the "Shift" button on the keyboard then drag. The window is shown below:



Settings

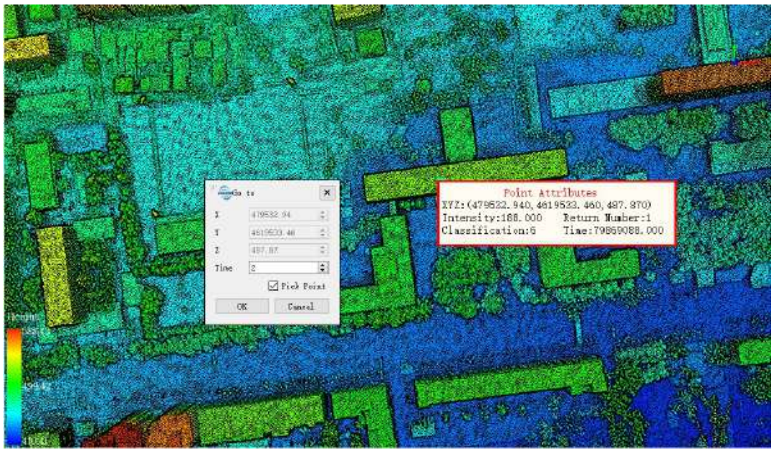
- **Prerequisite**: An active viewer.

> Note: This function cannot be apply to the profile window. Click this button again to exit.

Go To

Brief: This tool is applicable to point cloud data, raster data and model data. It's used to focus to the selected point.

Steps

1. Click the window that contains data with the left mouse button and set it as the active window.
2. Click the button on the toolbar and the mouse pointer turns to . Select a particular point in the scene. The attributes of this point will be shown in a label. At the same time, the "Go To" dialog pops up, as shown below.


3. If "Pick Point" is checked, the focus point is selected by mouse click. Otherwise, the coordinates of the focus point are selected by manual input.
4. Set the time to go to the selected point.
5. Click the "OK" button. The label will disappear, and a red dot will start to flash.
6. When it is focused to the selected point, the red dot disappear, as shown below.


Settings

- **X**: X component of the coordinate.
- **Y**: Y component of the coordinate.
- **Z**: Z component of the coordinate.

- **Time**: Time to go to the selected point.
- **Pick Point (Optional)**: If "Pick Point" is checked, the focus point is selected by mouse clicking.

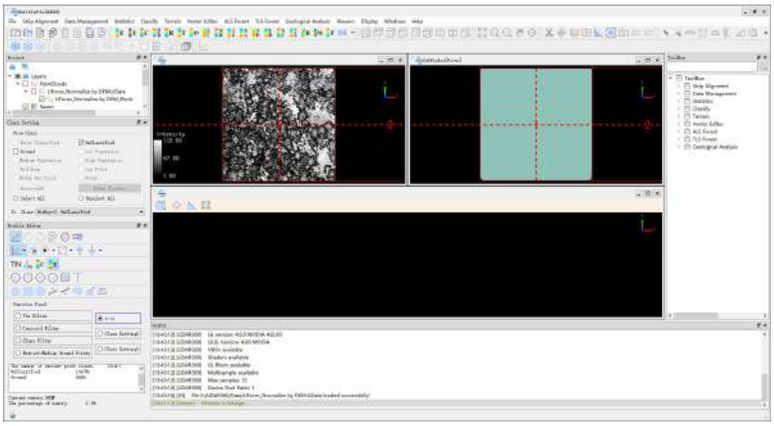
> Note: When there are multiple windows, this tool only works on the active window.

Draw Profile

Icon	Description	Hot Key
	Start Profile Editing Tool	Alt+S
	Move Profile	
	Rotate Profile Area	X or ← →
	Expant Profile Area	
	Move up Profile	↑
	Move down Profile	↓

Profile Tool

Description: Used to create a profile view of a certain area when select an area in the main window or TIN window.



Buffer

Description: Click the triangle icon right to the profile icon, and then click the buffer settings button, the buffer settings window will popup. This function can help users to fix the size of the buffer area.

Move Profile

Description: When the move profile tool is activated, move the mouse to the profile area and drag it. The profile area can be moved with the mouse. If the mouse drag outside the profile area drag it. The profile area can be moved with the mouse. If the mouse drag outside the profile area, it will not be moved with the mouse.

Rotate

Description: Support to rotate the profile area based on the angle that users enter in the rotate settings window. Click the triangle next to the icon and click the rotate settings button, the rotate settings window will popup.

Expand

Description: Used to expand the width of profile area by the entered number. If the entered number is negative, the profile area will be reduced.

Move Up the Profile

Description: Used to move up the current profile to create a new profile with the same size and right above the current profile.

Move Down the Profile

Description: Used to move down the current profile to create a new profile with the same size and right below the current profile. Click the triangle next to the icon and click the step length settings button, the step length settings window will popup. If the checkbox in front of the step length tag is checked, the step length will be fixed as 1.5 times step length and cannot be changed. If the checkbox is unchecked, users can change the step length based on their needs.

Linked Window

Function Description: Allows users to select a source point cloud window for cross-sectioning. During the cross-section viewing process, users can switch the source point cloud window as needed.

Profile Viewer Tool

By default, profile view shows the front view of selected rectangle area. Users can change the view mode in the drop-down menu .

Icon	Description	Hot Key
	Front View	
	Rear View	
	Left View	
	Right View	
	Rotate Scene	R
	Measuring Tools	
	Grid	
	Exaggerate Z	
	Export	

Strip Alignment

Airborne LiDAR measurement system is influenced by various error sources, of which the systematic errors (the largest error source) cause the systematic deviation of laser footprint coordinates. Mounting of LiDAR measurement system requires the axis of the scanning reference coordinate system and inertial platform reference coordinate system to be parallel. However, while mounting of system, it is not guaranteed that they are parallel, resulting in the so-called systematic boresight error. In the Strip Alignment module, airborne LiDAR point clouds of overlapping strips can be aligned through boresight calibration.

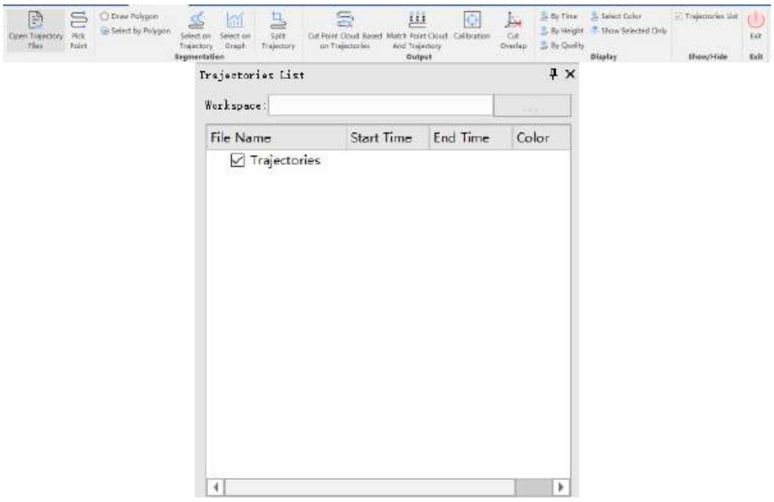
Boresight

Functional Overview

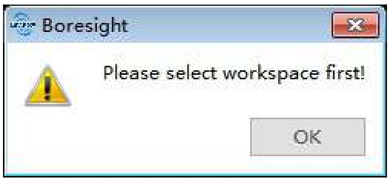
Brief: In the Boresight module, airborne LiDAR point clouds of overlapping strips can be aligned through boresight calibration. This module includes the following functions: loading/deleting/splitting trajectories, cutting point cloud according to trajectories, matching trajectory and point cloud, transforming point cloud based on boresight calibration parameters, eliminating point cloud redundancy (cut overlap), etc.

Usage

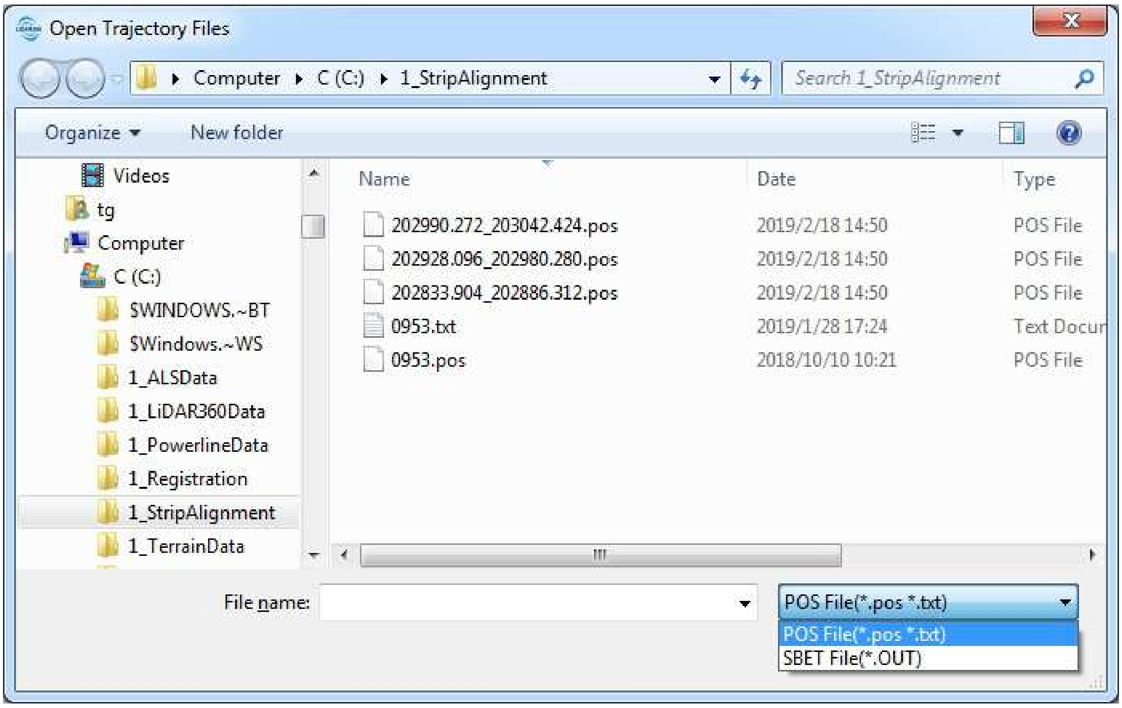
1. Open boresight module. Click *ToolBox* > *Strip Alignment* > *Boresight* or *Preprocessing* > *Boresight*, and the corresponding toolbox and Trajectories List Widget is shown as follows.



2. Open Trajectory Files: After clicking this button, an interface pops up to ask you to set the workspace.

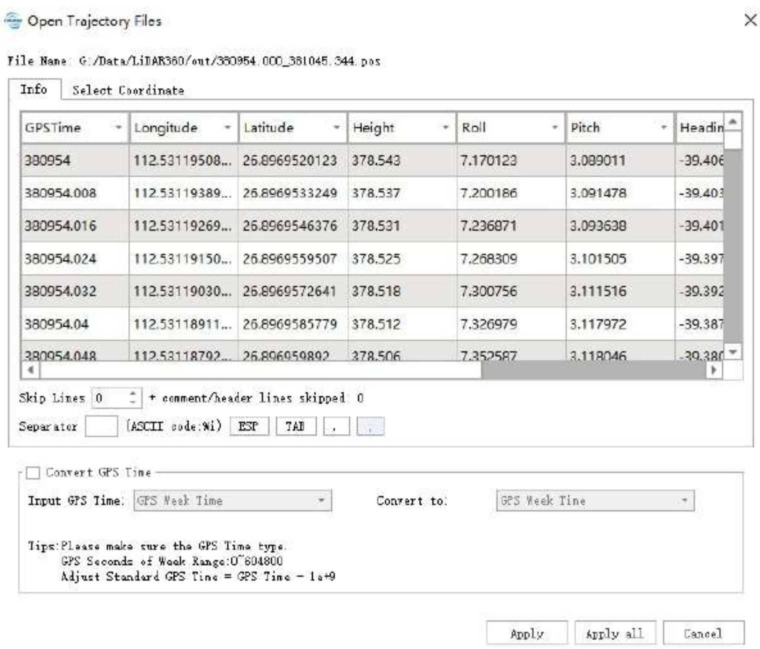


Set the workspace and a dialog for adding trajectories will be shown as follows.

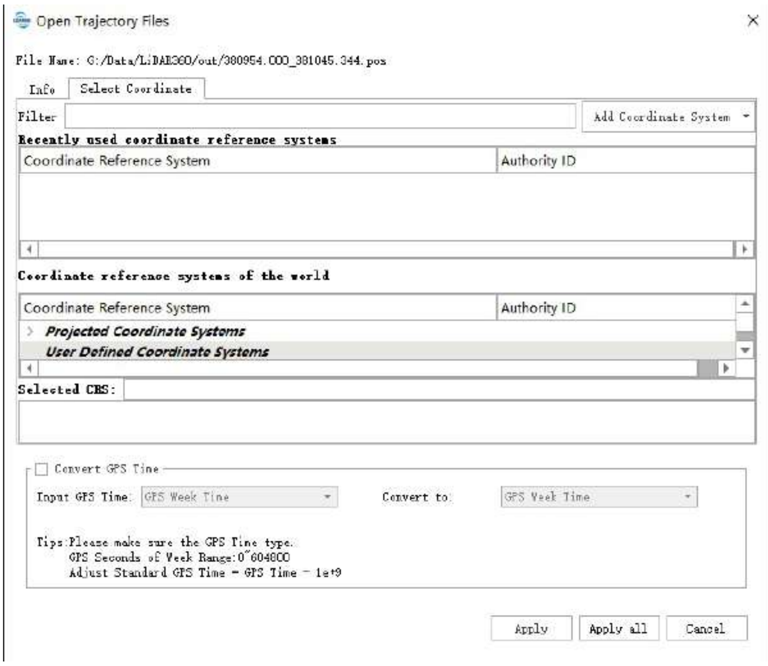


LiDAR360 supports two formats of trajectory files: POS(*.pos) (text format file) and SBET(*.out)] (binary format file).

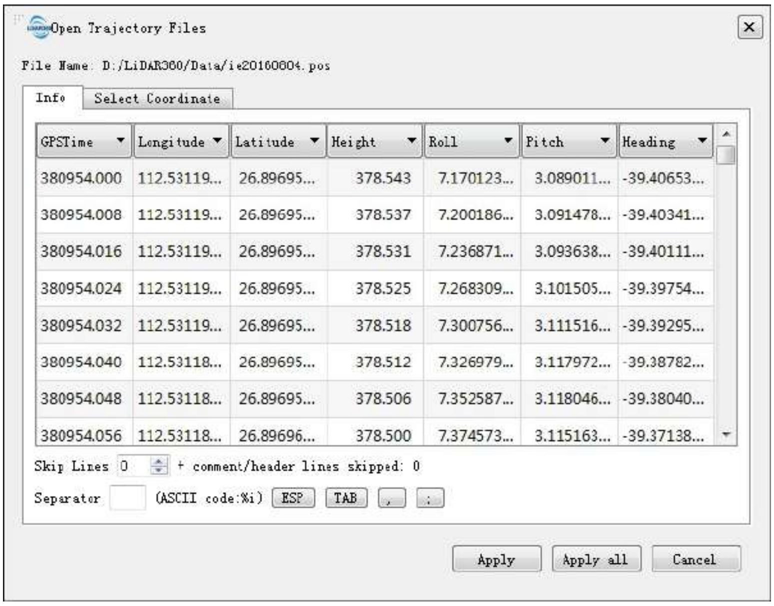
POS File Example 1: If the POS file does not contain GridX and GridY information, the user has to specify following headers (GPS time, longitude, latitude, height, roll angle, pitch angle and yaw angle) for data columns. An example is shown as follows.



It is required to select the correct projection coordinate system as the following picture.



POS File Example 2: If the POS file contains GridX and GridY information, the user has to specify following headers (GPS time, longitude, latitude, height, roll angle, pitch angle, yaw angle, GridX and GridY) for data columns. An example is shown as follows.

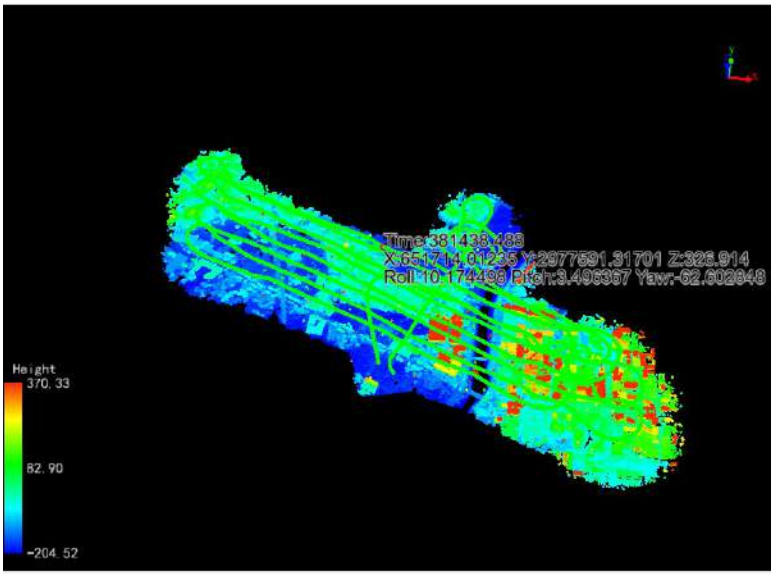


If the information of GridX or GridY is not correct, the POS File Example 1 can be used.

For both of the two examples, click "Apply" to make the current settings apply on current trajectory file, and all the trajectory files will be affected by the current settings if you click "Apply all".

The information of added trajectories will be shown in Trajectories List Widget. (Optional) Click ... to set the workspace.

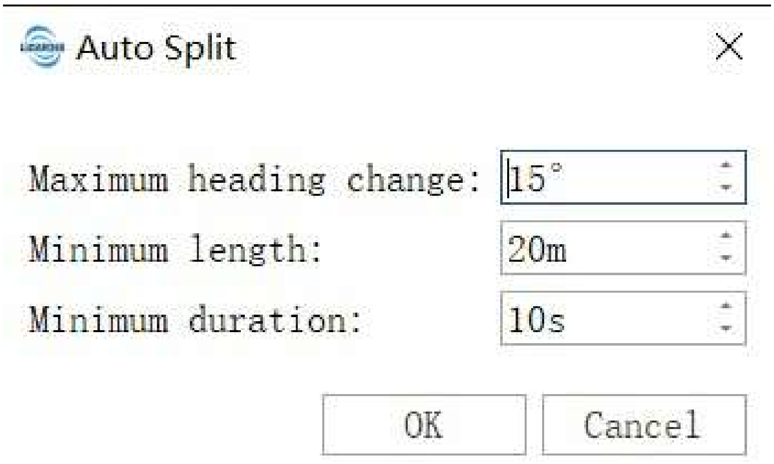
3. Pick Point: Left click a certain point in the trajectory and the information of this point will be displayed.



4. Segment Select

4.1 Auto Split Trajectory

- Click the *Automatic Trajectory Line Splitting* button to open the automatic splitting parameter setting dialog box.



- Set the parameters and click *OK* to start automatic splitting.
- Parameter Settings
- **Maximum Heading Change**: The change in heading angle of the automatically cut trajectory segment does not exceed the set value.
- **Minimum Length**: The minimum length of the automatically cut trajectory segment is not less than the set value.
- **Minimum Duration**: The duration of the automatically cut trajectory segment is not less than the set value.

4.2 Select by Polygon

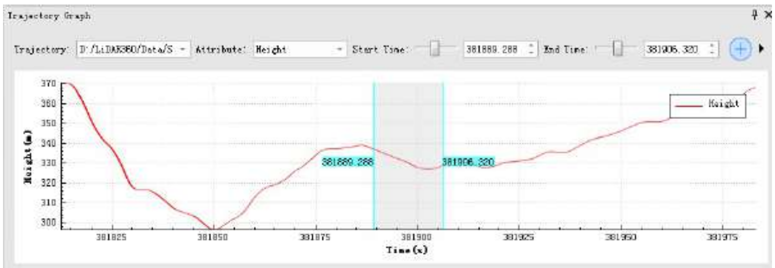
- Click Draw Polygon button.
- Left click to make an user-defined polygon in the view.
- Click Select by Polygon button, to select segment in the area of user-defined polygon.

4.3 Select by Bursh

- Click Select on Trajectory button.
- Left click to select the starting point in the trajectory.
- Along the trajectory, select the ending point, the trajectory between the starting point and the ending point will be selected.

4.4 Select by Trajectory Graph

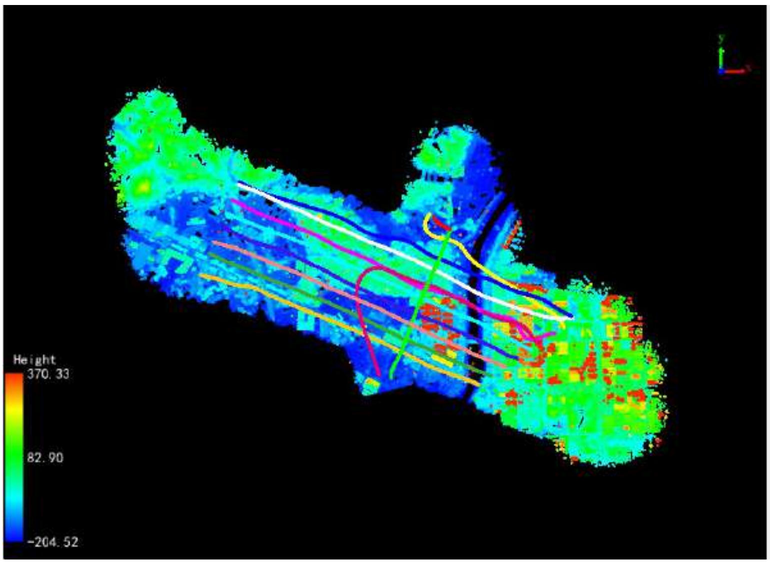
- Click Select on Graph button, the trajectory graph will pop up.



- (Optional) Select an trajectory in trajectory combox, the selcted segment of the trajectory and the trajectory will be shown in the graph.
- (Optional) Select an attribute. Users can select different display mode, including display by height, display by quality factory, display by roll, display by pitch, display by heading and display by velocity.
- Select both Start Time and End Time via scrollbars.
- Click to add a new segment.
- (Optional) Change display range. All the trajectory information at any time will be displayed in trajectory graph as a default. Scroll up and down of mouse wheel to zoom in and out the graph, click to restore the default display.

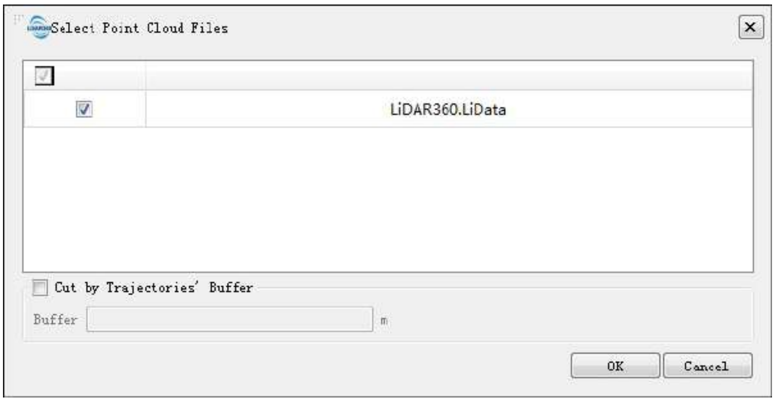
All the selected segment will be shown in Trajectories List Widget.

- Split Trajectories: Click this button to split the source trajectories by selected segments. New generated trajectories will be stored as POS file format in the workspace and named by the GPS starting and ending times of the trajectory. An example of new file name is "GPSStartTime_GPSEndTime.POS". The effect of cutting is shown as follows.



- Cut Point Cloud Based on Trajectories.

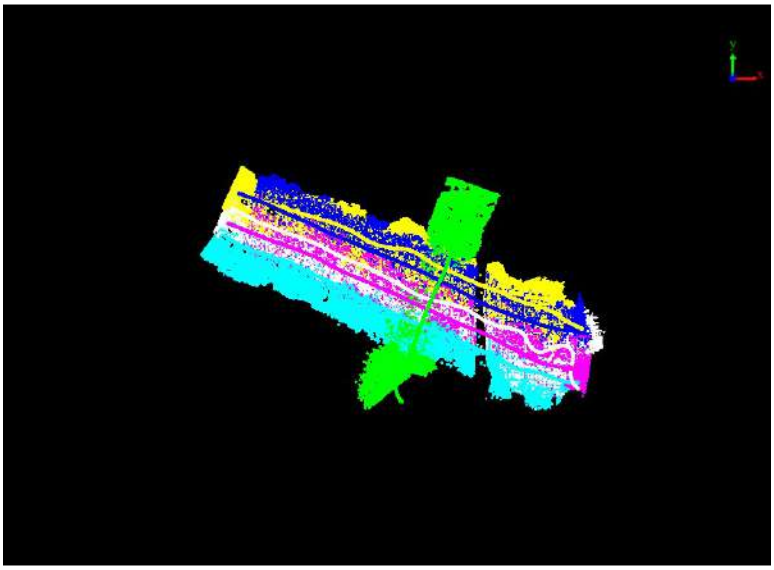
- 6.1 Click the button to pop up a dialog.



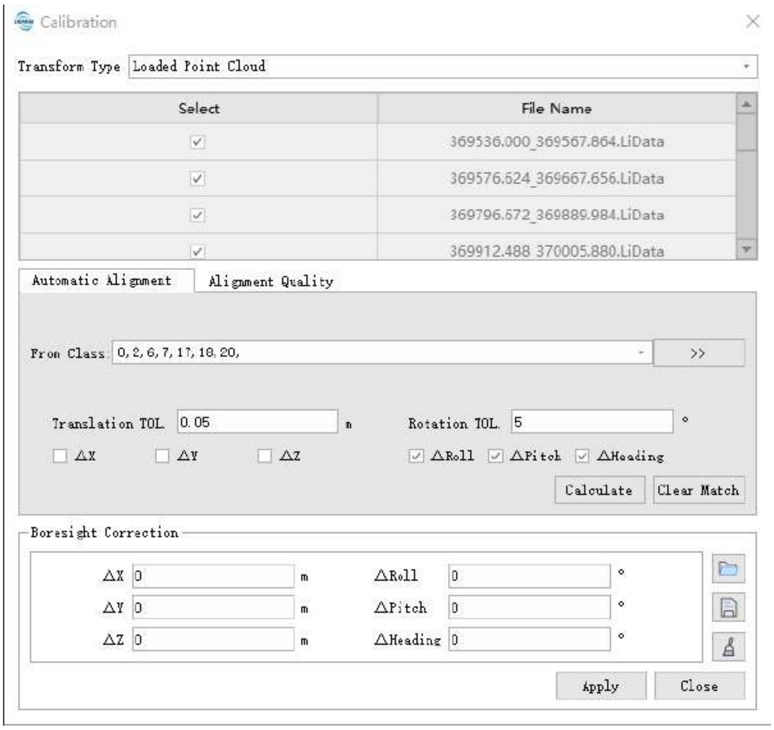
Select the point cloud data files to be cut. Check the “Cut by Trajectories' Buffer”and set the Buffer parameter as desired.

- 6.2 Click "OK". LiDAR360 will cut the point cloud by the trajectories and show the process speed accordingly. The cutting results will be saved and the name of the point cloud is identical to the corresponding POS file. A dialog pops up to ask the user whether to load the results or not.

- Match Point Cloud and Trajectory: Click this button, the point cloud data and the corresponding trajectory will be displayed in the same color.

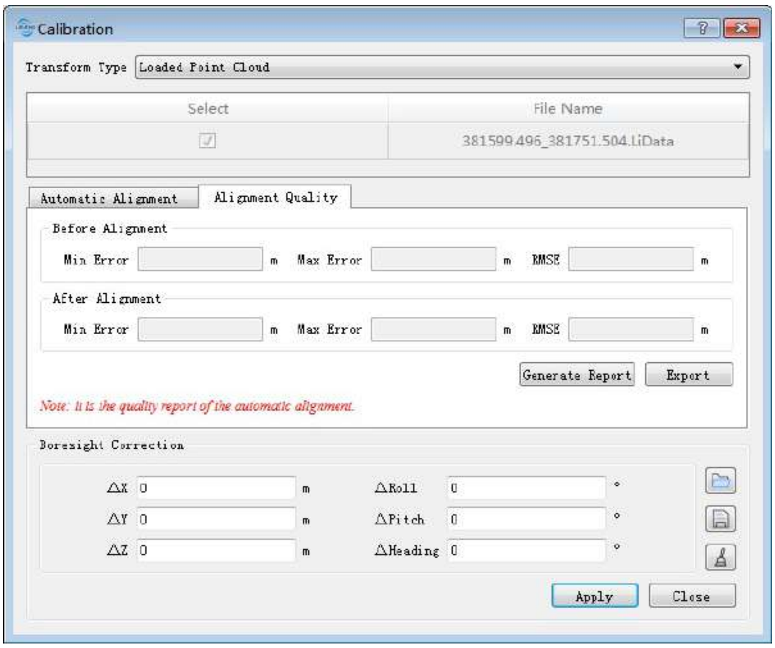


- Calibration: Click this button to pop up a dalog.



- 8.1 This function is applied to correct the Boresight Error, which can be calculated automatically (Automatic Alignment) or input manually (Manual Alignment).

  Automatic Alignment:
  Select the boresight error you want to correct, and set the Translation TOL and Rotation TOL. Click "Calculate" to generate the boresight error automatically and the result will be shown in the "Boresight Correction" part of the interface. Click "Clear Matching" to delete the matching information. After modifying relevant parameters, users can calculate the alignment parameters again. After the calculation of automatic alignment, LiDAR360 will figure out the Alignment Quality. The Alignment Quality will be shown in the "Alignment Quality" tab page. Click "Export" button to save the alignment quality information, and Click "Generate Report" button to generate the automatic alignment quality report. (Theory of correction based on automatic adjustment.)

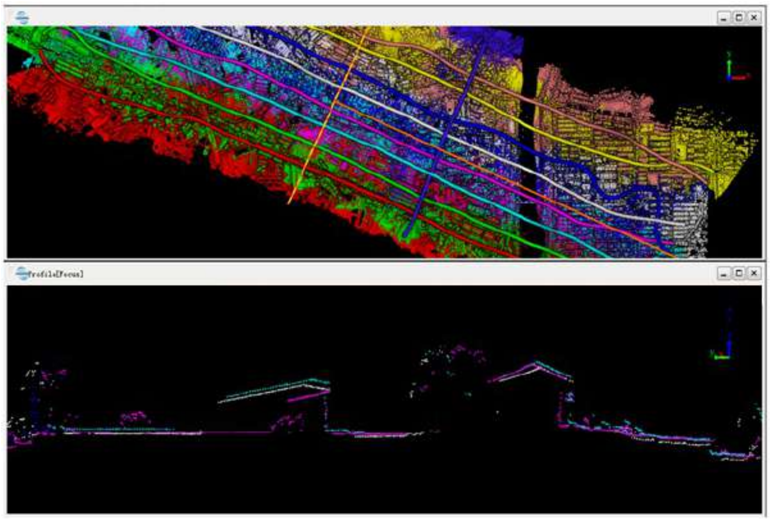


Manual Alignment:

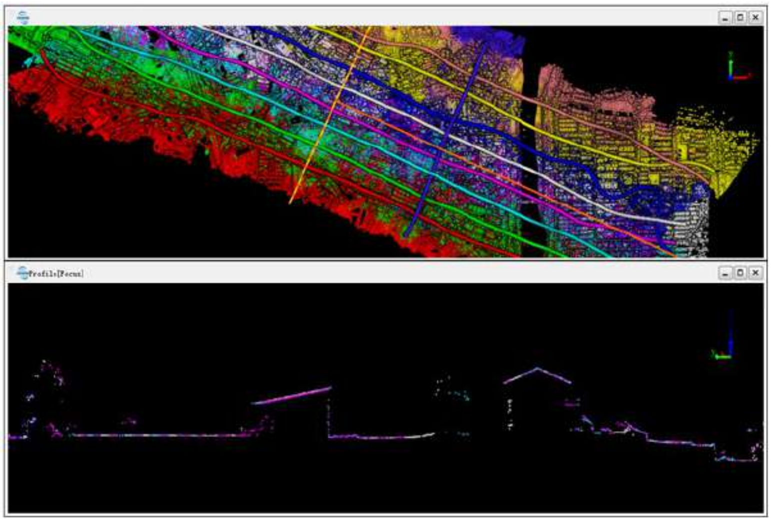
Adjust the boresight error manually. (Theory of correction based on manual measurement.)

Click "Apply" and the transformation of error correction will be applied on the selected Transform Type.

- 8.2 For both automatic and manual method, correction result can be checked in the profile window.



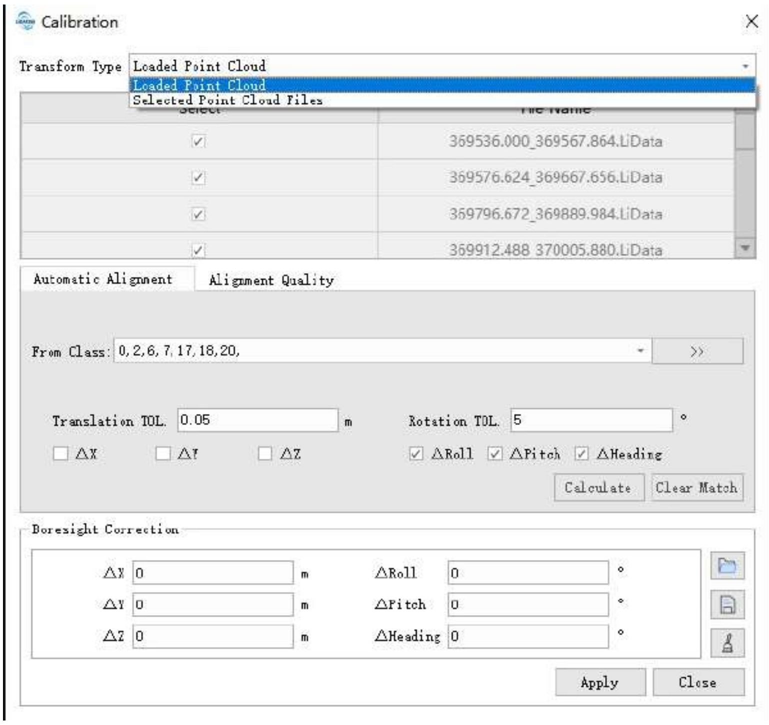
Before transformation



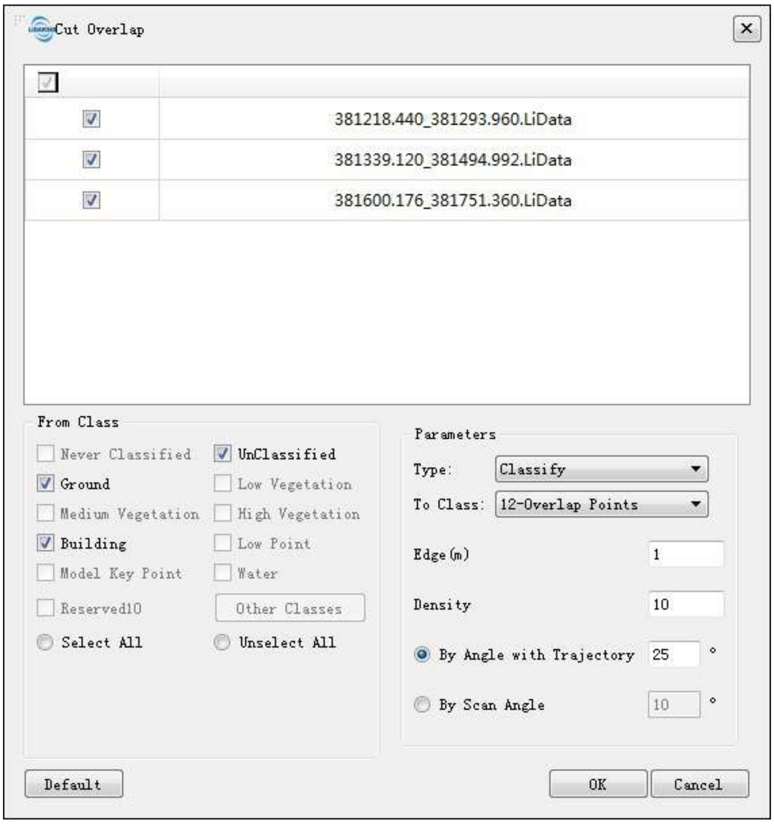
After transformation

- 8.3 Buttons on the right of the "Boresight correction" provide the following operations: "loading from file", "saving to a file" and "cleaning".

- 8.4 If the alignment effect is satisfied, you can switch Transform Type from the "Loaded Point Cloud" to "Selected Point Cloud Files". Click "Apply" to finish the transformation.



- Cut Overlap: Click this button to set the Cut Overlap Parameters. Click "OK" button, then the redundant points between the overlapped trajectories will be classified or deleted.



- Show Single Trajectory
    - Select a trajectory in the Trajectories List Widget, right click to pop up the Context Menu.
    - Click the Show Single Trajectory button, the color of the selected trajectory will be changed gradually, and then the trajecotry will twinkle several times.
- Delete Trajectories or Segments.

- Select the trajectories or segments in the Trajectories List Widget, right click to pop up the Context Menu.
  - Click the Delete button and the selected trajectories and segments will be removed.
- Show/Hide Trajectory or Segment Check/Uncheck the checkbox of the trajectory or segment in the Trajectories List Widget will Show/Hide the trajctory or segment.

- Trajectory Display the software supports muti-display modes: 13.1 Display by Time(Default)

  - Click button, the trajectory will be displayed in different color according to acquisition time, users can select the color bar in a pop-up window.

  13.2 Display by Height

  - Click button, the trajectory will be displayed in different color according to height information, users can select the color bar in a pop-up window.

  13.3 Display by Quality

  - Click button, the trajectory will be displayed in different color according to quality information, users can select the color bar in a pop-up window.

  13.4 Display by Specific Color

  - Click button, the trajectory will be displayed in specific color according to user-defined color, users can select the color in a pop-up window.

  13.5 Show selected segments

  - Check button, the unsegmented area will be hide, and only selected segment part will display.

Parameters Settings

- **Parameters of Cutting point cloud based on Trajectories**
  - **Buffer**: The remaining distance in both sides of trajectory while cutting.
- **Parameters of Transform**
  - **Boresight Error**
    - **ΔX/ΔY/ΔZ**: Correction parameters of boresight offset error.
    - **ΔRoll/ΔPitch/ΔHeading**: Correction parameters of boresight pose error.
  - **Automatic Alignment**: It matches corresponding points between strips automatically and calculates optimal correction value of boresight error via adjustment.
    - **Translation TOL**: The maximum translation tolerance of ΔX, ΔY, ΔZ. The default value is 0.05 m.
    - **Rotation TOL**: The maximum rotation tolerance of ΔRoll, ΔPitch, ΔHeading. The default value is 5°.
    - **Options**: Users can decide whether the 3 translations and 3 rotations are involved in adjustment to calculate correction values. For most aerial data, it is not recommended to correct ΔZ. In default state, all the rotations are selected while all the translations are unselected.
  - **Alignment Quality**: After automatic alignment calculation, LiDAR360 will figure out the alignment quality and show the result in the alignment quality result tab.
    - **Min Error**: Minimum error of matched corresponding points.
    - **Max Error**: Maximum error of matched corresponding points.
    - **RMSE**: RSE error of matched corresponding points.

- **Transform Type**:
    - **Loaded Point Cloud**: The loaded point cloud which is matched with trajectory in LiDAR360. When the transformation is applied, the corresponding point cloud is modified and the transformation effect can be shown in real-time. When boresight module exits, relevant transformation is eliminated without saving.
    - **Selected Point Cloud Files**: In this type, the file list area on the interface will be activated, users can choose which point cloud file(s) will participate in the transformation. Then the results will be saved to the file. It doesn't loss the information even though the boresight module exits.
- **Parameters of Cut Overlap:**
  - **Type**:
    - **Classify**: Classify points into target category and save the result in the original point cloud file.
    - **Delete**: Classify points into target category and then delete them from initial point cloud data (Note: If the input point cloud contains points of target category, they will also be removed).
  - **From Class**: Point cloud target category.
  - **Edge**: Grid unit size (length) of overlapping area. This parameter is used to rasterize the point cloud data of the overlapped area.
  - **Density**: Smallest point density of grid unit. If the point density is smaller than the defined threshold, these points will not be classified to redundant points.
  - **By Angle with Trajectory**: Angle between point cloud and trajectory. If this angle is larger than the threshold, the point cloud will be classified to redundant points.
  - **By Scan Angle**: If scanning angle is larger than the threshold, point cloud will be classified to redundant points. (Note: this function can only be used when scanning angle is recorded by point cloud.)

> Note:
> 1. The matching principle of a point cloud file and a trajectory is that: Each GPS time of all the points in the point cloud is inside the trajectory's GPS time span.
> 2. In most cases, the effect from X,Y or Z is tiny, while the ground control points are needed for higher precision of Z value. Therefore, only Roll, Pitch and Heading are selected to take apart in the adjusting by default.
> 3. The result may not satisfy if accuracy of the POS data file is not enough, and users can make an manual alignment based on the result.
> 4. For Overlapped function, at least two pairs of matched point cloud and trajectories are needed.

Boresight Error Calculation

The boresight error between **laser scanner coordinate system** and **GNSS/INS coordinate system** is the largest systematic error source of airborne LiDAR. The laser footprint error caused by the boresight error also depends on flight height and scanning angle.

LiDAR360 provides two approaches to eliminate the boresight error:

1. Correction based on manual measurement
2. Correction based on automatic adjustment

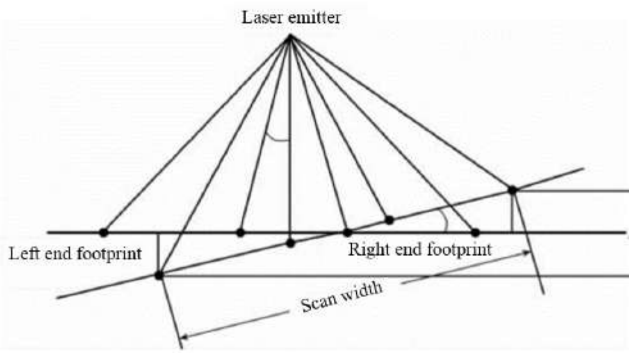
The principles of them are described as follows.

Manual Correction

Firstly, datasets for calibration need to be acquired during perpendicular and round flights (e.g., 4 flights: from east to west, from west to east, from north to south, from south to north). After data acquisition the boresight angle errors can be retrieved using the *Stepwise Geometric Method*(Zhang et al., 2010), which is based on measurements of overlapping laser footprints of regular objects (e.g., playground, building). Compared to angle errors, the translation errors are of less importance and hence omitted in the manual correction.

ΔRoll

Error of Roll angle leads to an oblique result of horizontal ground, which is shown in the following figure. The planar position of scanned object is shifted along scanline (perpendicular to flight direction).



Please measure the following parameters in the data acquired during round flights of same height.

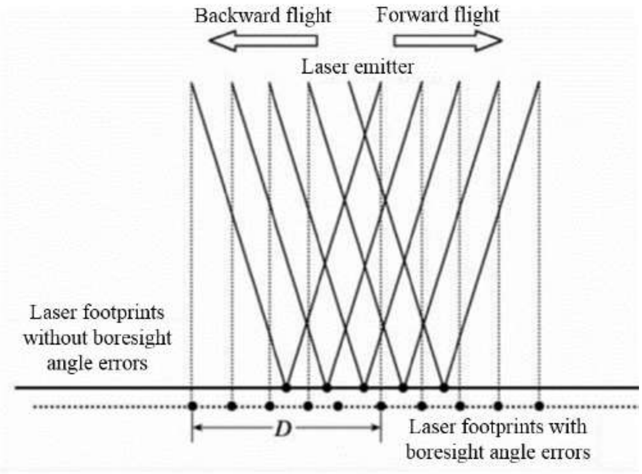
- Start a Profile Window perpendicular to the flight direction. Then measure the height difference Δh of quasi-homologous objects in two strips.
- Measure the horizontal distance r between the object and centerline of two strips in 2D-View Window.

The formula to calculate the Roll angle error is as follows:

$$\Delta Roll \approx arctan\left(\frac{\Delta h}{2r}\right)$$

ΔPitch

The position of scanned object is shifted along scanline due to Pitch angle error. The following figure shows the affected laser footprints.



Please measure the following parameters in the data acquired during round flights.

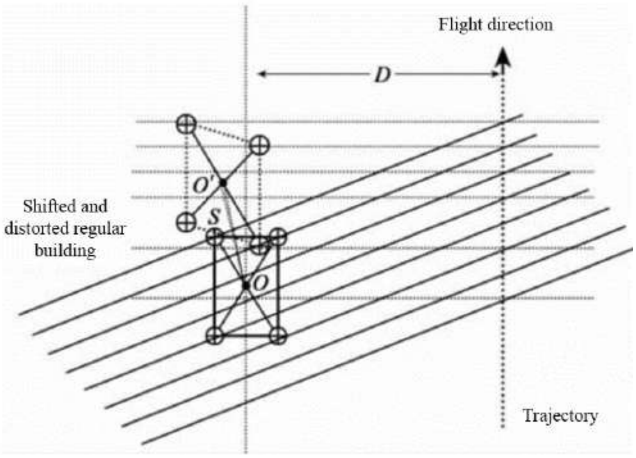
- Start a Profile Window along the flight direction. Then measure the distance D between centers of homologous objects along the flight direction.
- Estimate the average flight height H based on trajectory and object heights. The heights of round flights should be roughly equal.

The formula to calculate the Pitch angle error is as follows:

$$\Delta Pitch \approx arctan\left(\frac{D}{2H}\right)$$

ΔHeading

Error of Heading angle leads to object shifting and distortion. The following figure shows the erroneous result.



Please display data acquired during round flights in 2D-View Window.

- Measure the average distance S between centers of homologous objects in two strips.
- Measure the distance D between two strips.

The formula to calculate the Heading angle error is as follows:

$$\Delta Heading \approx arctan\left(\frac{S}{D}\right)$$

Automatic Correction

While the manual correction relies on well-trained operators, the automatic correction can reduce manual work significantly. Based on datasets with regular objects, the automatic process can replace manual correction and achieve equivalent or even better accuracy.

The algorithm is capable of correcting not only the angle errors, but also the translation errors. Users have the flexibility to check/uncheck every correction parameter. By default, the software only calculates the angle errors, because they have the largest weights. The principle of the algorithm is as follows:

1. Extract feature points and their normals in overlapping strips. Please refer to (Glira et al., 2015).
2. Match the feature points to get corresponding pairs.
3. Construct the correction model and calculate the distances of paired points along their normal.
4. Minimize the corresponding distances using least-square method, and get the correction parameters.

```
@inproceedings{
    author={Zhang Xiaohong, Forsberg Rene},
    title={Retrieval of Airborne Lidar Misalignments Based on the Stepwise Geometric Method},
    booktitle={Survey Review 42(316):176-192 April 2010},
    year={2010},
}
```

```
@inproceedings{
    author={Philipp Glira, Norbert Pfeifer, Christan Briese and Camillo Ressl},
    title={A Correspondence Framework for ALS Strip Adjustments based on Variants of the ICP Algorithm},
    booktitle={PFG Photogrammetrie, Fernerkundung, Geoinformation Jahrgang 2015 Heft 4},
    year={2015},
}
```

Trajectory Adjustment

After completing Boresight calibration, if there are still layering phenomena in the point cloud, it may be due to significant trajectory errors. The strip adjustment function can post-process the trajectory and point cloud to reduce errors. Strip adjustment also supports mixed adjustment of point cloud and imagery. When the point cloud and imagery are synchronously collected data sharing the same POS system, it improves layering between point clouds and between point clouds and imagery by adjusting trajectory, exterior orientation elements of imagery, and camera installation angle errors. The imagery project file is optional; when added, it performs mixed adjustment of point cloud and imagery, synchronously adjusting trajectory, camera installation angle, and exterior orientation. Without an imagery project file, only trajectory adjustment is performed.

The strip adjustment algorithm references the works of Glira et al., 2016; Glira et al., 2019. The basic algorithm process is as follows:

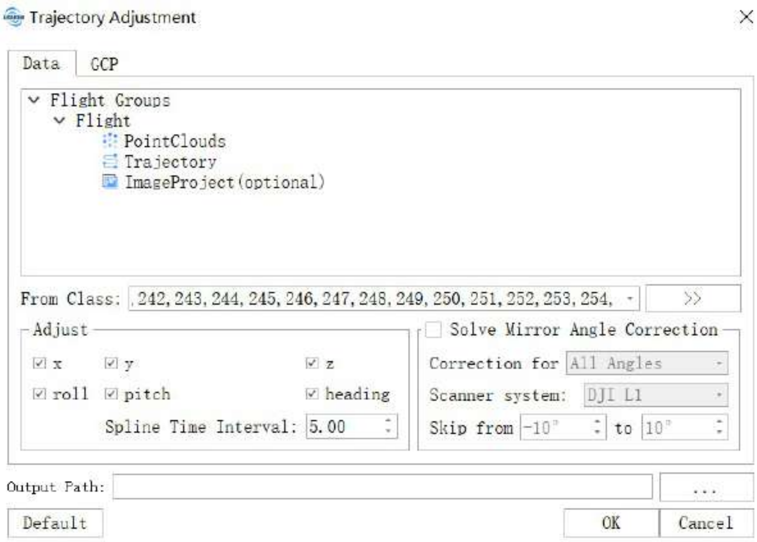
1. Extract **point cloud surface features** and normal vectors in overlapping areas of adjacent strips.
2. Extract **point cloud surface features** and normal vectors in overlapping areas of sparse imagery points and point clouds.
3. Match extracted feature surfaces in adjacent strips to obtain **point cloud surface feature matching pairs**.
4. Match sparse imagery points and feature surfaces to obtain **sparse imagery points and point cloud surface feature matching pairs**.
5. Establish a trajectory error correction model to calculate the **distance along the normal vector** of related point pairs.
6. Use the least squares method to **minimize** the distance between matching point pairs, obtaining the optimal solution.

> Note:
>
> 1. Sparse imagery point clouds and laser point cloud feature matching are susceptible to noise in sparse imagery point clouds, causing mismatches. For sparse imagery point clouds with significant noise, noise filtering such as filtering power line points, sparse vegetation points, etc., is recommended.
> 2. Sparse imagery points and the nearest laser point cloud surface features form a matching pair if certain matching conditions are met. Feature matching relies on good planar targets, including building tops, walls, and ground surfaces. In vegetation areas, it may not be possible to obtain sufficient matching features, leading to adjustment failure.

@inproceedings{ author={Philipp Glira, Norbert Pfeifer, and Gottfried Mandlburger}, title={Rigorous Strip Adjustment of UAV-based Laserscanning Data Including Time-Dependent Correction of Trajectory Errors}, booktitle={Photogrammetric Engineering & Remote Sensing}, year={2016}, }

Glira P, Pfeifer N, Mandlburger G. HYBRID ORIENTATION OF AIRBORNE LIDAR POINT CLOUDS AND AERIAL IMAGES[J]. 2019. DOI: 10.5194/isprs-annals-IV-2-W5-567-2019.

Click the *Preprocessing* > *Trajectory Adjustment* button to open the strip adjustment interface.

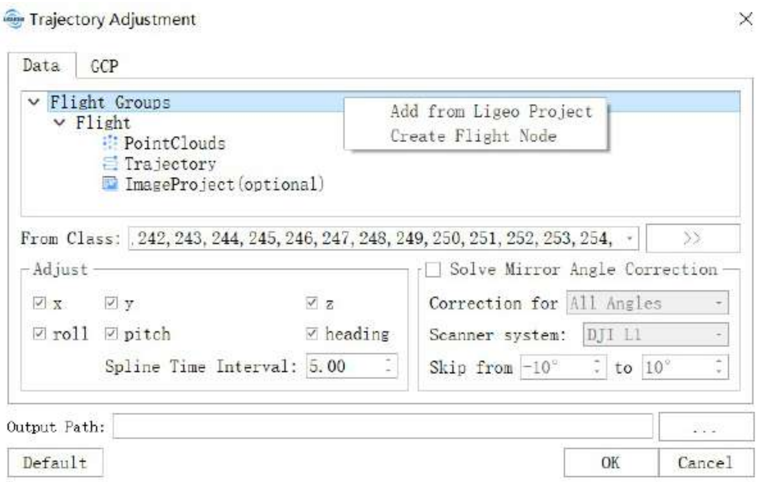


Main Interface

The interface has two pages: Point Clouds for inputting point clouds, trajectories, and optional imagery projects, and the GCP page for importing control point coordinate files.

Add Data

Two methods to add data:

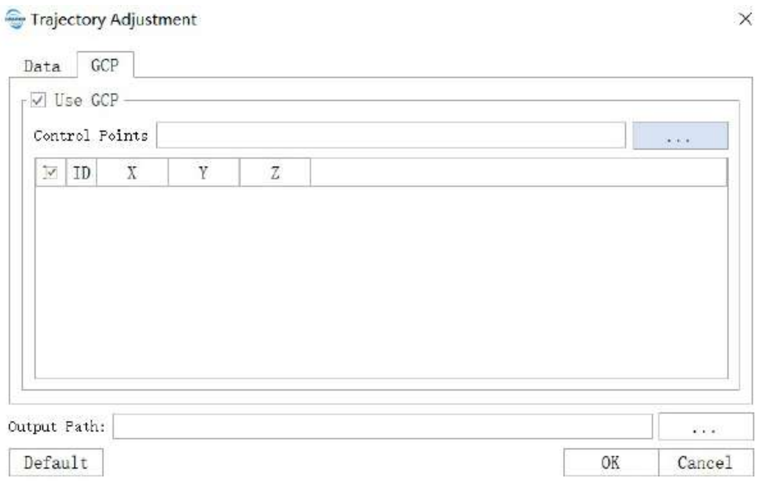


Import Data from Ligeo Project

- **Add from Ligeo Project**: Right-click **Flight Groups** to import the Ligeo project file, which automatically adds point cloud and trajectory files. Since the Ligeo project file does not record the imagery project file, it must be added under the **Image Project** node if needed.
- **Add Image Groups**: Create a mission directory tree under the **Flight Groups** directory tree, which includes nodes for point cloud, trajectory, and imagery project. Relevant data can be imported through the right-click menu of each node.

  > Note: If multiple point cloud or imagery project files are imported, the file names should be unique.

Use GCP



Open Control Point

Ground control points are optional and can be used to correct elevation direction deviations. The plane features of the point cloud in the control point area are extracted, with the goal of minimizing the distance from the control point to the point cloud plane feature. The control point file must contain at least the fields: point name, X, Y, Z, and the coordinate system must match the point cloud's coordinate system.

Parameters

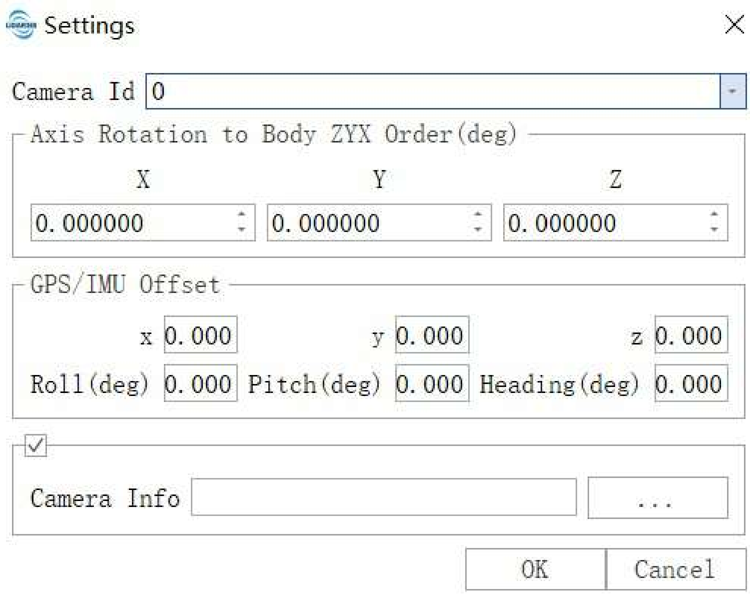
- **From Class**: The category of point cloud classification. The point cloud belonging to the selected category will be used to extract plane features.
- **Adjust**
  - **x/y/z**: Position components of the trajectory correction.
  - **roll/pitch/heading**: Angle components of the trajectory correction (roll, pitch, yaw).
  - **Spline Time Interval (s)**: The trajectory fluctuates based on the set time interval; the shorter the interval, the higher the fluctuation frequency. If the trajectory error fluctuates significantly in certain areas, a shorter interval can be set; otherwise, a longer interval can be used. The default value is effective for most data.
- **Mirror Angle Correction**
  - **Correction for**: Specify the correction range for the laser scanner's scanning mirror rotation angle.
    - **All angles**: Correct all rotation angles of the laser scanner's scanning mirror.
    - **Skip Centeral Part**: Correct rotation angles outside the specified angle range, determined by the skip value.
    - **Skip from**: Set the angle range of the scanner's scanning mirror that will not be corrected, and correct angles outside this range.
- **Output Path**: The storage path for the output files after the function runs. Output files include corrected point clouds, trajectories, imagery projects, and an HTML statistical report.

> Note: The trajectory input for this function should ideally be the complete trajectory file from the data collection. However, if a complete trajectory file is unavailable, multiple POS files from cropped trajectories can be used, but the point cloud and trajectory GPS times must match to avoid layering phenomena in local areas after correction.

Imagery Project Settings

Imagery data and point cloud data are synchronously collected and share the same POS system. Through strip adjustment, the trajectory and camera installation angle, and exterior orientation elements are corrected simultaneously. The imagery project file must record the camera's initial installation angle, installation error, position, and roll, pitch, and heading angles at the time of image exposure. If this information is not recorded in the imagery project, it can be set in the imagery project settings dialog.

Right-click the imagery project LIAEP file to open the imagery project settings dialog:



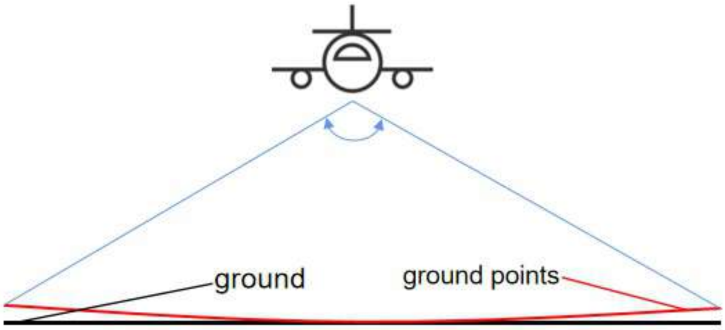
If the camera's initial installation angle, installation error, position, and roll, pitch, and heading angles at the time of image exposure are recorded in the imagery project, the corresponding values will be displayed. If not recorded, the default value is 0, and the values can be modified and updated in the imagery project file.

- **Camera Id**: Camera ID.
- **Axis Rotation to Body ZYX Order (deg)**: The installation angle of the camera relative to the IMU, generally obtained from the manufacturer.
- **GPS/IMU offset**: Initial value of the installation error of the camera relative to the IMU.
- **x, y, z**: The offset of the camera relative to the IMU (m), not adjustable.
- **Roll, Pitch, Heading**: The installation error of the camera relative to the IMU, adjustable.
- **Camera Info**: Import the exposure point file of all cameras from external sources. If there are multiple camera exposure point files, they should be merged into one file for import. The file should contain at least the fields: image name, longitude, latitude, roll, pitch, heading. The program matches based on the image name. If the exposure point file is already recorded in the imagery project, it does not need to be imported.

> Note:
>
> 1. The imagery project should be synchronously collected with the point cloud, sharing the same POS system.
> 2. The coverage range of the imagery project should match that of the point cloud. If the imagery project covers a much larger area than the point cloud, there may not be enough matching pairs, leading to adjustment failure.

Mirror Angle Error Correction

Mirror angle error refers to the scanning angle error of the laser scanner, caused by insufficient precision of scanner components or complex scanning structures. This phenomenon is more common in lightweight, low-cost scanners (such as the Risley Prism-Based Livox Mid-40). This error causes point cloud deformation, with more significant deformation at longer measurement distances, affecting the accuracy of point cloud positions and causing the cross-section point cloud along the scanning direction to curve, presenting a "smiling face" effect (as shown in smiling face). Correcting the mirror angle error can effectively eliminate the smiling face effect and improve the accuracy of the point cloud at the strip edges.



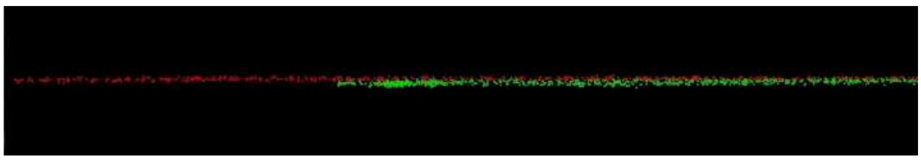
smiling face

Let the coordinates of the laser point in the scanner coordinate system be $X = \begin{bmatrix} x_{s} & y_{s} & z_{s} \end{bmatrix}^T$, the azimuth angle of the outgoing beam be Ω, and the elevation angle be Φ. Then $\Omega = arctan(y_{s} / x_{s})$, and $\Phi = arccos(z_{s} / ||X||_{2})$. Since the variation in the scan angle mainly causes changes in the azimuth angle and the elevation angle is hardly affected, the scan angle error mainly manifests as an azimuth angle error. Since the scan angle error varies with the scan angle, the azimuth angle is divided into N groups at 1° intervals, and the correction for the ith group is Δ_i. Then, the coordinates of the laser point corrected for the azimuth angle in the scanner coordinate system are:

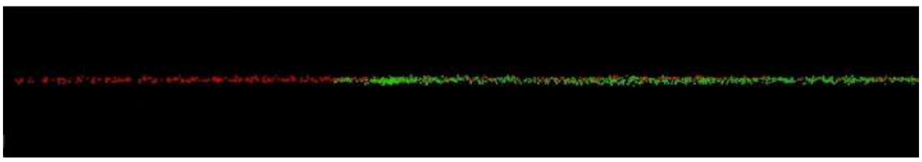
$\hat{X} = ||X||_{2} \begin{bmatrix} sin(\Omega+\Delta_i)cos\Phi & sin(\Omega+\Delta_i)sin\Phi & cos(\Omega+\Delta_i) \end{bmatrix}^T$

Mirror angle error correction is an optional function for strip adjustment. Currently, it supports DJI L1 and X3 series laser scanning systems, and it can correct all angles or angles outside a specified range.

Check *Mirror Angle Correction* and select *DJI L1* or *LiAir series* according to the type of laser scanning system used. If you need to correct all angles, select *All angles*. You can also set the start and end angles to be skipped in the skip angle range. Mirror angle correction is performed on a flight mission basis, not on a laser scanning system basis. For multiple flight missions, if *Mirror Angle Correction* is checked, mirror angle correction will be performed for each mission.

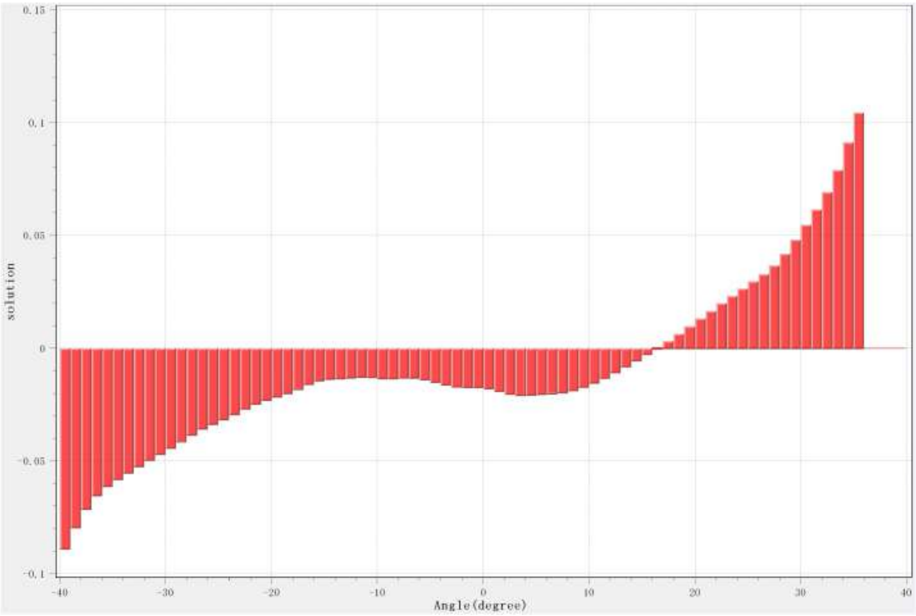


Before Mirror Angle Correction



After Mirror Angle Correction

If mirror angle correction is performed, the strip adjustment report *report.html* will output a histogram of the angle correction amounts.



Angle Correction Amount Histogram

> The histogram is obtained by smoothing the correction amount three times.

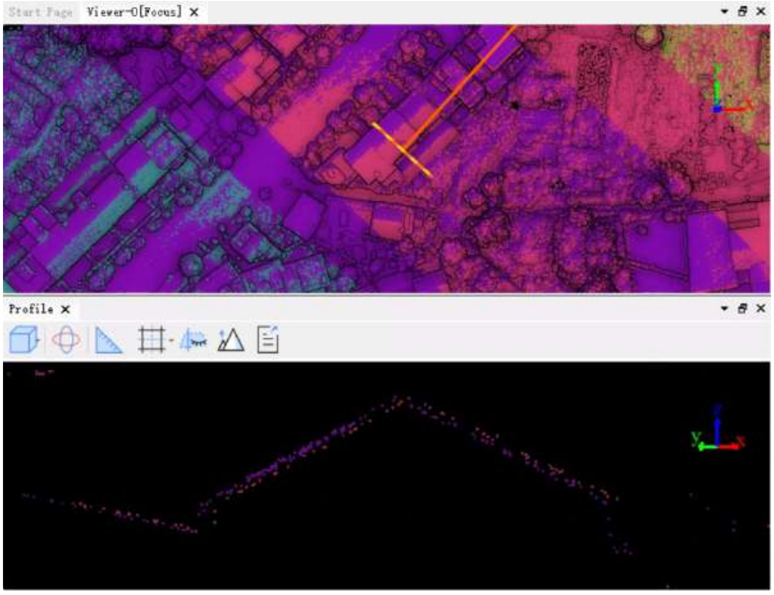
```
@inproceedings{
    author={Brazeal, Ryan G. , B. E. Wilkinson , and H. H. Hochmair},
    title={A Rigorous Observation Model for the Risley Prism-Based Livox Mid-40 Lidar Sensor},
    booktitle={Sensors},
    year={2021},
}
```

After adding all the data, click confirm, and the software will automatically perform the strip adjustment. After completion, the software will output the corrected point cloud, trajectory, image project, and statistical report, which are saved in the **output path/report** directory. Click *report.html* to open the statistical report and view the statistical charts and the updated camera installation angle errors.

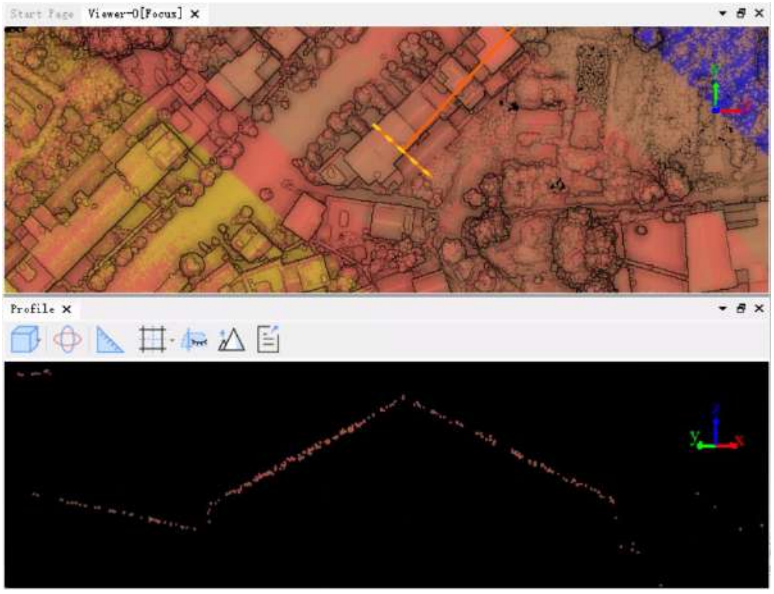
> Note: Each mission must add its corresponding trajectory and point cloud data, otherwise the adjustment calculation cannot be performed.

Since the corrected image project is a sparse point cloud, it cannot be effectively overlaid with the point cloud in the profile view. The corrected image project parameters can be used to generate a dense point cloud for overlaying with the laser point cloud.

Users can use the profile tool to view the correction results of the laser point cloud, as shown in the figure.

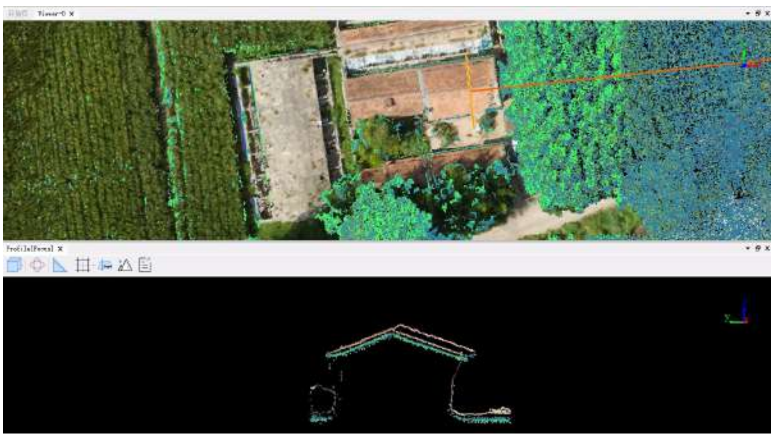


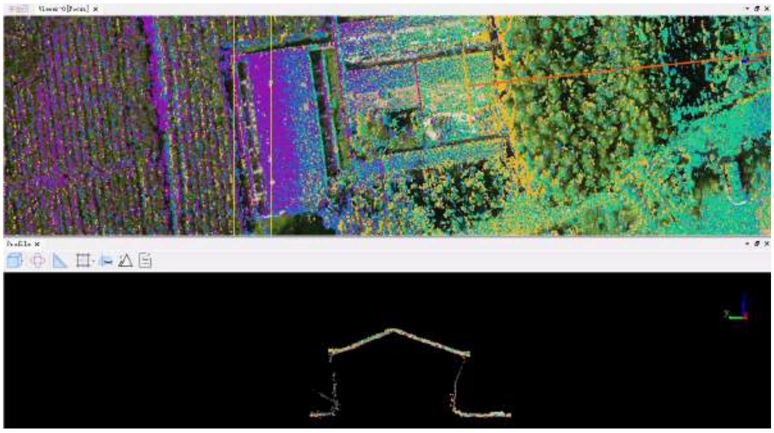
Layering of Laser Point Cloud Before Correction



Layering of Laser Point Cloud After Correction

When using the image project, overlay the dense point cloud before and after correction with the laser point cloud before and after correction, and use the profile tool to view the correction effect, as shown in the figure.


Layering of Laser Point Cloud and Image Dense Point Cloud Before Correction



Layering of Laser Point Cloud and Image Dense Point Cloud After Correction

Statistical Report

The content of the report varies based on the input. It includes basic information on the input data, parameter settings, trajectory maps, accuracy statistics before and after adjustment, residual histograms, scanner mirror angle error correction amounts, and camera installation error values. If only trajectory adjustment is performed, there will be no camera installation error values. If mirror angle error correction is not checked, there will be no scanner mirror angle error correction amounts.

Control Point Report

Summary

Control point report tool will create a report about elevation difference of laser point clouds and ground control points, which can be used to check the elevation accuracy of laser point clouds and improve the height accuracy of laser point clouds using calculated adjusted values.

Control point file is the text file separated by comma. Every row is composed by 3 columns which are X, Y and Z. At least 3 control points are requested to successfully create the control point report.

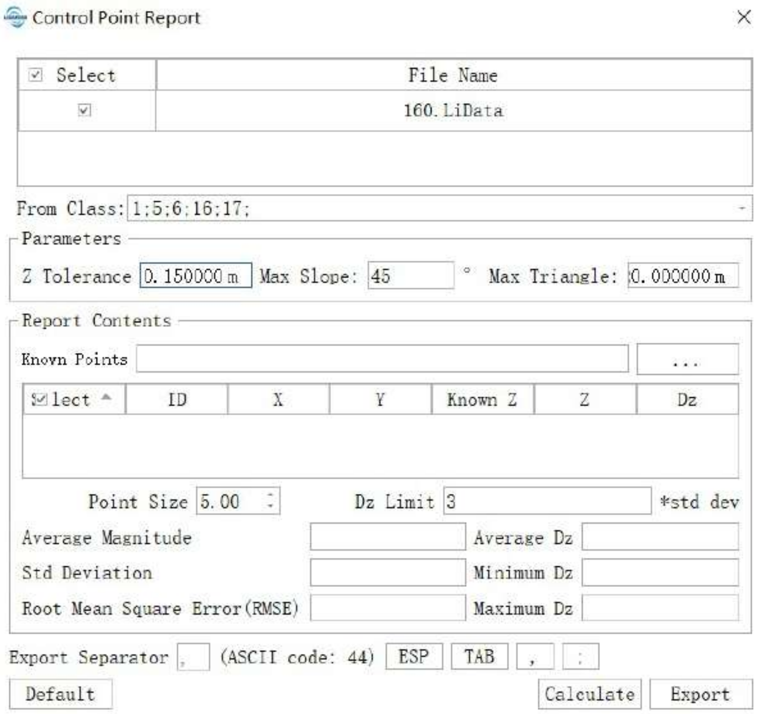
The output report displays information of elevation difference between used laser point clouds and control points of elevation, statistic information of height difference like average magnitude, standard deviation, root mean square, average of elevation difference as well as maximum and minimum height difference.

Usage

Click *Strip Alignment* > *Control Point Report*.

Select the path for the ASCII format control point file. The control point file reading settings will pop up, allowing you to set the column numbers for X, Y, and Z, as well as the number of rows to skip, and other information. You can refer to Add Data for guidance. Once the selection is complete, click Apply.

Click the Calculate button. After the calculation is complete, to view the position of each control point, double-click with the left mouse button on any position in the row where the control point is located to jump to that control point's position. You can more intuitively view the relative positions of the control points and the point cloud using the Profile Tool.



After the calculation, users can choose the control points, or change the maximum slope and maximum triangle edge length, and then click "Calculate" to re-calculate.

Settings

- **Select Data**: Input file can be single point cloud data file or point cloud data set.
- **From Class (default value is "all class")**: User needs to select the point cloud class which is used to check control point elevation accuracy, and normally point cloud of hard surface such as ground point, construction point, etc. are chosen.
- **Z Tolerance (default value is "0.15")**: The accuracy of the point cloud in the Z-axis direction. To avoid the distance between the points is too small leads to the excessive slope.
- **Max Slope (°, default value is "45")**: Maximum terrestrial gradient tolerance. If gradient is larger than this value, elevation difference will not be computed. Normally control points are chosen at the flat terrain, therefore if gradient is too steep, the result will be easily affected by wrong information.
- **Max Triangle (m，default value is "20")**: The longest side length of triangle is used to avoiding the large deviation caused by elevation interpolation. If the side length of triangle is too long, it is indicated that points of initial category in point cloud area where control points are located are less and calculated error of elevation difference is large.
- **Known Points**: Input control point file.
- **Point Size (default value is "0.10")**: The location of control points can be displayed in the window by double clicking the corresponding data in the list of control point report. The size of point controls the size of control points shown in the window.
- **Dz Limit (default value is "3")**: Set the tolerance of Dz. If Dz is not within the tolerance, show red in order to inspect elevation difference with large error between point cloud and control points. Maximum tolerance = Average Dz + Dz × Std Deviation. Minimum tolerance = Average Dz - Dz × Std Deviation.
- **Export Separator**：Set the exported control point report delimiter, including semicolon, comma, etc.
- **Calculate**: After setting parameters, click this button to calculate elevation difference.
- **Export**: Export the control point report file in txt format. The relevant information of the control point calculation is separated by the set delimiter. The export file contains the elevation error information of the point cloud data and the statistical information of Dz.

3D Control Point Report

Function Overview

Provides two inspection methods: elevation and XYZ.

When selecting elevation, the control point report tool generates a report on the elevation differences between the LiDAR point cloud and ground control points. This can be used to check the elevation accuracy of the LiDAR point cloud and improve its elevation accuracy using the calculated correction values. The output report shows the elevation differences between the used LiDAR point cloud and the elevation control points, as well as statistical information like the average magnitude of elevation differences, mean error, root mean square error, average elevation difference, and the maximum and minimum elevation differences.

When selecting XYZ, the control point report tool calculates elevation and horizontal differences based on the identified target positions. It generates a report on the elevation and horizontal differences between the LiDAR point cloud and the ground control points, which can be used to check the elevation and horizontal accuracy of the LiDAR data. The output report shows the elevation and horizontal differences between the used LiDAR point cloud and the elevation control points, as well as statistical information like the average magnitude, mean error, root mean square error, average elevation difference, and the maximum and minimum values.

Target refers to a ground marker made of special materials with specific patterns and shapes, typically placed in areas lacking distinctive features. It is used for accuracy checks in LiDAR and ground control in aerial photography. Targets can automatically identify their positions in point clouds or images, obtaining the 3D coordinates of the marker points or the image pixel coordinates, thereby reducing manual operations.

Target Types

Four types of targets are supported:

Square Checkerboard: A square black and white checkerboard pattern, with the target positioning point located at the center of the target.

Circular Checkerboard: A circular black and white checkerboard pattern, with the target positioning point located at the center of the circle.

2-Layer Concentric Circles: Consists of two concentric circles with alternating black and white colors. The outer circle is black, the inner circle is white, and the radius of the outer circle is twice that of the inner circle. The target positioning point is at the center.

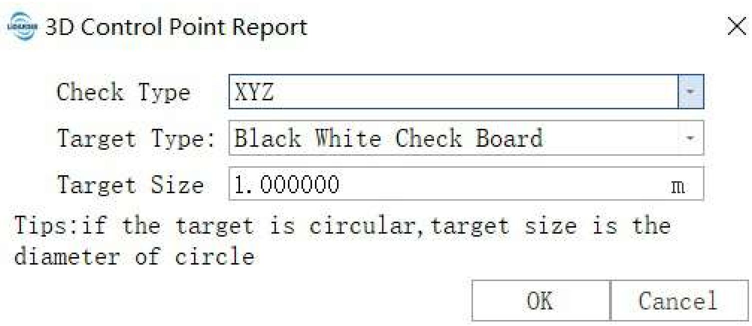
3-Layer Concentric Circles: Consists of three concentric circles with alternating black and white colors. The outer circle is black, the middle circle is white, and the innermost circle is black. The colors. The outer circle is black, the middle circle is white, and the innermost circle is black. The radius of the outermost circle is three times that of the innermost circle, and the middle circle's radius is twice that of the innermost circle. The target positioning point is at the center.

The control point file is a text file separated by commas or spaces, with each line containing X, Y, and Z information. At least three control points are needed to successfully create a control point report.

Usage

Click *Strip Alignment > 3D Control Point Report*

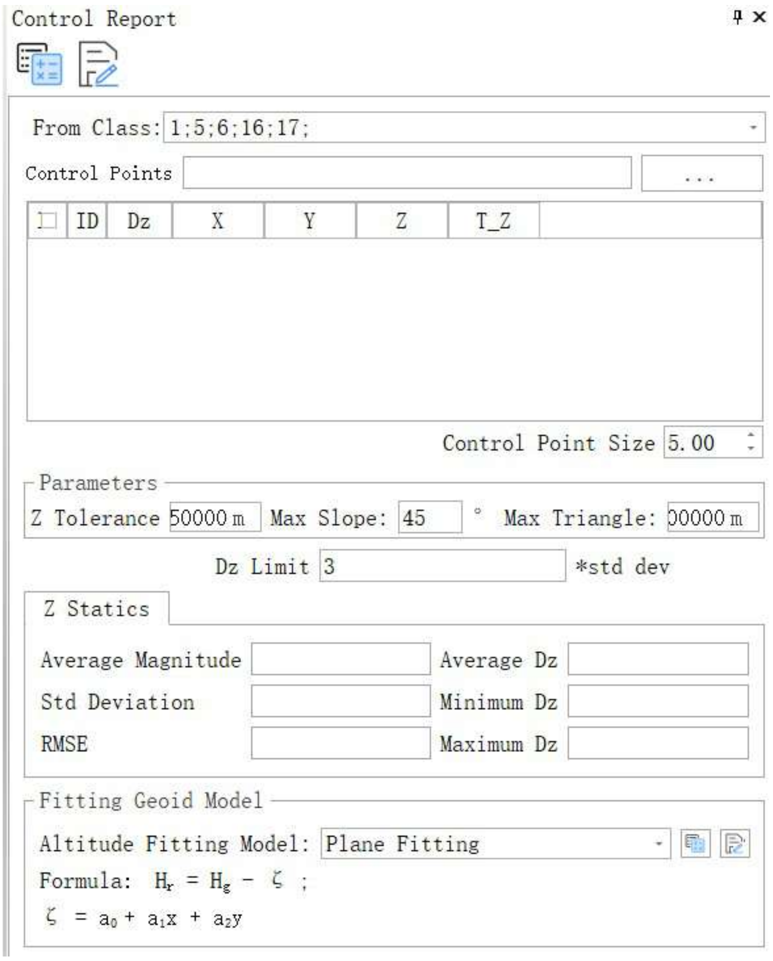
Choose the elevation check method if only control points are available. When targets are set up, you can choose the XYZ check method.



Elevation Check Method

Select the control point file path in ASCII format. A dialog for control point file reading settings will pop up, where you can set the columns for X, Y, and Z, and the number of skipped rows. After selecting, click Apply.

Click the Calculate button. Once the calculation is complete, you can double-click any row in the control point table to jump to the location of the control point. You can use the Profile Tool for a more intuitive view of the relative positions between control points and the point cloud.



After the calculation, you can select the corresponding control points or modify the maximum slope and maximum triangle edge length, then click the "Calculate" button to recalculate.

Parameter Settings

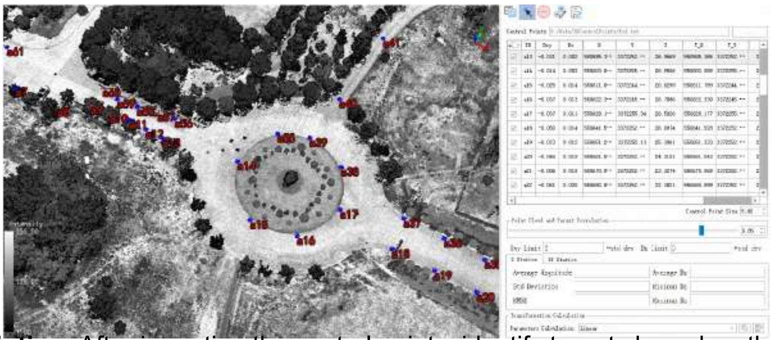
- **Input Point Cloud Data**: The input file can be a single point cloud data file or a point cloud dataset.
- **From Class (default "All Available Categories")**: The user needs to select the point cloud category to check the elevation accuracy using control points, generally choosing hard surface point clouds like ground points and building points.
- **Z Tolerance (default "0.15")**: Point cloud Z direction accuracy, to avoid excessively small distances between point clouds causing large slopes.
- **Max Slope (degrees) (default "45")**: The maximum terrain slope tolerance. If the slope exceeds this value, the elevation difference will not be calculated. Control points are generally placed on relatively flat terrain, so excessive slopes are prone to erroneous information.
- **Max Triangle (meters) (default "20")**: The maximum triangle length to avoid large deviations caused by elevation interpolation. A large triangle edge length indicates too few points in the **From Class** of the control point corresponding point cloud area, resulting in significant errors in the calculated elevation difference.
- **Control Points**: Input the control point file.
- **Control Point Size (default "5")**: The size of control points displayed in the window when double-clicking on data in the control point report list to locate the corresponding control points.
- **Dz Limit (default "3")**: Set the Dz tolerance. If not within this tolerance range, it is displayed in red to detect significant elevation differences between the LiDAR point cloud and control points. Maximum

tolerance = average elevation difference + Dz limit × mean error; minimum tolerance = average elevation difference - Dz limit × mean error.

- **Export Separator**: Set the separator for exporting the control point report, including semicolons and commas.
- **Calculate**: After setting the parameters, click this button to calculate the elevation difference.
- **Export**: Export the control point report in txt format. The exported file contains the elevation error information of the point cloud data and statistical information of Dz, separated by the specified separator.

XYZ Check Method

Select the control point file path in ASCII format. A dialog for control point file reading settings will pop up, where you can set the columns for X, Y, and Z, and the number of skipped rows. Refer to Open File for more details. After selecting, click Apply.



 **Feature Description**: After importing the control points, identify targets based on their positions.

 **Feature Description**: Select targets. If targets are not identified based on control points, manually identify the target center position in the viewer by adjusting the correlation parameters between the point cloud and the target.

 **Feature Description**: Delete target. Delete the selected row's target in the table.

 **Feature Description**: Modify target position. Adjust the target position in the table by fine-tuning the coordinates in the pop-up interface.

Lidar360 ControlPointReport

Feature Description: Delete target. Delete the selected row's target in the table.

Parameter Settings

- **Point Cloud to Target Correlation (default "0.85")**: Point cloud Z direction accuracy to avoid excessively small distances between point clouds causing large slopes.
- **Dz Limit (default "3")**: Set the Dz tolerance. If not within this tolerance range, it is displayed in red to detect significant elevation differences between the LiDAR point cloud and control points. Maximum tolerance = average elevation difference + Dz limit × mean error; minimum tolerance = average elevation difference - Dz limit × mean error.
- **Dxy Limit (default "3")**: Set the Dxy tolerance. If not within this tolerance range, it is displayed in red to detect significant horizontal differences between the LiDAR point cloud and control points. Maximum tolerance = average elevation difference + Dxy limit × mean error; minimum tolerance = average elevation difference - Dxy limit × mean error.

Transformation Relationship Save

The elevation check method provides four types of calculation methods for Height Fitting based on the calculation results: plane fitting, linear interpolation, quadratic surface fitting, and cubic surface fitting. The XYZ check method additionally provides Four Parameters Calculation, Seven Parameters Calculation, 3D Affine, and linear transformation. The linear transformation is based on the calculated delt X, delt Y, delt Z from the identified target positions and control point positions.

Lidar360 ControlPointReport

Feature Description: Calculate transformation relationship.

Feature Description: Save transformation relationship.

Refer to Create Transformations.

Trajectory Quality Analysis

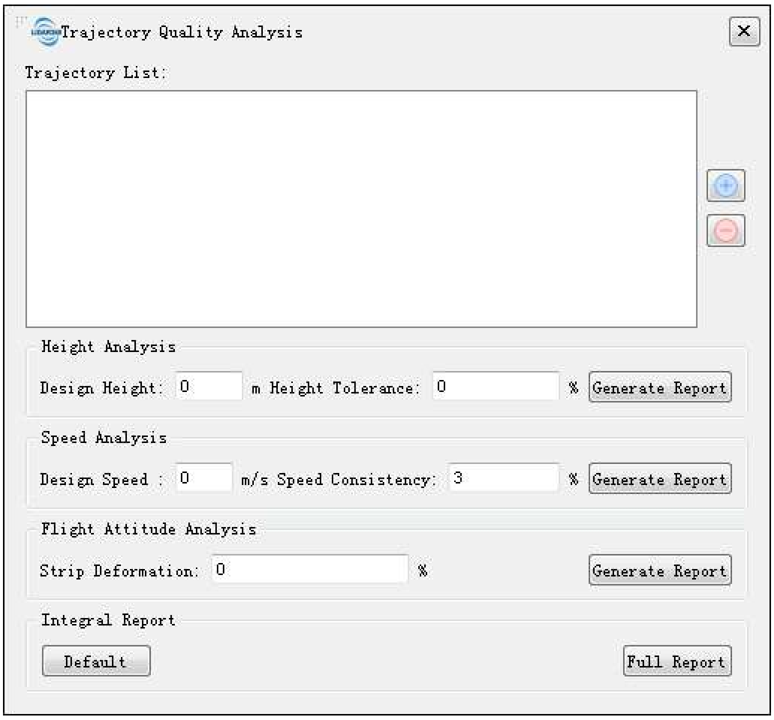
Summary

Trajectory Quality Analysis tool checks trajectory quality from the height analysis, speed analysis and flight attitude analysis.

Usage

Click *Strip Alignment* > *Trajectory Quality Analysis*.

Load trajectory file(s) and set the parameters of height analysis, speed analysis and fight attitude analysis. This tool supports generating reports for each analysis. Click **Export** to save the quality report in html format.



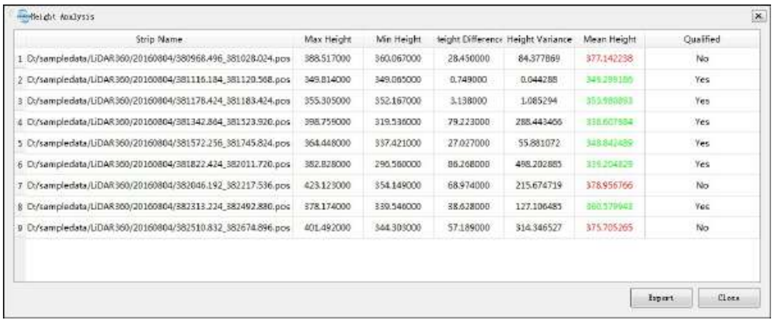
Click to load data. The supported formats of POS files contain *.OUT and *.pos. Click to clear loaded data.

Settings

- **Height Analysis**: Height analysis report contains strip name, max height, min height, height difference, height variance, mean height and qualified flag of each strip. The Yes is qualified, and No is unqualified. The mean height of qualified strip is displayed as green, or as red. Click **Export** button to save the result in html format.
- **Design Height (m, default value is "0")**: Designed height value.
- **Height Tolerance (%, default value is "0")**: The floating ratio of designed height. For example, 5% denotes that the qualified height range is between (1- 5%) × designed height and (1 + 5%) ×

designed height.

- **Generate Report**: Click **Generate Report** button to save height analysis quality report.

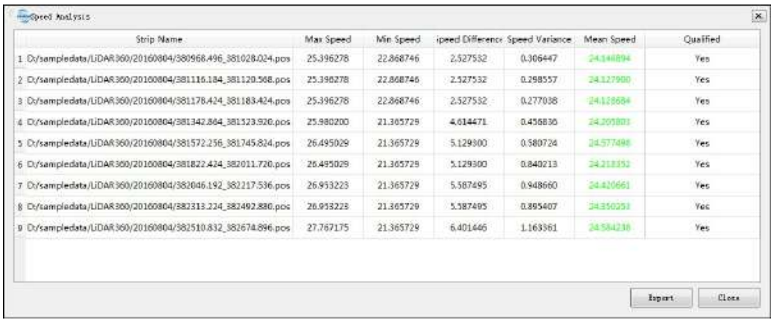


Height Analysis

	Strip Name	Max Height	Min Height	Height Difference	Height Variance	Mean Height	Qualified
1	D:/sampledata/LiDAR360/20160804/380968.496_381028.024.pos	388.517000	360.067000	28.450000	84.377869	377.142238	No
2	D:/sampledata/LiDAR360/20160804/381116.184_381120.568.pos	349.814000	349.065000	0.749000	0.044288	349.299180	Yes
3	D:/sampledata/LiDAR360/20160804/381178.424_381183.424.pos	355.305000	352.167000	3.138000	1.085294	353.980893	Yes
4	D:/sampledata/LiDAR360/20160804/381342.864_381523.920.pos	398.759000	319.536000	79.223000	288.443466	338.607984	Yes
5	D:/sampledata/LiDAR360/20160804/381572.256_381745.824.pos	364.448000	337.421000	27.027000	55.881072	348.842489	Yes
6	D:/sampledata/LiDAR360/20160804/381822.424_382011.720.pos	382.828000	296.560000	86.268000	498.202885	339.204829	Yes
7	D:/sampledata/LiDAR360/20160804/382046.192_382217.536.pos	423.123000	354.149000	68.974000	215.674719	378.956766	No
8	D:/sampledata/LiDAR360/20160804/382313.224_382492.880.pos	378.174000	339.546000	38.628000	127.106485	360.579943	Yes
9	D:/sampledata/LiDAR360/20160804/382510.832_382674.896.pos	401.492000	344.303000	57.189000	314.346527	375.705265	No

Export Close

- **Speed Analysis**: The speed analysis report contains strip name, max speed, min speed, speed difference, speed variance, mean speed and qualified flag of each strip. The mean speed of qualified strip is displayed as green, or as red. Click *Export* button to save the result in HTML format.
- **Design Speed (m/s, default value is "0")**: Designed speed value.
- **Speed Consistency (%, default value is "0")**: The floating ratio of designed speed. For example, 5% denotes that the qualified speed range is between (1- 5%) × designed speed and (1 + 5%) × designed speed.
- **Generate Report**: Click **Generate Report** button to save speed analysis quality report.

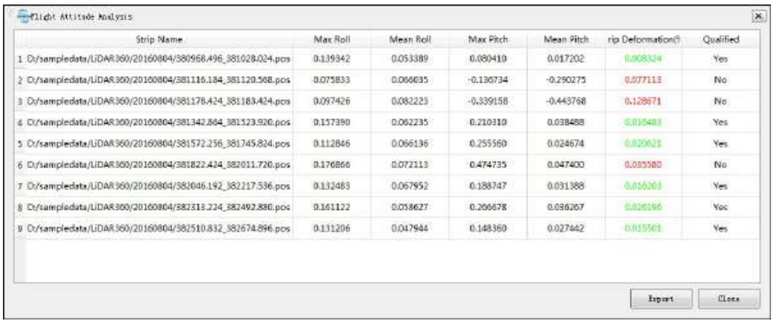


Speed Analysis

	Strip Name	Max Speed	Min Speed	Speed Difference	Speed Variance	Mean Speed	Qualified
1	D:/sampledata/LiDAR360/20160804/380968.496_381028.024.pos	25.396278	22.868746	2.527532	0.306447	24.146894	Yes
2	D:/sampledata/LiDAR360/20160804/381116.184_381120.568.pos	25.396278	22.868746	2.527532	0.298557	24.127900	Yes
3	D:/sampledata/LiDAR360/20160804/381178.424_381183.424.pos	25.396278	22.868746	2.527532	0.277038	24.128684	Yes
4	D:/sampledata/LiDAR360/20160804/381342.864_381523.920.pos	25.980200	21.365729	4.614471	0.456836	24.205803	Yes
5	D:/sampledata/LiDAR360/20160804/381572.256_381745.824.pos	26.495029	21.365729	5.129300	0.580724	24.577498	Yes
6	D:/sampledata/LiDAR360/20160804/381822.424_382011.720.pos	26.495029	21.365729	5.129300	0.840213	24.213352	Yes
7	D:/sampledata/LiDAR360/20160804/382046.192_382217.536.pos	26.953223	21.365729	5.587495	0.948660	24.420661	Yes
8	D:/sampledata/LiDAR360/20160804/382313.224_382492.880.pos	26.953223	21.365729	5.587495	0.895407	24.350251	Yes
9	D:/sampledata/LiDAR360/20160804/382510.832_382674.896.pos	27.767175	21.365729	6.401446	1.163361	24.584238	Yes

Export Close

- **Flight Attitude Analysis**: Flight attitude analysis report contains strip name, max roll, mean roll, max pitch, mean pitch, strip deformation (%) and qualified flag of each strip. The Yes is qualified, and No is unqualified. The strip deformation of qualified strip is displayed as green, or as red. Click **Export** button to save the result in html format.
- **Strip Deformation (%, default value is "3")**: According to the related data standard, the qualified threshold of strip deformation is generally set to 3%.
- **Generate Report**: Click **Generate Report** button to save flight attitude analysis quality report.



Flight Attitude Analysis

	Strip Name	Max Roll	Mean Roll	Max Pitch	Mean Pitch	Strip Deformation(%)	Qualified
1	D:/sampledata/LiDAR360/20160804/380968.496_381028.024.pos	0.139342	0.053389	0.080410	0.017202	0.008324	Yes
2	D:/sampledata/LiDAR360/20160804/381116.184_381120.568.pos	0.075833	0.066035	-0.136734	-0.290275	0.077113	No
3	D:/sampledata/LiDAR360/20160804/381178.424_381183.424.pos	0.097426	0.082223	-0.339158	-0.443768	0.128671	No
4	D:/sampledata/LiDAR360/20160804/381342.864_381523.920.pos	0.157390	0.062235	0.210310	0.038488	0.010483	Yes
5	D:/sampledata/LiDAR360/20160804/381572.256_381745.824.pos	0.112846	0.066136	0.255560	0.024674	0.020621	Yes
6	D:/sampledata/LiDAR360/20160804/381822.424_382011.720.pos	0.176866	0.072113	0.474735	0.047400	0.035580	No
7	D:/sampledata/LiDAR360/20160804/382046.192_382217.536.pos	0.132483	0.067952	0.188747	0.031388	0.016203	Yes
8	D:/sampledata/LiDAR360/20160804/382313.224_382492.880.pos	0.161122	0.058627	0.206678	0.036267	0.026196	Yes
9	D:/sampledata/LiDAR360/20160804/382510.832_382674.896.pos	0.131206	0.047944	0.148360	0.027442	0.015501	Yes

Export Close

- **Integral Report**: Click **Full Report** button to save the integral report.

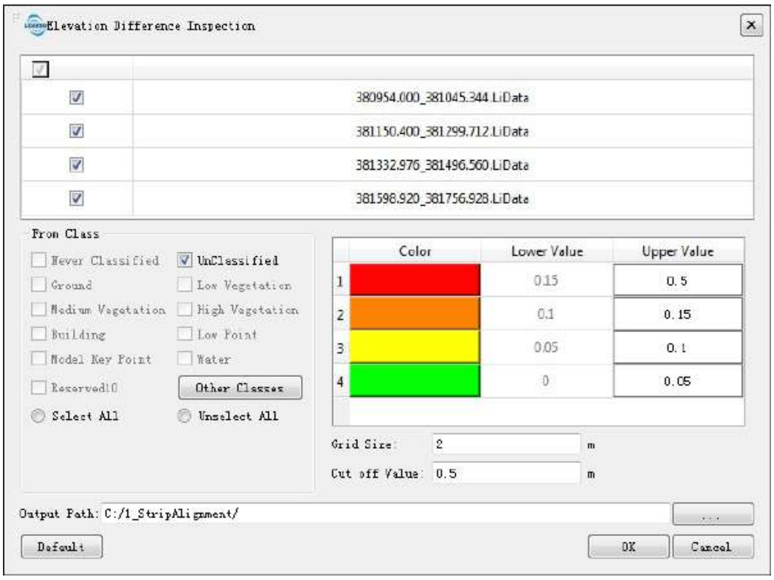
Elevation Difference Inspection

Summary

Elevation Difference Inspection tool analyzes the elevation difference between point clouds.

Usage

Click *Strip Alignment* > *Elevation Difference Inspection*



Settings

- **Input Data**: Input files can be single point cloud file or point cloud dataset. The data to be processed must be opened in the LiDAR360.
- **From Class**: The point cloud classes used in the elevation difference quality inspection.
- **Grid Size (m, default value is "2")**: The grid size in the point cloud gridding.
- **Cut off Value (m, default value is "0.5")**: The data will be ignored if the height difference is greater than this threshold. Considering the moving vehicles in the scanning area which can be scanned at different time, it is necessary to set the cut off value for height difference.
- **Output Path**: The results of elevation difference quality inspection will be exported to the selected folder, which contains the result of each strip density quality inspection and integral report in HTML format.

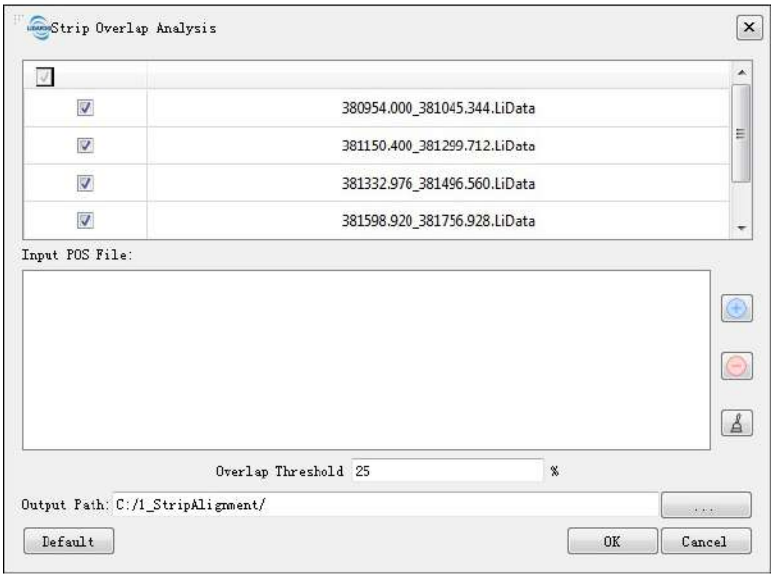
Strip Overlap Analysis

Summary

Strip Overlap Analysis tool analyzes the overlap between point clouds.

Usage

Click *Strip Alignment* > *Strip Overlap Analysis*



Settings

- **Input Data**: The input files should be 2 or more point cloud files. The data to be processed must be pre-opened in LiDAR360.
- **Input POS File**: Click to load the data. The supported formats of POS files include *.OUT and *.pos. Click to clear the loaded data. Click to clear the loaded POS file(s).
- **Overlap Threshold (%, default value is "25")**: Set the threshold to define the overlap region between two adjacent strips.
- **Output Path**: The results of overlap quality inspection will be exported to the selected folder, which contains the results of each strip overlap quality inspection in SHP format and a comprehensive report in html format.

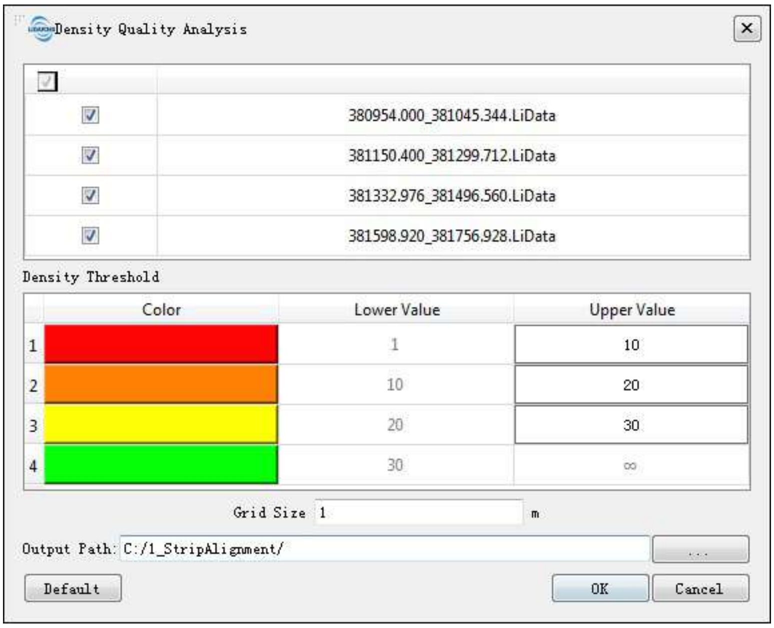
Density Quality Analysis

Summary

Density Quality Analysis tool analyzes the density of point cloud.

Usage

Click *Strip Alignment* > *Density Quality Analysis*



Settings

- **Input Data**: Input file can be single point cloud file or point cloud dataset. The data to be processed must be opened in the LiDAR360.
- **Density Threshold**: The point density between the adjacent thresholds is shown in the corresponding color.
- **Grid Size (m, default value is "1")**: The grid size in the point cloud gridding.
- **Output path**: Select the output path of density inspection quality report. The folder contains the results of each strip density quality inspection and integral report in html format.

DJI L1/L2 Reconstruction

Function Overview

The DJI L1/L2 reconstruction feature supports the processing of Zenmuse L1 and Zenmuse L2 data. It converts the raw LiDAR files from the L1 and L2 into various formats of 3D point cloud data (PNTS/LAS/PLY/PCD/S3MB). It supports point cloud classification, generates Digital Elevation Models (DEMs), and allows users to import the results into downstream analysis software for applications in surveying, power industry, etc.

Usage

Click *Strip Alignment > DJI L1/L2 Reconstruction*.

Parameter Settings

- **Project Directory**: Users can click the button on the right to select the directory of the raw LiDAR data for L1 and L2.
- **Scene**:
  - Output Point Cloud: Users can select this option to output point cloud results.
  - Refine calibrate: Users can select this option to perform refine calibrate.
- **Point Cloud Processing**:
  - ICP optimise: Users can select this option to optimise the LiDAR point cloud using the ICP (Iterative Closest Point) algorithm.
  - Smooth point: Users can select this option to smooth the LiDAR point cloud.
  - Colorize points: Users can select this option to color the LiDAR point cloud.
  - LiDAR Point Max Distance: Set the maximum effective distance for the LiDAR point cloud. Points beyond this distance will be clipped.
- **Output Coordinate System**: Users can select the coordinate system for the output results. If not specified, the algorithm will automatically determine it.
- **Output Format**:
  - LAS: Users can select this option to generate point cloud model results in LAS format.
  - PLY: Users can select this option to generate point cloud model results in PLY format.
  - PNTS: Users can select this option to generate point cloud LOD models in PNTS format, viewable with Terra software or Cesium web viewer.
  - PCD: Users can select this option to generate point cloud model results in PCD format.
  - S3MB: Users can select this option to generate point cloud model results in S3MB format, viewable with SuperMap software.
  - Merge Point: Users can select this option to merge the point cloud into a single file.
- **Configuration authentication information**:

- **App Key**: Enter the App Key authorized by DJI Terra.
- **Secret Key**: Enter the Secret Key authorized by DJI Terra.

- **Output Path**: The folder path where the generated and downloaded results will be saved during the execution of the function.

Data Management

The data management module includes basic processing tools for point clouds and images, mainly including pre-processing of point clouds, mutual conversion between point cloud and image formats, projection tools, etc.

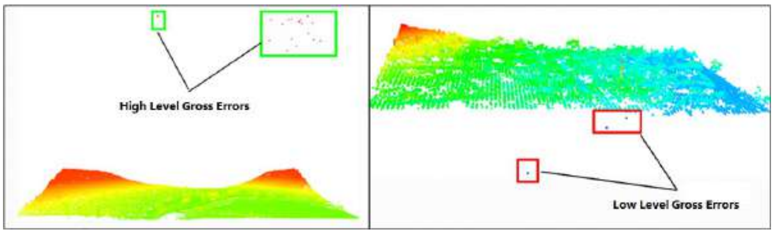
Point Cloud Tools Index

Point Cloud Tools include the following point cloud processing functions.

Remove Outliers

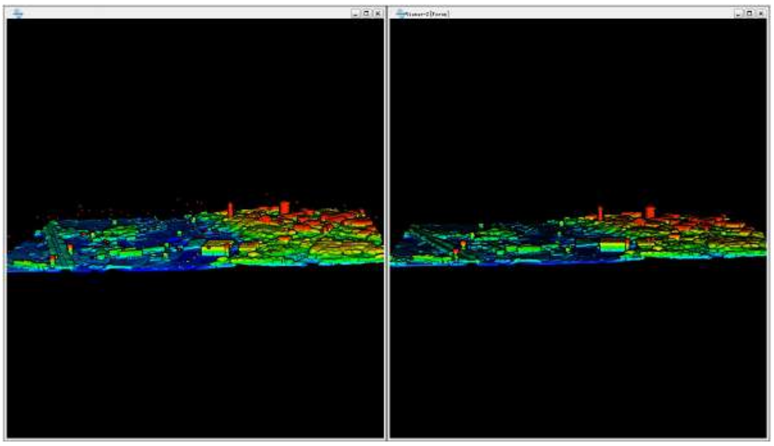
Summary

Common noises include high level gross errors and low level gross errors. As shown below, high level gross error is usually caused by the returns of high-flying objects (such as birds or aircraft) during the process of data collection; low-level gross error are returns with extremely low attitudes caused by the multipath effect of a laser pulse. The Outlier Removal tool aims to remove these errors as much as possible and therefore improve the data quality.



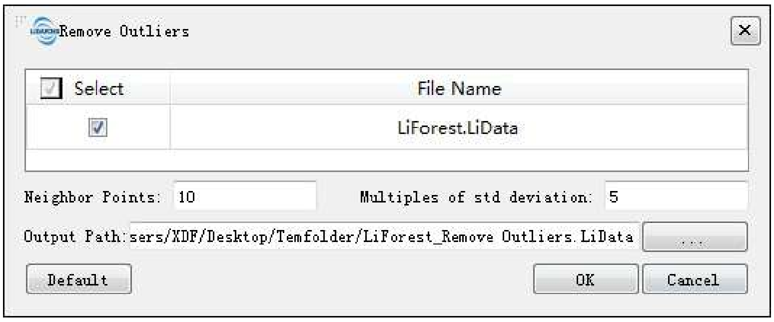
The algorithm will first search for each point's neighboring points within a user-defined area and calculate the average distance from the point to its neighboring points. Then, the mean and standard deviation of these average distances for all points are calculated. If the average distance of a point to its neighbors is larger than maximum distance (maximum distance = mean + n * standard deviation, where n is a user-defined multiple number), it will be considered as an outlier and be removed from the original point cloud.

Effect picture:



Usage

Click *Data Management* > *Point Cloud Tools* > *Remove Outliers*



Settings

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

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

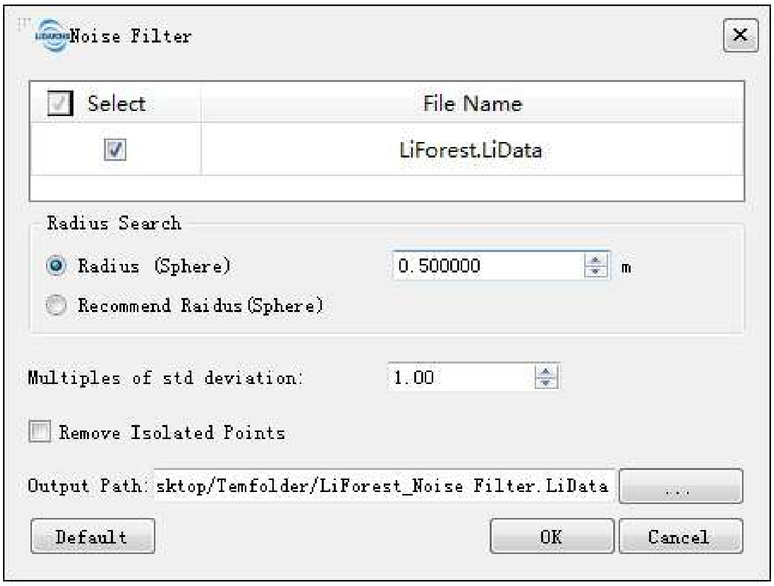
Noise Filter

Summary

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

Usage

Click *Data Management* > *Point Cloud Tools* > *Noise Filter*.



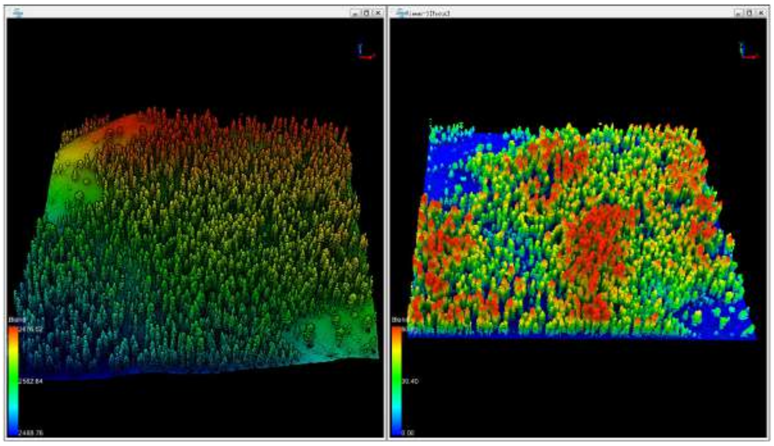
Settings

- **Input Data**: The input data can be a single point cloud file, or can be a point cloud data collection; these data to be processed must be pre-opened in LiDAR360 software.
- **Radius(Default value is “0.5m”)**: Set the radius of the fitting plane. This function can be used when users know the approximate density of the point cloud data.
- **Recommended Radius**: Automatically calculate the appropriate searching radius according to the input point cloud data.
- **Multiples of std deviation(Default value is “1.0”)**: Using the relative error (sigma) as a parameter for outliers removal. The algorithm will automatically calculate the standard deviation (stddev) of a point P's surrounding fitting plane. If the distance, d, from this point to that plane is less than sigma * stddev, this point, P, will be kept. The reduction of this relative error results in removing more points. Conversely, more points will be retained. The change of this parameter will not ifnluence the efficiency.
- **Remove Isolated Points**: The point will be treated as an isolated point when there is less than 4 points within the distance of the searching radius (cannot create a fitting plane with less than 4 points).
- **Output Path**: The point cloud data will be save at this path after the outliers are removed.

Normalize by DEM

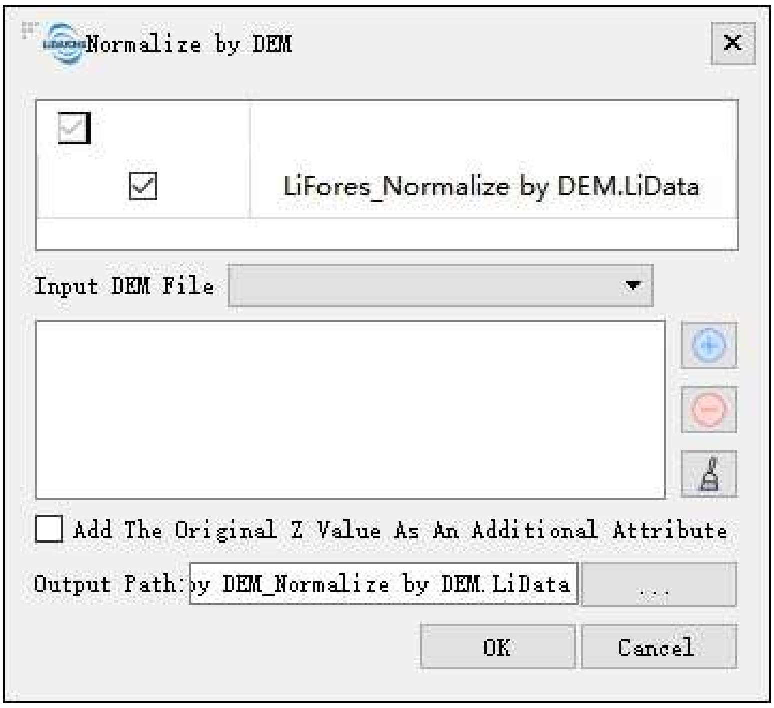
Summary

The normalization tool can remove the influence of terrain relief on the elevation value of the point cloud data. This function requires that the extent of the DEM overlaps with the extent of the point cloud data. The normalization process is performed by subtracting the corresponding terrain elevation of the DEM from each point's Z value. The output of this function is similar to Normalize by Ground Class.



Usage

Click *Data Management* > *Point Cloud Tools* > *Normalize by DEM*



Settings

- **Input Data**: The input file can be a single point cloud data file or multiple data files. File Format: *.LiData.
- **Input DEM File**: Users can select single or multiple single-band TIFF image files from the drop-down list. File format: *.tif.
- : Users can add external DEM file data.
- : Users can select a file in the list and click this button to remove the file from the list.
- : Click this button to clear all the data in the list.
- **Add Z Value to Additional Attributes**: Add the Z value of the current point cloud to the additional attribute table. If this option is not checked, then the normalized point cloud cannot be denormalized.
- **Output Path**: Path of the output file. After the algorithm being executed, the new normalized file will be generated. When entering more than one file, the path will need to be set as a folder.

> Note: The newly generated LiData files in normalized format default to the highest version supported by the current software (supporting registered attributes).

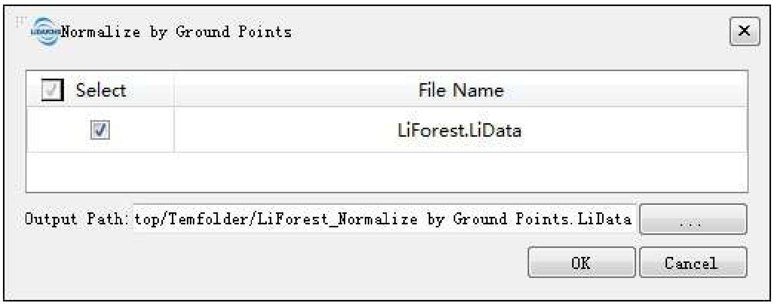
Normalize by Ground Points

Functional Overview

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

Usage

Click *Data Management* > *Point Cloud Tools* > *Normalize by Ground Points*



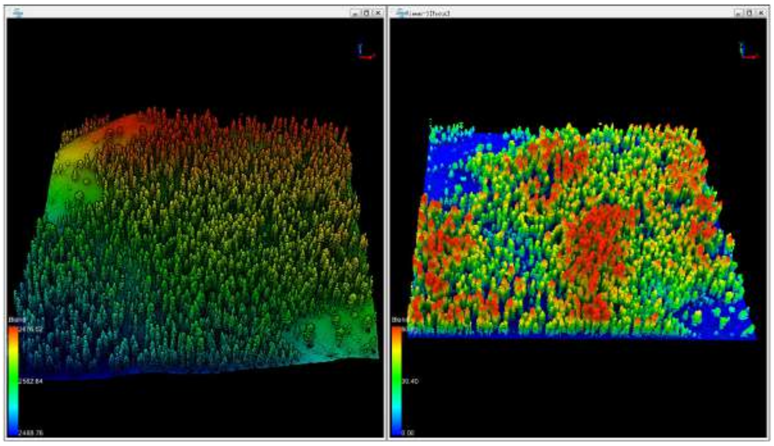
Parameters Settings

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

Denormalize

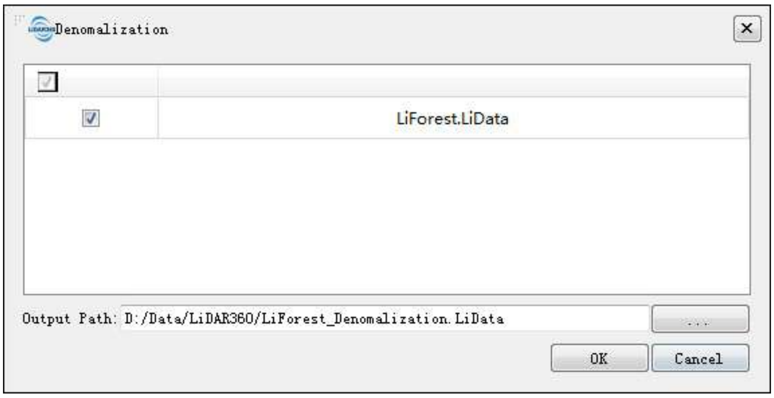
Summary

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



Usage

Click *Data Management* > *Point Cloud Tools* > *Denormalize*



Settings

- **Input Data**: The input data could be one or more point cloud files. The input data should be in *.LiData format.
- **Output Path**: The path of the output data after denormalization. When number of input data sets is more than one, the output path should be a folder.

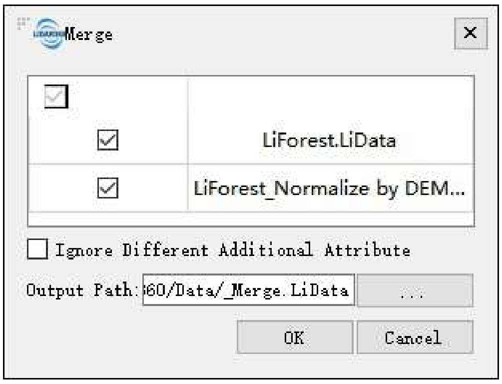
Merge

Summary

Merge multiple point cloud files into a single point cloud file. This function is the reverse operation of Tile by Range and Tile by Point Number.

Usage

Click *Data Management* > *Point Cloud Tools* > *Merge*



Settings

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

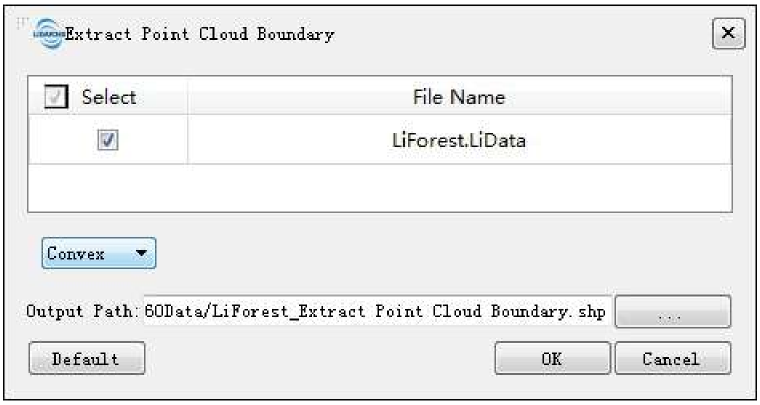
Extract Point Cloud Boundary

Functional Overview

Use a regular hexagon to extract the boundary of a point cloud data. At present, three types of boundaries are supported, namely hexagonal boundary, convex hull and concave hull. The ultimate output file is the final border vector file.

Usage

Click *Data Management* > *Point Cloud Tools* > *Extract Point Cloud Boundary*



Parameters Settings

- **Input point cloud data**: Input one or more point cloud data files. File format: *.LiData.
- **Extraction Methods**: Choose the method to extract the boundary. Currently supports three methods, hexagon, convex hull, and concave hull.
    - **Hexagon**: Use regular hexagons to extract the boundaries of the point cloud data. Based on **the hexagon height** to determine the size of the hexagon. According to the bounding box of point cloud data, draw each hexagon. If the number of point clouds in the hexagon is greater than or equal to the set **minimum number of points, draw the hexagon**, merge the connected regular hexagon, and output the final boundary vector file.
        - **Hexagon Height (m) (default value is "10")**: The height of the hexagon, which is used to set the size of a hexagon.
        - **Minimum Number of Points (default value is "1")**: The threshold of points in a hexagon below which no boundaries are drawn.
    - **Convex Hull (default)**: Extract the convex hull of the point cloud in the X-Y plane (the z-value is ignored). The output file is the vector file (polygon) of the convex hull of the original point cloud.
    - **Concave Hull**: Extract the concave hull of the point cloud in the X-Y plane (the Z-value is ignored). The output file is the vector file (polygon) of the concave hull of the original point cloud.
        - **Maximum Side Length (m) (default value is "2")**: Each edge of concave hull has the maximum distance (Len) in the xy plane. As this value becomes larger, more long edges will be preserved and the resulting boundary will become more similar to the convex hull. On the contrary, more boundary details will be retained by the short edges, and the computational

efficiency will be reduced. At this time, the program will automatically estimate the average distance between all the points, and set the parameter as twice of the average distance.

- **Output to**:
    - **Single file (default)**: When checked, the boundary vector files generated by each LiData file will be merged into the same vector file.
    - **Multi-file**: Each LiData file will generate its own boundary file.
- **Output Path**: The path of the output file as a new vector file generated after the execution of the function.

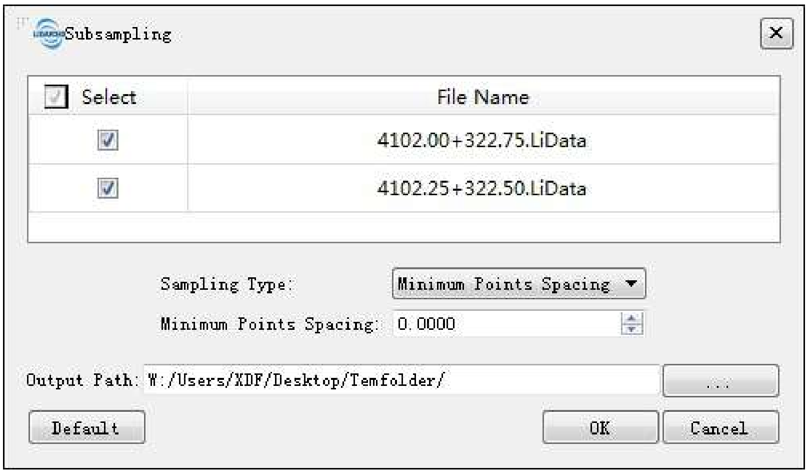
Subsampling

Summary

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

Usage

Click *Data Management* > *Point Cloud Tools* > *Subsampling*



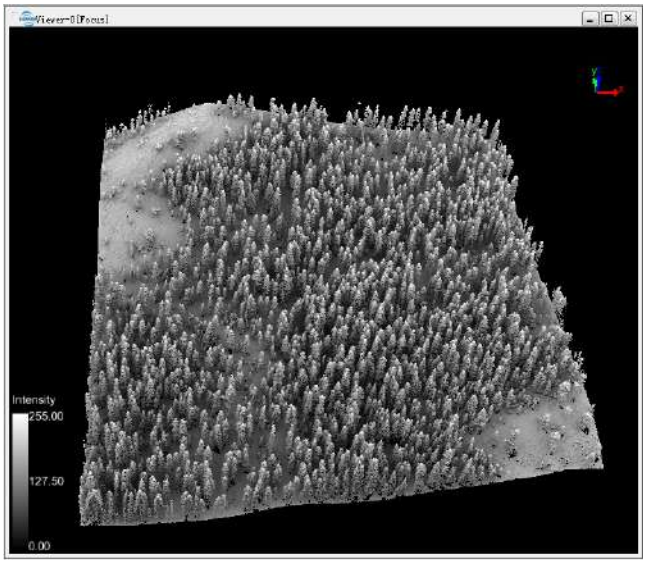
Settings

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

PCV

Summary

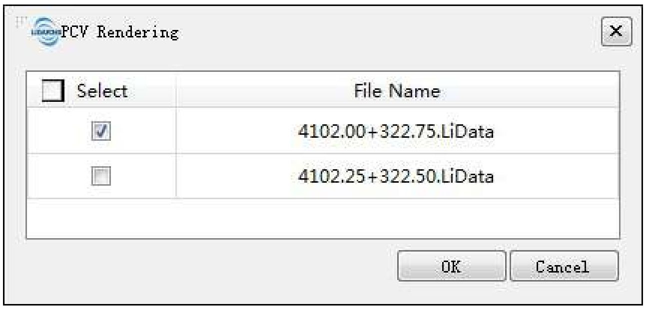
PCV tool can be used to improve visualization effect of a point cloud. The principle of the PCV (Portion of Visible Sky) algorithm is to evenly distribute multiple light sources at the top of the hemisphere or sphere calculate, and then calculate the cumulative number of times when the light can be irradiated by illuminating each point in a point cloud. The last statistical result will be used as the intensity of the point cloud. The rendering effect of the point cloud intensity after PCV calculation is shown in the figure below.



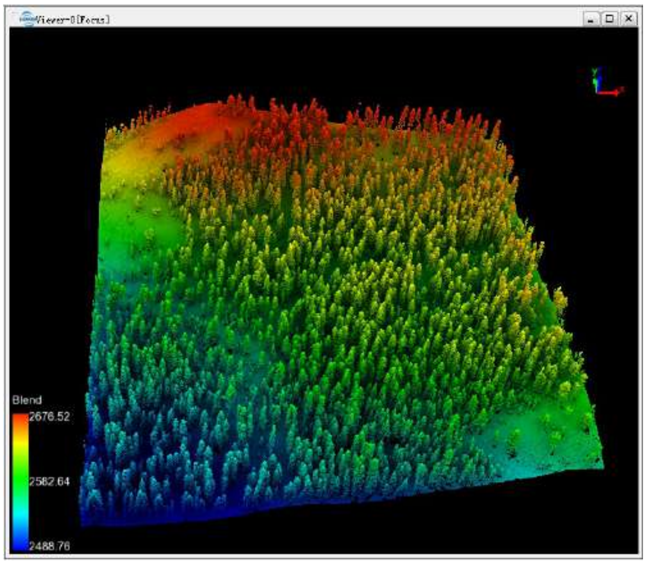
Usage

Click *Data Management* > *Point Cloud Tools* > *PCV*

Select the point cloud data that needs to be processed by PCV.



> After the PCV process, the intensity range of a point cloud will be 0-255. By clicking on display by intensity or clicking display by the mixed mode, users can see the boundary between different land covers more clearly in the point cloud after PCV process. The figure below shows an example of intensity and height blend displaying effect after PCV calculation.


Viewer-0 [Focus]
Blend
2676.52
2582.64
2488.76

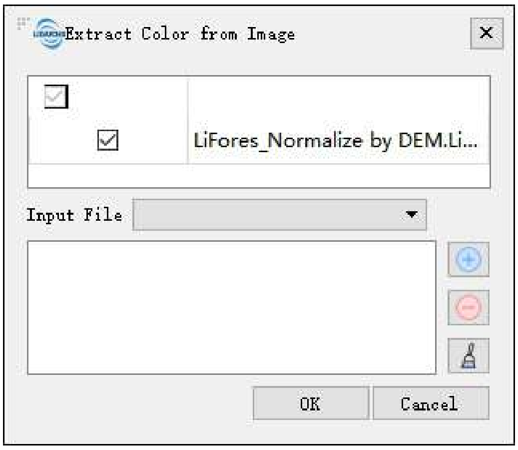
Extract Color from Image

Summary

Extract Color from Image tool can extract RGB information from multi-band imagery and assigned them to each point in a point cloud. The user is required to enter one or more multi-band images that overlapping with the extend of the point cloud data.

Usage

Click *Data Management* > *Point Cloud Tools* > *Extract Color from Image*



Settings

- **Select File**: Input one or more point cloud data files. File format: *.LiData.
- **Ortho Image/Image Project**: Choose whether to map using multi-band imagery or image projects.
- **Overwrite Color**: Specify whether to overwrite points that already have color value attributes.
- **Input Files**:
  - If selecting multi-band imagery, you need to input multi-band imagery data with the same geographic location as the point cloud. If the imagery data is already open in the software, select it from the dropdown menu. Alternatively, click to open external imagery data. Click to remove selected data, and click to clear the imagery data list. After processing, the display mode of the point cloud data will automatically switch to RGB display (you can also click the button on the toolbar to display the point cloud in RGB). File format: *.tif.
  - If selecting image projects, select the image project file (*.liaep) from the dropdown menu. For information on importing image projects, refer to the Import Data page for adding aerial triangulation results data.

Subdivision

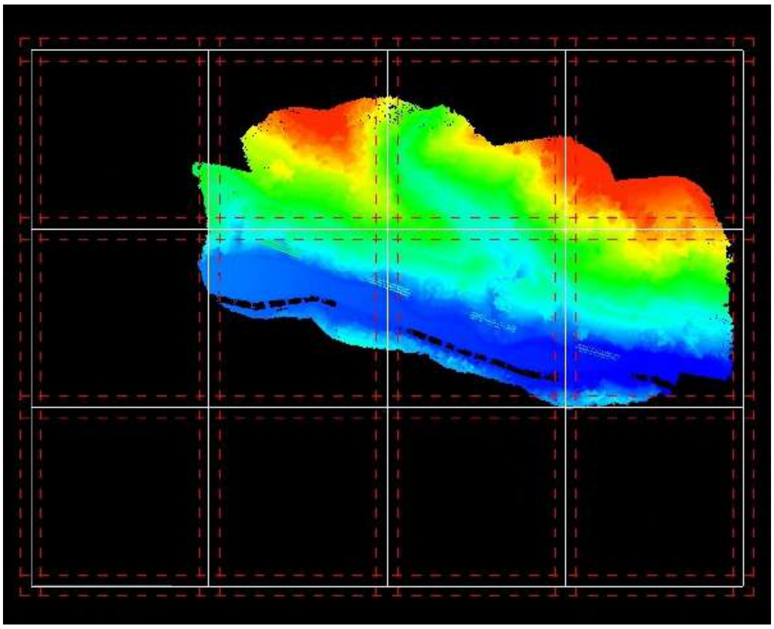
Summary

Subdivision tool can divide a point cloud to a series of frames based on the user defined scales. Currently, for rectangle-division, LiDAR360 supports four subdivision scales: a scale of 1:5000 (taken to 1 km), 1:2000 (taken to 0.1 km), 1:1000 (taken to 0.1 km), and 1:500 (taken to 0.01km). The frame ID will be named by the coordinate number, which is composed of the southwest corner of the frame (Y coordinate + X coordinate). After the operation is completed, one or multiple LiData file will be generated. A vector file in shp format will also be generated recording each subdivided frame.

Usage

Click *Data Management* > *Point Cloud Tools* > *Subdivision*

After completing the parameter settings, click the **Preview** button to check the subdivision grid. The white solid lines indicate the block boundaries, while the red dashed lines represent the buffer zone. Only rectangular subdivision previews are supported, and previewing is only allowed when all selected point cloud files are within the current window.



Settings

- **Input LiData File**: Input the point cloud data to be subdivided. If the point cloud data is already open in the software, click the drop-down button to select the data; or you can click to open external point cloud data. Click to remove the selected data. Click to clear the data list. File format: *.LiData.
- **Subdivision by Rectangle**: Use "Subdivision by Rectangle". Scale is 1:500 by default.
- **Subdivision by Latitude and Longitude**: Use "Subdivisions of Graticules". Scale is 1:500 by default.
- **Scale**: Users can choose the scale of the division. For rectangle-division, LiDAR provides four scales, including 1:500, 1:1000, 1:2000, 1:5000. For subdivisions of graticules, LiDAR provides eleven scales, including 1:500, 1:1000, 1:2000, 1:5000, 1:10000, 1:25000, 1：50000, 1:100000, 1:250000, 1:500000, 1:1000000.
- **Buffer (default is "0")**: The size of buffer around each data.
- **Ignore Different Additional Attribute**: In the case of processing multiple input point clouds, if this option is checked, only the same additional attributes will be merged. Otherwise, as long as there are different additional attributes, the merge cannot be performed.
- **Output Path**: The path for the output folder. After running this function, the new divisions will be generated.

Transform GPS Time

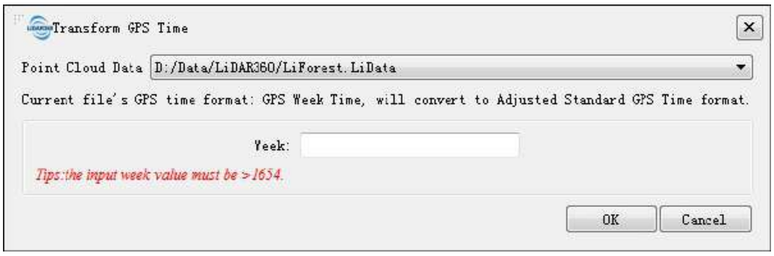
Function Description

This function converts the GPS time information in LiData from GPS coordinate time to GPS week second, or from GPS week second to GPS coordinate time. There are two ways to record the GPS time information, GPS week second and GPS coordinate time (GPS standard time minus 10^9).

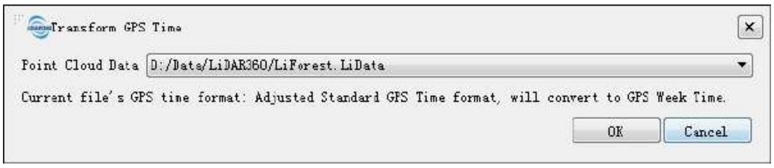
Usage

Click Data Management > *Point Cloud Tools* > *Transform GPS Time.*

If the GPS time of the selected data is GPS weeks and seconds, the pop-up interface is as shown in the figure below：



If the GPS time of the selected data is GPS coordinated, the pop-up interface is as shown in the figure below：



Settings

- **Input Data**：Enter a LiData point cloud data file.

If the GPS time of the input point cloud data is GPS weeks and seconds, it needs to be converted to GPS coordination time.

- **Collection Date**：If the GPS time information of the input point cloud is in GPS week second and to be transformed into GPS coordinate time, the week number is necessary. And the week number must be greater than 1654. Otherwise, when being subtracted by 10^9, the GPS time will be negative. If the GPS time information of the input point cloud is in GPS coordinate time and to be transformed into GPS week second, the week number is unnecessary.

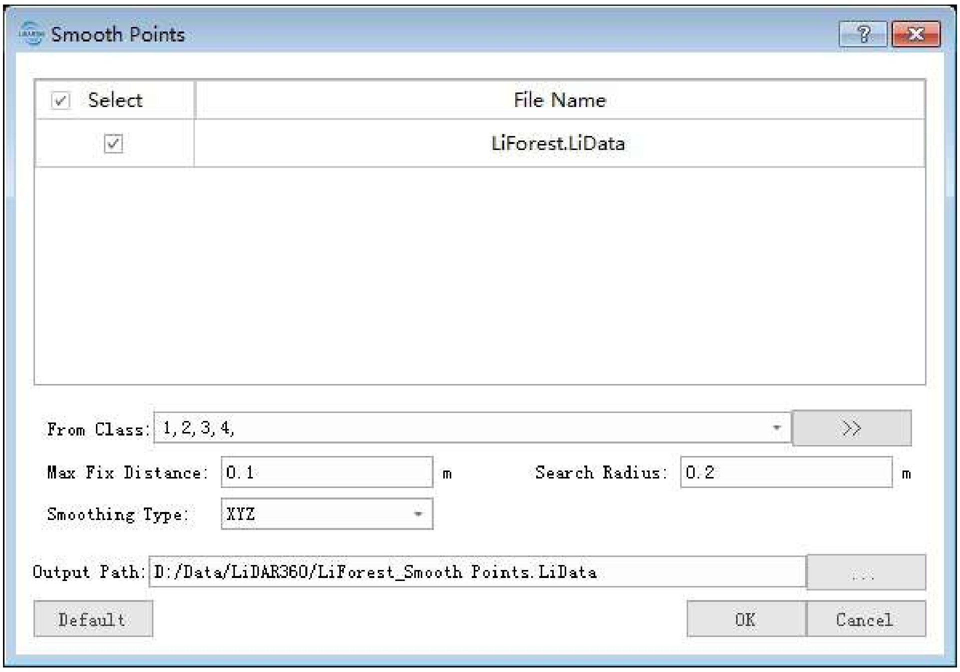
Smooth Points

Summary

Smooth the laser point cloud based on the neighbouring points, making the point cloud look more consistent.

Usage

Click *Data Management* > *Point Cloud Tools* > *Smooth Points*.



Settings

- **Input Point Cloud Data**: Input the LiData point cloud data to be smoothed.
- **Smoothing Type**:
    - **XYZ（Default）**: In the 3D smoothing process, the points on the vertical plane are smoothed in the XY direction, and the points on the horizontal plane are smoothed in the Z direction.
    - **Elevation**: Fit the best fitting plane equation according to the neighboring points, and adjust the elevation of the center point to better fit the plane equation. If the adjustment distance exceeds the repair threshold, no adjustment will be made.
- **Search Radius (m) (Default Value is “0.2”)**: Search radius of neighboring points.
- **Max Fix Distance (m) (Default Value is “0.1”)**: The maximum single-point adjustment distance threshold, beyond which it will not be adjusted.
- **Output Path**: Set the output path for smooth point cloud files generated by the smoothing process.
- **Default**: Restore the setting parameters to the default values.
- **Parallel settings**: Whether to use the multi-core CPU parallel computing mechanism for smoothing.

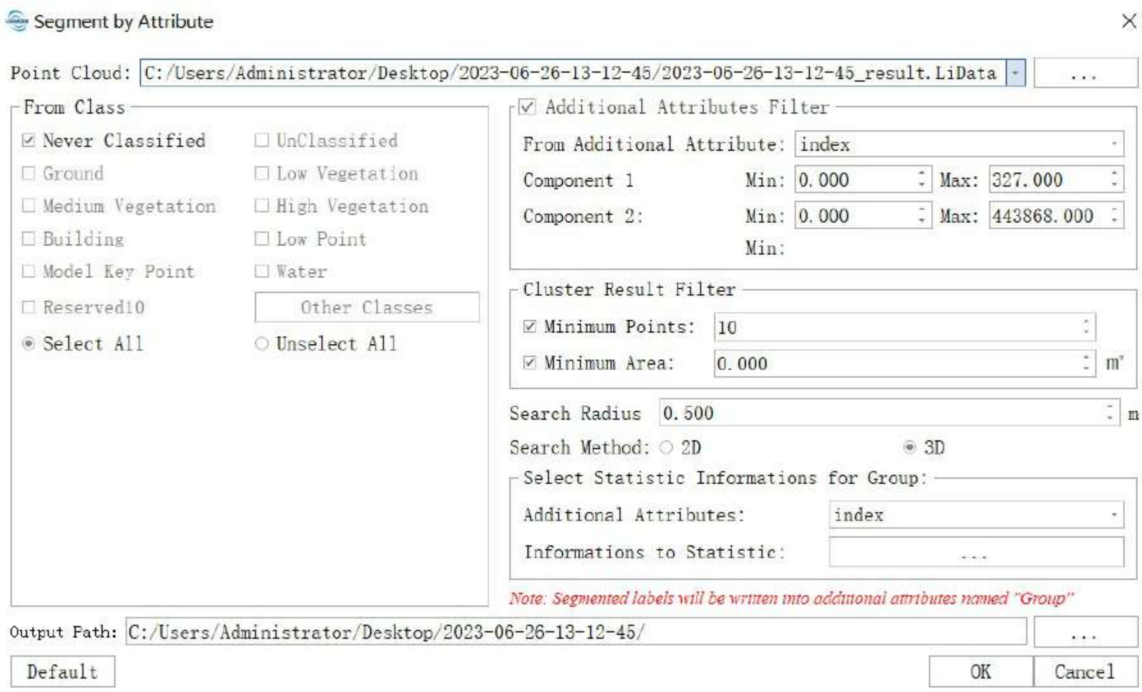
Segment by Attribute

Functional Overview

This function first extracts the point clouds of interest through attribute filtering and category filtering, then uses Euclidean clustering to segment the point clouds of interest into several clusters, further filters these clusters, and finally counts the information of these clusters and outputs a statistical report. This function does not generate new point cloud files but writes the labels of the clusters into an additional attribute named "Group" in the point cloud (this additional attribute is an integer single component that cannot be defined as other types or components). Point clouds with identical labels greater than 0 are considered as belonging to the same cluster, such as 1, 2...; -1 is an invalid label indicating that this point does not belong to any cluster. Users can further process the point clouds of interest through Extract by Attribute, Classify by Attribute or Extract by Group. This function generates a CSV format statistical report. Each row in this report represents a cluster, and users can freely choose which statistics information they want.

Usage

Click *Data Management > Point Cloud Tools > Cluster by Attribute*



Parameter Settings

- **Point Cloud** : The input file must be a single LiData point cloud data file. Users can select already opened LiDAR360 software's point cloud data or open other LiData files from disk.
- **From Class**: Initial categories are used to extract interested category points.
- **Additional Attributes Filter**: Used for extracting interested range within additional attributes. Refer to Extract by Additional Attributes. If not selected, this filtering will not be performed.
- **Cluster Result Filter**:

- **Minimum Point (default is "10")**: If checked, the following filter will be performed: if the number of points in a cluster is less than this value, the cluster will be classified as an invalid cluster and its label will be set to -1.
- **Minimum Area (default is "0")**: If checked, the following filter will be performed: if the projection area of a cluster is less than this value, it will be considered as an invalid cluster and its label set to -1.

- **Search Radius (default is "0.5")**: Euclidean clustering threshold. Point clouds with Euclidean distance smaller than this threshold are grouped into one cluster. The calculation of Euclidean distance depends on search method. The larger this value, the fewer clusters you get.
- **Search Method**: Dimension used for calculating Euclidean distances between point clouds during clustering process; currently supports 2D and 3D. When Search Method is “2D”, only horizontal direction's euclidean distance would be considered during clustering process.
  - **3D (Default)**: Calculation formula for Euclidean distance: $d = \sqrt{(x0-x1)^2+(y0-y1)^2+(z0-z1)^2}$
  - **2D**: Calculation formula for Euclidean distance: $d = \sqrt{(x0-x1)^2+(y0-y1)^2}$
- **Select Statistic Informations for Group (checked by default)**：If checked，add statistics about certain additional attributes in output CSV report。
  - **Additional Attributes** ：Select name of additional attribute，refer to Extract by Additional Attributes.
  - **Informations to Statistic** ：Choose one or more items from name, convex hull area, point cloud number, position, AABB bounding box, minimum value, maximum value, average value and standard deviation for statistics. Name，convex hull area，point cloud number，position and AABB bounding box are required.
- **Default**: Set parameters to default values.
- **Output Path**: Output path of CSV statistical report.

Vegetation Index

Functional Description

Vegetation index can reflect the background influences of plant canopy, such as soil, wet ground, snow, dead leaves, roughness, etc., and is related to vegetation cover.

Principle Description

Normalized difference vegetation index

The normalized difference vegetation index(NDVI) requires near-infrared (NIR) and red (R) attributes in point cloud data. The specific calculation method is as follows:

$$ND = \frac{NIR - R}{NIR + R} \qquad -1 \leq ND \leq 1$$

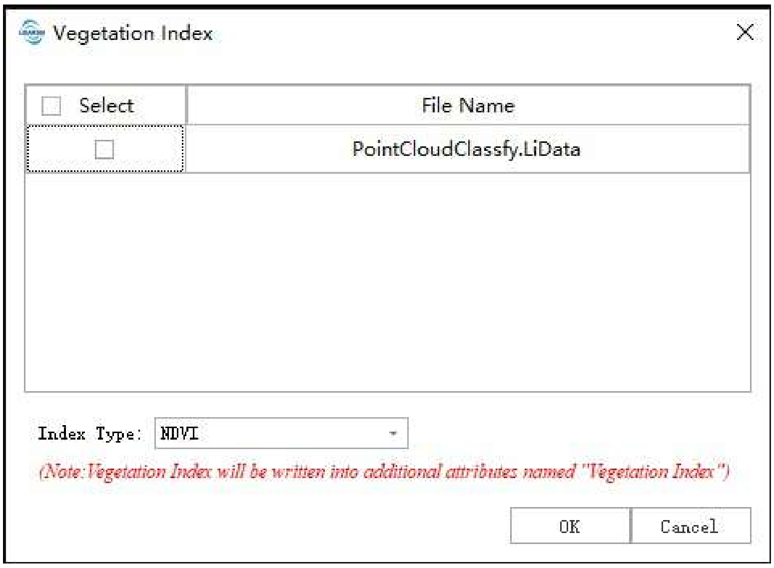
Visual band difference vegetation index

The visual band difference vegetation index requires that red (R), green (G) and blue (B) attributes exist in the point cloud data and are not all 0. The specific calculation method is as follows:

$$VBD = \frac{2 * G - R - B}{2 * G + R + B} \qquad -1 \leq VBD \leq 1$$

Usage

Click *Data Management* > *Point cloud tool* > *Vegetation index*, Set parameters click *OK* to generate Vegetation Index, vegetation index will be written into the point cloud additional attribute "Vegetation Index".



Parameters Settings

- **Input data**: The input file can be a single data file or a point cloud data set; the file(s) to be processed must be opened in the LiDAR360 software.
- **Index Type**:
  - **NDVI**: The vegetation index is calculated from near - infrared (NIR) and red light (R) attributes.
  - **Visual band difference**: The vegetation index is calculated according to the red (R), green (G) and blue (B) attributes.

> Note: If the point cloud does not have RGB attribute or both are 0, the two planting indexes cannot be calculated. Normalized vegetation index cannot be calculated if the point cloud does not have Near Infrared attribute.

Adjust Point Cloud Colors

Feature Description

By selecting the point cloud, determining the input color and output color, the colors of points that meet the similarity and category criteria can be modified.

Selection Tools

Polygon Selection

Draw a polygon by clicking the left mouse button, double-click to confirm the point cloud area to be modified.

Circular Selection

Click the left mouse button to determine the center of the circle, move the mouse to set the radius, double-click to confirm the point cloud area to be modified.

Rectangular Selection

Click the left mouse button to determine the starting point of the rectangle, move the mouse, and double-click to confirm the point cloud area to be modified.

Add Area

Confirm the point cloud area whose color needs to be modified.

Clear Area

Clear the point cloud area whose color needs to be modified, and redefine the range of the point cloud to be modified.

Input Color

Click on the point cloud with the mouse to get the color of the point cloud as the input color, or select the color through the color panel as the input color.

Input Color Similarity Threshold

The colors of points that are within the similarity threshold of the input color will be modified.

Output Color

Click on the point cloud with the mouse to get the color of the point cloud as the output color, or select the color through the color panel as the output color.

Undo

Undo the last color modification.

Redo

Redo the last color modification.

Category

Only the point cloud that meets the category criteria will have its color modified.

Preview

Change the colors of the point cloud within the area that meets the category and color threshold to the output color.

Save

Save the modified colors to a file.

Unit Conversion

Description

The Unit Conversion tool is used for converting measurement units of point clouds, vectors, models, image projects, rasters, TreeDB, and tables (csv files).

Usage

Click Data Management > Unit Conversion

Parameters

- : Users can add files to be converted from external sources.
- : Users can select a file from the list and click this button to remove it from the list.
- : Clicking this button will clear all data from the list.
- **Source Unit**: Select the measurement unit of the files to be converted.
- **Target Unit**: Select the measurement unit for the converted files.
- **Output Path**: Specify the folder path where the converted files will be saved. A new file with the converted units will be generated after the function is executed.

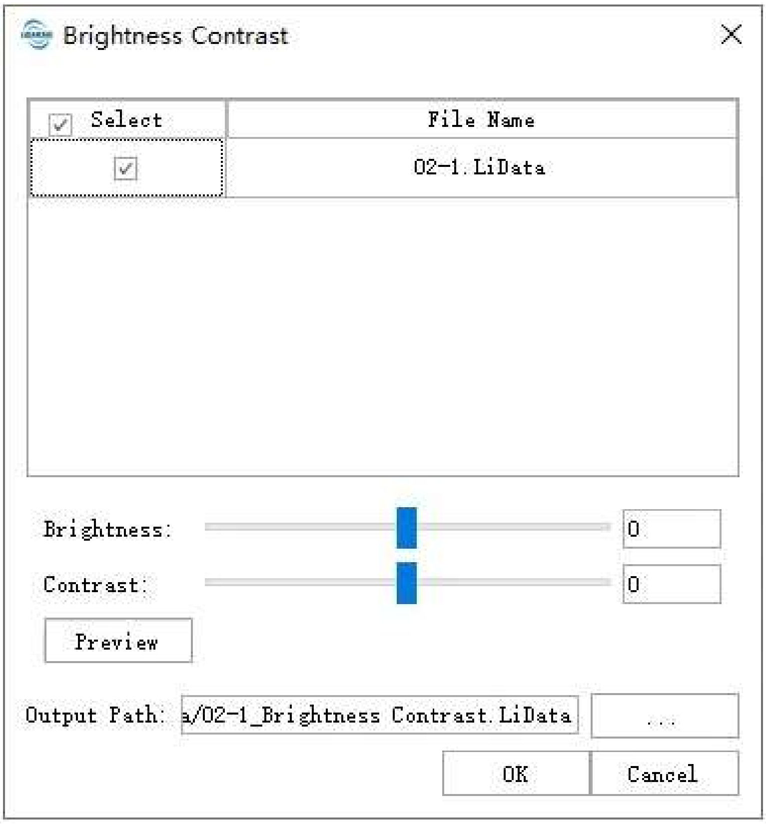
Brightness Contrast

Functional Overview

For point clouds with color information, adjust the brightness and contrast of the point cloud.

Usage

Click *Data Management* > *Point Cloud Tools* > *Brightness Contrast*. Adjust the two parameters until the preview effect meets expectations, then click "OK" to process.



Parameters Settings

- **Input Data**: The input file can be a single point cloud data file or a point cloud dataset. Only point clouds containing color information will be displayed in the list.
- **Brightness**: Adjust the brightness by sliding the bar or entering the values directly. The range is from -100 to +100.
- **Contrast**: Adjust the contrast by sliding the bar or entering the values directly. The range is from -50 to +50.
- **Preview**: Click the button after adjusting the parameters to preview the current effect in the window. Preview only affects the selected point cloud data.
- **Output Path**: The output path of the resulting point cloud file.

  > This function only applies to point clouds containing color information.

Raster Tools

Raster tools include **Band Calculation**, **Raster Mosaic**, and **Raster Subdivision**.

- Raster Mosaic
- Raster Subdivision
- Raster Calculator

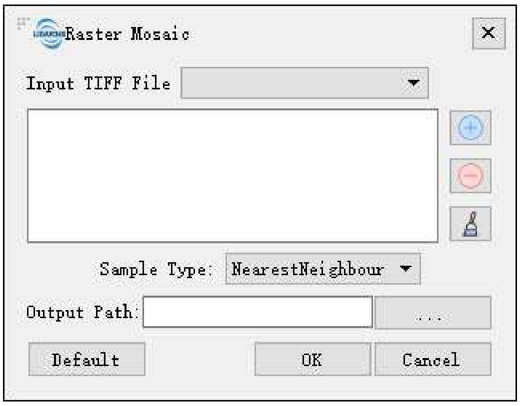
Raster Mosaic

Summary

Raster Mosaic refers to the technical process of merging two or more images to one image. LiDAR360 provides seven sampling methods: **Nearest Neighbour**, **Bilinear**, **Cubic**, **CubicSpline**, **Lanczos**, **Average**, and **Mode**. It is the reverse operation of Raster Subdivision.

Usage

Click *Data Management* > *Raster Tools* > *Raster Mosaic*



Settings

- **Input Tiff File**: Users can select multiple files (>1) to be processed from the drop-down list.
- : Users can add multiple external images for image mosaic.
- : Users can select a file in the list and click this button to remove the file from the list.
- : Click this button to clear all images in the list.
- **Sample Type**: Users need to select the sampling type from the drop-down list.
  - **NearestNeighbour (default)**: Nearest neighbor, sampled from the nearest neighbor.
  - **Bilinear**: Bilinear sampling (2 x 2 cores).
  - **Cubic**: Cubic convolution approximation (4 x 4 kernels).
  - **CubicSpline**: Cubic B-spline approximation (4×4 kernel).
  - **Lanczos**: Lanczos Window Sine Interpolation (6×6 Cores). Lanczos can be used as a low pass filter or to smoothly interpolate the value of a digital signal between its samples.
  - **Average**: Calculate the average of all non-value pixels.
  - **Mode**: Select the most frequently occurring value for all sampling points.
- **Output path**: Path of the output folder. After the function is executed, a new file will be generated.

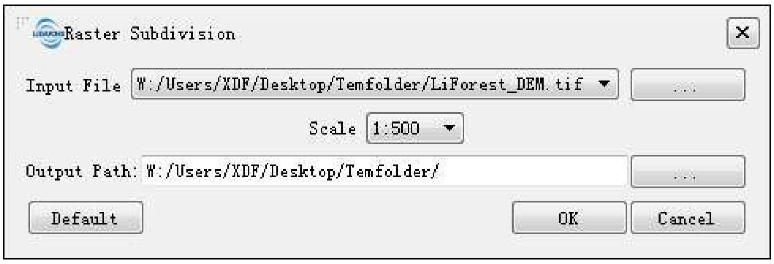
Raster Subdivision

Summary

The raster subdivision tool is the inverse operation of Raster Mosaic. The frame number adapts the coordinate number and consists of the southwest corner of the frame (Y coordinate + X coordinate). A scale of 1:5000 is taken into 1 km, 1:2000 and 1:1000 to 0.1 km, and 1:500 to 0.01 km.

Usage

Click *Data Management* > *Raster Tools* > *Raster Subdivision*



Settings

- **Input File**: Users can select the file to be processed from the drop-down list.
- : Users can add external files that need to be processed.
- **Scale (default "1:500")**: Users can select a targeted scale. LiDAR360 has four scale options: 1:500, 1:1000, 1:2000, and 1:5000.
- **Output path**: Path of the output folder. After the function is executed, new subdivided files will be generated.

Raster Calculator

This tool is a spatial analysis tool. User can input raster data algebraic expressions, use operators and functions to do mathematical calculations, in order to establish selection queries, or enter raster data algebraic syntax.

Advantages of the raster calculator tool:

1. Supports the use of variables in raster data.
2. The ability to apply operators to three or more inputs of an expression.
3. The raster calculator is used to execute single-line algebraic expressions through a simple, calculator-like tool interface, using a variety of tools and operators. When multiple tools or operators are used in an expression, the speed of execution is generally faster than executing each operator or tool separately.

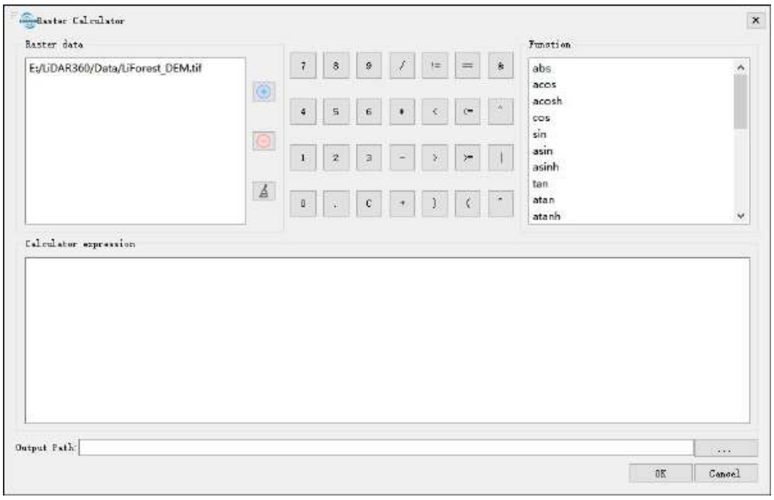
Principle Description

Use the calculator buttons to enter values in expressions. Use the operator buttons to enter mathematical operators (addition, division, etc.) and logical operators (greater than, equal to, etc.) into expressions. By clicking these buttons, user can enter a number or operator into the current position of the pointer in the expression. The following are some examples of expressions that can be executed in the raster calculator tool. In these expressions, the raster layer name is enclosed in quotation marks, such as "dist".

```
("pop" > 150) & ("dist" > 10)
(("reclass_rd_dist" * 3) + ("reclass_landuse" * 2) + "reclass_elev") /6.0
("Band4" - "Band3") / Float("Band4" + "Band3")
Con("elev" <= 3000, 1, 0)
Con(IsNull("elev"),0, "elev")
Con(("landuse1" == 1) & ("landuse2" == 5), "landuse1" + "landuse2", 99)
Con(Raster('elev') != 0,'elev')
(Con('elev', 'elev', "", "elev_feet <> 0")) + Raster("tree_height")
Con("inRas" < 45,1, Con(("inRas" >= 45) & ("inRas" < 47),2, Con(("inRas" >= 47)&("inRas" < 49),3, Con("inRas"
 >= 49,4))))
```

Usage

Click *Data Management* > *Raster Tools* > *Raster Calculator*.



Settings

- **Raster Data**：Display the raster data path (.tif) contained in the main interface, user can import the local raster data, perform expression calculations, and remove or clear the imported data.
- **Calculator Buttons**：Contains number buttions 1, 2, 9, 0, etc. Algebraic operator buttions +, -, *, /, etc. Logical operators >, <, ==, etc. The major operation and logical buttons are as follows:

/ Divided by	!= Not equal to	== Equal to	& AND
X Multiplied by	< Less than	<= less="" than="" or="" equal<="" td="">	| OR
- Minus	> Greater than	>= Greater than or equal	^ Exclusive OR
+ Plus	~ NOT	C Clear expression	

- **Function**：Lists the functions of abs, tan, cos, log, etc.

Function	Description
abs	Absolute value function
acos	Arc-cosine function
acosh	Hyperbolic arc-cosine fuction
cosh	Hyperbolic cosine function
asin	Arc-sine function
asinh	Hyperbolic arc-sine function
sin	Sine function
tan	Tangent function
atanh	Hyperbolic arc-tangent function
atan	Arc-tangent function
cot	Cotangent function
pow	Exponent power function
log	Logarithm function
sqrt	Square root function

sinc	Sinc function

- **Calculator Expression**：Use the calculator button to enter values in the expression, or use operators, functions, etc. to form a raster calculation expression with raster data, and display it in this panel.
- **Output Path**：Select the output directory or .tif file path. This output raster represents the result value of the calculation expression.

Vector Tools

- Smooth Line

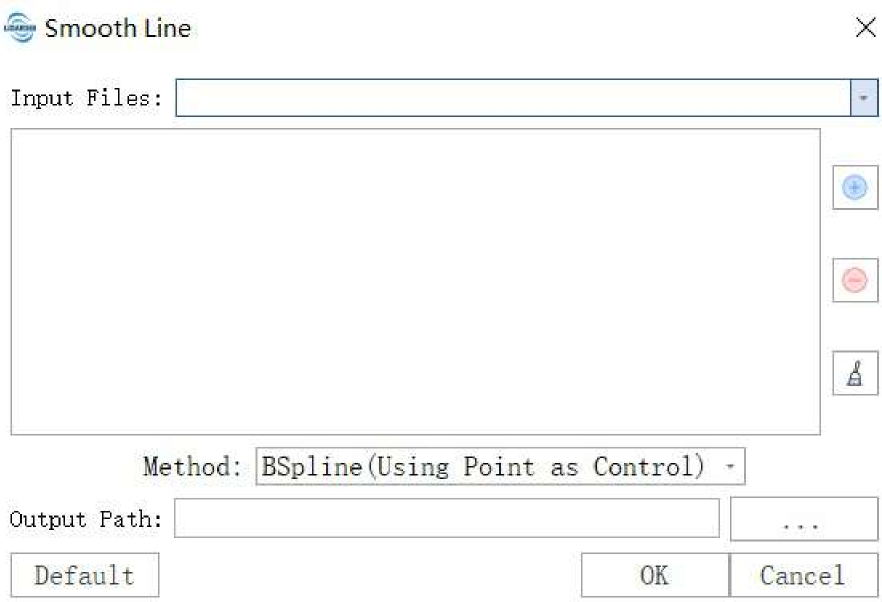
Smooth Line

Functional Overview

The Line Smoothing tool can smooth the input vector lines. It supports 2D and 3D polylines and polygons.

Usage

Click *Data Management* > *Vector Tools* > *Smooth Line*

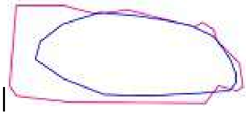


Parameters Settings

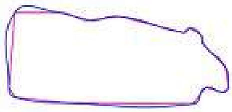
- **Input Files List**: Input one or more vector files. Input file format: *.gpkg,* .shp, *.dxf.
- **Method**: Multiple methods are available to smooth the vector lines.
- **Mean smoothing (3\5\7 neighborhood)**: Use a 3\5\7 neighborhood for mean smoothing on each point of the vector line.
- **Bezier (Using Line as control)**: Smooth using segmented Bezier curves, generating a new vector line without control points.
- **Bezier (Using Point as control)**: Smooth using segmented Bezier curves, generating a new vector line with control points.
- **B-Spline (Using Point as control)**: Smooth using cubic B-spline curves, generating a new vector line passing through some of the control points.
- **B-Spline Fit**: Fit using cubic B-spline curves, generating a new vector line without any control points.
- **Output Path**: Output folder path. The newly generated vector lines will be output to this folder.

|Method|Effect(The red line is the original polyline while blue is after being smoothed)|

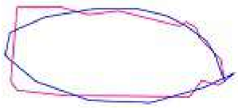
|:----:|:----:|

|Mean Smoothing(3\5\7 neighborhood)|  |

|Bezier Smoothing(line control)|  |

|Bezier Smoothing (point control)|  |

|B-Spline (point control)|  |

|B-Spline Fit|  |

Tile

The Tile Tool Includes Tile by Range, Tile by Point Number, Tile by Draw Line, and Tile by Polygon.

- Tile by Range
- Tile by Point Number
- Tile by Draw Line
- Tile by Polygon

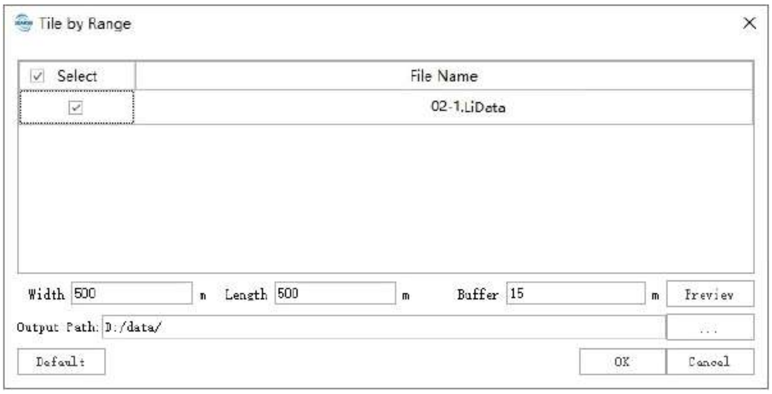
Tile by Range

Overview

Tile by Range divides the point cloud into a series of small data files based on the user-defined width ,length and buffer size of the small data file. This process begins from the lower-left corner of the input file's bounding box,the result includes the point cloud blocks data(including buffer) and the blocks boundary stored in the shp file(not including buffer). The output of this function is similar to Tile by Point Number.

Usage

Click *Data Management > Point Cloud Tools > Tile by Range*



After completing the parameter settings, click the Preview button to check the subdivision grid for each point cloud. The white solid lines indicate the block boundaries, while the red dashed lines represent the buffer zone.

Settings

- **Input Data**: The input file can be a single point cloud data file or multiple data files. File Format: *.LiData.
- **Width (m)(default value is "500")**: The width of the data block size, which is the length in the X-axis direction.
- **Height (m)(default value is "500")**: The length of the data block size, which is the length in the Y-axis direction.
- **Buffer (m)(default value is "0")**: The extend size of every block in all directions.
- **Output path**: Path of the output folder. After the function being executed, new files will be generated which include the point cloud blocks data(including buffer) and the blocks boundary stored in the shp file(not including buffer).

Tile by Point Number

Summary

Tile by Point Number divides the point cloud into a series of small data files with a total of user-defined number of points. This process begins from the lower-left corner of the input file's bounding box. The output of this function is similar to Tile by Range.

Principle

The **Point Number** entered by the user will be recalculated according to the actual number of blocks in the actual chunking process. The formula is as follows:

$$N_{block} = \begin{cases} N_s/N_u, & N_s\%N_u = 0 \\ N_s/N_u + 1, & N_s\%N_u > 0 \end{cases}$$

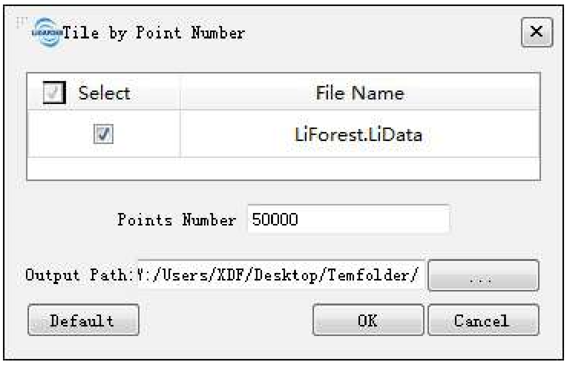
$$N_{real} = \begin{cases} N_s/N_{block} \\ N_s/N_{block} + 1 \end{cases}$$

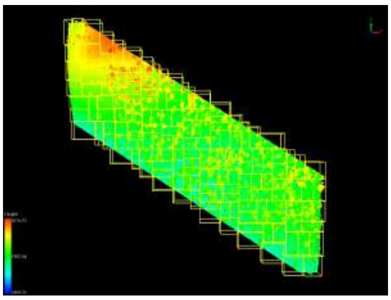
Among them, N_{block} is the number of tiles, N_{real} is the actual point number of a tile, N_s is the total number of points in the original file, and N_u is the user-defined point number.

Original point cloud data and point cloud data after chunking by point (different yellow bounding boxes represent different point cloud data).

Usage

Click *Data Management* > *Point Cloud Tools* > *Tile by Point Number*





Settings

- **Input Data**: The input file can be a single point cloud data file or multiple data files. File Format: *.LiData.
- **Point Number (default value is "50000")**: Set the number of points in each tile. Note that the actual number of points in a tile may be slightly different from the user-defined number.
- **Output path**: Path of the output folder. After the function being executed, new files will be generated.

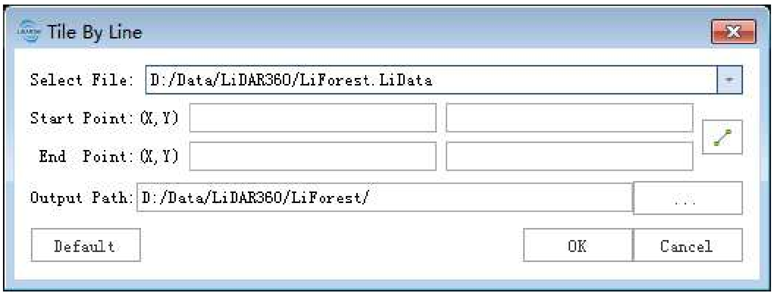
Tile by Draw Line

Summary

Draw a straight line on the Viewer and divide the point cloud data into two pieces by using this line as the boundary.

Usage

Click*Data Management* > *Point Cloud Tools* > *Tile by Line*



Enter the coordinates for the start and end points, or update the start and end points by drawing a line on the Viewer after clicking the buttons on the right side of the dialog box.

Settings

- **Input Data**：The input file should be a single point cloud data file. File Format: *.LiData.
- **Initial Point**（**X,Y**）：The XY of initial point.
- **Termination Point**（**X,Y**）：The XY of termination point.
- **Out Path**：Path of the output folder，After the function is executed, the divided point cloud file and the shp file of the block boundary are generated. The block boundary stored in the shp file does not include the size of the buffer.

Tile by Polygon

Summary

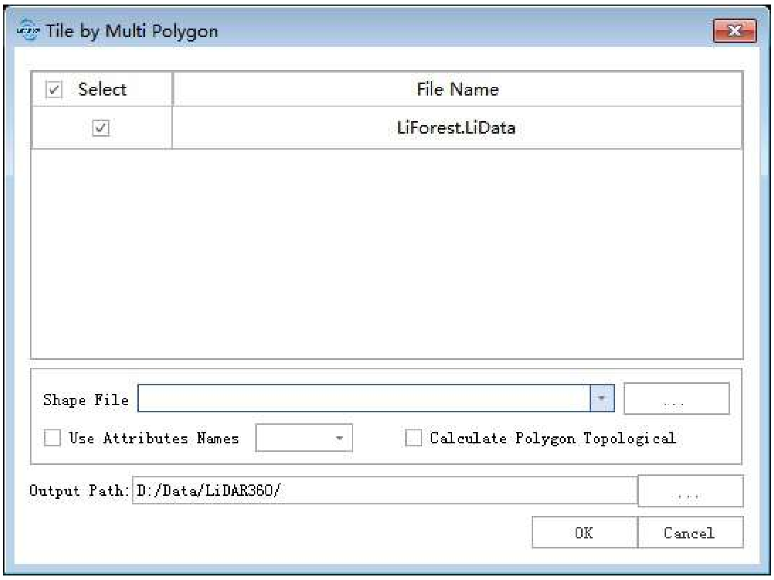
Read the polygon information in the vector data by the polygon block function, extract the points inside the polygon, the points outside the polygon or the points between the polygons to generate a new point cloud file.

For Polygon objects in vector data, a new file is generated for the points within each polygon, and a point cloud file is generated for the points that are not in any polygon.

For MultiPolygon objects in vector data, the point cloud file of the inner polygon will be generated as point cloud data, the set of points between the inner polygon and the outer polygon will be generated as point cloud data, and all points outside the Polygon will be generated as point cloud files.

Usage

Click*Data Management > Point Cloud Tools > Tile by Polygon*



Settings

- **Input Point Cloud Data**： The input file can be a single point cloud data file or multiple data files. File Format:*.LiData.
- **Vector File**: User can select the Vector File loaded into LiDAR360 software from the drop-down menu, or select  button to load an external vector data file.
- **Named With The Attribute Name of The shp File**： Select the attribute field of Polygon and name the newly generated block data.
- **Calculate the topological relationship of polygons**: For the Polygon object in the vector data, calculate the topological relationship between polygons. If checked, the topological relationship between polygons will be calculated; if not checked, all points in the Polygon will be generated into a new file.

- **Output Path**：Path of the output folder, the point cloud file after the function is executed will be output under this file path.

Projections and Transformations

Different kinds of transforms for point clouds, e.g. reprojection, elevation adjustment, are provided. The calculation of parameters necessary for transforms, e.g. seven-parameter solution, four-parameter solution, are provided as well.

- Define Projection
- Reproject
- Transformation
- Convert Projection Surface
- Elevation Adjustment
- Transformations Calculation
- Fitting Geoid Model
- ICP Registration
- Convert ASCII to BLH
- Manual Registration
- Manual Rotation and Translation
- The Geoid Model
- Create Transformations

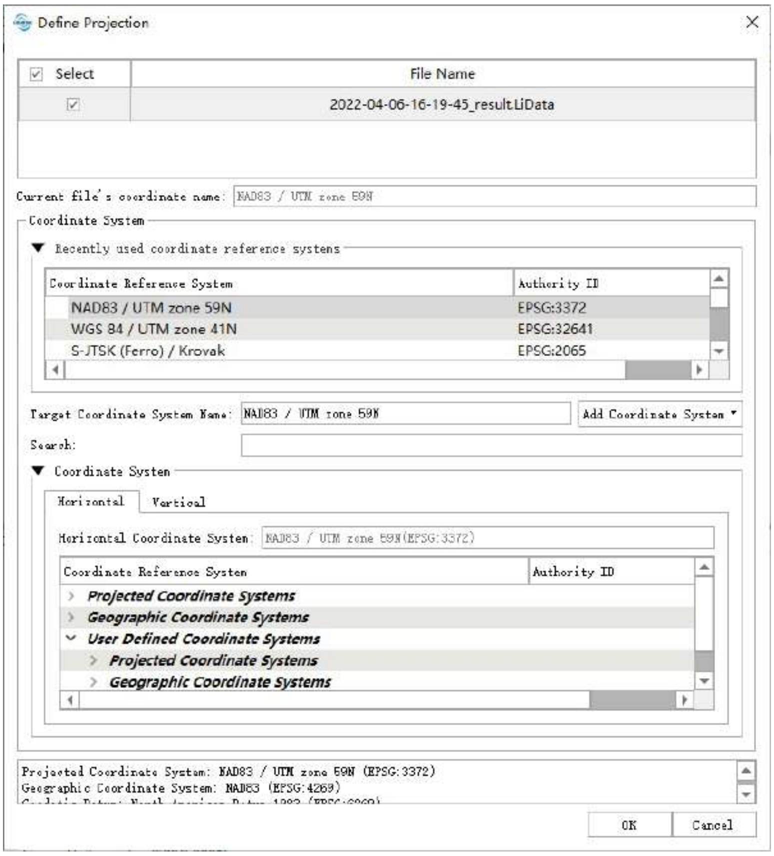
Define Projection

Summary

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

Usage

Click *Data Management* > *Projections and Transformations* > *Define Projection*



Settings

- **Select Data**: Input one or more point cloud data files. File format: *.LiData.
- **Current File's Coordinate name**: Displays the name of the coordinate system defined for the current point cloud.
- **Target Coordinate System**: Displays the name of the currently set coordinate system, allowing for custom coordinate system names.
- **Detail**: Displays detailed information about the currently set coordinate system.

- **Add**: Users can add coordinate systems from external sources or create custom coordinate systems. LiDAR360 software provides three methods to add external coordinate systems:
    - **Add Geographic Coordinate System**
    - **Add Projected Coordinate System**
    - **Add Vertical Coordinate System**
- **Filter**: Users can input the defined coordinate system by entering keywords related to the coordinate system, corresponding systems can be filtered from the World Coordinate Systems list (e.g., to set the point cloud coordinate system to WGS 84 / UTM Zone 49N, you can enter "UTM 49N" in the filter option for quick selection, or enter its EPSG code: 32649 for rapid searching). Alternatively, you can click the **Add** button to import coordinate systems from external sources.

- **Set Coordinate System**: Select the coordinate system to be set, which can be selected from the default coordinate system or a custom coordinate system. First, select the horizontal coordinate system on the **Horizontal** tab. At this time, the vertical coordinate system defaults to the height of the ellipsoid. If you need to define a vertical coordinate system, you can switch to the **Vertical** tab to select a vertical coordinate system, and the two together form a combined coordinate system.The coordinate system selection interface includes the following sections:

- **Recent**: Records the coordinate systems recently used.

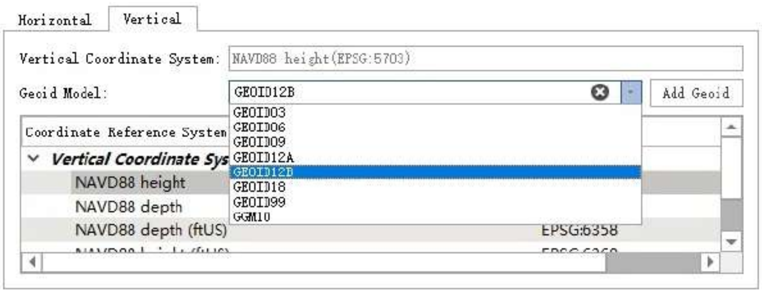
- **Favourites**: Shows coordinate systems added to favorites.
- **Projected Coordinate Systems**: Includes all default projected coordinate systems.
- **Geographic Coordinate Systems**: Includes all default geographic coordinate systems.
- **User Defined Coordinate Systems**: Includes all custom coordinate systems.
- **Layers**: Includes coordinate systems from loaded data.
- **Vertical Coordinate Systems**: Includes all default vertical coordinate systems.
- **User Defined Vertical Coordinate Systems**: Includes all custom vertical coordinate systems.
- **Layers**: Includes coordinate systems from imported data, grouped by coordinate system name.

> Right-click any coordinate system and select the **Add to Favourites** option from the context menu to add the selected coordinate system to your favorites. To remove a coordinate system from favorites, right-click the favorite system and choose **Remove from Favourites**.
>
> Delete: Right-click on any custom coordinate system and select the **Delete** option to remove it. If the deleted custom vertical coordinate system is used in any custom combined coordinate systems, you will be prompted to decide whether to delete the combined coordinate systems as well. Choosing "Yes" will delete all associated coordinate systems, while "No" will leave the combined coordinate systems unaffected.
>
> Copy and Modify: Right-click on any coordinate system and select the **Copy and Modify** option to modify the coordinate system parameters and save it as a new custom coordinate system. This action will pre-fill the modification interface with the parameters of the currently selected coordinate system. For detailed settings, refer to the section on adding coordinate systems.

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

- When this option is not set, if the vertical datum grid data is provided, the appropriate geoid model will be automatically searched and used during coordinate transformation. If not provided, the geoid model will not be used during coordinate transformation (except for the EGM2008 model included in the software installation package).
- If the selected vertical coordinate system can use the geoid model supported by default, it can be selected from the drop-down list, and the model will be specified when performing coordinate transformation. For the geoid models currently supported by default in LiDAR360 software, please refer to the Geoid Model page.
- If the geoid model to be used is not included in the package provided by PROJ, the user can place the model file to be used under the geoid folder in the software installation directory, and then add the geoid model by clicking the **Add Geoid** button to add the file (support tiff and gtx formats), the model will be added to the drop-down list, after selecting the added model, the custom geoid model will be used in the coordinate transformation process.

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

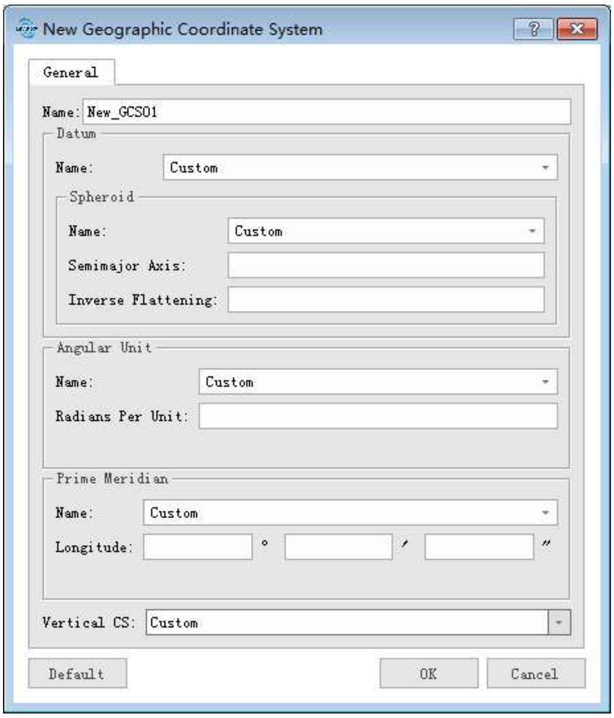


- **Selected Coordinate System**: The information of the coordinate system selected by the user will be displayed here.
- After clicking **OK**, the WKT information representing the projection will be written to LiData.

Vertical Coordinate System: Users can customize the vertical coordinate system. Vertical coordinate system transformations and grid projections require vertical datum grid data for valid transformations. EGM2008 geoid model elevation conversion is provided by default in the software. If you need to support other regional geoid models, you need to download the corresponding grid data files. The grid files are provided by the official PROJ library, which can be downloaded here proj-data-1.8, or on the official website https ://proj.org/download.html to download the latest version of proj-data. After downloading, unzip the file to the geoid folder in the software installation directory, the default is C:\Program Files\GreenValley Suite\LiDAR360\5.0.0.0\geoid, "5.0.0.0" is the version number, and there will be differences between different versions.

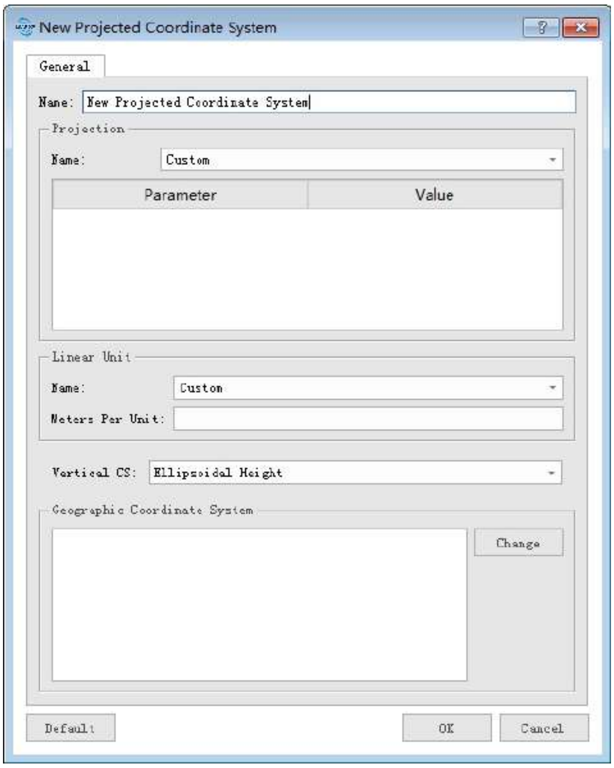
> Note: The custom geoid model must be placed in the geoid folder to take effect.

Add Geographic Coordinate System

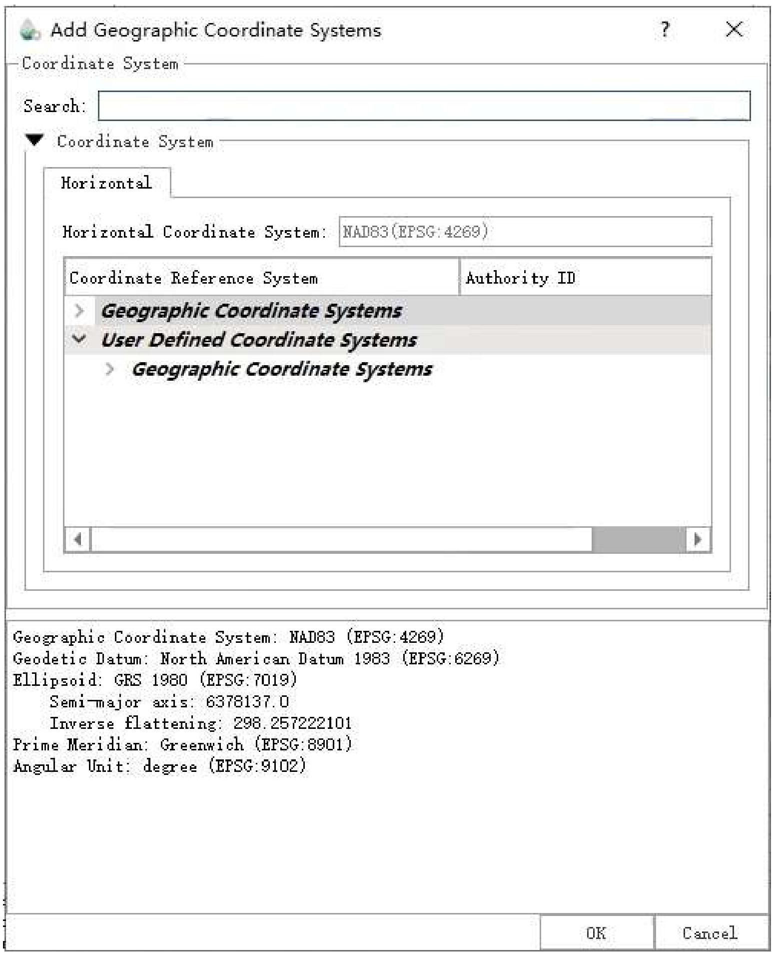


- **Name**: Users can customize the name of a geographic coordinate system .
- **Datum**:
  - **Name**: Users can customize a datum name or select a known datum from the drop-down list.
  - **Spheroid**: If a user selects the datum from the drop-down list, the ellipsoid parameter does not require to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

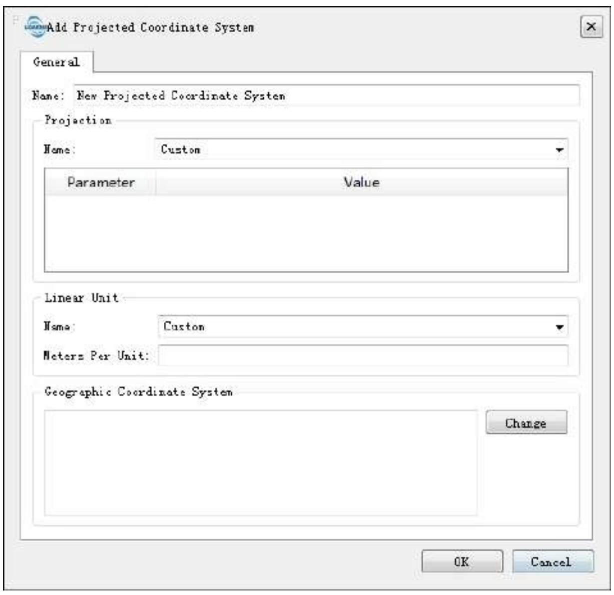
Add Projected Coordinate System



- **Name**: Users can customize the name of a projection coordinate system.
- **Projection**:
    - **Name**: Users can customize a projection name or select a known projection from the drop-down list. It should be noted that user-defined projections can be modified from a projection template selected from the drop-down list.
- **Linear Unit**:
    - **Name**: Users can customize the linear unit name or select the linear unit name from the drop-down list.
    - **Meters per Unit**: Input meters per unit.
    - **Geographic Coordinate System**: User can click Modify to select a geographic coordinate system.
    - **Change**: When the user clicks the Change button, the following dialog box will pop up to modify the geographic coordinate system. Users can filter out the desired coordinate system through the Filter operation. Supports selection of default geographic coordinate system and custom geographic coordinate system.



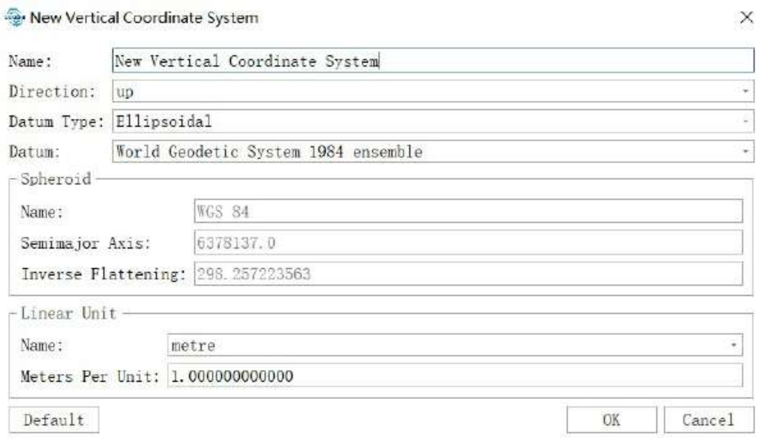
Add Vertical Coordinate System



- **Name**: Users can customize the name of the projection coordinate system.
- **Direction**: Set the direction of the coordinate axis, either upward or downward.
- **Datum Type**: Set the datum type of the vertical coordinate system. "Ellipsoidal" is for ellipsoidal datum, and "Gravity-related" is for gravity field datum.
- **Datum**: When the datum type is set to Ellipsoidal, the ellipsoidal datum can be set through this option. Once selected, the ellipsoid parameters will be automatically filled into the ellipsoid parameter list

below.

- **Vertical Datum**: When the datum type is set to Gravity-related, the vertical datum can be set through this option.
- **Linear Unit**:
  - **Name**: Users can customize the name of the linear unit or select a name from the drop-down list.
  - **Meters per Unit**: Input the number of meters per unit.
- **Spheroid**: Users can click to modify and select the geographic coordinate system.
- **Default**: Restore default parameters.



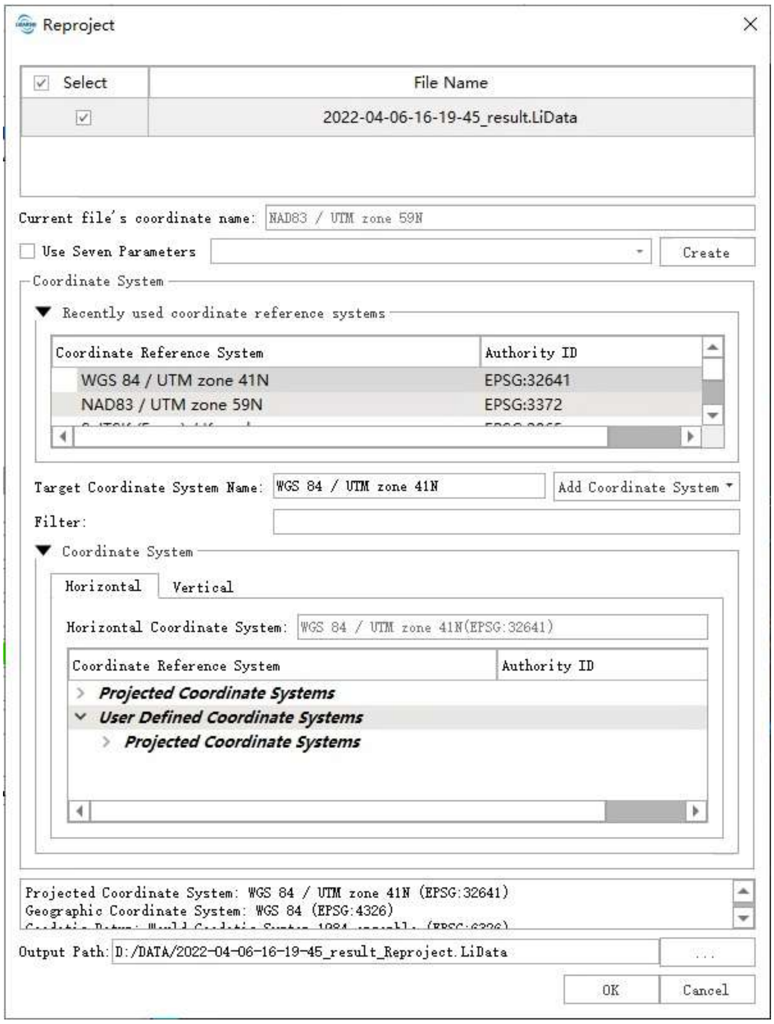
Reproject

Function Description

The point cloud can be reprojected among geographic coordinate systems and projected coordinate systems. When different geographic coordinate systems are converted to each other, LiDAR360 provides the seven parameter transformation model to adjust the differences between ellipsoids and reference planes.

Usage

Click *Data Management* > *Projections and Transformations* > *Reproject*



Settings

- **Select Input File**: The input file can be a single point cloud data file or multiple data files. File Format: *.LiData.
- **Current File's Coordinate Name**: Display the coordinate system name of the current point cloud.
- **Use Seven Parameters**: Users can choose whether to use seven parameters for re-projection. If

checked, seven parameters will be used; vice versa.

- **Seven Parameters Setting**: Select the **Use Seven Parameters** option and click the **Create** button to create a seven-parameter conversion relationship. Please refer to Create Transformations.
- **Filtering**: Users need to enter a customized coordinate system. By entering the coordinate system keywords, the corresponding coordinate system can be filtered from the **Coordinate reference systems of the world** table (for example: to set the point cloud coordinate system to WGS 84 / UTM Zone 49N, users can enter UTM 49N in the filter for fast screening, or enter its EPSG number 32649 for quick search.) Users can also import external coordinate system by clicking the **Add Coordinate** System button.
- **Add coordinate systems**: For details, refer to Define Projection parameter settings.
- **Output path**: Path of the output file. New files will be generated after the function being executed.

> Note: The reprojected point cloud must already have projection information. To check whether the current point cloud has projection information, users can select the data in the data list and its projection information will be displayed in the current file coordinate system

Transformation

Function Description

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

Usage

Click *Data Management> Projection and Coordinate Conversion> Coordinate Conversion*

Usage

1. **Input Data**： The input file can be a single point cloud data file or multiple data files. File Format: *.LiData.
2. Select or customize **transformation method**: available for reference, Create Transformtions.
3. **Output Path**：Path of the output folder. After the algorithm being executed, new files after coordinate conversion will be generated.

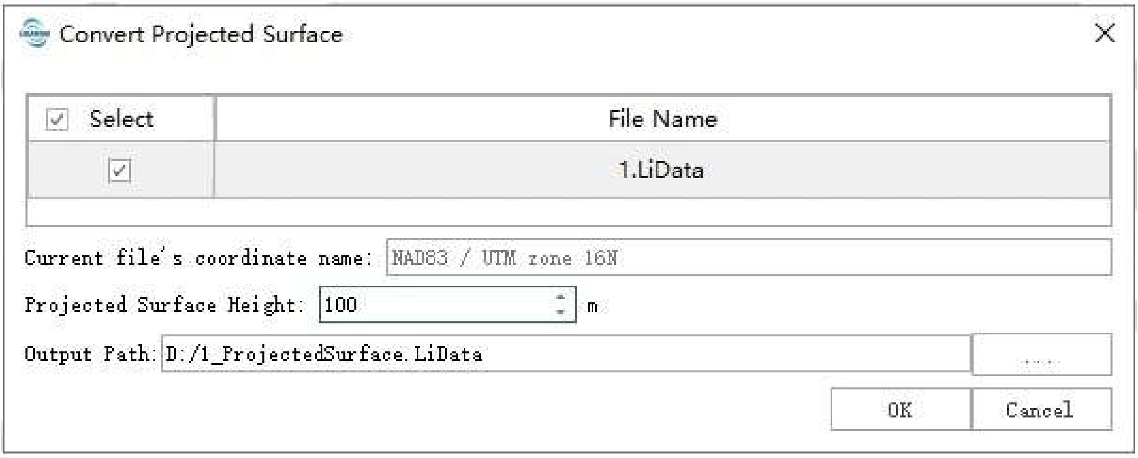
Convert Projection Surface

Functional Overview

For Gauss projection and UTM projection, the length deformation value will increase when the projection area is far away from the central longitude of the projection belt. Convert Projection Surface function **can be used to reduce length deformation**.

Usage

Click *Data Management* > *Projection and Coordinate Conversion* > *Convert Projection Surface*



Steps

1. **Input point cloud data**: The input file with projection information can be a single point cloud data file or multiple data files. File format: *.LiData. Click the point cloud file to display the coordinate system of the point cloud on the interface.

   > Only point cloud files whose horizontal coordinate system is Gauss projection or UTM projection are supported.

2. Set **projection surface height**: Set the elevation of the projection. The positive value is the distance above the ellipsoid and the negative value is the distance below the ellipsoid, the unit is in meters.

3. **Output Path**: Output folder path, after the algorithm is executed to generate a new file after coordinate conversion.

Principle

1. Calculate the radius of curvature of the earth Ra where the point cloud is located. Where a is the semi-major axis of the ellipsoid, e is the eccentricity of the ellipsoid, and B is the average latitude of the measurement area of the point cloud.

$$N = \frac{a}{(1 - e^2 sin^2 B)^{1/2}}$$
$$M = \frac{a(1 - e^2)}{(1 - e^2 sin^2 B)^{3/2}}$$
$$R_a = \sqrt{MN} = \frac{a\sqrt{(1 - e^2)}}{1 - e^2 \frac{1 - cos(2B)}{2}}$$

2. Calculate the scale factor q. In the formula, Hb is set as the elevation parameter of the projection.

$$q = \frac{H_b}{R_a}$$

3. Calculate the modified coordinates, where Xc and Yc are the modified coordinates, X and Y are the original coordinates, and X0 and Y0 are the projection center coordinates of the coordinate system.

$$x_c = x + q(x - x_0)$$
$$y_c = y + q(y - y_0)$$

Elevation Adjustment

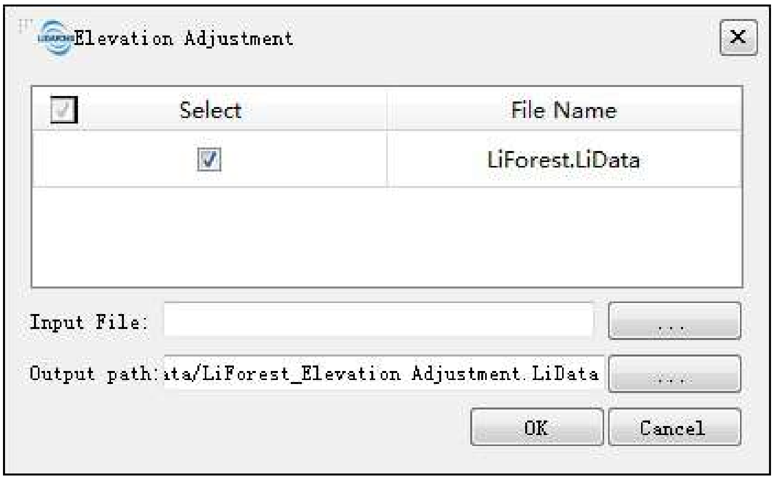
Summary

The elevation of raw laser data is normally represented by ellipsoidal height. Normally these values need to be transformed to values in regional height system or local height system. For large area, the adjustment of elevation can not be defined as a mathematical formula. Hence, the model of elevation adjustment is required to be defined. The algorithm is to build triangular network model using known control point data as well as to interpolate and correct local points using height anomaly between different height systems. Adjusted elevation result can be checked according to the Control Point Report.

It is recommended to use control point data containing entire project region to adjust model in order to provide more accurate elevation information to the project border.

Usage

Click *Data Management* > *Projections and Transformations* > *Elevation Adjustment*



Setting

- **Input Data**: Input files can be single point cloud file or point cloud data set.
- **Input File**: Users need to input adjustment model file of control points. This file data can be generated by tool of Control Point Report.
- **Output path**: Output adjusted point cloud data.

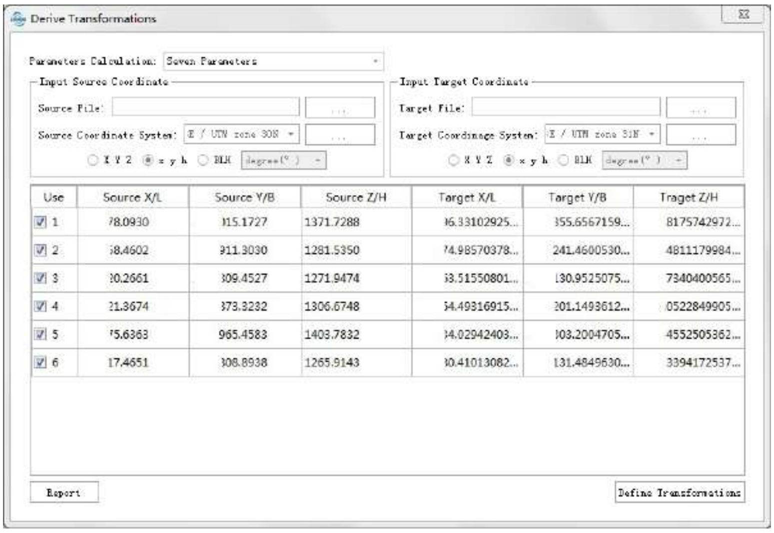
Conversion relationship calculation

Function Description

This function calculates the required conversion relationship based on the control point pair, and can save the calculated conversion relationship to the history file for reuse. Currently supports four-parameter and seven-parameter solutions.

Usage

Click *Data Management> Projection and Coordinate Conversion> Conversion Relationship Calculation*



Parameter Settings

- **Parameter Solution**: The method of parameter calculation supports **four parameters** and **seven parameters**.
- **Source/target coordinate file**: Input the control point source coordinate file, the format can be referred to Seven Parameter format.
- **Source/target coordinate system**: When the coordinate form is **x y h / BLH**, the coordinate system can be selected, please refer to Define Projection.
- **Source/target coordinate form**:
  - **X Y Z (Earth-centered coordinate system)**: Select this item when the input data is in the projected coordinate system. At this time, the "source/target coordinate system" cannot be selected. Directly calculate the seven-parameter transformation between the source coordinate and the target coordinate. If the user selects **four parameters**, it will be forced to switch to **X Y Z**
  - **X Y H / BLH**: Select this item when the input data is in the projection/geographical coordinate system. At this time, you need to select the "source/target coordinate system". After the selection is completed, it will automatically switch once according to the selected coordinate system. When selecting BLH, you can select degrees or degrees: minutes: seconds.
    - **x y h**: The "source/target coordinates" will be projected to geographic coordinates according to the projected coordinate system selected by the user, and then the geographic

coordinates will be transferred to the geocentric coordinate system according to the ellipsoid information in the projected coordinate system, and finally the seven parameters between the two geocentric coordinate systems will be solved.
    - **BLH**: The user needs to select "Geographic Coordinate System", the program transfers the coordinates to the geocentric coordinate system according to the ellipsoid information in the geographic coordinate system, and finally solves the seven parameters between the two geocentric coordinate systems.
- **Report**：Calculate the conversion relationship and generate a report, which contains the calculated conversion relationship parameters and the back-projection report of various control points. The report can be saved as HTML and ASCII format.
- **Define conversion relationship**：Please refer to Create Transformation.

Seven-Parameter Solution

Using Bursa model to calculate the seven parameters for the transformation among different coordinate systems according to three or more pairs of tie-points. The parameters include three translation parameters, dx, dy, dz, three rotation parameters, rx, ry, rz, and a scaling factor, m. The calculation formula is shown below. Bursa seven parameters formula can be written as follow:

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

Convert the above formula to a equation set, which is：

$$\begin{cases} X_B = dx + mX_A - ryZ_A + rzY_A \\ Y_B = dy + mY_A + rxZ_A - rzX_A \\ Z_B = dz + mZ_A - rxY_A + ryX_A \end{cases}$$

In the equation set, there are seven unknown parameters. If there are three or more than pairs of tie-points, these seven parameters can be solved through the above equation set using the least square method.

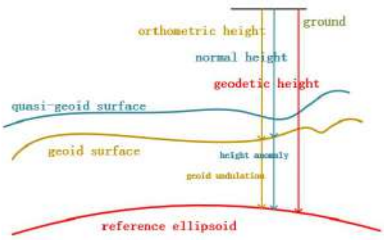
Four-Parameter Solution

According to two or more control points, users can calculate the four parameters that used to perform coordinate system transformation. The four parameters include two translation parameters, dx and dy, a rotation parameter, T, and a scaling factor, K.

Fitting Geoid Model

Functional Overview

There are mainly three elevation datums involved in surveying: the physical surface of the earth - the geoidal surface, the mathematical surface of the earth - the reference ellipsoid, and an abstract geoid surface. These three elevation datums correspond to three commonly used elevation systems, namely: orthogonal, geodetic elevation and quasi-geodetic elevation system (normal elevation system). The relationship between these elevation datums is shown in the following figure. The elevation system currently used in China is the normal elevation system, while the GPS elevation is usually the geodetic elevation system. Since the reference ellipsoid does not coincide with the geoid, there is an elevation anomaly between the geodetic height and the normal height. The so-called GPS height fitting is the process of obtaining the abnormal height and then obtaining the normal height to establish a geoid surface. Analytical interpolation is the most commonly used method for elevation fitting. The main idea is to approximate the quasi-geoid with a curved surface to establish the closest quasi-geoid in the survey area, so as to calculate the elevation anomaly of any point in the survey area. But the analytical interpolation method requires that the height difference of the combined measurement points should not be too large, and the distance between the two points should not be too long, otherwise the terrain fluctuation between these points cannot be truly reflected.



The elevation fitting function mainly uses the geodetic height and normal height to solve the elevation anomaly process. LiDar360 currently supports four elevation fitting methods. They are plane fitting, linear interpolation, Conicoid fitting and cubic surface fitting. When choosing these mathematical models, first consider the distribution of GPS points (control points) and terrain conditions. Depending on the method and formula used, the minimum number of control points required for elevation fitting varies. These four methods all calculate the fitting parameters according to the least squares solution.

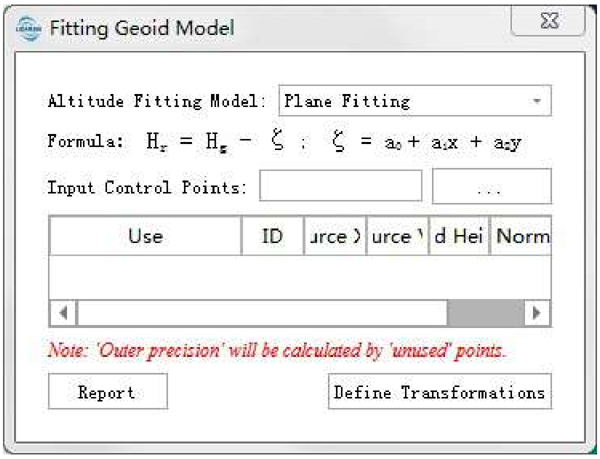
- **Plane Fitting**: At least 3 pairs of control points are required. The plane fitting method can be used in a small area or in a plain area to quickly obtain the required parameters with a small number of control points.
  - The plane fitting formula is $\zeta = a0 + a1x + a2y$;
- **Polynomial Curve Fitting**. At least 4 pairs of control points are required. This method can be used if the GPS points are distributed in a line, and the geoid-like surface along the line is a continuous and smooth curve. But when using this method, the measured line should not be too long (usually controlled within 300 meters). If the fitting range is too large, the abnormal elevation changes of the points will be more complicated, and the resulting fitting errors will be larger.
  - The linear interpolation formula is $\zeta = a0 + a1x + a2x^2 + a3x^3$;
- **Quadric Surface Fitting**. At least 6 pairs of control points are required. In the small area GPS network, the geoid is regarded as Conicoid, and the elevation anomaly is expressed as a quadratic

function of plane coordinates (x, y). The parameters are then fitted by least squares. Then each parameter is fitted by the least square method.

  - Conicoid fitting formula is $\zeta = a0 + a1x + a2y + a3xy + a4x^2 + a5y^2$;
- **Trigonometric Fitting** requires at least 10 pairs of control points. The principle is similar to Conicoid fitting, which expresses the elevation anomaly as a cubic function of plane coordinates (x, y).
  - The cubic surface fitting formula is $\zeta = a0 + a1x + a2y + a3xy + a4x^2 + a5y^2 + a6x^2y + a7xy^2 + a8x^3 + a9y^3$;

Usage

Click *Data Management* > *Projection and Coordinate Conversion* > *Elevation Fitting*



Parameters Settings

- **Altitude Fitting Model**: The four above-mentioned elevation fitting models can be selected from the drop-down menu. After you switch models, the formulas used by that model are also switched and displayed on the interface.
- **Input Control Points**: Import control point files in ascAll format. The control point file that needs to be selected must contain at least four columns, namely X, Y, geodetic height, and normal height. The imported control points will be displayed on the interface in the form of a table. The first column is "Whether it is a checkpoint". When this column is checked, the control point will be used for checking and not participating in the fitting. The number of control points involved in the fit must meet the minimum number of control points required for the elevation fit model. The control points participating in the fitting will be used to calculate the "inner coincidence accuracy", and the control points used for checking will be used to calculate the "outer coincidence accuracy". The same formula used for the accuracy of the inside and outside is: $\mu=\sqrt{[VV]/(n-1)}$.
- **Report**: Use control points to perform elevation fitting and display the fitting results and accuracy assessments on the interface. The report can be saved in html format.
- **Define Transformations**: Use control points to perform elevation fitting and save the fitting results for next use. Refer to creating a transformation relationship.

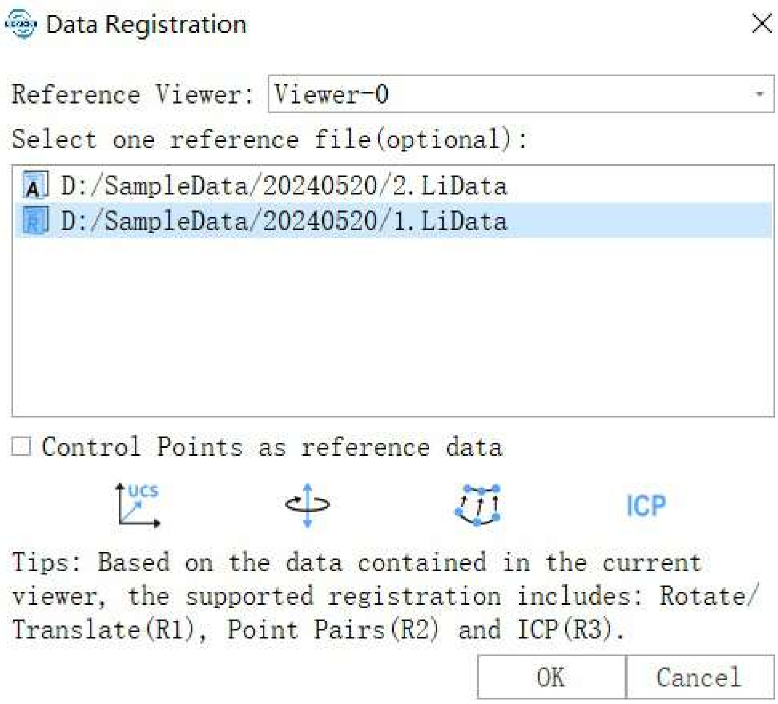
Data Registration

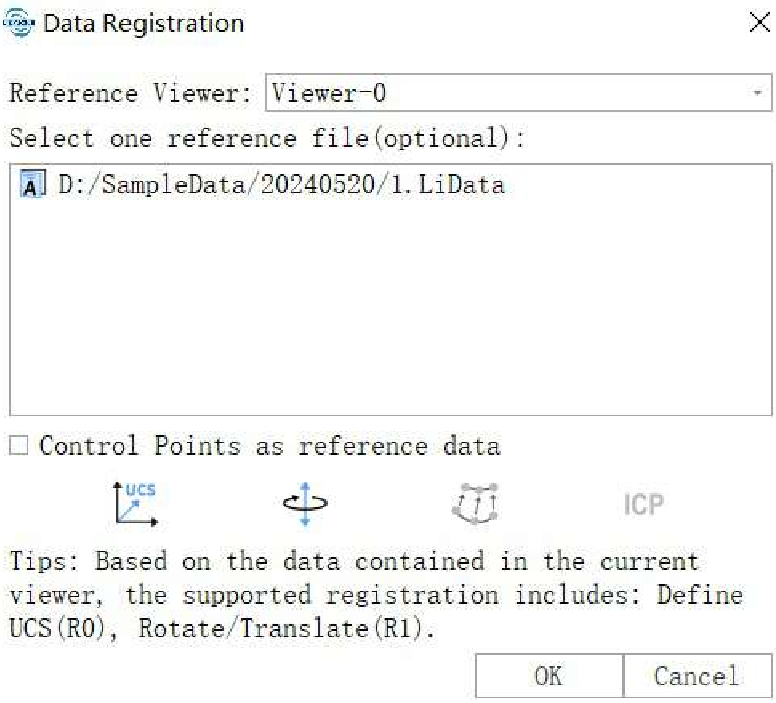
Function Overview

The data registration function allows users to perform registration operations on point cloud, model, raster, and other types of data.

Steps

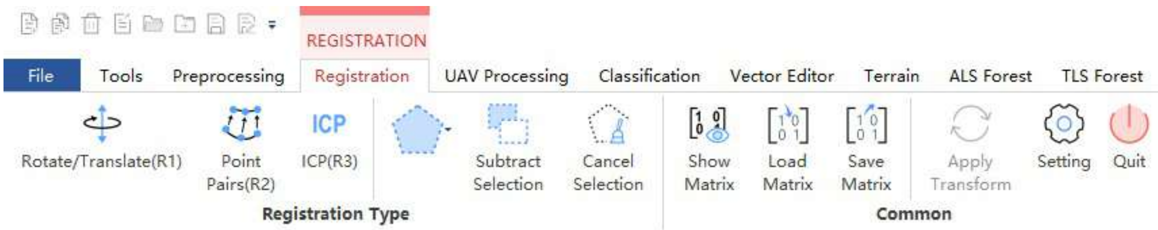
1. Open a new window and drag the data to be registered, along with the reference data (if available), into this window. Click on the data registration tool to open the following dialog box. Depending on the data in the view, the available registration functions will be displayed. Icons in gray indicate that the registration function is not available.





When selecting control points as the reference file, only point-to-point registration is supported.

1. In the dialog box that appears, select the registration window and choose the point cloud to be registered. If there is a reference point cloud, select the reference point cloud. On the registration page, select the appropriate registration function, which includes the following:
    - Define UCS
    - Manual Rotation And Translation
    - Rectif
    - ICP regression



Selection Tool

The selection tool allows you to select the data range on the reference or target point cloud for ICP registration. Specifically, in the ICP registration state, you can check the *Use Selected Range* option to use the data in the selected area for registration.

Transformation Matrix Tool

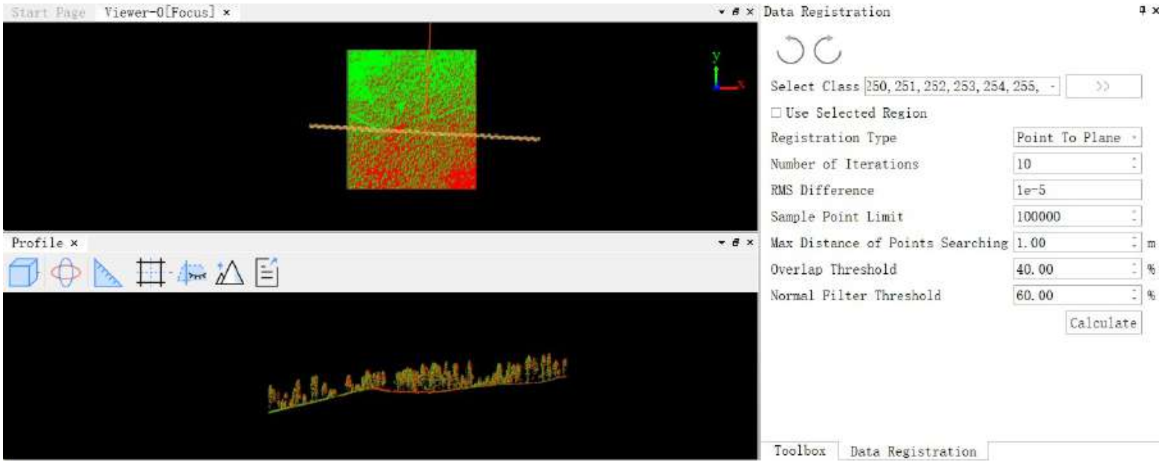
- **Display Transformation Matrix**: The manual rotation and translation, manual point matching, and ICP registration tools all generate a transformation matrix. The display transformation matrix function shows the contents of this transformation matrix.

- **Load Transformation Matrix**: Load the contents of a transformation matrix from an external file.
- **Save Transformation Matrix**: Save the contents of the transformation matrix to a file.

Other Common Functions

- **Apply Transformation**: Apply the transformation matrix to the data to be registered, generating the resulting file with completed registration.
- **Select Reference File**: If a reference file was not selected when entering data registration, you can set it using this function. Switching the reference file is not supported.
- **Exit**: Exit the data registration tool.

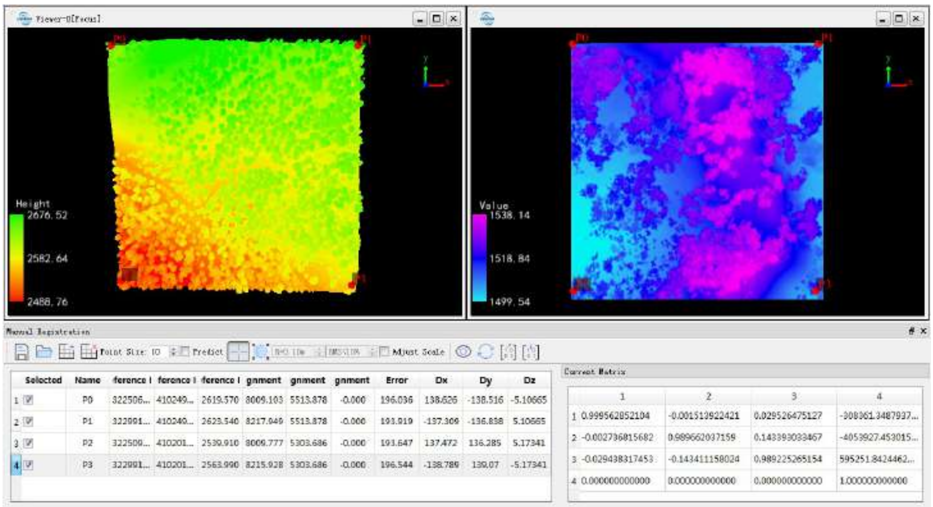
In the registration state, Profile View is supported to check the registration status of the data to be registered.



> Note:
>
> 1. At least one window (excluding the profile window) must be open, and the window must contain at least one point cloud data to activate this function.
> 2. When the registration window contains only one point cloud or model data, only UCS definition and rotation/translation registration can be performed.
> 3. When the registration window contains image data, only point-to-point registration can be performed.
> 4. ICP registration must set a reference file.
> 5. Point-to-point registration must set a reference file. If no reference file is available, imported control points can be used as reference points.

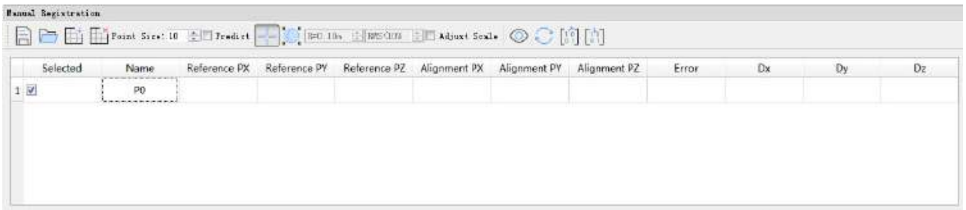
Manual Select Point Pairs Registration

The manual point selection registration tool can be used for data correction between point clouds, images, and model data. Place the reference data and the data to be registered in two windows, select or fit spheres (for point cloud data) in both windows to obtain at least three pairs of corresponding points. Calculate the coordinate transformation matrix between the two datasets through these corresponding points to perform coordinate correction on the data. By selecting point pairs, users can view residuals between datasets in a list of corresponding points.



Point Pair Registration by Reference Files

1. Select or fit spheres (for point cloud data) in two windows to obtain at least three pairs of corresponding points. These corresponding points are used to calculate the coordinate transformation matrix between the two datasets for coordinate correction. By selecting point pairs, the user can view the residuals between the datasets in the corresponding point pairs list.



2. **(Optional)** Users can click Open Data to load known corresponding points from external sources, thus skipping steps 4, 5, and 6.

3. Click Select Corresponding Points or Select Target Sphere to select a point as the corresponding point in both the reference and registration windows.
4. After selecting a pair of corresponding points, click the Add Row button to add an empty row.
5. Repeat steps 3 and 4 to select at least three pairs of corresponding points.
6. **(Optional)** If a certain pair of corresponding points is not needed for coordinate transformation, there are two ways to achieve this: (1) Uncheck the row in the list; (2) Click Delete Corresponding Points to remove point pairs with large residuals.
7. **(Optional)** If the residual of a pair of corresponding points is large or if you want to correct the

coordinates of the point pair, there are two ways to achieve this: (1) Select the corresponding point pair in the list and reselect the point in either the reference or registration window; (2) Double-click the coordinate value you want to modify and edit it directly.

8. **(Optional)** To view a particular pair of corresponding points, double-click the pair in the list to center it in the window.
9. **(Optional)** Click Save Data to save the selected corresponding points.

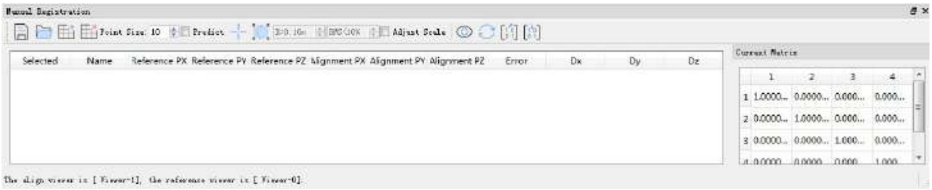
Registration by Control Points

1. Click **Load Pair Points**. Control point information is displayed in both the reference and registration windows. Users can toggle the display of control points in the registration window. Select the corresponding control points in the table and choose corresponding points in the registration window. These corresponding points are used to calculate the coordinate transformation matrix between the two datasets for coordinate correction. By selecting point pairs, users can view the residuals between the datasets in the corresponding point pairs list.

Lidar360 Rectify

The other operations are the same as the point-to-point registration steps using reference files.

Toolbar



In the toolbar, the buttons from left to right are: Load Pair Points, Save Data, Add Point, Delete Points, Pick Points, Pick Registration Sphere.

Load Pair Points: Import an existing corresponding points list file from external sources. For details on the format of the corresponding points list, refer to the appendix Homologous Points File Format.

Save Data: Save the selected corresponding points list as a txt file.

Add Point: After selecting a pair of corresponding points, click this button to add a new row in the table window.

Delete Point: To delete a row of corresponding points, select the row and click this button.

Pick Points: Select the corresponding points in the reference and registration windows.

Pick Registration Sphere: If using target spheres for matching in the point cloud, this tool can be used to select and automatically fit the target sphere. The coordinates of the corresponding point will be the center of the sphere.

In the lower toolbar, the buttons from left to right are Point Size, Adjust Scale, Predict.

Point Size: The size of the selected points in the window.

Adjust Scale (default is "unchecked"): If the scales between the datasets are different, this option should be checked. If unchecked, the point clouds are considered to undergo a rigid transformation, and the calculated rotation matrix will not include a scaling factor. If checked, the rotation matrix will include a scaling factor.

Predict (default is "unchecked"): Check this box and after selecting more than three pairs of corresponding points, the corresponding point in the reference window can be predicted after selecting the corresponding point in the registration window.

Manual Rotation and Translation

Description

The manual rotation and translation tool supports point cloud and model data types. By rotating and translating the corresponding data in the window, it obtains the corresponding transformation matrix. After applying this transformation, the transformed result data will be obtained.

Tools

In the data registration interface, the buttons from left to right are: Match Data Center, Pause Rotation and Translation, Set Rotation Center, and Reset to Initial State.

The translation and rotation angle information is calculated based on the rotation center, which corresponds to the world coordinates of the geometric center of the bounding box of the data to be registered.

Match Center: This function is available when there is a reference point cloud. It calculates the center of the reference point cloud and the center of the point cloud to be processed, and translates the point cloud to be processed to the location of the reference data.
Pause: Pause the rotation and translation operations of the data to be processed. In this state, you can view the differences between the current transformation and the reference data from various angles.
Set Rotation Center: Click this function to bring up a dialog box to set the coordinates of the rotation center.
Restore: Reset the data to be processed to the initial state and change the rotation transformation matrix to the identity matrix.
Apply Transform: After applying the transformation matrix, you can save the transformed point cloud result to the original file or generate a new file. Transformation formula: Given the coordinates of the point before transformation P0(X0,Y0,Z0). The transformation matrix M is as follows:

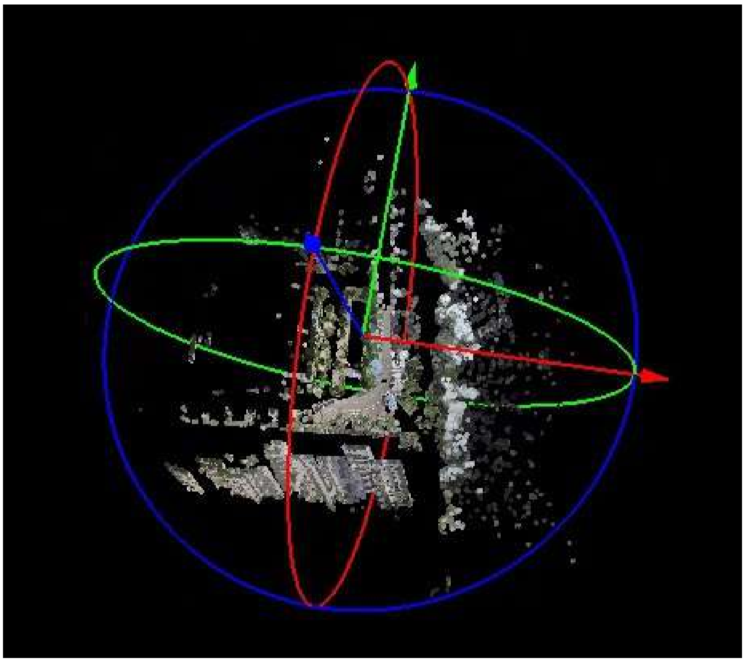
```
a00 a01 a02 a03

a10 a11 a12 a13

a20 a21 a22 a23

a30 a31 a32 a33
```

```
The transformation formula is [X1 Y1 Z1 1] = [X0 Y0 Z0 1]*M /(a03*X0 + a13*Y0 +a23*Z0);

The coordinates after transformation are P1(X1,Y1,Z1).
```

TX: After checking this item, you can perform translation operations along the X-axis. The translation amount in the X direction will be displayed on delt X.

TY: After checking this item, you can perform translation operations along the Y-axis. The translation amount in the Y direction will be displayed on delt Y.
TZ: After checking this item, you can perform translation operations along the Z-axis. The translation amount in the Z direction will be displayed on delt Z.
Rotate: There are four options: X, Y, Z, and XYZ. If X is selected, rotation operations can only be performed along the X-axis; if Y is selected, rotation operations can only be performed along the Y-axis; if Z is selected, rotation operations can only be performed along the Z-axis; if XYZ is selected, rotation operations can be performed along the X, Y, and Z axes.

Rotate Ball



1. As shown in the figure, hold the left mouse button and drag the red, green, and blue arrow axes to translate the point cloud along the X, Y, and Z axes, respectively. The dragged axis will be highlighted. Hold the left mouse button and drag the red, green, and blue track circles to rotate the point cloud around the X, Y, and Z axes, respectively. The dragged track circle will be highlighted.

ICP Registration

Summary

By setting benchmark point cloud and point cloud to be registered, perform the point cloud registration through ICP (Iterative Closest Points) algorithm. The basic steps of this algorithm are shown as follow.

Supposing that there are two given 3D-point collections, X1 and X2, the steps of ICP registration are shown as follows:

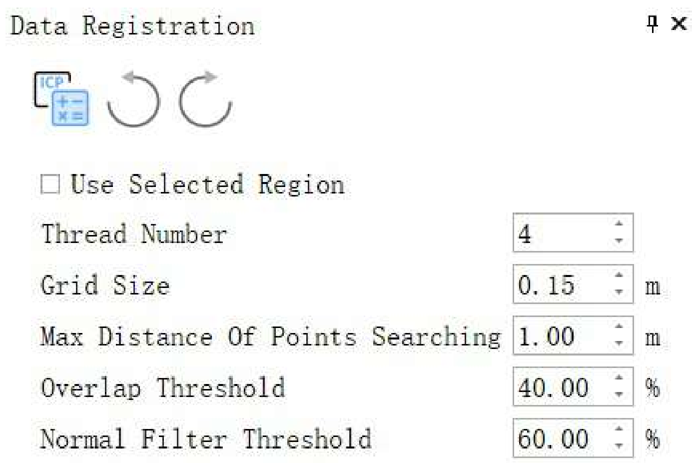
Step 1. For each point in X2, calculate and find its closest point in X1;

Step 2. Estimate the combination of rotation and translation using a root mean square point to point distance metric minimization technique which will best align each source point to its match found in the previous step.

Step 3. Transform X2 using the obtained transformation.

Step 4. Iterate the above steps until the average distance from X2 to X1 is less than a given threshold.

After entering the Registration Tool, click the "ICP Registration" button to enter the ICP Registration Tool, and the ICP Toolbar will pop up. If there is no reference file, a dialog box will appear for selecting a reference file.



Setting

Parameter Settings

- **Select Class**: Choose the initial category for the point cloud ICP registration. By default, all categories are selected.
- **Use Selected Region**: You can use the selection tools on the page, such as the polygon cut tool , to select the overlapping region of the point cloud. Check the *Use Selected Area* option to use the overlapping area for registration. If unchecked, the global point cloud will be used for registration. It is recommended to check this option.

- **Registration Type**: Includes two types: point-to-point and point-to-plane. The former estimates the transformation parameters by minimizing the Euclidean distance between matching points, while the latter solves for the optimal transformation matrix by minimizing the normal (plane) distance between source and target points. If the point cloud data contains ground, walls, or other planar features, the latter should be prioritized.
- **Number of Iterations (default is "10")**: The algorithm supports multi-threaded accelerated computation. If the parameter value exceeds the maximum number of threads of the computer, the algorithm will use the maximum number of threads by default.
- **Sample Point Limit (default is 100,000)**: This parameter is used for downsampling the original data. The algorithm will randomly sample the original point cloud data to the specified number of points, obtaining a subset of the original point cloud. The subsequent registration process will be based on this subset.
- **Max Distance of Points Searching (default is 1m)**: This parameter is used for nearest neighbor search to construct matching pairs. When coarse registration accuracy is low, for example, if the distance between two walls exceeds 5 meters after coarse registration, and the maximum point search distance is set to 1 meter, the algorithm will fail to associate features of the two walls, resulting in matching errors. In this case, increase the maximum point search distance, for example, to 10 meters, and run a global optimization. This will bring the point clouds closer, but local details may not be well stitched. Then set the maximum point search distance to 1 meter for global optimization. Generally, running the optimization 2-3 times is recommended to achieve good stitching results.
- **Overlap Threshold (default is 40%)**: This parameter is used for deciding which stations to include in the adjustment network during the inter-station nearest neighbor search. It works in conjunction with the maximum inter-station search distance. When the distance between two stations is less than the maximum inter-station search distance, the overlap rate between the two station point clouds must also meet the requirement. If the overlap rate exceeds the set overlap threshold, these two stations will be included in the adjustment network.
- **Normal Filter Threshold (default is 60%)**: This parameter is used for filtering the extracted line and plane features. The algorithm will retain features based on this parameter's proportion for computation. The larger the parameter, the more features will participate in the calculation, resulting in better optimization effects but longer computation times. It is recommended to set this parameter above 50%.

> Note:

```
1. This function does not support model-to-model registration.

2. If the files to be registered and the reference files are far apart in position, coarse registration (rota
tion and translation registration, and point-to-point registration) should be performed first.
```

Define UCS

Define UCS is used to set a custom coordinate system for point cloud or model data. In the window, you can determine the coordinate system's origin and the X, Y, Z axes, establishing a transformation matrix. Applying the transformation matrix will change the data's coordinates. If the transformation matrix is not applied, it will reset the current transformation matrix at the end.

Usage Instructions

1. Use the left mouse button to select the origin of the coordinate system on the point cloud or model.
2. Move the mouse to the appropriate position and click the left mouse button to set the X axis of the coordinate system.
3. Move the mouse to the appropriate position and double-click the left mouse button to set the Y and Z axes of the coordinate system.
4. Click "Apply Transformation Matrix" to change the data's coordinates and save to a file.

Geoid Model

When performing coordinate transformation in the projection library, vertical coordinate system transformation and grid projection need to provide vertical datum grid data for effective transformation. EGM2008 geoid model elevation conversion is provided by default in the software. If you need to support other regional geoid models, you need to download the corresponding grid data files. The grid files are provided by the official PROJ library, which can be downloaded here proj-data-1.8, or on the official website https ://proj.org/download.html to download the latest version of proj-data. After downloading, unzip the file to the geoid folder in the software installation directory, the default is C:\Program Files\GreenValley Suite\LiDAR360\5.0.0.0\geoid, "5.0.0.0" is the version number, and there will be differences between different versions.

Supported Geoid Model

Geoid Model	Vertical coordinate system
GEOID99	NAVD88 height - EPSG:5703
	NAVD88 height (ft) - EPSG:8228
	NAVD88 height (ftUS) - EPSG:6360
GEOID03	NAVD88 height - EPSG:5703
	NAVD88 height (ft) - EPSG:8228
	NAVD88 height (ftUS) - EPSG:6360
GEOID06	NAVD88 height - EPSG:5703
	NAVD88 height (ft) - EPSG:8228
	NAVD88 height (ftUS) - EPSG:6360
GEOID09	NAVD88 height - EPSG:5703
	NAVD88 height (ft) - EPSG:8228
	NAVD88 height (ftUS) - EPSG:6360
GEOID12A	NAVD88 height - EPSG:5703
	NAVD88 height (ft) - EPSG:8228
	NAVD88 height (ftUS) - EPSG:6360
	NMVD03 height - EPSG:6640
	PRVD02 height - EPSG:6641
	VIVD09 height - EPSG:6642
	ASVD02 height - EPSG:6643
	GUVD04 height - EPSG:6644
GEOID12B	NAVD88 height - EPSG:5703
	NAVD88 height (ft) - EPSG:8228
	NAVD88 height (ftUS) - EPSG:6360
	NMVD03 height - EPSG:6640

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

Create Transformation Relationship

Functional Overview

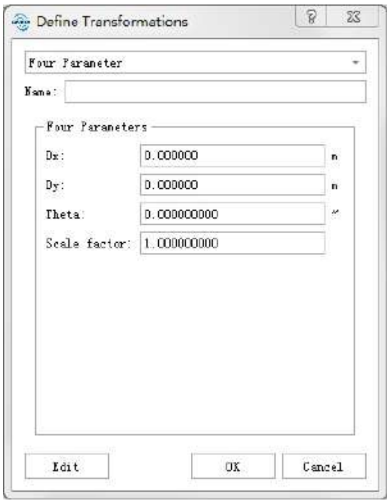
The user can define and edit a variety of transformation relationships that can be saved to a history file for the user to reuse. These transformation relationships can be defined during "transformation relationship calculations" as well as when importing data.

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

Supported conversion relationship:

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

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



The four-parameter formula can be written as follows:

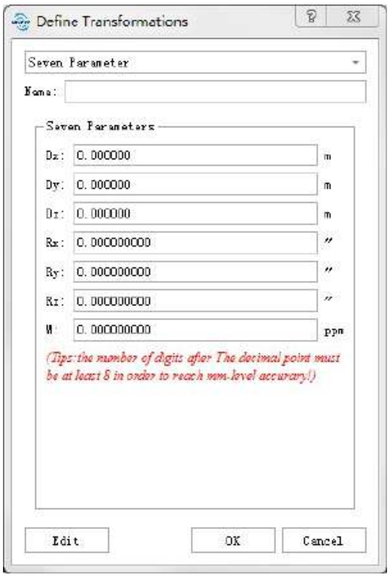
$$X_2 = X_0 + X_1 mCOS\,\alpha - Y_1 m\sin\alpha$$
$$Y_2 = Y_0 + Y_1 mCOS\,\alpha + X_1 m\sin\alpha$$

It can be solved according to the following formula:

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

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

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



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

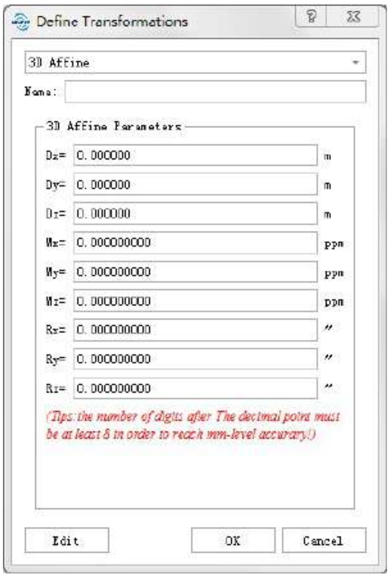
$$\begin{bmatrix} X_B \\ Y_B \\ Z_B \end{bmatrix} = \begin{bmatrix} X_A \\ Y_A \\ Z_A \end{bmatrix} + \begin{bmatrix} 1 & 0 & 0 & X_A & 0 & -Z_A & Y_A \\ 0 & 1 & 0 & Y_A & Z_A & 0 & -X_A \\ 0 & 0 & 1 & Z_A & -Y_A & X_A & 0 \end{bmatrix} \begin{bmatrix} dx \\ dy \\ dz \\ m \\ rx \\ ry \\ rz \end{bmatrix}$$

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

```
Dx = Dx;
Dy = Dy;
Dz = Dz;
Rx(″) = Rx(″);
Rx(″) = Rx(″);
Rx(″) = Rx(″);
Mx(ppm) = m(ppm);
My(ppm) = m(ppm);
```

```
Mz(ppm) = m(ppm);
```

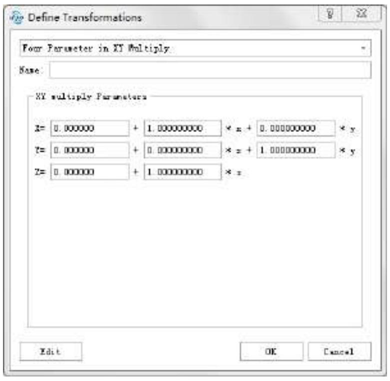
- **3D affine transformation**:



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

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

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



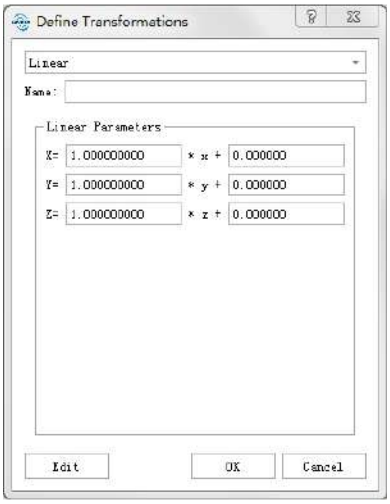
Two-dimensional transformation parameters can be expressed in the following formula:

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

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

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

- **Linear transformation**: The linear transformation is used to pan and zoom a point cloud. A panning parameter and a scaling parameter need to be set to the X, Y, Z coordinates, respectively.

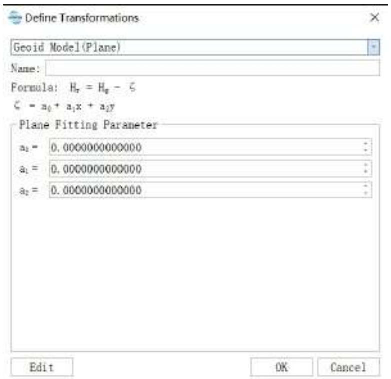


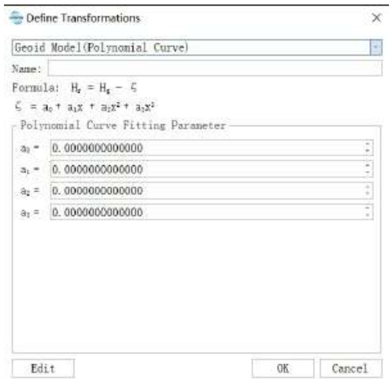
Linear transformation is calculated using the following formula:

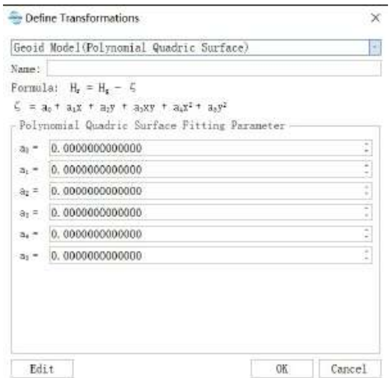
$$\begin{cases} X = S_x * x + P_x \\ Y = S_y * y + P_y \\ Z = S_z * z + P_z \end{cases}$$

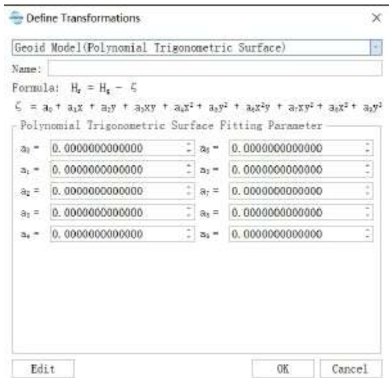
Among them: Sx, Sy. Sz are the scaling factors for the x, y, and z coordinates. Px, Py, and Pz are the panning parameters for the x, y, and z coordinates. x, y, and z are the original coordinates, and X, Y, and Z are coordinates obtained after the linear transformation.

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


Define Transformations
Geoid Model(Plane)
Name:
Formula: Hr = Hg - ζ
Plane Fitting Parameter
Edit
OK
Cancel


Define Transformations
Geoid Model(Polynomial Curve)
Name:
Formula: Hr = Hg - ζ
Polynomial Curve Fitting Parameter
Edit
OK
Cancel


Define Transformations
Geoid Model(Polynomial Quadric Surface)
Name:
Formula: Hr = Hg - ζ
Polynomial Quadric Surface Fitting Parameter
Edit
OK
Cancel


Define Transformations
Geoid Model(Polynomial Trigonometric Surface)
Name:
Formula: Hr = Hg - ζ
Polynomial Trigonometric Surface Fitting Parameter
Edit
OK
Cancel

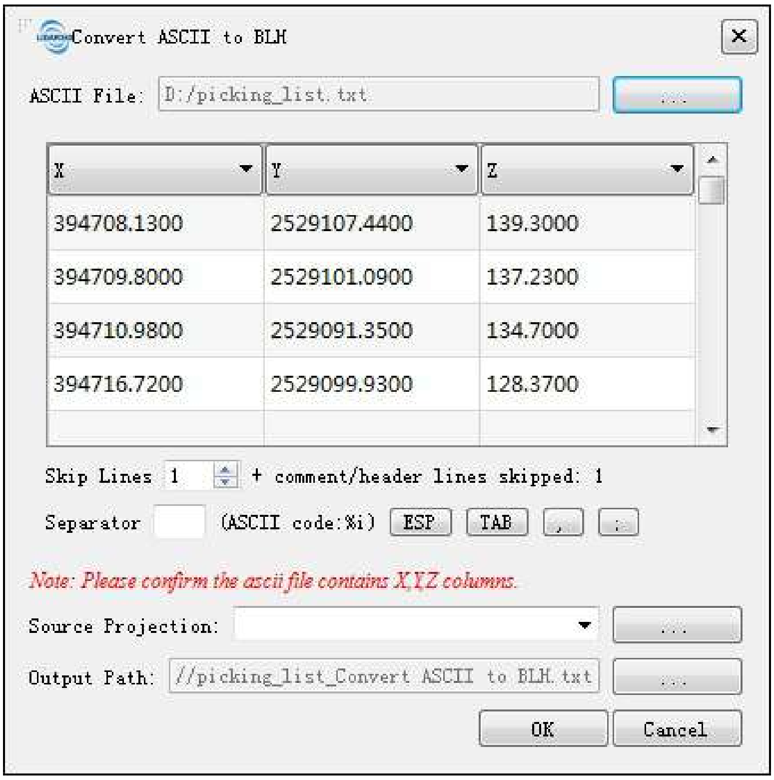
ASCII Data Reprojection

Function Overview

Reproject the X, Y, Z coordinate data in the ASCII file from the current coordinate system to the target coordinate system while keeping other data unchanged.

Usage

Click *Data Management* > *Projections and Transformations* > *ASCII Data Reprojection*



Parameter Settings

- **Input File**: Input an ASCII data file. File formats: .txt, .csv, .treedb.
- **Current Coordinate System**: Select the coordinate system information of the current ASCII data.
- **Target Coordinate System**: Select the target coordinate system information.
- **Use Seven Parameters**: Select the **Use Seven Parameters** option and click the **Create** button to create a seven-parameter transformation relationship. Refer to Create Transformations for more information.
- **Output Path**: Output the reprojected ASCII file to the specified path.

Clip

The clipping tools include **Clip by Circle**, **Clip by Rectangle** and **Clip by Polygon**.

- Clip by Circle
- Clip by Rectangle
- Clip by Polygon

Point Cloud Conversion

Point Cloud Format Conversion Tool Provides Conversion Between Point Cloud and Other File Formats.

- Convert to ASCII
- Convert to TIFF
- Convert to Shape
- Convert to DXF
- Convert to Las
- Convert to E57
- Convert LiData to LiData
- Convert Las to LiData

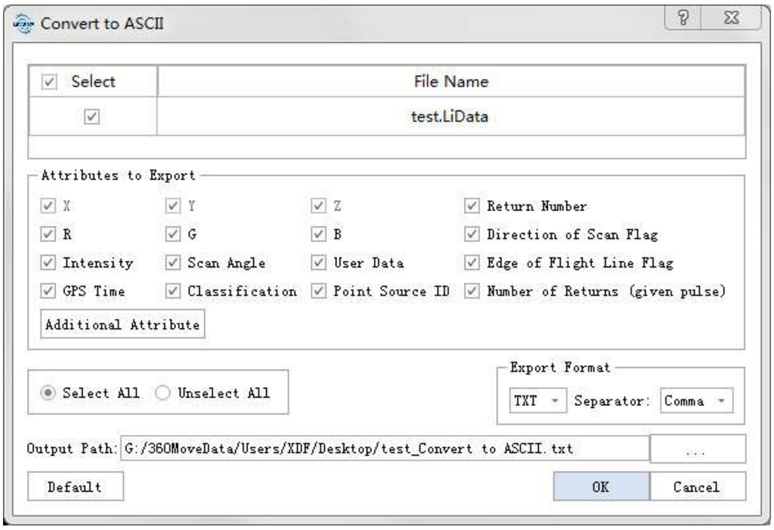
Convert to ASCII

Summary

The Convert to ASCII tool can convert the LiData point cloud to ASCII format, a text format that can be easily viewed in a text editor.

Usage

Click *Data Management* > *Conversion* > *Convert to ASCII*



Settings

- **Input Data**: The input file can be a single point cloud data file or multiple data files. File Format: *.LiData.
- **Attributes to Export**: Users can select the attributes that need to be exported. All Las attributes are supported.
- **Export Format**: LiDAR360 supports two suffix formats: txt and csv, and support separators are: commas, spaces and TAB.
- **Output path**: Path of the output folder. After the conversion is executed, the converted new file(s) will be generated.

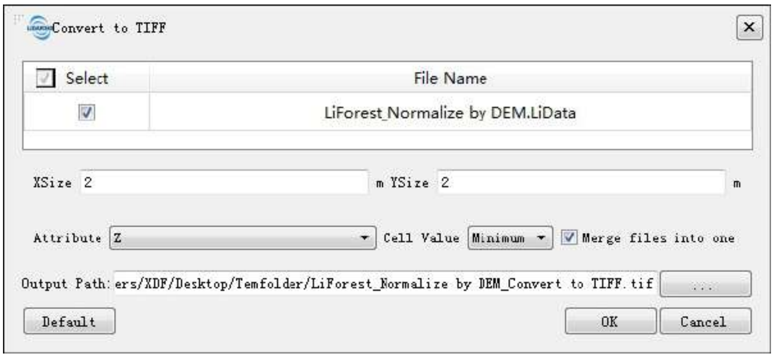
Convert to TIFF

Summary

The Convert to TIFF tool can convert LiData point cloud data to raster images according to the **Attribute** of the points in **Cell Value** method.

Usage

Click *Data Management* > *Conversion* > *Convert to TIFF*



Settings

- **Input Data**: The input file can be a single point cloud data file or multiple data files. File Format: *.LiData.
- **XSize (m, default value is "2")**: The X pixel resolution of the output raster.
- **YSize (m, default value is "2")**: The Y pixel resolution of the output raster.
- **Attribute (default is "Z")**: Select the point attribute used to calculate the raster cell value.
  - **Z (default)**: Generate the raster data using the Z value of the point cloud data.
  - **Intensity**: Generate raster data using the intensity values of the point cloud data.
  - **Scan angle**: Generate raster data using the scan angle of point cloud data.
  - **Return number**: Generate raster data using the return number of the point cloud data.
  - **Number of returns of given pulse**: Raster data is generated using the number of returns of given pulse of the point cloud data.
- **Cell value**: Choose how to fill raster pixel values.
  - **Minimum (default)**: Use the smallest attribute value as the value of a raster cell.
  - **Average**: Use the average of attribute values of all points within a raster cell as its value.
  - **Maximum**: Use the largest attribute value as the value of a raster cell.
- **Merge files into one**: Users can set merging all raster into one data file. If not checked, each point cloud data will be generated into a separate raster file.
- **Output path**: Path of output folder. After the function is executed, the converted new file is generated.

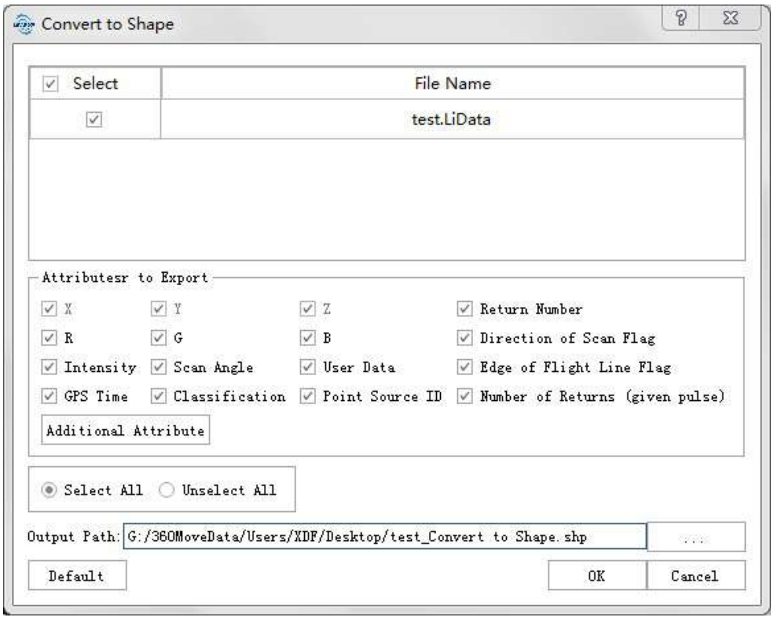
Convert to Shape

Summary

The Convert to Shape tool can convert point cloud files in LiData format to point files in vector format.

Usage

Click *Data Management* > *Conversion* > *Convert to Shape*



Settings

- **Input data**: The input file can be a single point cloud data file or multiple data files. File Format: *.LiData.
- **Attributes to Export**: Users can select the attributes that need to be exported, and all Las attributes are supported. The selected attributes will be written in the attributes of the shape file data.
- **Output path**: Path of the output folder. After the conversion being executed, the converted new file will be generated.

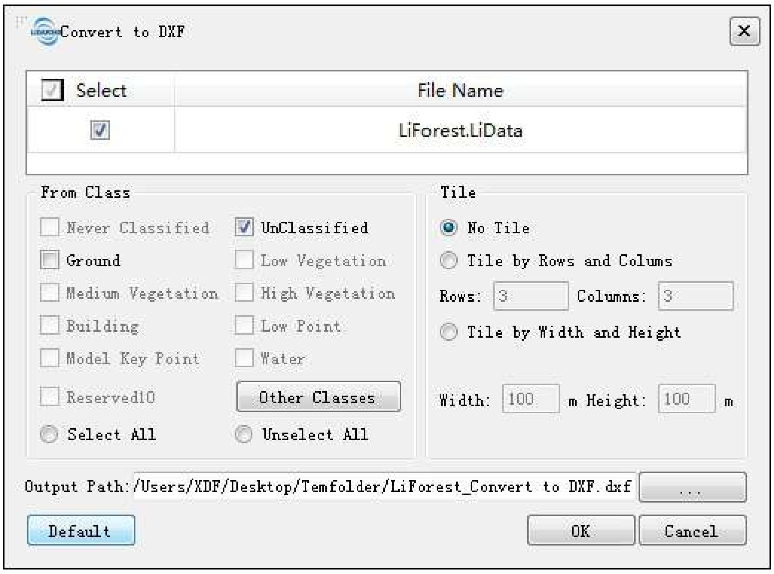
Convert to DXF

Summary

The Convert to DXF tool can convert LiData format point cloud files to point vector data in DXF format. LiDAR360 software exports DXF files in ASCII format.

Usage

Click *Data Management* > *Conversion* > *Convert to DXF*



Settings

- **Input Data**: The input file can be a single point cloud data file or multiple data files. File Format: *.LiData.
- **From Class**: Source class (es), supports up to 256 different classes.
- **Tile**: Since the amount of point cloud data is often large, the software provides an optional tile mode that can split the input data file to a number of tiles.
  - **No Tile**: Export a single DXF file.
  - **Tile by Rows and Columns**: According to the boundary of the point cloud data, the point cloud will be split into a total number of rows * columns data blocks evenly, and a single DXF file will be generated.
    - **Rows (default value is "3")**: User-defined number of rows that the point cloud want to be split into.
    - **Columns (default value is "3")**: User-defined number of columns that the point cloud want to be split into.
  - **Tile by Width and Height**: The point cloud will be split into a number of tiles from the lower left corner by the specified width and height, one DXF file per block.
    - **Width (m, default value is "100")**: User-defined block width.
    - **Height (m, default value is "100")**: User-defined block height.

- **Output path**: Path of the output folder. After the conversion being executed, the converted new file(s) will be generated.

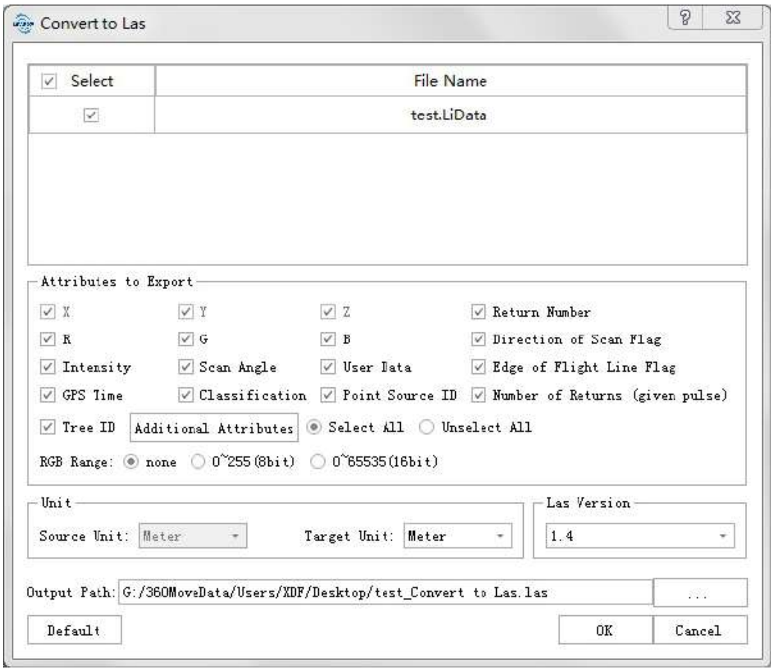
Convert to Las

Summary

Convert to Las can convert point clouds in LiData format to LAS format, namely the standard Lidar point cloud data format.

Usage

Click *Data Management* > *Conversion* > *Convert to Las*



Settings

- **Select Input File**: The input file can be a single point cloud data file or multiple data files. File Format: *.LiData.
- **Attributes to Export**: Users can select the attributes that need to be exported. All Las attributes are supported.
- **RGB Range**: There are three options for the RGB range.
  - **none**：Output according to the original RGB range of the selected file.
  - **0~255 (8 bits)**: Output with the RGB mapped to 0~255. If the color range of the file selected is 0~1 or 0~65535, it can be mapped to 0~255 for output.
  - **0~65535 (16 bits)**：Output with the RGB mapped to 0~65535. If the color range of the file selected is 0~1 or 0~255, it can be mapped to 0~65535 for output.
- **Source Unit**: The unit of the LiData to be exported as LAS file. As the only supported unit in LiDAR360 is meters currently, this option is fixed as "meter".
- **Target Unit**: The unit of the exporting LAS file. Choose from meters, decimeters, centimeters, millimeters, feet, and inches.
- **LAS Version (Default value is 1.4)**: The version of the exporting LAS file. Choose from 1.2 to 1.4.

Please refer to LiData Format for details. When exporting from a higher version to a lower version, the exceeded field range limitation will be set as 0.

- **Output Path**: Path of the output folder. After the conversion being executed, the converted new file(s) will be generated.

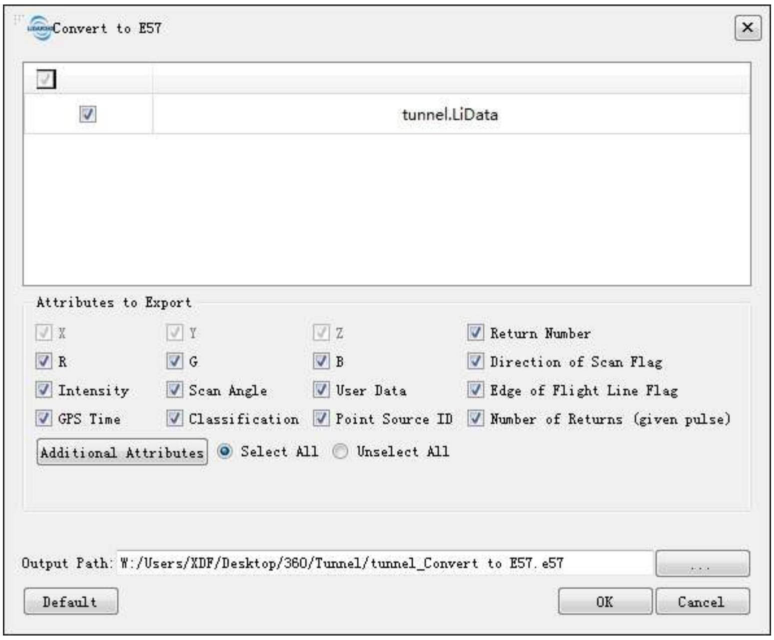
Convert to E57

Functional Overview

Convert to E57 can convert point clouds in LiData format to E57 format.

Usage

Click *Data Management* > *Point Cloud Conversion* > *Convert to E57*



Parameters Settings

- **Select Input File**: The input file can be a single point cloud data file or multiple data files. File Format: *.LiData.
- **Attributes to Export**: Users can select the attributes that need to be exported. The basic and additional attributes are supported.
- **Output Path**: Path of the output folder. After the conversion being executed, the converted new file(s) will be generated.

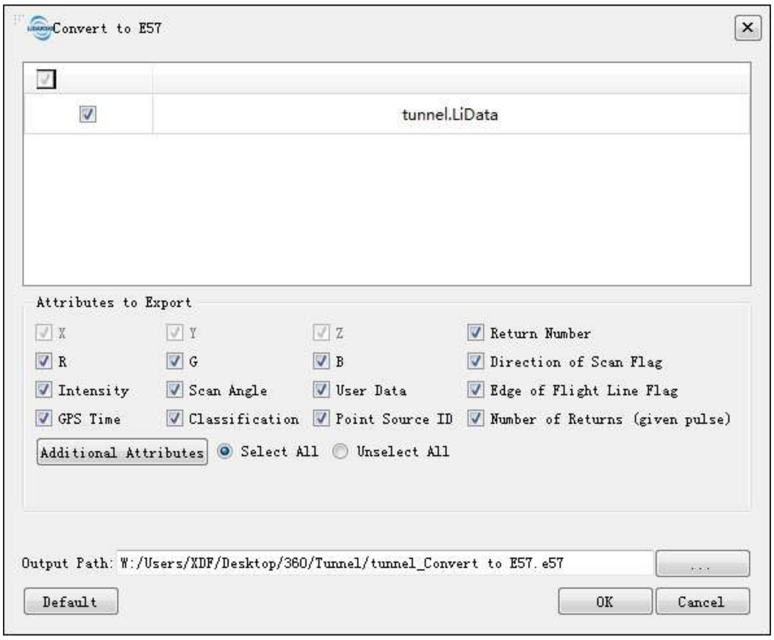
Convert to PLY

Functional Overview

Convert to PLY can convert point clouds in LiData format to PLY format.

Usage

Click *Data Management* > *Point Cloud Conversion* > *Convert to E57*



Parameters Settings

- **Select Input File**: The input file can be a single point cloud data file or multiple data files. File Format: *.LiData.
- **Attributes to Export**: Users can select the attributes that need to be exported. The basic and additional attributes are supported.
- **Output Path**: Path of the output folder. After the conversion being executed, the converted new file(s) will be generated.

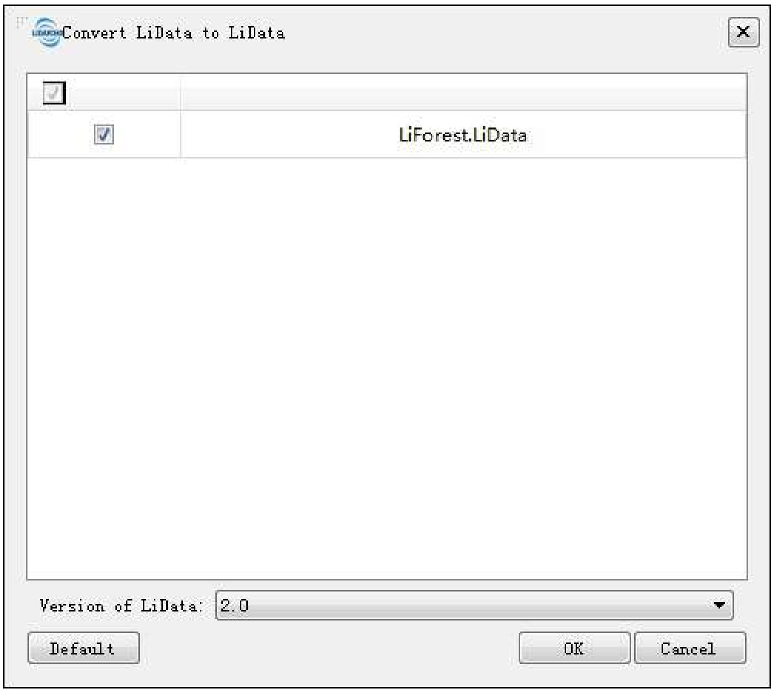
Convert LiData to LiData

Summary

Convert the version of LiData file. Currently, the tool supports the conversion between LiData 1.9 and LiData 2.0. LiData 2.0 supports the additional attributes and expands the scope of some attribute fields (i.e., the number of classes has been expanded from 32 to 256). It is required to use LiData 2.0 file to run some of the functions, i.e., change detection, deviation analysis.

Usage

Click *Data Management* > *Conversion* > *Convert LiData to LiData*



Settings

- **Input Data**: Input data could be one or more point cloud files. File format: *.LiData.
- **LiData Version (2.0 by default)**: The version of output LiData file. It is supported to export LiData in the version of 1.9 or 2.0.

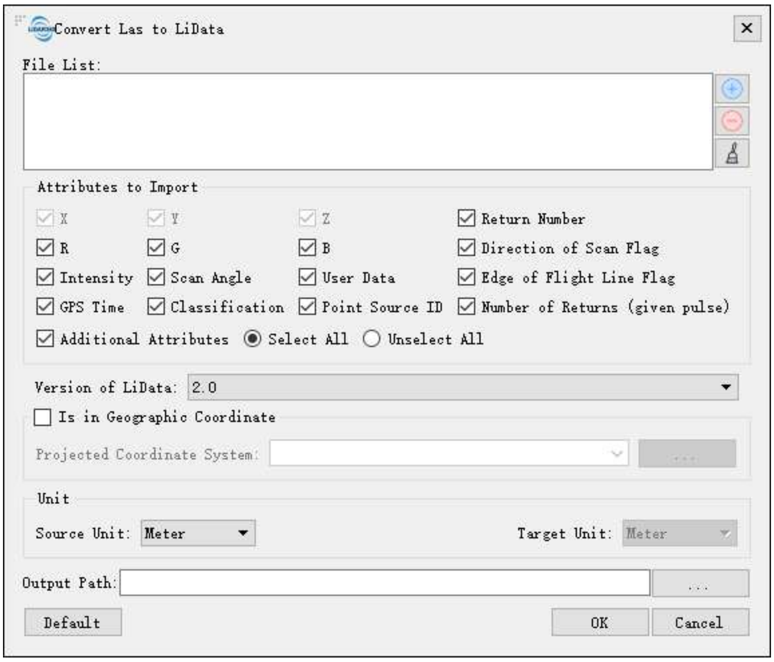
Convert Las to LiData

Summary

Convert to LiData can convert point cloud data in LAS or LAZ formats to LiDAR360 customized point cloud format (LiData format).

Usage

Click *Data Management* > *Conversion* > *Convert Las to LiData*



Settings

- **File List**: Import the file that needs to be converted. The file formats currently supported are : .las, .laz. Users need to click to select the point cloud data; users can select one or more file(s) in the existing list and click to delete them from the list; users can click to remove all files in the list.
- **Attributes to Export**: Users can select the attributes that need to be exported. All Las attributes are supported.
- **Source Unit**: The unit of the importing point cloud file. Choose from meter, decimeters, centimeters, millimeters, feet, and inches.
- **Target Unit**: The unit of the exporting LiData. As the only unit supported in LiDAR360 is meter, this option is fixed as "meter".
- **LiDAR Version (The most updated version of LiData by default)**: The version of the exporting LiData. Choose from 1.9 and 2.0
  - Note: LiData 2.0 supports all the features of LAS 1.4. LiData 1.9 is mainly used to support LAS 1.0 to LAS 1.3. Please refer to LiData Format for details. When exporting from a higher version to a lower version, the exceeded field range limitation will be set as 0.
- **Coordinate Transformation**: Please refer to Import Data.

- **Output Path**: Path of the output folder. After the conversion is executed, the converted new file(s) will be generated.

Raster Conversion

Raster format conversion tool provides conversion between raster data and other file formats.

- Convert TIFF to LiModel
- Convert to Texture LiModel
- Convert TIFF to LiData
- Convert TIFF to ASCII

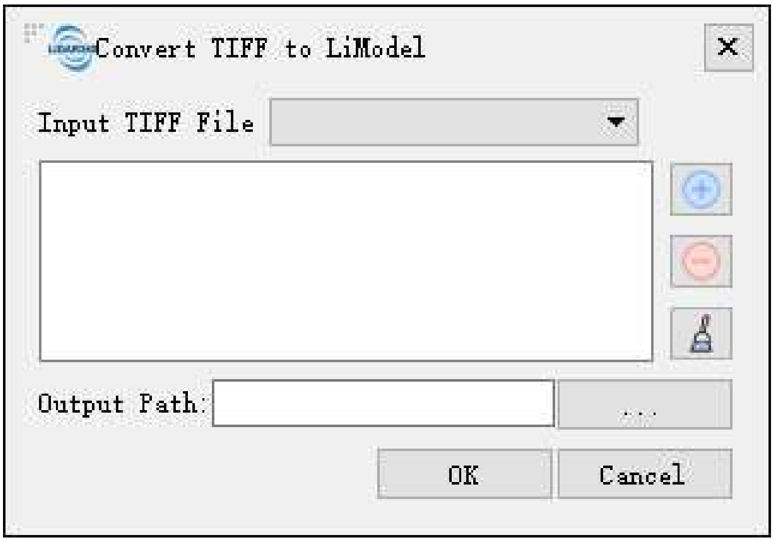
Convert TIFF to LiModel

Summary

Convert TIFF to LiModel tool can convert single-band raster data (DEM, DSM, CHM, etc.) to LiModel format. The LiModel format is a LiDAR360 software customized format that allows the converted single-band raster data to be displayed and edited in a 3D window.

Usage

Click *Data Management* > *Conversion* > *Convert TIFF to LiMode*



Settings

- **Input Tiff File**: Users can select the file to be processed from the drop-down list.
- : Users can add files to be converted that have not been opened.
- : Users can select a file in the list and click this button to remove the file from the list.
- : Click this button to clear all files in the list.
- **Output path**: Path of the output folder. After the conversion being executed, the converted new file(s) will be generated.

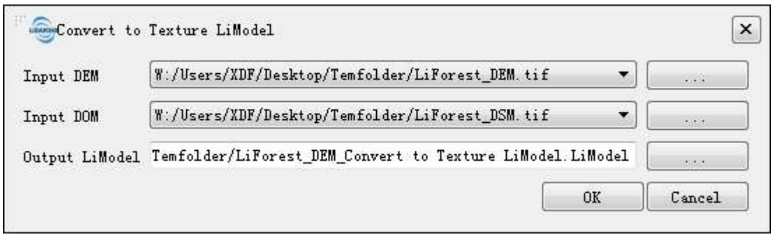
Convert to Texture LiModel

Summary

Convert to Textured LiModel tool is based on the Convert Image to LiModel tool, which maps color values of a DOM data to the LiModel model for display. This function only supports single file data conversion.

Usage

Click *Data Management* > *Conversion* > *Convert to Texture LiModel*



Settings

- **Input DEM**: Users can select a file to be converted from the drop-down list or use the button ... to load an external data file. Single-band raster data is required.
- **Input DOM**: Users can select a desired file from the drop-down list or use the button ... to load an external data file. The selected color image needs to have at least three bands.
- **Output LiModel**: Path of the output file. After the conversion being executed, the converted new file will be generated.

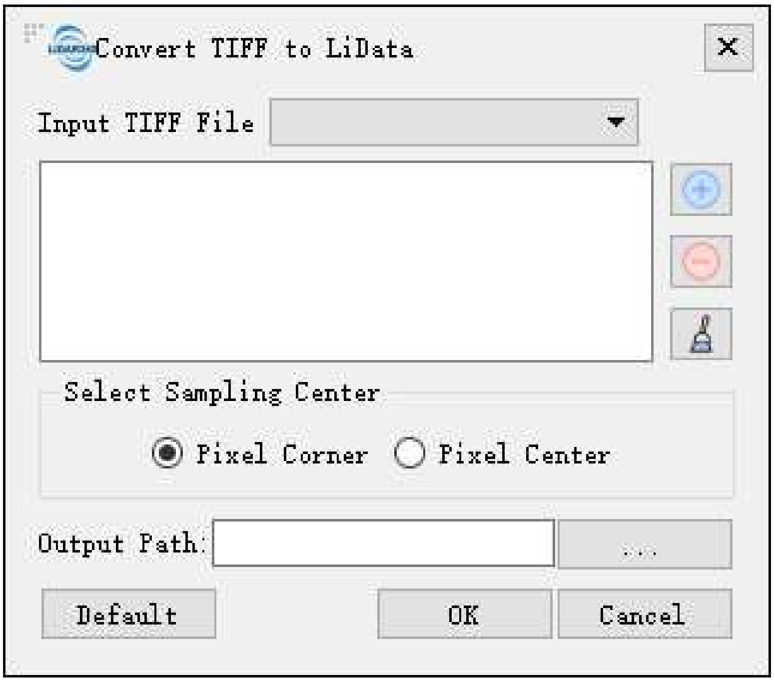
Convert TIFF to LiData

Summary

Convert TIFF to LiData tool can convert single-band raster data (DEM, DSM, CHM, etc.) to LiData. This tool is particular useful when the users can harldy acuquire point clouds in a certain area but have accurate raster products.Tthis function can help to simulate the point cloud data in that region missing LiDAR points.

Usage

Click *Data Management* > *Conversion* > *Convert TIFF to LiData*.



Settings

- **Input TIFF File**: Users can select the files need to be processed from the drop-down list. The files will be added to the list of files to be processed.
- : Users can insert external files to be processed.
- : Users can select one file in the list of files to be processed, and click this button to remove this file from that list.
- : Click this button to clear the list of files to be processed.
- **Select Sampling Center**: Users can choose either the corner (default) or the center points of the pixels as the sampling center.
- **Output Path**: Path of output folder. After the function is executed, the converted new file is generated.

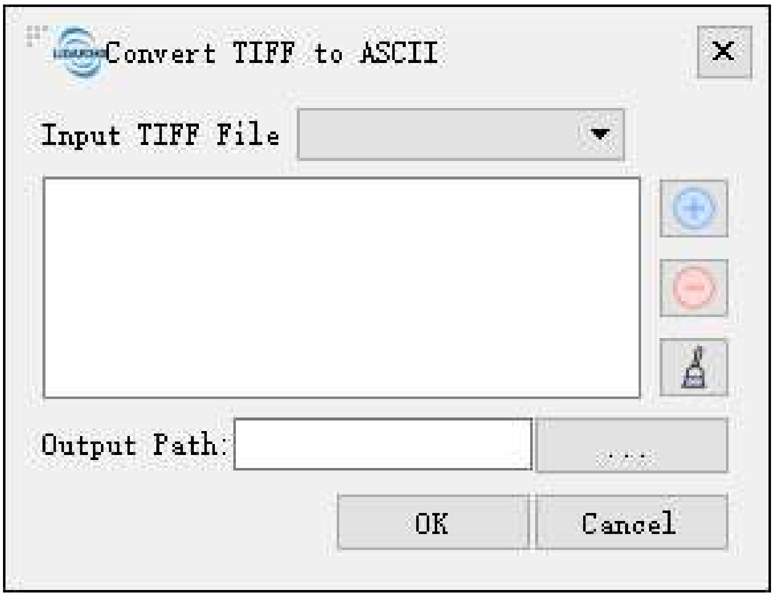
Convert TIFF to ASCII

Summary

Convert TIFF to ASCII tool can be used to convert TIFF format raster images to files in ASCII format.

Usage

Click *Data Management* > *Conversion* > *Convert TIFF to ASCII*



Settings

- **Input Tiff File**: Users can select the file to be processed from the drop-down list.
- : Users can add external file(s) to be converted.
- : Users can select one or files in the list and click this button to remove them from the list.
- : Click this button to clear all files in the list.
- **Output path**: Path of the output folder. After the conversion being executed, the converted new file(s) will be generated.

Model Conversion

Model Conversion tool provides conversion between model files and other format.

- Convert LiModel to TIFF
- Convert LiTin to DXF
- Convert LiBuilding to OBJ
- Convert LiBuilding to FBX
- Convert LiBuilding to CityJson

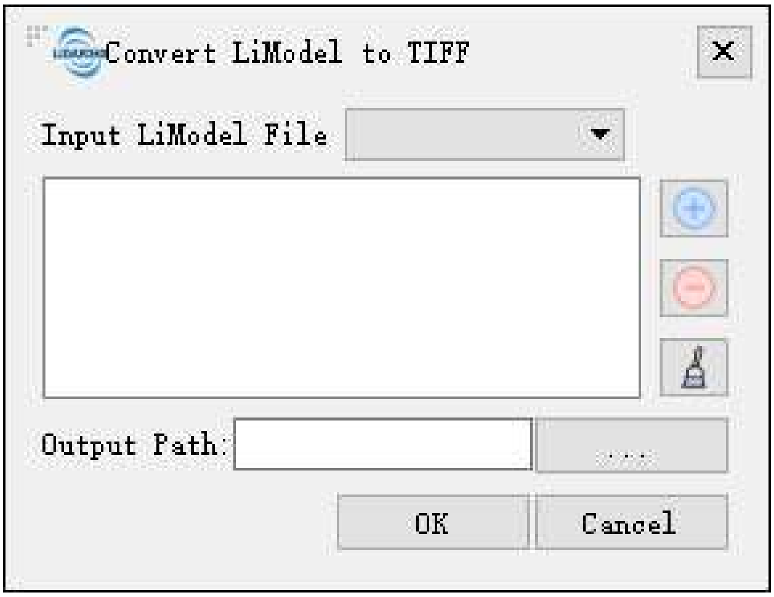
Convert LiModel to TIFF

Summary

Convert LiModel to TIFF tool converts LiModel format files to raster images in TIFF format. After editing LiModel data, users can use this function to convert the data to TIFF format.

Usage

Click *Data Management* > *Conversion* > *Convert LiModel to TIFF*



Settings

- **Input LiModel File**: User can select a file to be processed from the drop-down list.
- : Users can add external data files to be converted.
- : Users can select a file in the list and click this button to remove the file from the list.
- : Click this button to clear all the data in the list.
- **Output path**: Path of the output folder. After the conversion being executed, the converted new file will be generated.

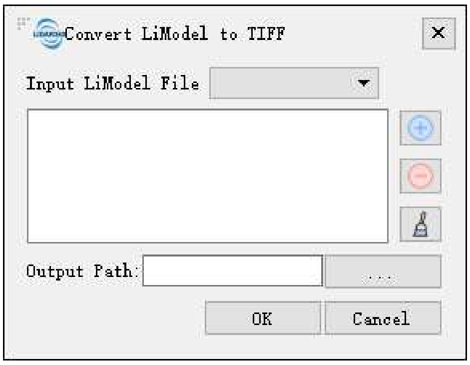
Convert LiModel to OBJ

Function Overview

The Convert LiModel to OBJ tool converts LiModel format files into OBJ model format.

Usage

Click *Data Management* > *Model Conversion* > *Convert LiModel to OBJ*.



Parameter Settings

- **File List**: List of files to process.
- : Add files to convert from external sources.
- : Select a file in the list and click this button to remove it.
- : Click this button to clear all files from the list.
- **Output Path**: Directory path for the output files. After execution, the converted files will be generated in this folder.

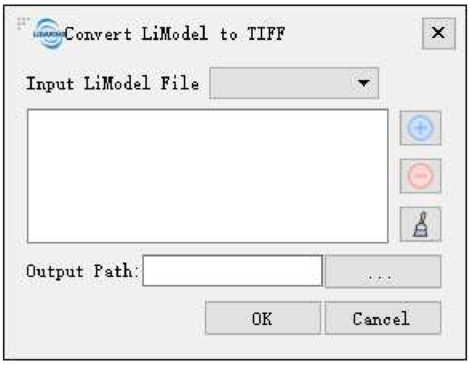
Convert LiModel to OSGB

Function Overview

The Convert LiModel to OSGB tool converts LiModel format files into OSGB format. This function outputs OSGB files at different LOD (Level of Detail) levels according to the LiModel's LOD structure. The root node is named after the output directory, and a metadata.xml file recording projection information is also generated.

Usage

Click *Data Management* > *Model Conversion* > *Convert LiModel to OSGB*.



Parameter Settings

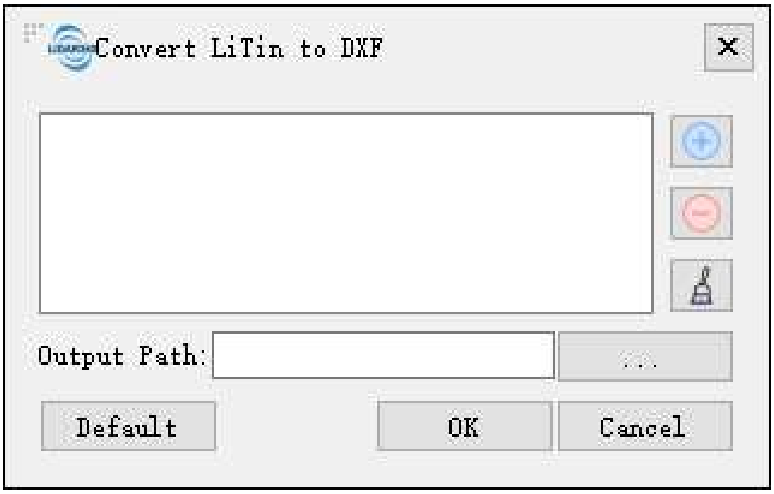
- **Select file list**: List of files to process.
- : Add files to convert from external sources.
- : Select a file in the list and click this button to remove it.
- : Click this button to clear all files from the list.
- **Output Path**: Directory path for the output files. After execution, the converted files will be generated in this folder.

Convert LiTin to DXF

Convert the TIN in LiTin format to Drawing Exchange Format, or DXF format.

Usage

Click *Data Management* > *Conversion* > *LiTin to DXF*.



Settings

- **File List**: Input the TIN files to be converted. The supported format is LiTin. Users can click to add external data, or users can select the file in the file list and click button to remove the file from the list. Click button to clear all the files from the file list.
- **Output Path**: select the output path to store the newly generated DXF file.

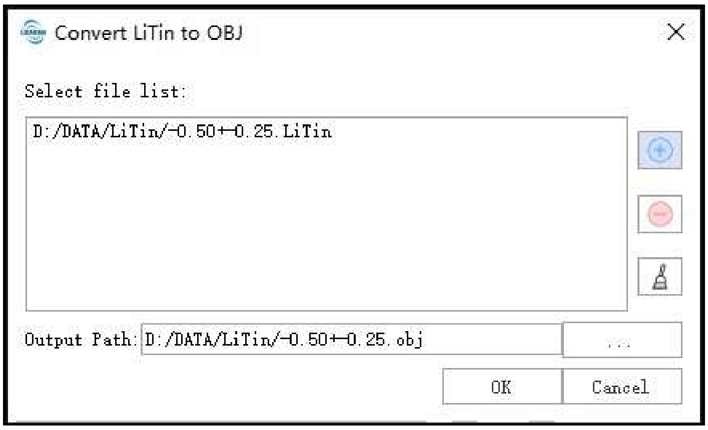
Convert LiTin to OSGB

Function Overview

The Convert LiTin to OSGB tool transforms TIN (Triangular Irregular Network) data in LiTin format into OSGB format files.

Usage

Click *Data Management* > *Model Conversion* > *Convert LiTin to OSGB*.



Parameter Settings

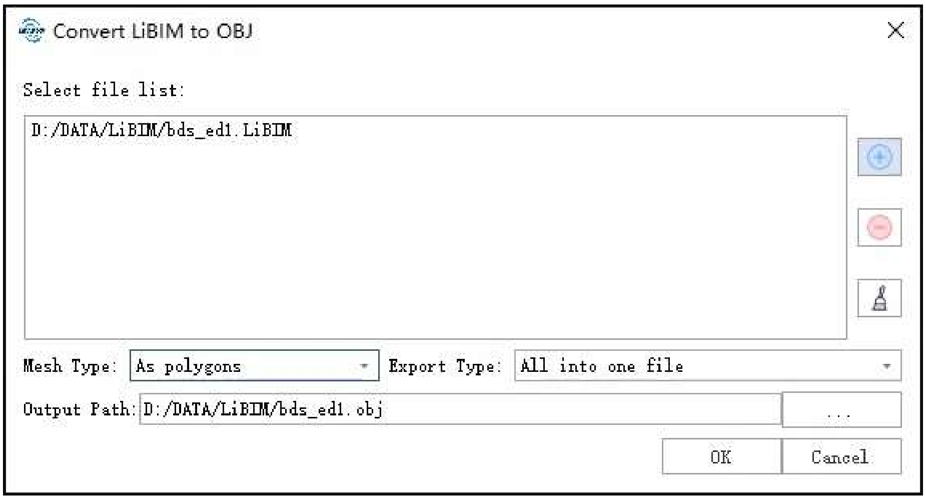
- **File List**: Input the TIN files you wish to convert. Supported formats include *.LiTin.
  - Use to add files from external sources.
  - Select a file in the list and click to remove it.
  - Click to clear all files from the list.
- **Output Path**: Select the directory path for the output files. After the algorithm executes, OSGB files will be generated in this directory.

Convert LiBIM to OBJ

Convert LiBIM format data into standard 3D model file in .obj format.

Usage

Click *Data Management* > *3D Building Tools* > *Convert LiBIM to OBJ*.



Parameters

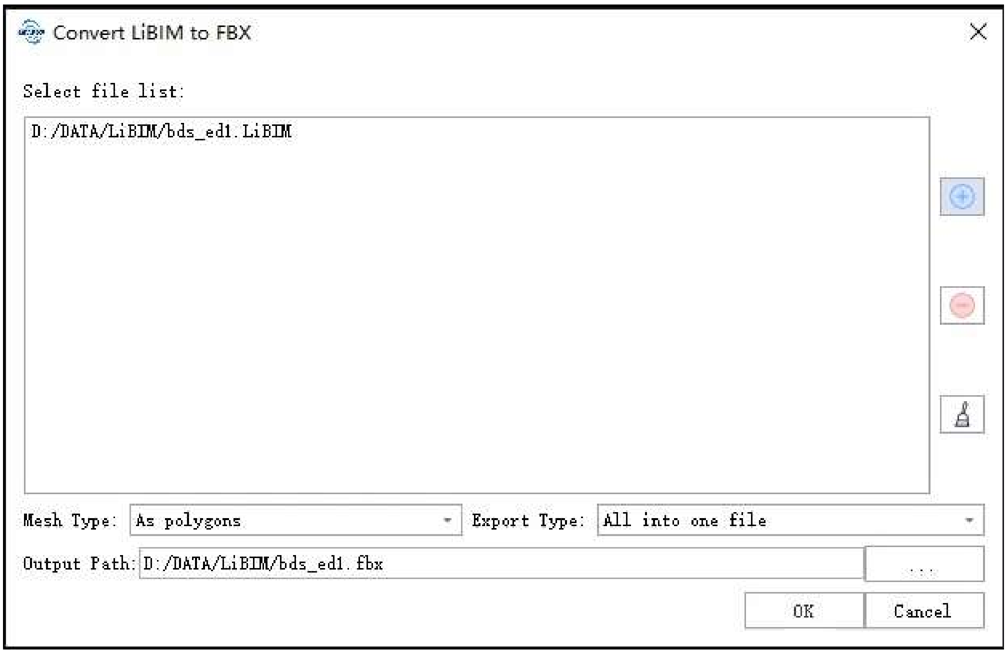
- **File list**: Input LiBIM files to be converted. Users need to click the button to add the file data to be converted from outside; Users can select a file in the list and click button to remove the file from the list. Click button to clear all the data in the list.
- **Mesh Type**: Choose between polygons and triangles.
- **Export Type**: Choose to export all building models as a single file, or export each building model as a separate file.
- **Output path**: Select the path of the output file directory. After the algorithm is executed, the corresponding obj file is generated.

Convert LiBIM to FBX

Convert LiBIM format data into standard 3D model file in .fbx format.

Usage

Click *Data Management* > *3D Building Tools* > *Convert LiBIM to FBX*.



Parameters Settings

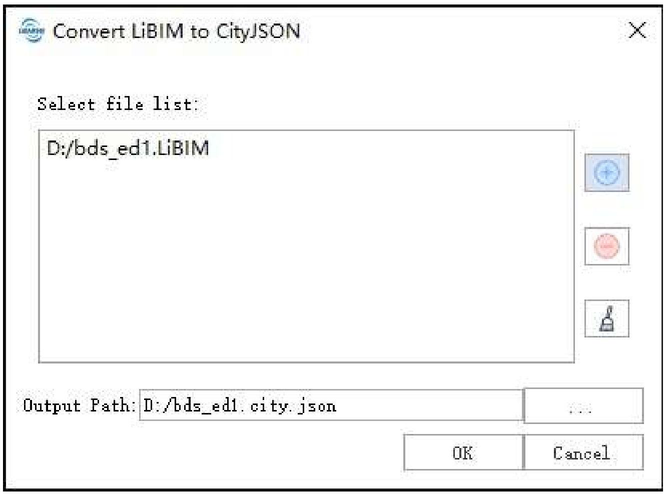
- **File list**: Input LiBIM files to be converted. Users need to click the button to add the file data to be converted from outside; Users can select a file in the list and click button to remove the file from the list. Click button to clear all the data in the list.
- **Mesh Type**: Choose between polygons and triangles.
- **Export Type**: Choose to export all building models as a single file, or export each building model as a separate file.
- **Output path**: Select the path of the output file directory. After the algorithm is executed, the corresponding fbx file is generated.

Convert LiBIM to CityJSON

Convert LiBIM format data into standard city.json format file.

Usage

Click *Data Management* > *3D Building Tools* > *Convert LiBIM to CityJSON*.



Parameters Settings

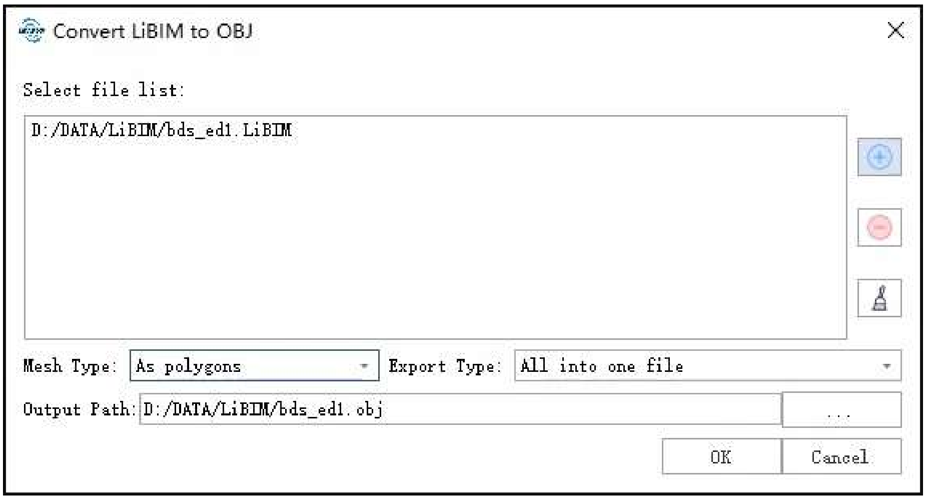
- **File list**: Input LiBIM files to be converted. Users need to click the button to add the file data to be converted from outside; Users can select a file in the list and click button to remove the file from the list. Click button to clear all the data in the list.
- **Output path**: Select the path of the output file directory. After the algorithm is executed, the corresponding city.json file is generated.

Convert LiBIM to Collada

Convert LiBIM format data to standard 3D model .dae format files.

Usage

Click *Data Management* > *Model Conversion* > *Convert LiBIM to Collada*.



Parameter Settings

- **File List**: Input the .LiBIM files to be converted.
  - Click the button to add files from an external source.
  - Select a file in the list and click to remove the selected file.
  - Click to clear all data from the list.
- **Output Each Building Model as a Separate File**: If checked, each building model in the LiBIM will be output as a separate file. If unchecked, all building models in the LiBIM will be output to a single file.
- **Output Path**: Select the directory path for the output file. After the algorithm execution, the corresponding .dae files will be generated.

Vector Conversion

Vector Conversion Tool Provides Mutual Conversion of Vector Format Files.

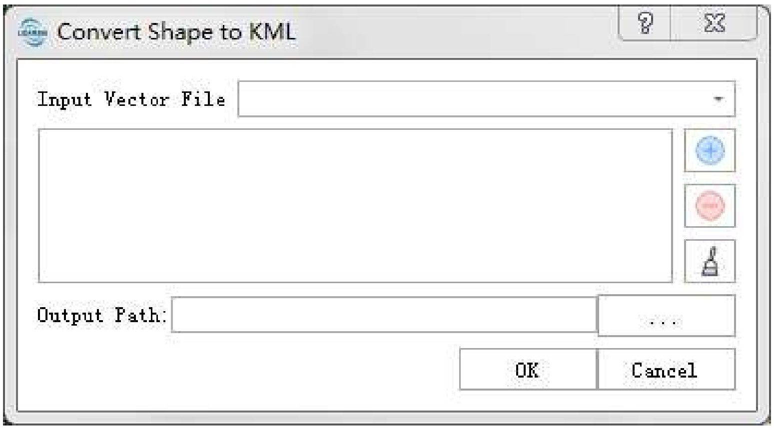
- Convert Shape to KML
- Convert KML to Shape

Convert Shape to KML

The data in shape format will be converted to KML format. KML format reference：https://wiki.openstreetmap.org/wiki/KML

Usage

Click *Data Management* > *Conversion* > *Convert Shape to KML*.



Parameters

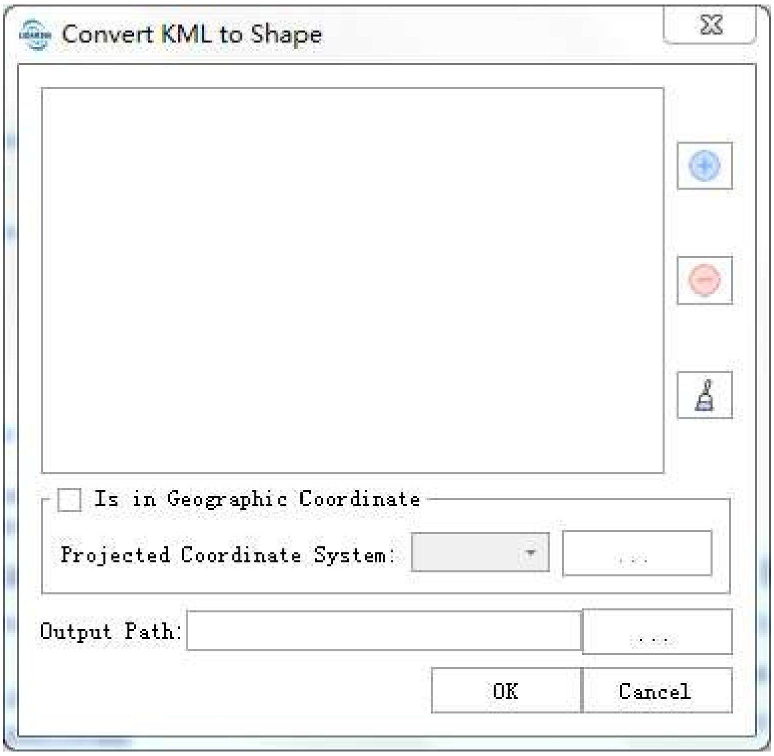
- **Input Shape File**: Input the vector files that need to be converted.
- **Projection Coordinate System**: KML files contain geographical coordinates (latitude and longitude) for points, lines, and polygons. By selecting a projection coordinate system, you can convert the geographical coordinates in the KML file to projected coordinates. For information on choosing a coordinate system, refer to Define Projection. If no projection coordinate system is selected, the coordinates will be output as they are.
- **Output Path**: Choose the directory path for the output files, specifying where the KML files will be saved.

Convert KML to Shape

The data in KML format will be converted to shape format. KML format reference：https://wiki.openstreetmap.org/wiki/KML

Usage

Click *Data Management* > *Conversion* > *Convert KML to Shape*.



Settings

- **Input KML File**： Users can select the files need to be processed from the drop-down list. The files will be added to the list of files to be processed.
- ： Users can insert external files to be processed.
- ： Users can select one file in the list of files to be processed, and click this button to remove this file from that list.
- ： Click this button to clear the list of files to be processed.
- **Under Geographic Coordinate System**: Check this option to convert the geographic coordinates in the kml file to projected coordinates during export to shp. For the selection of projected coordinate system, refer to Define Projection
- **Output Path**： Path of output folder. After the function is executed, the converted Shape file is generated.

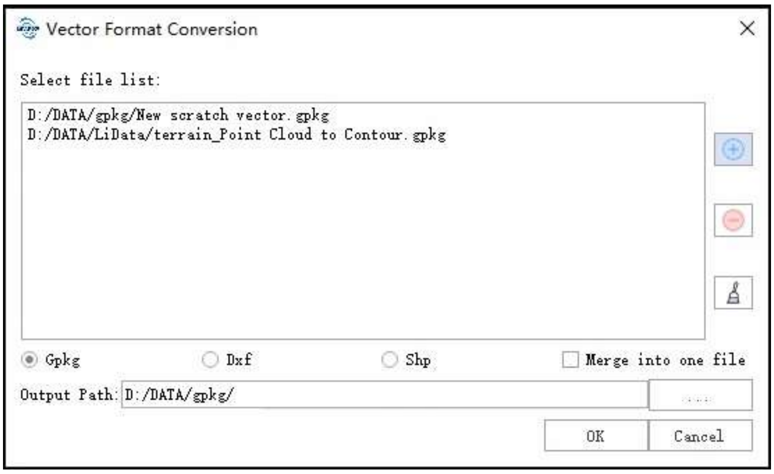
Vector Format Conversion

Description

Convert vector data among gpkg, dxf, and shp formats.

Usage

Click *Data Management* > *Vector Conversion* > *Vector Format Conversion*.



Parameter Settings

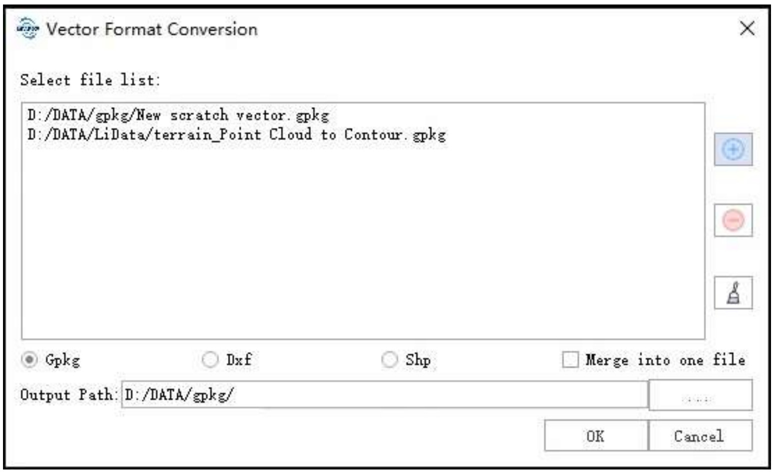
- **File List**: Input the vector files that need to be converted. Users need to click the button to add the file data that needs to be converted from outside. Users can select a file data in the list and click the button to remove it from the list. Clicking on this button clears all data in the list.
- **Output format**
- **Gpkg:** Output vector files in gpkg format
- **Dxf:** Output vector files in dxf format
- **Shp:** Output vector files in shp format
- **Output as one file**: If selected, convert all files in the list into one output format of a vector file; otherwise, each file is converted into an output format of a selected output type.
- **Output path**: Selects the directory path for outputting files. After conversion is complete, corresponding vector files are generated.

ASCII to Vector

Convert ASCII files to vector format, supporting txt, csv, and treedb files.

Usage

Click *Data Management* > *Vector Conversion* >*Convert ASCII to Vector*.



Parameter Settings

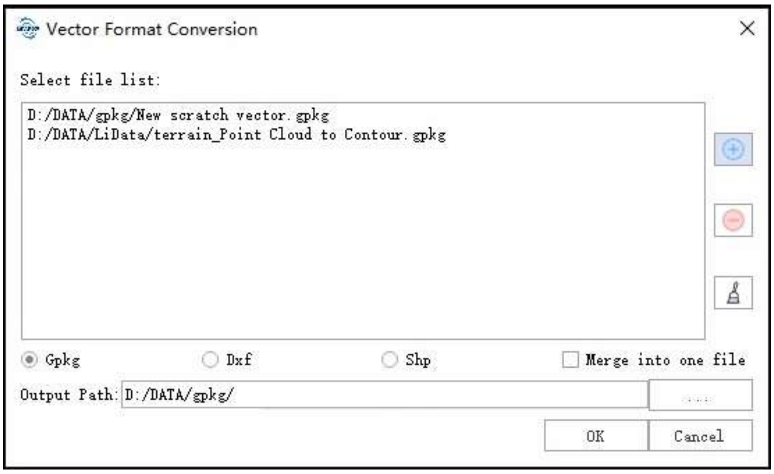
- **File List**: Input the vector files to be converted. Users need to click the button to add files from external sources; users can select a file in the list and click the button to remove it from the list. Click the button to clear all files from the list.
- **Geometry Type**: Select the geometry type of the input file. Internally, FID and geometry type will determine how the data is converted.
- **Output File Type**: Select the format of the output file, supporting gpkg, dxf, and shp.
- **Output Path**: Select the directory path for the output files. After conversion, the corresponding vector files will be generated.

Convert Vector to ASCII

Convert vector data (gpkg, dxf, shp formats) to ASCII format (txt, csv).

Usage

Click *Data Management* > *Vector Conversion* > *Convert Vector to ASCII*.



Parameter Settings

- **File List**: Input the vector files you want to convert.
  - Click to add files from external sources.
  - Select a file in the list and click to remove it.
  - Click to clear all files from the list.
- **HeaderRow**: Output header row including FID, X, Y, Z, and attribute table field names.
- **Attributes**: Output attributes. If selected, attribute fields will be output after basic element coordinates.
- **Delimiter**: Select the delimiter for the output file: space, comma, semicolon, or tab.
- **Output Format**:
  - **txt**: Output as a text file in txt format.
  - **csv**: Output as a text file in csv format.
- **Output Path**: Choose the directory path for the output files. After conversion, the corresponding files will be generated here.

Extract

The extraction tool can filter the data based on several criteria and generate new data files.

- Extract by Class
- Extract by Elevation
- Extract by Intensity
- Extract by Return
- Extract by GPS Time
- Extract by Additional Attributes
- Extract by Segmentation

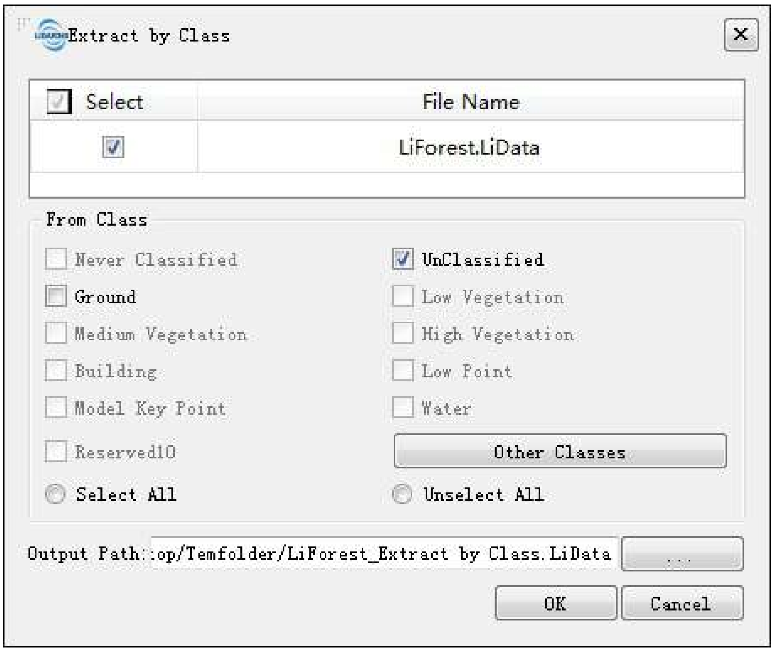
Extract by Class

Summary

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

Usage

Click *Data Management* > *Extract* > *Extract by Class*



Settings

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

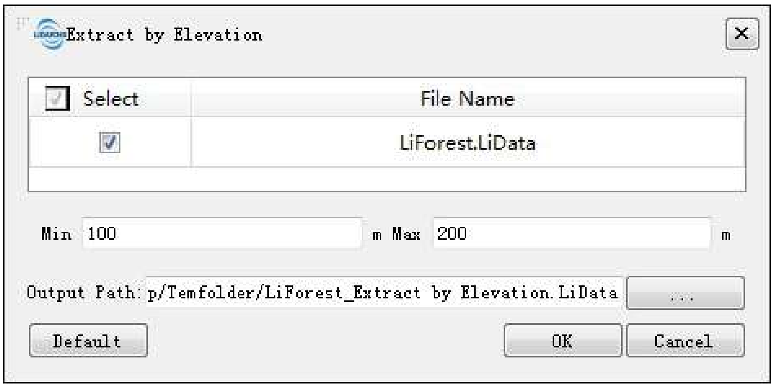
Extract by Elevation

Summary

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

Usage

Click *Data Management* > *Extract* > *Extract by Elevation* S



Settings

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

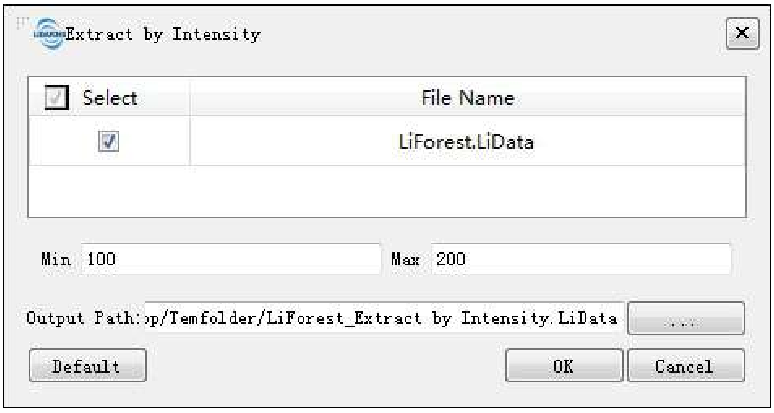
Extract by Intensity

Summary

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

Usage

Click *Data Management* > *Extract* > *Extract by Intensity*



Settings

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

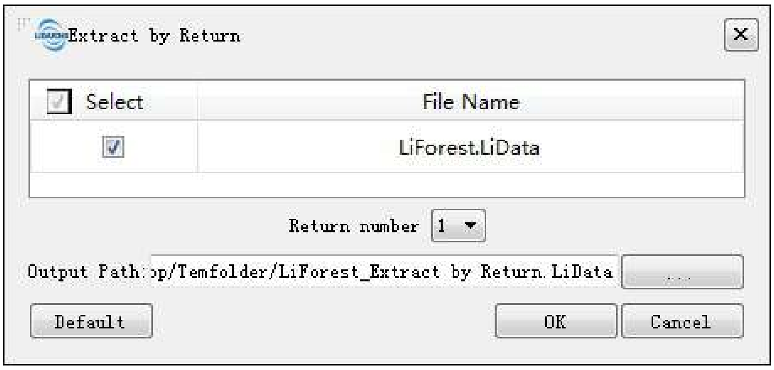
Extract by Return

Summary

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

Usage

Click *Data Management* > *Extract* > *Extract by Return*



Settings

- **Input Data**: Input one or more point cloud data files. File Format: *.LiData.
- **Return number**: Users need to select the return number of the point cloud data to be extracted. Click on the drop-down menu to select the return number, including: first return, last return, 2nd return, 3rd return, 4th return, and returns after 4th.
- **Output path**: Path of the output folder. After the function being executed, the extracted new file(s) will be generated.

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

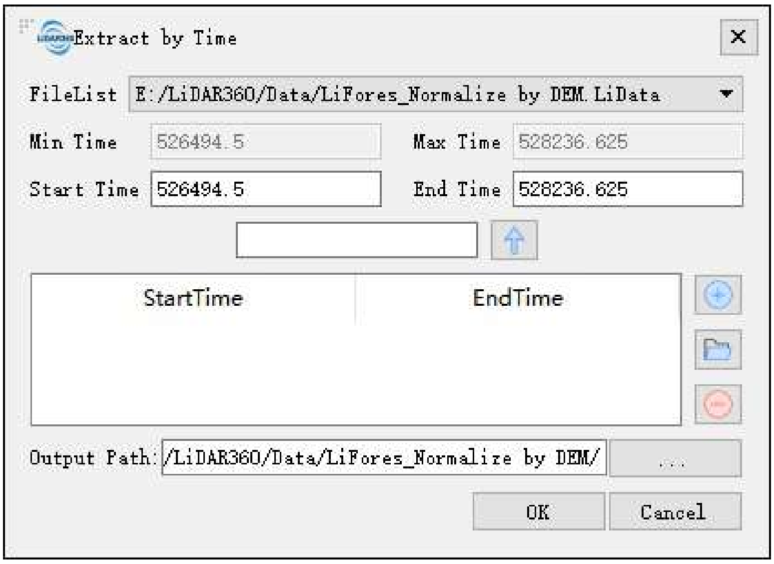
Extract by Time

Summary

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

Usage

Click *Data Management* > *Extract* > *Extract by Time*



Settings

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

- **Output Path**: Path of the output folder. After the function being executed, the extracted new file will be generated.

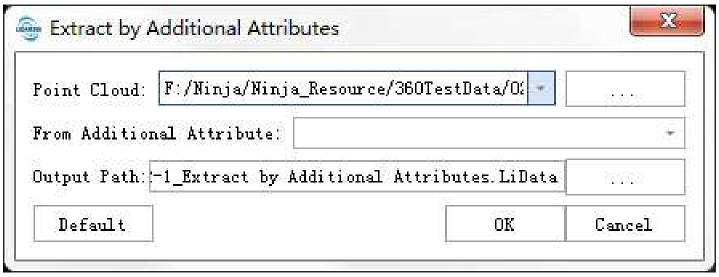
Extract by Additional Attributes

Functional Overview

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

Usage

Click on *Data Management* > *Extract* > *Extract by Additional Attributes*



Parameters Settings

- **Point cloud Data**: Select the point cloud data file, currently only one point cloud data is supported. File format: *.LiData.
- **From Additional Attribute**: User needs to select the attached property name for filtering from the checkbox. This function cannot be run if the point cloud data does not contain additional attributes.
- **Component Selection**: supports one component or multiple components, the number of components depends on the additional properties of the point cloud. Each component corresponds to an optional set of ranges. The point cloud and additional attributes within the range will be extracted to generate a new point cloud file.
- **Output Path**: The path of the extracted output file after the function is executed.

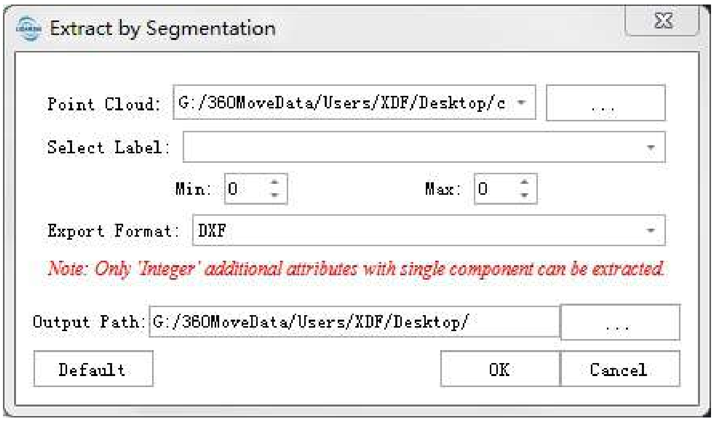
Extract by Segmentation

Functional Overview

The Extract by segment tool extracts point clouds based on user-selected ranges of integer single-component additional attributes. This feature currently only supports the processing of a single point cloud.

Usage

Click *Data Management* > *Point Cloud Tools* > *Extract by segment*



Parameters Settings

- **Point Cloud**: Select the point cloud data file, currently only one point cloud data is supported. File format: *.LiData.
- **Select Label**: The user needs to select the additional attribute name from the checkbox as the label. The label must be integer but component. This function cannot be run if the point cloud data does not contain additional attributes of integer single component. The selected additional attributes will be used as tags, and points with the same tags will be extracted as a file. Assuming that LiData contains an integer single-component additional attribute A,in which the A attribute value of 10 points is "1", and the A attribute value of 20 points is “2”,then the function will generate two files C after running.(including 10 points, the attribute is "1") and D (including 20 points, the attribute is "2");
- **Range Selection**: Select the range of labels. Only point clouds whose labels are within this range are extracted.
- **Export Format**: The format for exporting point clouds, currently DXF and Las are supported.
- **Output Path**: The path of the extracted output file after the function is executed.

Statistics

LiDAR360 can be used for statistical analysis of point cloud data and raster data.

- Grid Statistics
- Volume Statistics
- Raster Statistics

Grid Statistics

Functional Overview

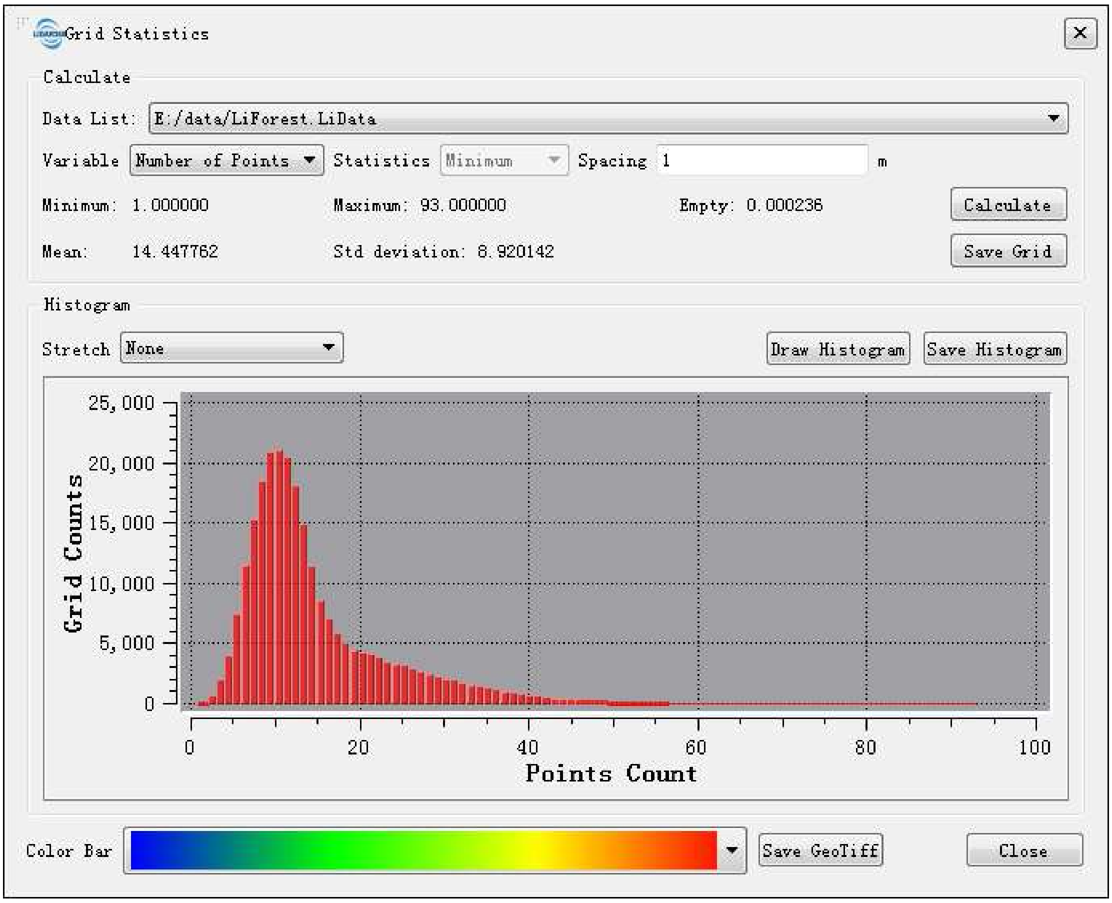
LiDAR360 supports fast grid statistics analysis on point cloud data. The tool supports meshing of point cloud data to count the number of points, density, and Z attributes in the grid. It can be used to view the point cloud data flight and processing quality, as well as analyze the geographical features of the flight area. If only one type of data in point cloud data is counted, the Extract tool can be used to extract point clouds and perform statistics.

Steps

1. Select the point cloud data file to analyze from the **Data List** drop-down box.
2. Select **Variable** to be analyzed. Users can select the variables**Z** to be analyzed from the drop-down box.
3. Set **Spacing** size.
4. Click the **Calculate** button.
5. Click **Draw Histogram** to view the histogram of the grid statistics. **(Optional)** You can also use **Stretch** method to stretch the histogram.
6. **(Optional)**Users can click **Save Histogram** to save the displayed histogram as a PDF file.
7. **(Optional)** Users can click **Save Grid** to save the statistics grid as a single-band grayscale image, **or select Color Bar** and click **Save GeoTiff** to map single-band grayscale images to multiband color images. It should be noted that if the user selects **Stretch** action, the single-band image will be mapped after stretching.

Usage

Click *Statistics > Grid Statistics*



Parameters Settings

- **Data List**: Users need to select point cloud data to be counted from the drop-down list.
- **Variable**: Users need to select the variable to be analyzed from the drop-down list. Fill the statistic value of the variable in the grid.
  - **Number of Points (default)**: Count the number of points in the grid.
  - **Density**: Calculate the density of the point cloud within the grid, which is obtained by dividing the number of points by the grid area.
  - **Z**: To calculate the user-defined Z-value of the point cloud in the grid, users need to select the statistics list.
- **Statistics**: If you select **Z** variable in the **Variable** list, users need to select the statistical method for **Z** variable.
  - **Minimum (default)**: The minimum value of Z variable in the statistics grid.
  - **Maximum**: The maximum value of Z variable in the statistics grid.
  - **Mean**: The average value of Z variable in the statistics grid.
  - **Range**: The range of Z variable in the statistics grid (range = maximum - minimum).
  - **Deviation**: The standard deviation of Z variable in the statistics grid.
- **Spacing (m, default value: "1")**: The size of the statistics grid.
- **Calculate**: After setting the parameters, click this button to calculate.
- **Save Grid**: Save the statistics grid as a single- band grayscale image in the format of *.tif or* .csv.
- **Stretch**: Users can choose **whether to stretch** Draw Histogram **or** Save GeoTiff.
  - **None (default)**: Do not perform any stretching operation.
  - **Minimum and Maximum**: Use the maximum and minimum values of the statistical grid data for

stretching. Linear stretch is applied based on the minimum and maximum pixel values, which will be regarded as the endpoints of the histogram. For example, the minimum and maximum values of a image are 2488 and 2656, linear stretch will rearrange the values based on a linear equation calculated from the maximum and minimum values to re-distribute pixel values from 0 to 255.
    - **Minimum (default value is the minimum of the grids)**: The minimum value of the stretch.
    - **Maximum (default value is the maximum of the grids)**: The maximum value of the stretch.
  - **Std Deviation**: The stretching operation is performed using the standard deviation of the statistical grid data. The quasi-difference stretch type applies the linear stretching to the maximum value (maximum = mean + **n** *standard deviation) and minimum value (minimum = mean -* ***n*** standard deviation).
    - **n**: The multiple of the standard deviation.
      - **Percent Clip**: Percent stretch uses statistical grid data. Percent Stretch applies a linear stretch between the pixel values of the minimum and maximum cropping percentages. For example, the minimum and maximum values of a image are 2488 and 2656 respectively, and the user-defined minimum and maximum values of clipping percentage are 0.02 and 0.98 respectively. Values less than 0.02 stand for values between 2488 and 2492, and values above 0.98 stand for values between 2652 and 2656. The value between 2488-2492 will become 0, while the value between 2652-2656 will become 255. Other values will be distributed between 0-255.
    - **Minimum**: The minimum stretch percentage, range 0~1.0.
    - **Maximum**: The maximum stretch percentage, range 0~1.0.
- **Draw Histogram**: Draw a histogram of grid statistics in the canvas.
- **Save Histogram**: Save the drawn histogram in the PDF format.
- **Color Bar**: LiDAR360 provides 11 color bar options for users to choose.
- **Save GeoTiff**: Save statistical grid data **as a multi-band color** image based on the user-selected Color Bar.
- **Close**: Close the tool.

Note: The Save Grid function saves the original statistical result as a single-band TIFF file. Save GeoTiff function saves the result after histogram stretch and performs RGB three-channel mapping according to the selected color bar.

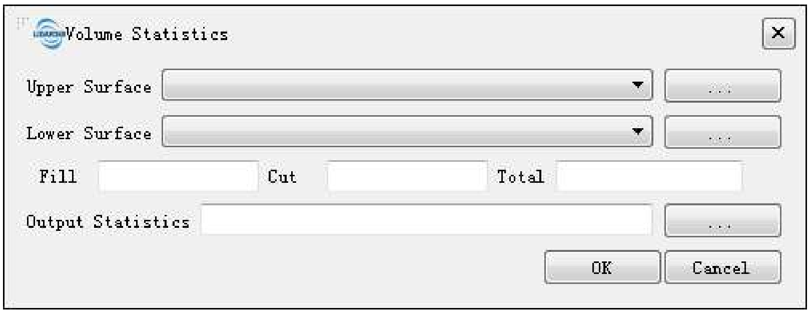
Volume Statistics

Summary

LiDAR360 supports the use of surface model data (single-band tif data) with spatial overlaps to calculate volume statistics. By subtracting the lower surface from the upper surface, the statistical filling amount (**Fill**), excavation amount (**Cut**) and total fill volume (**Total**) (calculated by subtracting fill volume from volume of excavation) can be obtained.

Usage

Click *Statistics* > *Volume Statistics*



Settings

- **Upper Surface**: Users can select the *.tif format file from the drop-down list, or click ... to select external file.
- **Lower Surface**: Users can select the *.tif fornat file from the drop-down list or click ... to select external file.
- **Output Statistics**: Specify the path to output the statistical result as a *.txt format file. The file contains the path of the upper and lower surface files, spatial resolution, XSize, YSize, and the amount of the filler.

> Note: The upper and lower surface files are single-band data, they must have the same spatial resolution and the spatial ranges have intersections. The unit of the point cloud data processed by LiDAR360 is meter, and the calculated fill volume, excavation volume, and total amount are in cubic meter.

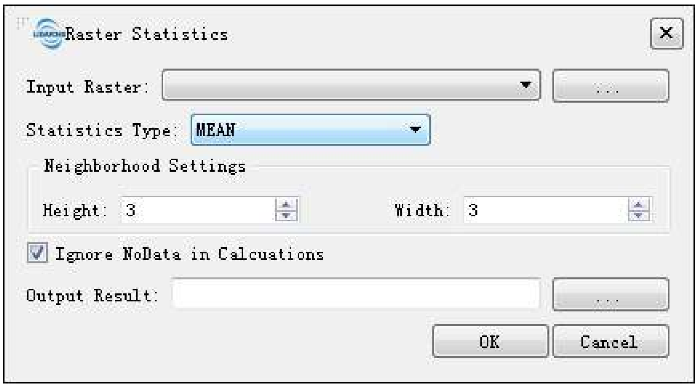
Raster Statistics

Summary

This function can perform neighborhood calcuation for each cell in a tif format raseter data. The output can be different kinds of statistics of the neighborhood. The statistics include mean, maximum, minimum, range, standard deviation, and sum.

Usage

Click *Statistics* > *Raster Statistics*



Settings

- **Statistics Type ("Mean" by default)**: Users can choose any type of statistics in the drop-down menu. The statistics include mean, maximum, minimum, range, standard deviation, and sum.
    - **Mean**: Calculate the mean value for the neighborhood of each cell to be as the value of the cell at the same location as the orignal cell.
        - **Maximum**: Calculate the maximum value for the neighborhood of each cell to be as the value of the cell at the same location as the orignal cell.
        - **Minimum**: Calculate the minimum value for the neighborhood of each cell to be as the value of the cell at the same location as the orignal cell.
        - **Range**: Calculate the range for the neighborhood of each cell to be as the value of the cell at the same location as the orignal cell.
        - **STD**: Calculate the standard deviation for the neighborhood of each cell to be as the value of the cell at the same location as the orignal cell.
        - **Sum**: Calculate the sum for the neighborhood of each cell to be as the value of the cell at the same location as the orignal cell.
- **Neighborhood Settings**: Settings of the neighborhood size.
    - **Height**: Height of the neighborhood.
    - **Width**: Width of the neighborhood.
- **Ignore NoData in Calculation (Checked by default)**: If this option is checked, the software will will ignore the NoData cells in the neighborhoods during the calcuation. If this option is not checked, software will define the cell with NoData in its neighborhood as NoData. If this option is checked, for those NoData cells, the software will calcuate the statistics with the values from their neighborhoods.

If the all the cells in the neighborhood of a cell are NoData, whether or not this option is checked, this cell will be defined as NoData during the calculation.

- **Output Result**: The path for the result of raster statistics.

Classification

LiDAR360 can be used to classify unclassified point cloud, or to reclassify points that have already been classified.

- Slope Filter
- Conicoid Filter
- Classify Ground Points
- Classify Ground Points by CSF
- Extract Median Ground Points
- Classify by Attribute
- Classify by Additional Attributes
- Classify Low Points
- Classify Below Surface Points
- Classify Isolated Points
- Classify Noise Points
- Classify Air Points
- Classify by Height Above Ground
- Classify by Min Elevation Difference
- Classify Closeby Points
- Classified by Region
- Classify Buildings
- Classify Model Key Points
- Classify Water Points
- Classify by Machine Learning
- Classify by Trained ML Model
- Classify Ground by Selected
- Classify by Deep Learning
- Custom Deep Learning Classification
- Classify by Interactive Edit

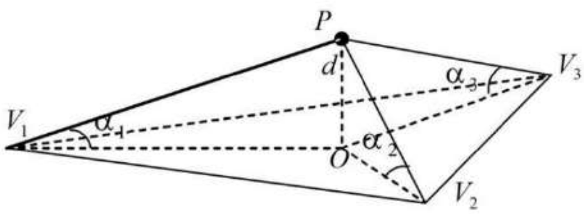
Classify Ground Points

Summary

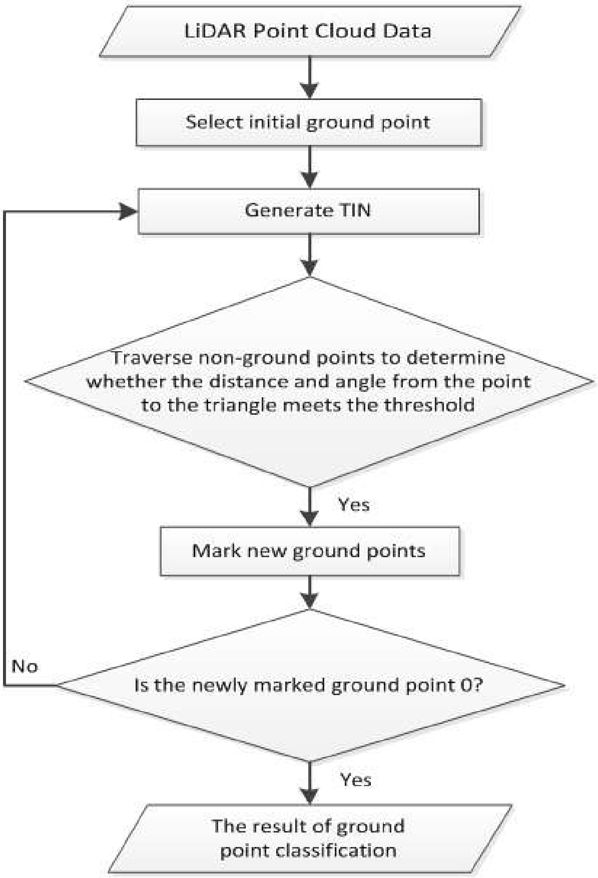
Ground points classification is an important operation of point cloud preprocessing, which can be implemented in LiDAR360 with an improved progressive TIN densification filtering algorithm (Zhao et al.,2016).

The algorithm first generates a sparse triangulated irregular network(TIN) through seed points, and then iteratively processes layer-by-layer densification until all ground points have been classified. The specific steps of the algorithm are described as follows:

1. The initial seed point selection. If the point cloud contains buildings, the **maximum building size** is taken as the grid size to grid the point cloud, and for the point cloud without the building, the default value is used as the grid size. Take the lowest point in the grid as the starting seed point.
2. Build a TIN. The initial TIN was constructed using the initial seed point.
3. Iterative densification process. Traverse all the unclassified points, query the triangles that each point belongs to in the horizontal projection plane; Calculate the distance(d) from the point to the triangle and the max angle between the point and three vertices with the triangles plane. As shown in the figure below, the distance and max angle are compared with the **Iteration distance** and **Iteration angle**, separately. If it is less than the corresponding threshold, then the point is classified as a ground point and added to the triangulation. Repeat this process until all ground points have been classified.

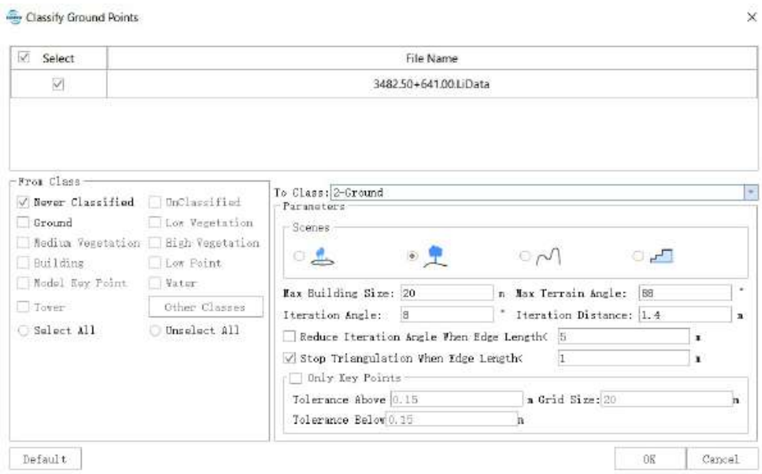


The flow of algorithm is shown in the figure.



Usage

Click *Classify* > *Classify Ground Points*



Settings

- **Input Data**: The input file can be a single point cloud data or a point cloud dataset, which must be opened in the LiDAR360 software.
- **From Class**: Source class(es).
- **To Class**: Target class.
- **Scenario (optional)**: For different terrain scenarios, you can choose flat terrain, gentle slope terrain,

steep slope terrain, and mountain terrain. Check different terrain features, corresponding to different default parameters.

- **Max Building Size (m, default value is "20")**: The maximum length of the building edge that exists in the point cloud scan. If this parameter is set too small, the flat roof of the building may be mistaken for the terrain. When there is a building in the point cloud data, the maximum building size can be measured by using the Length Measurement in the menu bar. The value of this parameter should be greater than the measured value. For point cloud data without buildings, this parameter can use the default value of 20m.
- **Max Terrain Angle (°, default value is "88")**: The maximum slope of the terrain shown in the point cloud. This parameter can determine whether the points nearby the ground points belong to the ground or not. Usually, the parameter can be set as default.
- **Iteration Angle (°, default value is "8")**: The allowable range of angles between unclassified points and ground points. The area with large topographic relief can be appropriately set larger and adjusted accordingly with the iteration distance. Generally, it is set to 6 to 12 degrees.
- **Iteration Distance (m, default value is "1.4")**: Distance threshold between the unclassified points and the triangle in the triangle mesh. When the terrain undulations are large, it can be appropriately enlarged and adjusted accordingly to the iteration angle, which is generally set at 1.2-1.6 meters.
- **Reduce Iteration Angle When Edge Length < (m, default value is "5", Optional)**: When the triangle length of the points to be classified is smaller than the threshold, the iteration angle should be decreased. If this parameter is checked, it means that when the triangle side length of the point to be classified in the triangulation network is less than the threshold value, the iteration Angle can be reduced accordingly to obtain smoother ground points. When a sparse ground point needs to be obtained, the threshold may be increased accordingly.
- **Stop Triangulation When Edge Length < (m, default value is "2", Optional)**: When the triangle length of the point to be classified corresponds to the length of the triangle is less than the threshold, the densification of triangulation network is stopped. This value can prevent the locally generated ground point from being too dense. When this value increase, the ground points will be sparse, and vice versa.
- **Only Key Points (Optional)**: On the basis of ground point filtering, the key points of the model are further extracted as the ground point category, which can preserve the key points on the terrain and comparatively extract the points in the gentle ground area. For the specific usage of this function, please seeClassify Model Key Points.

> Note: Because the actual terrain is complex and changeable, when using this function to perform ground point classification, different parameters needs to be adjusted in order to achieve relatively ideal results. In addition, the classification result in local area can be reclassified by Classify Ground by Selected and the Classify by Interactive Editing tool.

```
@inproceedings{
    author={Zhao X Q, Guo Q H, Su Y J and Xue B L},
    title={Improved progressive TIN densification filtering algorithm for airborne LiDAR data in forested areas},
    booktitle={ISPRS Journal of Photogrammetry and Remote Sensing,117:79-91},
    year={2016}
}
```

Classify Ground Points by CSF

Functional Overview

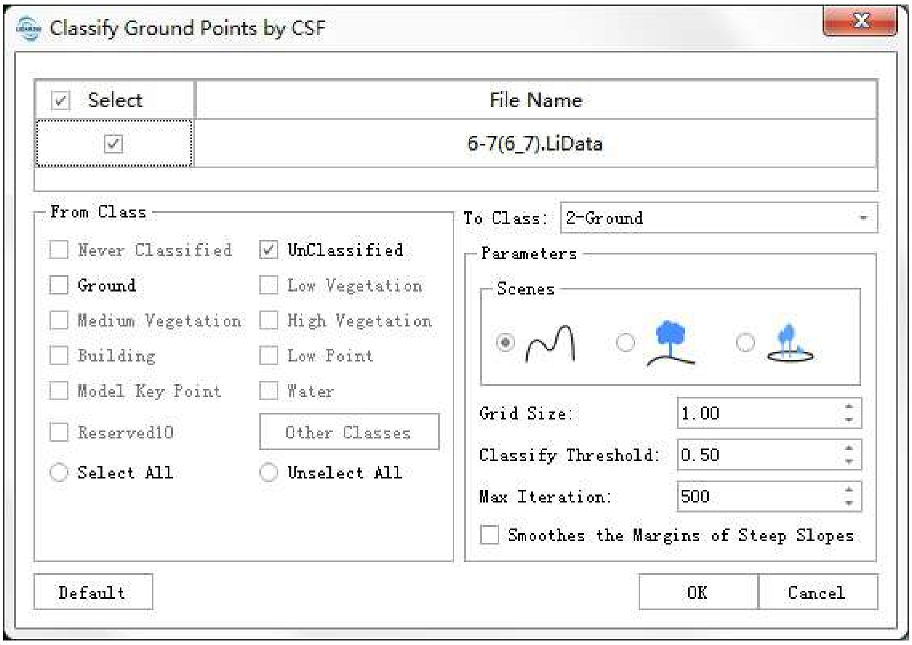
Ground point classification is the basic operation of point cloud data processing. This function adoptsGround Point Filtering Algorithm Based on Cloth Simulation(Cloth Simulation Filter, IPTD(Zhang et al.,2016)).

This algorithm can be divided into the following steps:

1. Invert the Z direction of the point cloud. The point cloud is divided into a grid with a certain resolution in the XY direction, and the nodes of the grid are used as a simulated cloth.
2. Set the initial height of the cloth to the point cloud maximum and start the iteration. During each iteration, the cloth will "sink" to the point cloud with a certain gravity, and the positional relationship between the cloth nodes and the point cloud is calculated at the same time. Nodes that have landed on the point cloud will not be movable during the next iteration. An immovable node will slow the sinking of surrounding nodes according to the stiffness value r.
3. After a certain number of iterations, calculate the positional relationship between each point and the cloth; points whose distance from the cloth in the Z direction is less than a certain threshold will be classified into target categories.

Usage

Click*Classification > Classify Ground Points by CSF*.



Parameters Settings

- **Input Data**: The input file can be a single data file or a point cloud data set; the file(s) to be processed must be opened in the LiDAR360 software.

- **From Class**: Categories to be classified.
- **To Class:** Classification target category.
- **Scene**: There are three types of scenes: steep slope, gentle slope and flat ground.
- **Grid Size (meters) (default "1.0")**: The resolution of the cloth nodes. 1.0 is suitable for most point clouds. For data with large terrain fluctuations, this value can be appropriately reduced.
- **Classify Threshold (meters) (default "0.5")**: After the iteration is complete, the points whose distance from the cloth in the Z direction is less than this threshold will be classified into the target category.
- **Max Iteration (default "500")**: The algorithm completes the iteration when it reaches the maximum number of iterations or when all cloth nodes are immovable.
- **Smoothes the Margins of Steep Slopes**: When the cloth is located on a steep slope, due to the internal constraints between the cloth nodes, it cannot fit the ground well, and the algorithm may generate large errors. Checking this option can eliminate the effects of steep slopes to a certain extent. If the scene does not contain steep slopes, you can uncheck it.

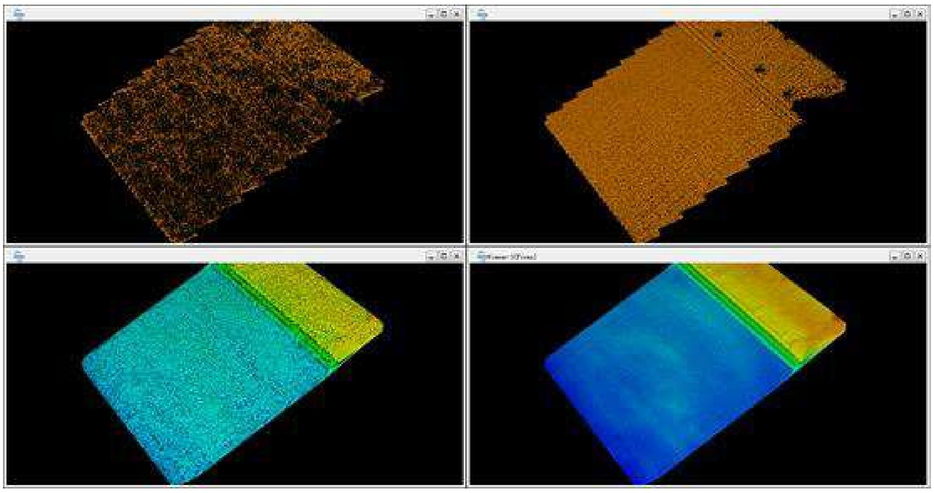
```
@inproceedings{
    author={Zhang W, Qi J, Wan P, Wang H, Xie D, Wang X, Yan G},
    title={An Easy-to-Use Airborne LiDAR Data Filtering Method Based on Cloth Simulation.},
    booktitle={Remote Sensing. 8(6):501.},
    year={2016}
}
```

Extract Median Ground Points

Summary

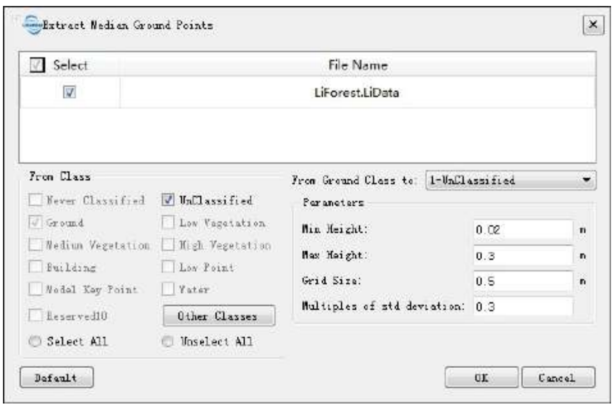
Point clouds generated from data collected by small aircrafts and unmanned aerial vehicles (UAVs) typically have many ground points and individual point densities in these regions can be very high. Traditional point cloud filtering methods used to thin these high density regions often produce ground point subsets with vertical thicknesses that can be measured in centimeters. When these filtered point clouds are used to create 3D mesh objects (e.g. TINs), the results tend to be poor as the 3D modeled surfaces have an uneven appearance.

The Extract Median Ground Points tool can be used to obtain a ground point cloud that is thinner and smoother than those produced using traditional point cloud filtering methods. Generating this type of ground point cloud allows for consideration of the points found in a denser and thicker ground point cloud. **This method belongs to the optimization step after the initial ground point has been extracted. Therefore, the point cloud data must first be classified using ground point classification method.** The comparison results before and after using the median ground point classification method are as follows:



Usage

Navigate to and click on *Classify* > *Extract Median Ground Points*



Settings

- **Input Data**: Single or multiple point cloud data files can be input into the tool. Each point cloud must contain points classified as ground points and have been loaded in the LiDAR360 software.
- **From Class**: Source class(es) that represent ground points or other points are included in the median ground points finding operation.
- **From Ground Class to**: Classify target ground points that do not meet the median rule to this class.
- **Min Height (meters, default value is "0.02")**: The minimum height above the lowest ground point elevation found in the input file(s)from which ground points will be taken and used to find the median ground point elevation. Ground points falling below the Min Height will not be used to find the median ground point elevation value. The default value is set to 0.02 meters.
- **Max Height (meters, default value is "0.3")**: The maximum height above the lowest ground point elevation found in the input file(s)from which ground points will be taken and used to find the median ground point elevation. Ground points falling above the Max Height will not be used to find the median ground point elevation value. The default Max Height value is set to 0.3 meters.
- **Grid Size (meters, default value is "0.5")**: The grid size defines the horizontal extents of the square area from which ground points will be extracted from the input file(s)and then used to find a median ground point elevation value for each grid cell. When the number of points in a grid cell falls below a user-defined threshold, the median ground point elevation for that grid cell will not be included in the output dataset. The default Grid Size is 0.5 meters.
- **Multiples of Std Deviation (default value is"0.3")**: The number and thickness of the extracted point cloud ground points are controlled by the user-defined Multiples of Std Deviation parameter. The default value is 0.3, which means that 22% of ground points are extracted as ground points. Moreover, a value of 0.5 Multiples of Std Deviation corresponds to 40% of all points classified as ground points being extracted. 0, 7 corresponds to 50%, 0.9 corresponds to 62%, and 1.5 corresponds to 86%.
- **DefaultValue**: Click to set all Extract Median Ground Points tool parameters to their default values.

> Note: This classification algorithm is only applicable to thicker ground point data scanned by small aircrafts and unmanned aerial vehicles (UAVs). This data must be classified by ground points first, as an optimization step.

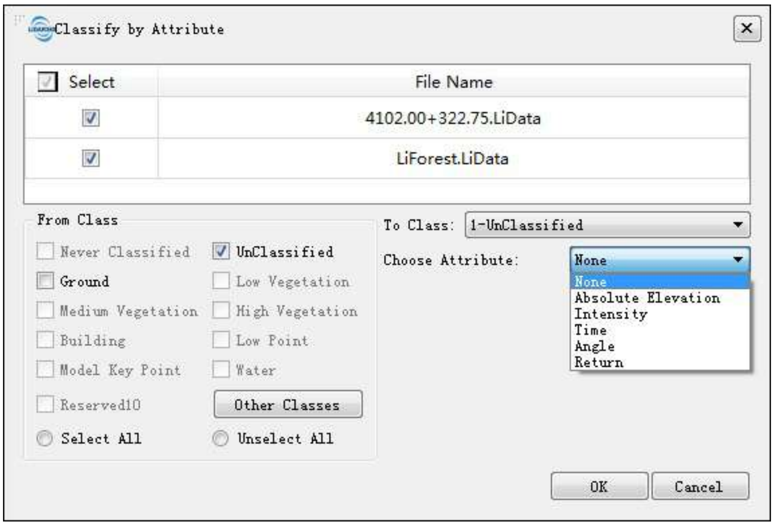
Classify by Attribute

Summary

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

Usage

Click *Classify* > *Classify by Attribute*



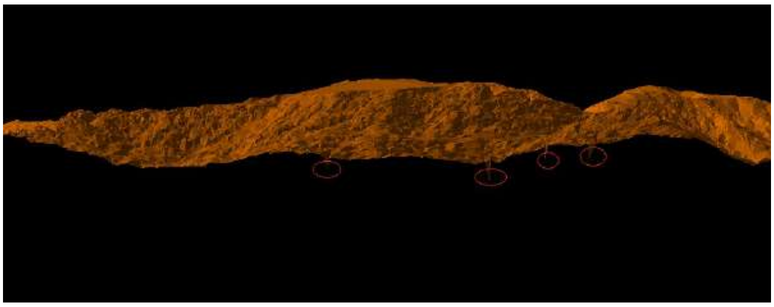
Settings

- **Input Data**: The input file can be a single point cloud data or a point cloud dataset, which must be opened in the LiDAR360 software.
- **From Class**: Source class(es).
- **To Class**: Target class.
- **Choose Attribute**: The selected attribute will be used to classify the point cloud.
  - **None (default)**: The default setting will change all points in the **From Class** option to **To Class**.
  - **Absolute Elevation**: Classified by elevation range. If the elevation value of a point is between the specified range, it will be assigned to the **To Class**.
  - **Intensity**: Classified by intensity range. If the intensity value of a point is between the specified range, it will be assigned to the **To Class**.
  - **Time**: Classified by GPS time. If the GPS time value of a point is between the specified range, it will be assigned to the **To Class**.
  - **Angle**: Classified by scan angle. If the scan angle value of a point is between the specified range, it will be divided into the **To Class**.
  - **Return**: Classified by the return number. If the return number at a point is between the specified range, it will be assigned to the **To Class**.

Classify Low Points

Summary

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



The distribution of low points is divided into individual points or clusters. The algorithmic flow of this function is:

1. Traverse the point cloud and search for points to be classified within a certain range of **radius** of the current single point or point cluster.
2. Calculate the maximum **height difference** between the current point and the neighboring point, which is compared with the threshold.
3. If the value is greater than the threshold, the current point is considered to be a low point, otherwise, it is not classified as a low point.

Usage

Click *Classify > Classify Low Points*

Settings

- **Input Data**: The input file can be a single point cloud data or a point cloud dataset, which must be opened in the LiDAR360 software.
- **From Class**: Source class(es).
- **To Class**: Target class.
- **Points Number (default value is“1”)**: When the number of points is set to 1, the single low point is classified. If it is greater than 1, cluster low points are classified.
- **Radius (m, default value is"5")**: The radius threshold between the unclassified point and the neighboring point.
- **Height (m, default value is"0.5")**: The **height difference** threshold between the unclassified point and the neighboring point.
- **DefaultValue**: Click this button to set all parameters as default.

Classify Below Surface Points

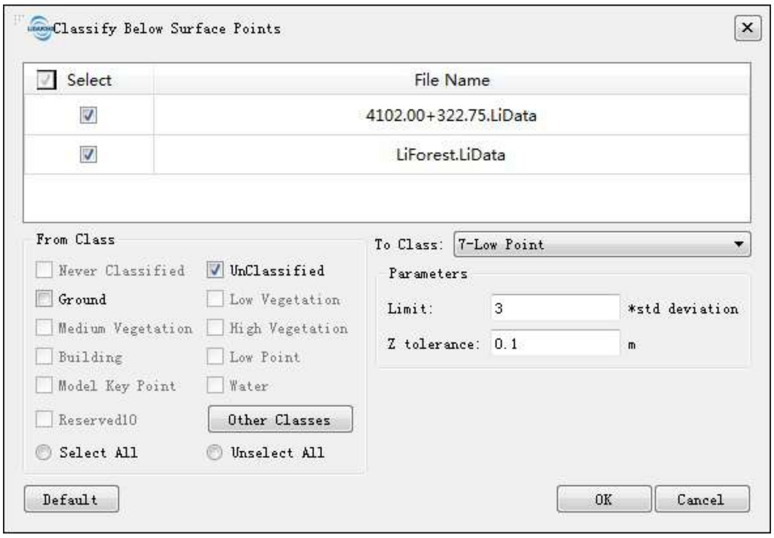
Summary

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

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

Usage

Click *Classify* > *Classify Below Surface Points*



Settings

- **Input Data**: The input file can be a single point cloud data or a point cloud dataset, which must be opened in the LiDAR360 software.
- **From Class**: Source class(es).
- **To Class**: Target class.
- **Limit (default value is"3")**: The multiple of the mean squared error of the neighboring point fitting plane of the unclassified points. The larger the value, the less points will be classified into the target class.

- **Z tolerance (m, default value is"3")**: The threshold of height difference. The point to fit plane distance less than this value is not classified. The larger the value, the less points will be classified into target class.
- **DefaultValue**: Click this button to set all parameters as default.

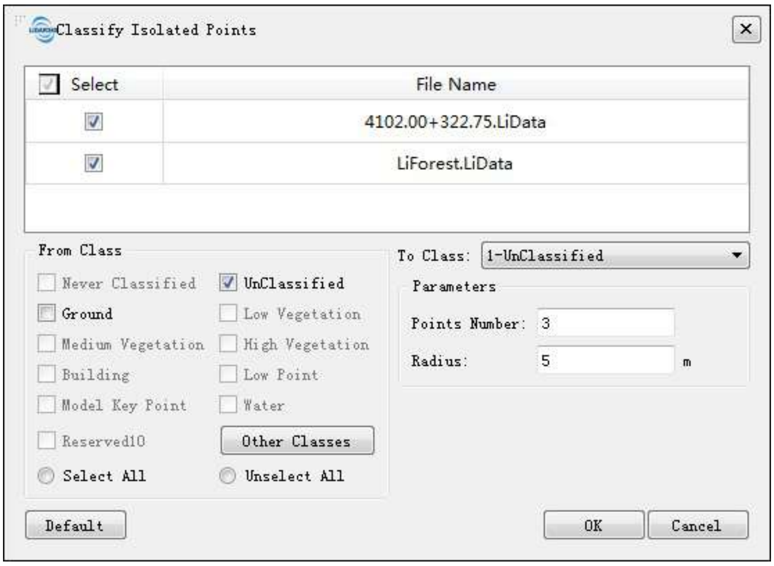
Classify Isolated Points

Summary

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

Usage

Click *Classify* > *Classify Isolated Points*



Settings

- **Input Data**: The input file can be a single point cloud data or a point cloud dataset, which must be opened in the LiDAR360 software.
- **From Class**: Source class(es).
- **To Class**: Target class.
- **Points Number (default value is"3")**: If the number of points in the neighboring radius are less than or equal to the value, the point is considered as an isolated point.
- **Radius (m, default value is"5")**: Neighboring search radius.
- **DefaultValue**: Click this button to set all parameters as default.

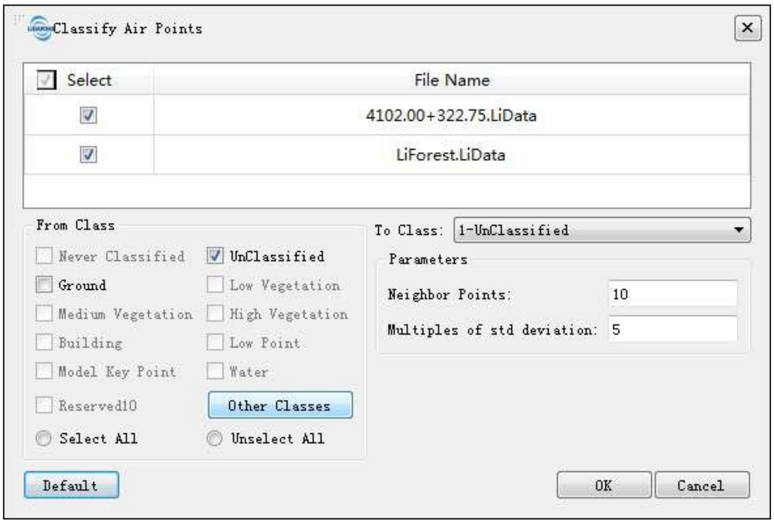
Classify Air Points

Summary

This function classifies points that are significantly higher than the surrounding points into airborne noise points. The idea of the algorithm is the same as the principle of Outlier Removal in the data management module.

Usage

Click *Classify > Classify Air Points*



Settings

- **Input Data**: The input file can be a single point cloud data or a point cloud dataset, which must be opened in the LiDAR360 software.
- **From Class**: Source class(es).
- **To Class**: Target class.
- **Neighbor Points (default value is"10")**: The number of neighbors that will be used to determine whether a point is a noise in the sky. Calculate the distance between each point to the nearest point and calculate the standard deviation of the nearest distances.
- **Multiples of std deviation (default value is"5")**: If the deviation of points beyond the minimum allowable threshold, they are considered as noise in the sky. The larger the threshold, the less the noise will be divided into.
- **DefaultValue**: Click this button to set all parameters as default.

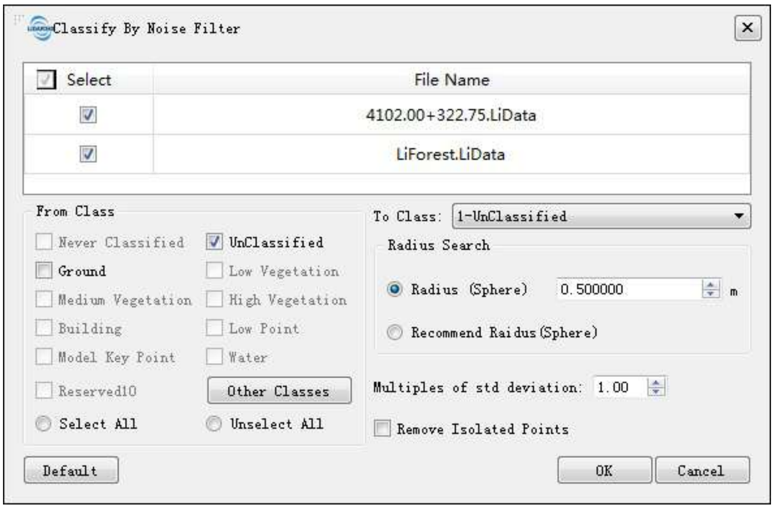
Classify Noise Points

Summary

Classify the outliers in the point cloud data as a certain class in the format of *.LiData.

Usage

Click *Classify* > *Classify Noise Points*.



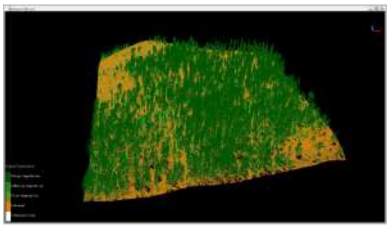
Settings

- **Input Data**: The input data can be a single point cloud file, or can be a point cloud data collection; the data to be processed must be pre-opened in LiDAR360 software.
- **Radius(Default value is "0.5m")**: Set the radius of the fitting plane. This function can be used when users know the approximate density of the point cloud data.
- **Recommended Radius**: Automatically calculate the appropriate searching radius according to the input point cloud data.
- **Multiples of std deviation(Default value is "1.0")**: Using the relative error (sigma) as a parameter for outliers removal. The algorithm will automatically calculate the standard deviation (stddev) of a point P's surrounding fitting plane. If the distance, d, from this point to that plane is less than sigma * stddev, this point, P, will be kept. The reduction of this relative error results in removing more points. Conversely, more points will be retained. The change of this parameter will not ifnluence the efficiency.
- **Remove Isolated Points**: A point will be treated as an isolated point when there is less than 4 points within a distance of the searching radius (cannot create a fitting plane with less than 4 points).

Classify by Height Above Ground

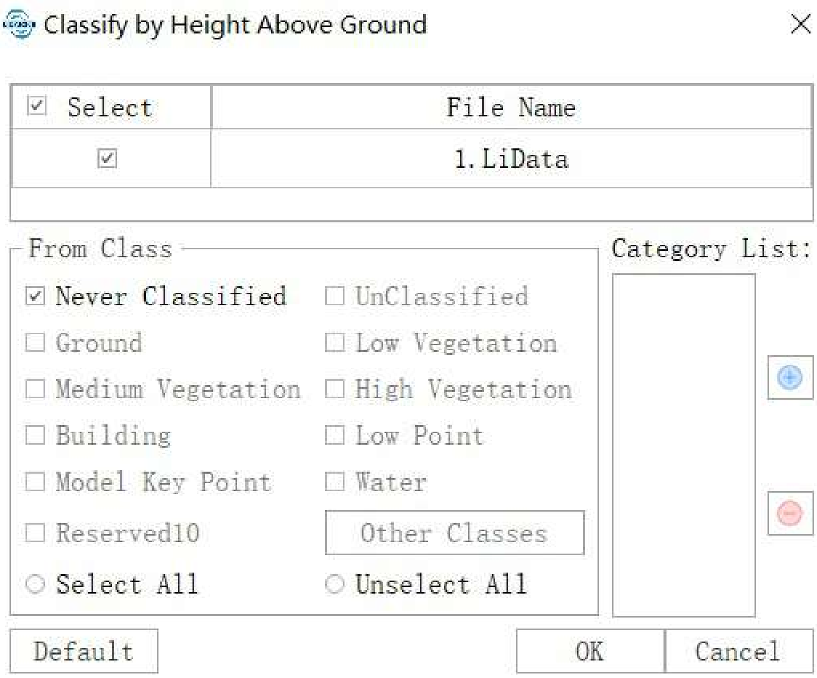
Functional Overview

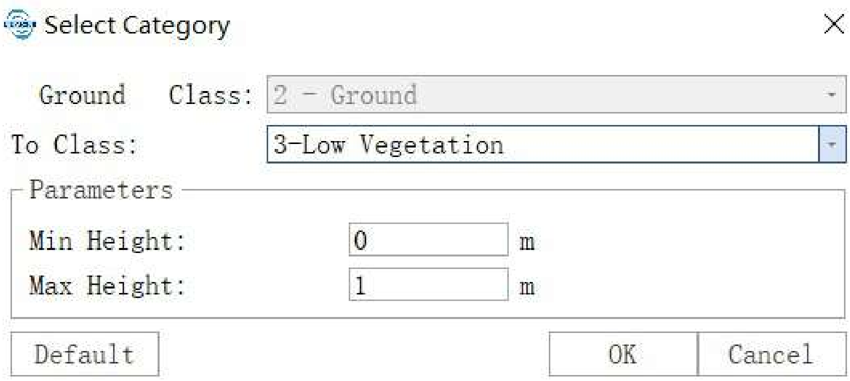
Classify points on the surface of terrain at certain heights. This function can quickly classify vegetation at different heights. For example, this classification can be done three times to distinguish low vegetation (0-1m), medium vegetation (1-10m) and high vegetation (10-100m), as shown in the figure below.



Usage

Click *Classification* > *Classify by Height Above Ground*.





Parameters Setting

- **Input Data**: Make sure that each input point cloud data has been classified with ground points; Input files can be single point cloud data files or point cloud datasets; The data to be processed must be opened in LiDAR360 software.

> Category List (Support setting multiple height ranges for classification)

- **From Class**: The category to be classified.
- **Ground Point Class**: 2-Ground Points
- **To Class**: Target category for classification.
- **Minimum Height(m)(Default is "0")**: Minimum height difference above ground points in the area to be classified.
- **Maximum Height(m)(Default is "1")**: Maximum height difference above ground points in the area to be classified.
- **Default**: Click this button to restore all parameters to default values.

> Note: This function requires ground point categories in the point cloud.

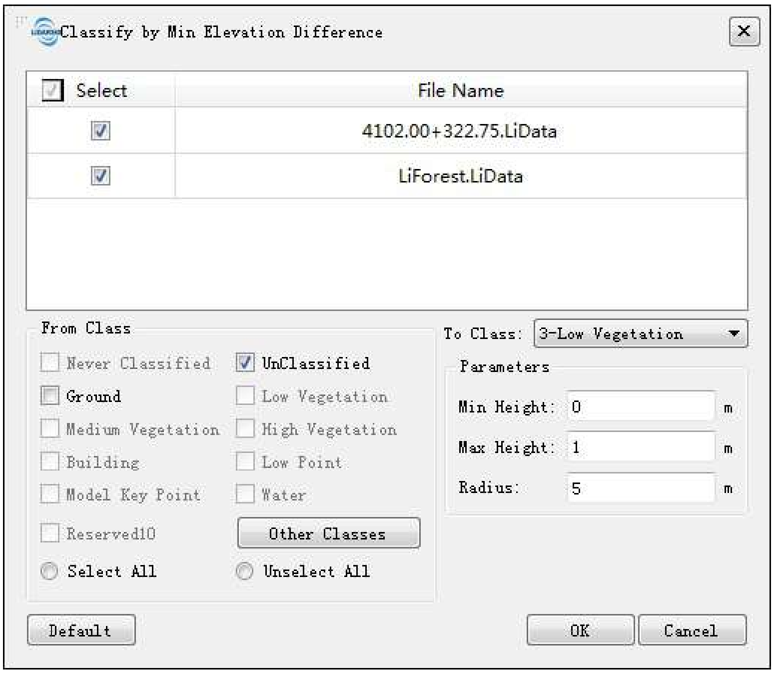
Classify by Min Elevation Difference

Summary

For each point in the data, this function calculates the elevation difference between the lowest point within a specified radius around it and itself. If the elevation difference is between **Min Elevation Difference** and **Max Elevation Difference**, that point will be marked as **Target Class**.

Usage

Click *Classify* > *Classify by Min Elevation Difference*



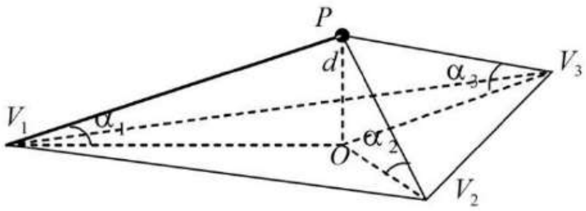
Settings

- **Input Data**: The input file can be a single point cloud data or a point cloud data set, which must be opened in the LiDAR360 software.
- **From Class**: Source class(es).
- **To Class**: Target class.
- **Min Elevation Difference (m, default value is"0")**: Threshold for the minimum elevation difference.
- **Max Elevation Difference (m, default value is"1")**: Threshold for the maximum elevation difference.
- **Radius (m, default value is"5")**: The radius of the area needs to be classified at the current point, which needs to be classified.
- **DefaultValue**: Click this button to set all parameters as default.

Classify Close by Points

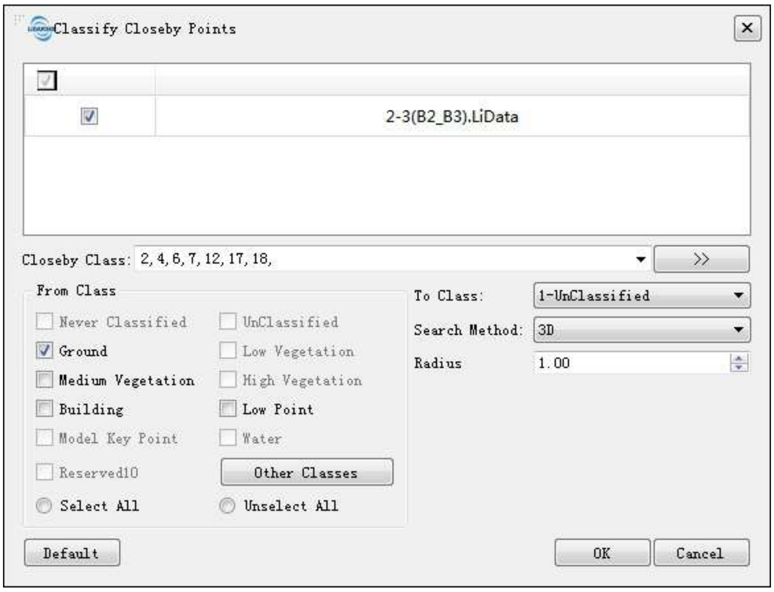
Summary

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



Usage

Click *Classify* > *Classify Closeby Points*.



Settings

- **Input Data**: The input data could be one point cloud file or a set of point cloud files. The data to be processed must be opened in LiDAR360 software.
- **Neighbor Class**: The points of this class which is close to the source points will be classified.
- **Source Class**: The points close to this class will be classified.
- **Target Class**: Target class of the classification.
- **Search Method**: The method for searching in the neighborhood. Support 2D or 3D neighborhood.
  - **Radius**: Neighborhood search radius.

Classify by Range

Summary

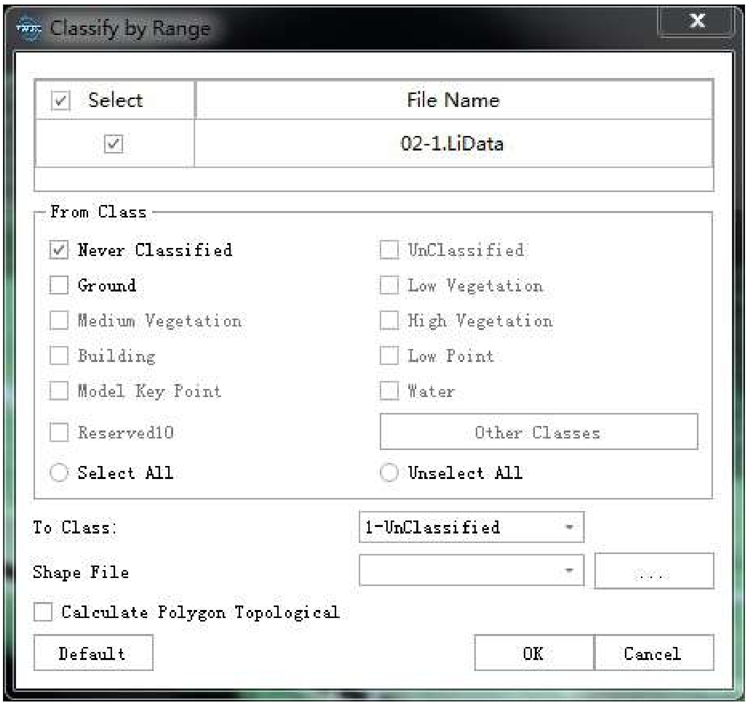
Initial points within the polygon are classified into target categories.

For Polygonal objects in the vector data, if the calculation of topological relationship is not checked, the points of all initial categories in the polygon will be divided into the target category; if the calculation of topological relationship is checked, the points in the interval between the inner polygon and the outer polygon will be The points of the initial class are divided into the target class, and the points in the inner polygon remain unchanged.

For Multi-Polygonal object in the vector data, the points of the initial category in the interval between the inner polygon and the outer polygon are divided into the target category, and the points in the inner polygon remain unchanged.

Usage

Click *Classify* > *Classify by Range*.



Settings

- **Input Data**: The input file can be a single point cloud data or a point cloud dataset, which must be opened in the LiDAR360 software.
- **From Class**: Categories to be classified.
- **To Class**: Classification target category.
- **Vector File**: User can select the Vector File loaded into LiDAR360 software from the drop-down menu, or select ... button to load an external vector data file.

- **Whether to calculate the topological relationship of polygons**: For the Polygonal object in the vector data, calculate the topological relationship between polygons. If checked, the topological relationship between polygons will be calculated; if not checked, all points in the Polygon will be generated into a new file.

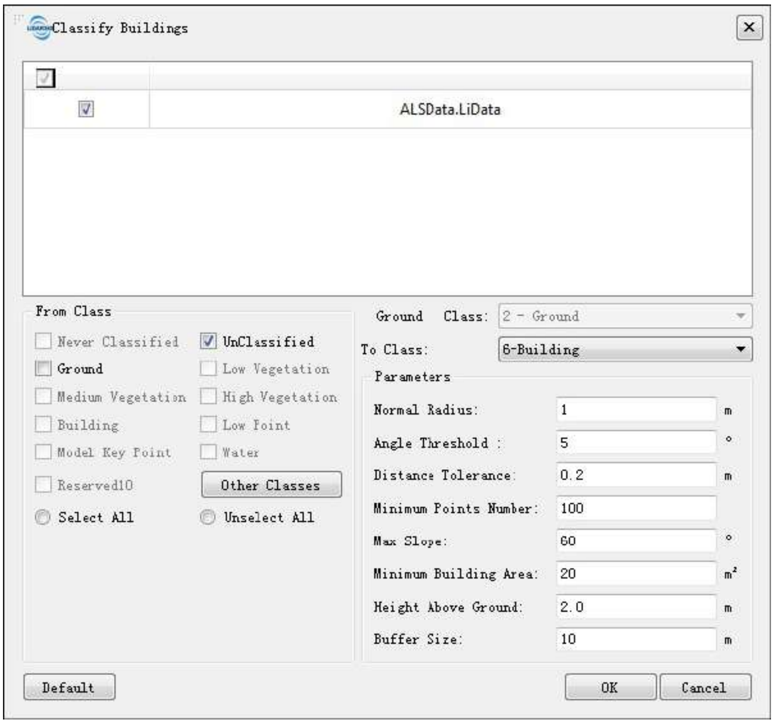
Classify Buildings

Summary

This function classifies buildings in point cloud data.

Usage

Click *Classify* > *Classify Buildings*



Settings

- **Input Data**: The input file can be a single point cloud data file or a point cloud dataset, which must be opened in the LiDAR360 software and has already been classified by ground points.
- **From Class**: Source class(es).
- **Ground Class**: The default ground point is 2-class.
- **Building Class**: Specify the building class number.
- **Wall Class**: Specify the wall class number.
- **Search Radius (default "1")**: The radius of the neighborhood used for calculating the point cloud normal vector, typically set to 4-6 times the point spacing.
- **Angle Threshold (°) (default "5")**: The angle threshold between two points during plane clustering. Points with angles smaller than this value are considered part of the same cluster.
- **Distance Tolerance (default "0.2")**: The distance threshold from a point to a plane during plane clustering. Points with distances smaller than this value are considered part of the same cluster, generally set to slightly more than the point spacing.
- **Min Points Number(default "100")**: The minimum number of points required for a building face.

- **Max Slope (°) (default "60")**: Angles greater than this value are considered not to be building rooftops but potentially walls or other categories.
- **Min Building Area (default "20")**: The minimum area for a building rooftop. Areas smaller than this value are considered not part of the rooftop.
- **Height Above Ground (default "2")**: Used for preliminary filtering of rooftop points.
- **Buffer Size (default "10 m")**: The buffer zone for point cloud segmentation.
- **Default**: Click this button to restore all parameters to their default values.

> Note:
>
> 1.This function requires that the point cloud has already undergone ground point classification.
> 2.In the input data, points classified as roof or wall categories will not be reclassified.

Classify Model Key Points

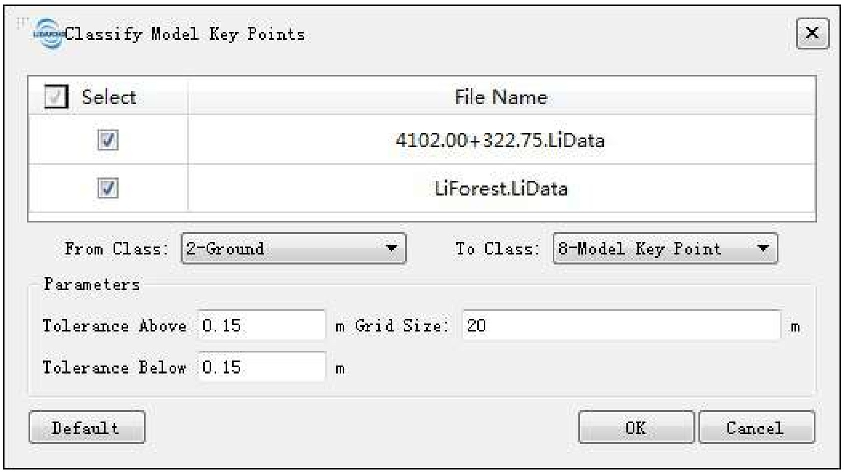
Functional Overview

This function can thin a certain level of the classified points. It is generally used to generate a sparse point set that retains the key points in the complex terrain area and thin the points in the flat area from the extracted dense ground points.

The idea of the algorithm is: first, meshing of point cloud data, and then use the seed points in the grid to establish the initial triangulation network. According to the upper and lower boundary thresholds, the points that meet the conditions are added to the triangulation network. The process is iterated until all of the key points of terrain model are classified. In the following figure, the yellow point is the ground point and the purple point is the key point of terrain model.

Usage

Click*Classify > Model Key Points*.



Parameters Settings

- **Input data**: The input file can be a single data file or a point cloud data set; the file(s) to be processed must be opened in the LiDAR360 software.
- **Original category**: Categories to be classified.
- **Target category:** Classification target category.
- **Tolerance Above (m, default value is"0.15")**: The maximum allowable elevation tolerance value on the triangulation network model composed of original points. If it exceeds this threshold, it is regarded as the key point. Simply saying, the larger the value is set, the more sparse the extracted key points of the model will be; otherwise, the denser it will be.
- **Tolerance Below (m, default value is"0.15")**: Maximum allowable elevation tolerance in the triangulation network model composed of original points. If it exceeds this threshold, it is regarded as

the key point. Simply put, the larger the value is set, the more sparse the extracted key points of the model will be; otherwise, the denser it will be.

- **Grid Size (m, default value is"20")**: The value is used to ensure the density of key points extracted from the model. For example, if you want to ensure that there is at least one point in the grid every 20 meters, this value is set to 20.
- **Default Value**: Click this tool option to restore all default parameters.

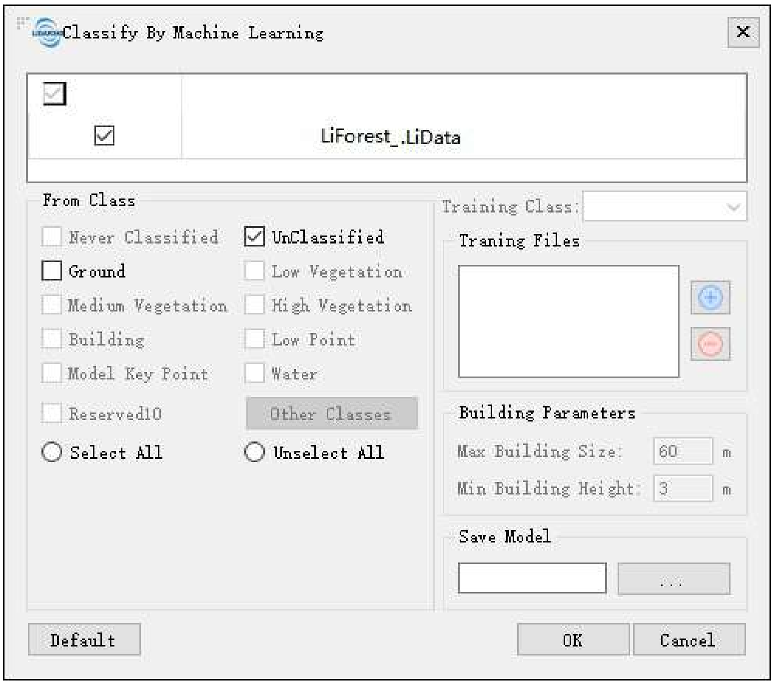
Classify by Machine Learning

Summary

This feature uses random forests (machine learning) method to classify point cloud data. In the same batch of data, it is necessary to manually edit the categories of a small amount of data. After the model is trained, a large amount of data is processed in batches, and it is used to reduce the amount of labor. This feature supports two kinds of processes, one is through selecting training samples, generating training models, and classifying the data, the other is classifying the data directly using the existing models.

Usage

Click *Classify* > *Classify by Machine Learning*



Settings

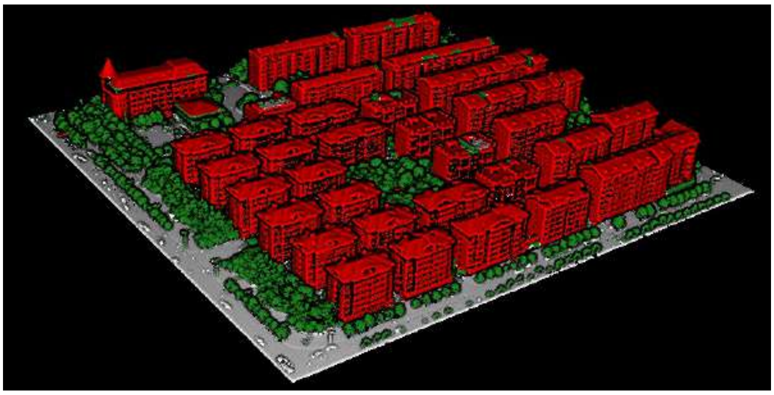
- **Input Data**: The input file can be a single point cloud data or a point cloud dataset, which must be opened in the LiDAR360 software.
- **From Class**: Source class(es).
- **Training Class**: The classes that are of interest to the user are trained, which will also be included in the classification results. At least two types of training categories should be selected, of which one must be unclassified
- **Training Files**: Click to load training data. Click to remove the selected data. You can train multiple files. The categories in the training data are edited manually.
- **Building Parameters**: The parameters are set only when buildings are included in the training class, which were used to set the maximum building size and minimum building height, respectively.
    - **Max Building Size (m, default value is"60")**: The largest building size in the data to be

processed.

  - **Min Building Height (m, default value is"3")**: The smallest building size in the data to be processed.
- **Save Model**: After training, save the model in the file path with the custom **.vcm** file format . This model file can be used as input data for Run Machine Learning Classification Model.
- **DefaultValue**: Click this button to set all parameter as default.

Training a small amount of manually edited data, including unclassified, vegetation, building, and all involved in training, as shown below:

Handle a large number of data results, including unclassified, vegetation, building, as shown below:



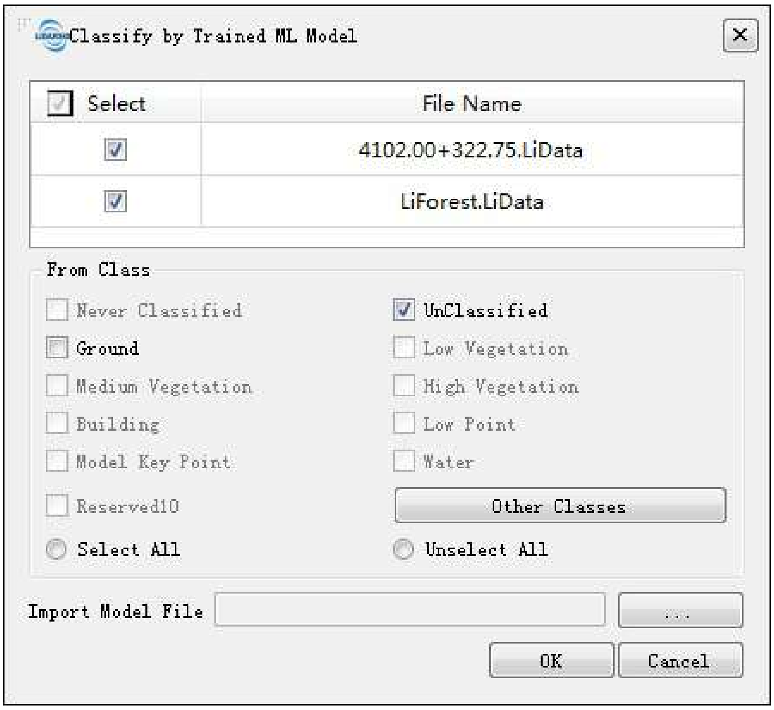
Classify by Trained ML Model

Summary

By directly importing the machine learning model with the suffix vcm for classification to classify the point cloud. It should be noted that the source cloud data trained for the imported machine learning model should contain relevant geometric features of the unclassified data.

Usage

Click *Classify > Classify by Trained ML Model*



Settings

- **Input Data**: The input file can be a single point cloud data or a point cloud data set, which must be opened in the LiDAR360 software.
- **From Class**: Source class(es).
- **Import Model File**: Import the trained model file, whose suffix is vcm. This file is generated by Classify by Machine Learning.

Classify Ground by Selected

Functional Overview

Due to the complex and varied terrain of point cloud data, it is often difficult to achieve good classification results using a set of parameters when using Classify Ground Points, especially for the mixed areas (mountains, plains, etc.). Therefore, in the case that the complex terrain cannot get a good classification effect in one classification, the ground point classification toolbar of the selected region can be considered to reclassify the ground point in the region with unsatisfactory local classification effect.

Usage

Load the data first inside the window, Click *Classify* > *Classify Ground by Selected*, the classification toolbar appears at the top of the data window, as shown below.

From left to right: *polygon selection, rectangle selection, sphere selection, subtract selection, clear selection, classify by attribute, Conicoid Filter, slope filter, TIN filtering, extract median ground points, recover selection, and exit.* Select and Clip tools can be found in Select and Clip

> Note: The ground point classification algorithms provided by the selected area point classification include: Conicoid Filter, slope filter, TIN filtering. Different methods should be chosen according to the terrain. Generally, the effect of TIN filter is the most stable and therefore we recommend this first. In addition, it would be better to handle each file separately.

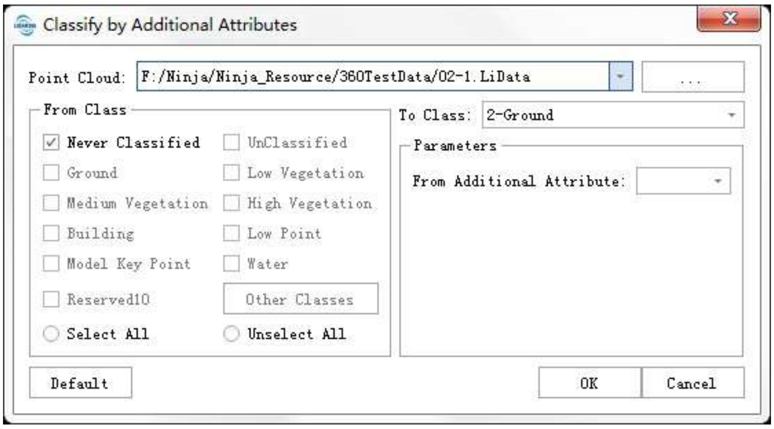
Classify by Additional Attributes

Functional Overview

With this function, a category in the point cloud can be classified into another category according to the additional attribute range. An attached property supports one or more components.

Usage

Click *Category > Sort by Additional Properties*.



parameter settings

- **input data**: The input file must be a single point cloud data file. Users can select point cloud data already opened in LiDAR360 software, or open other LiData files on disk.
- **Initial Category**: Category to be classified.
- **Target Category**: Categorize the target category.
- **Additional attributes and range selection**: refer to Extract by Additional Attributes

Classify by Deep Learning

Functional Overview

This function uses a deep learning segmentation model to classify point cloud data, suitable for uav (unmanned aerial vehicle) data, mountainous or rural scenes. Now supports 2 categories: vegetation, buildings. Developed based on a sparse convolutional model, the algorithm ensures high efficiency while bringing better detail performance.

Usage

Click on*Classification > Classify by Deep Learning*

Parameters Settings

- **Data to be processed**: Select single or multiple data loaded in the software.
- **From Class**: You can select the category that needs to be classified, and the unselected category will not be covered by the model classification result, which is especially effective when there are some

fine categories in the data.

- **Sence**:Choose Urban or Rural sence, to use different deep learning model.
- **Classify xx**: Check to enable this category, you can select the number corresponding to this category.
- **Height Above Ground**: Select the ground point of category 2 as the reference ground point and fill in the ground height. This parameter can effectively prevent the ground point from being wrongly classified as a building.
- **Use GPU**:Using GPU acceleration to improve the performance of classification.

  > Note: The output of this function will overwrite the original data file, users who need it should back up the data by themselves. Due to limited training data scenarios, deep learning models may perform poorly on certain scenarios and certain types of data.

Classify by Custom Deep Learning

This function uses a deep learning segmentation model to classify point cloud data. This function uses supervised classification, in the same batch of data, need to manually edit a small number of data categories, after training the model to batch process a large number of data. Two processes are supported: select and generate training models, processing data to be classified; Use the existing models to process the data to be classified.

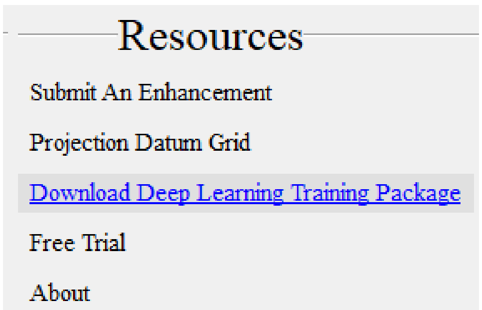
Requirement

GPU Requirement	Description
GPU Type	the CUDA type of minimum computing capability is 3.5. So 6.1 or higher is recommend. Refer to Compute Capability
GPU Driver	NVIDIA GPU drivers — version 452.39 or higher
GPU Memory	minimum: 6GB recommend: 8GB or higher, depending on deep learning model structure and batch size

Download Deep Learning Training Package

1. Click "Download Deep Learning Training Package" on the Start Page, get a python38.7z file.

2. Extract the installation package and copy it into the "\LiDAR360\8.0.0\python38" folder under the installation path.



Usage

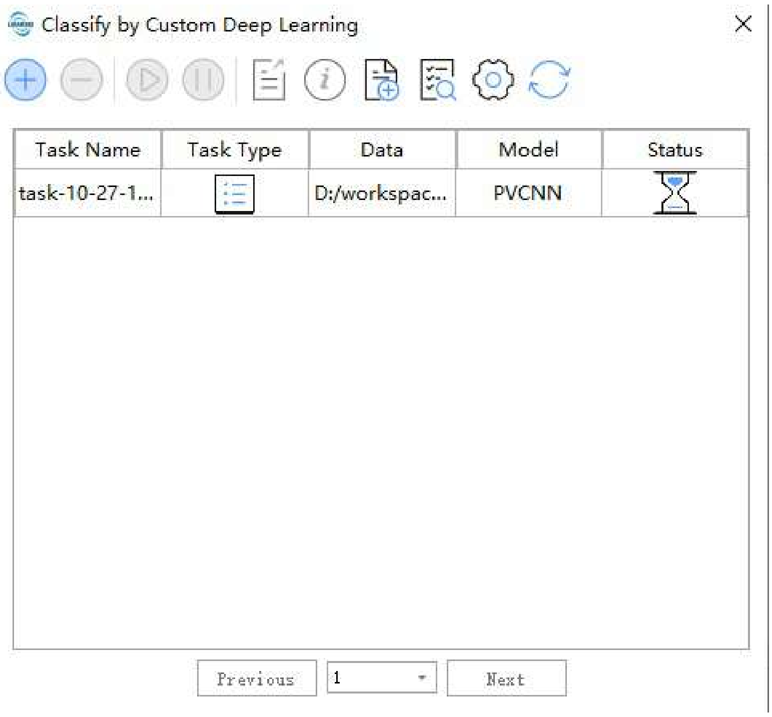
Click *Classification >Custom Deep Learning Classification*

Usage Procedure: Data preparation -->Training point cloud classification model -->Use training model for classification

Please note that when you enablethe back-end for the first time.

The following is the task management page, which can display the status/management of training point cloud classification model tasks and use the training model to classify tasks.

Tool bar from left to right **Create a task,delete a task, start a task, pause a task, export a task, task information,import task,model management,setting,refresh**



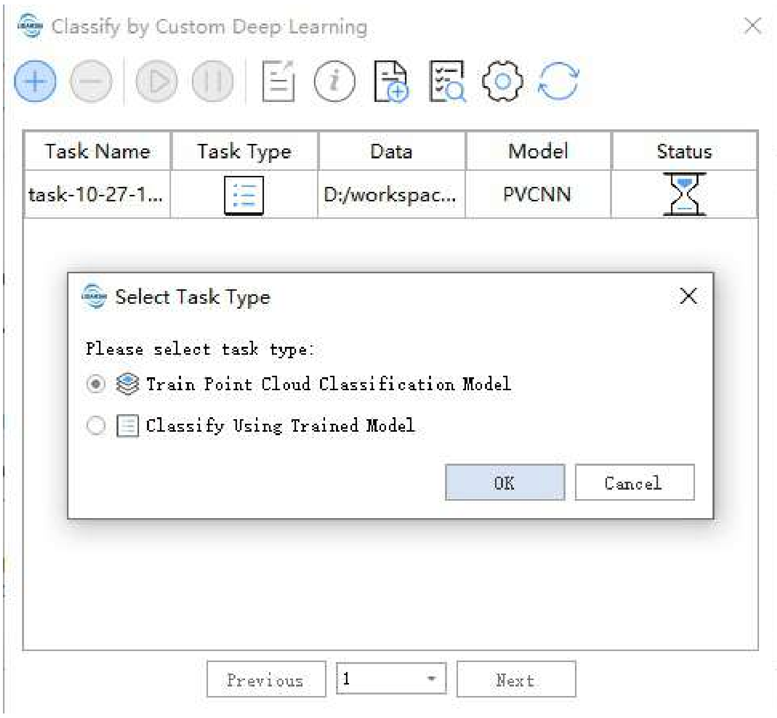
The task management page, it can display state/manage training point cloud classification model tasks and use training model classification tasks. Note the connection to the back end should be set at the first startup.

Data Preparation

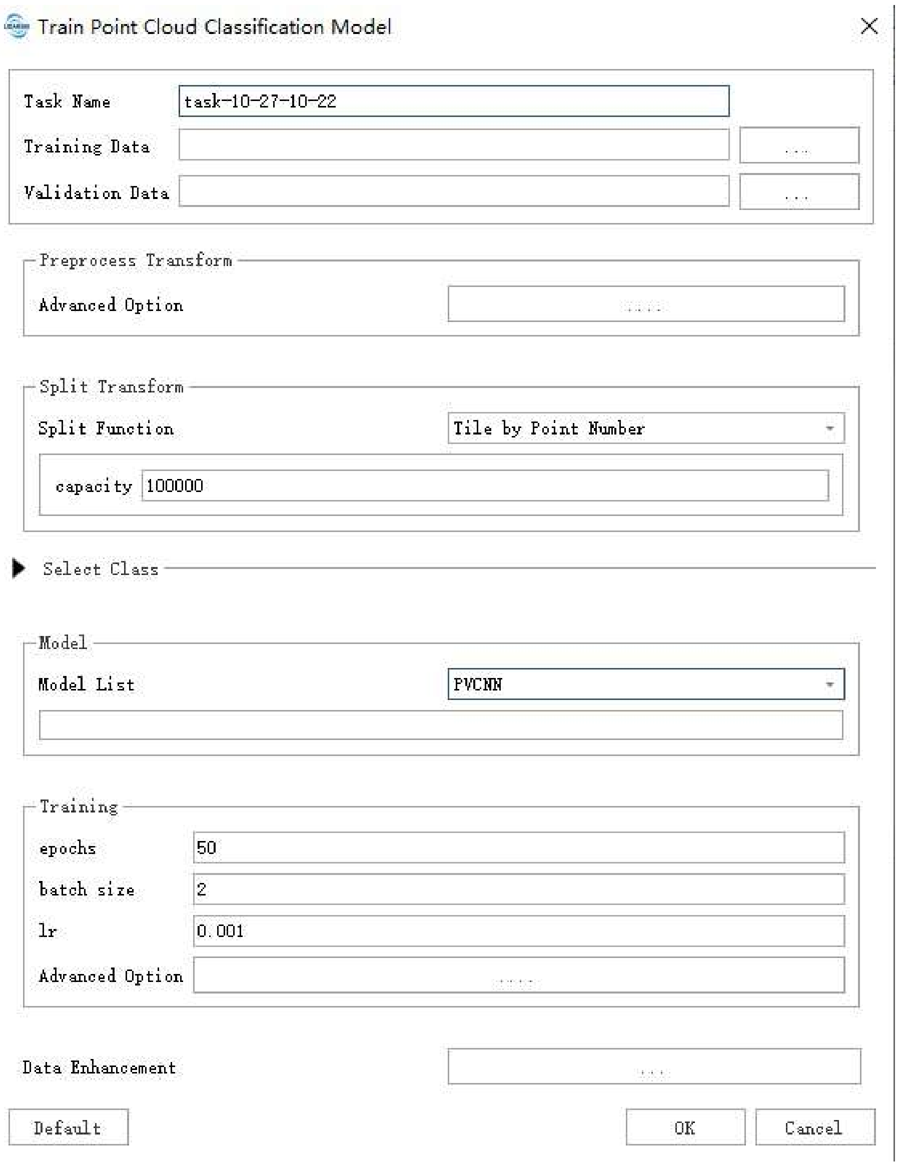
The data required for training (labeled data) should be prepared before training, and you can use the profile editing function provided by lidar360 to edit the category data. The training data should be real scene data, and data in las, lidata and laz formats are allowed. The amount of training data is adjusted according to different scenarios and algorithm processing methods. In principle, the more annotated data involved in the training, the better. We recommend that at least 100m*100m data be prepared for use to obtain a good use experience.Please note that data annotation must be consistent with the principle, and the setting of categories should not be changed according to the scene. Mislabeled data can have a negative impact.

Training point cloud classification model

Click *Classification* > *Classify by Deep Learning-> Training point cloud classification model*



Classify by Custom Deep Learning
Task Name
Task Type
Data
Model
Status
task-10-27-1...
D:/workspac...
PVCNN
Select Task Type
Please select task type:
Train Point Cloud Classification Model
Classify Using Trained Model
OK
Cancel
Previous
1
Next



After filling in the data path as required, you can use the default parameters.

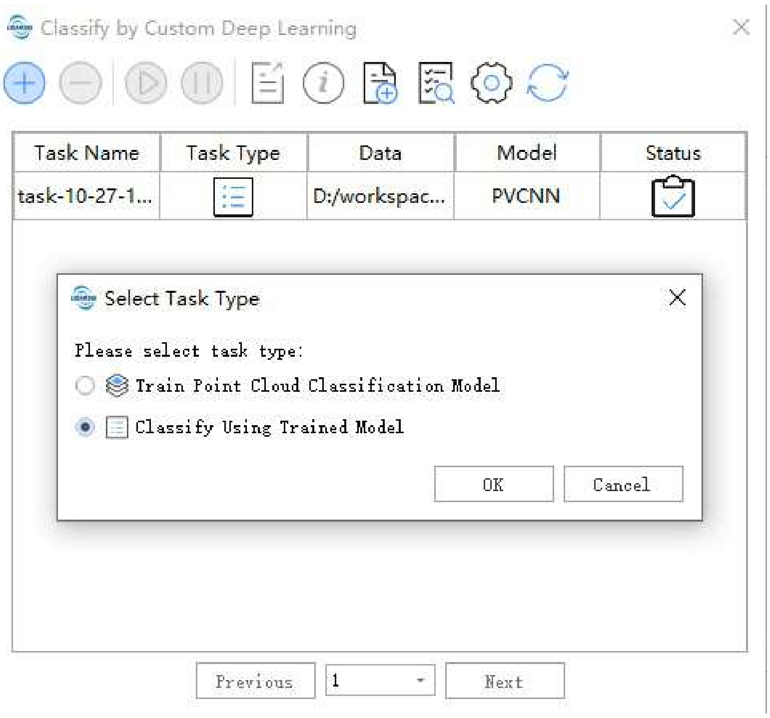
- **Task Name**: The task name will be used as the trained model name.
- **Training Data**: If the data marked by categories is placed in the same folder, all data in the supported formats of the folder will be read.
- **Validation Data**: Put the data marked by categories in the same folder. The data in this folder will be used as indicators such as the accuracy of sample calculation.
- **Preprocess Transform**:
  - **Split Function**: Data preprocessing, you can select data preprocessing function here, usually include statistical filtering, voxel filtering, etc.
- **Split Transform**:
  - **Split Function**: Data segmentation, in order to avoid the use of too much memory, users can segment the data to the appropriate size, the size is related to data distribution, algorithm and so on. We will continue to describe how to choose the appropriate size later.
- Select Class: Category mapping, where one or more source classes correspond to target classes, can save a lot of work when adjusting algorithm classification categories and correcting category definitions.

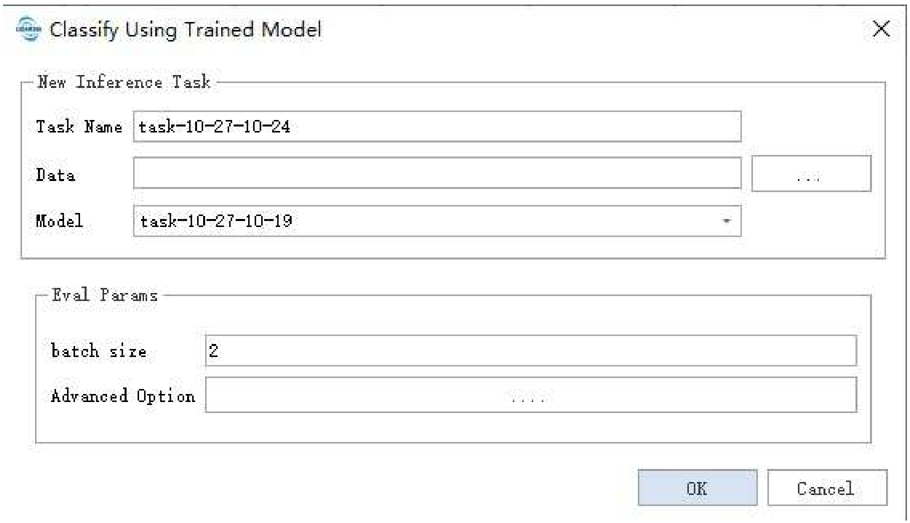
- **Model**:
  - **Model List**: The model temporarily provides three classical models: PVCNN, KPConv and MinkUNet (not the original implementation, with a small number of changes). For these three algorithms, we have provided corresponding recommended parameters settings. You can use the Default button in the lower left corner to automatically fill in subsequent parameters.
- **Training**:
  - **epochs**: Iteration period. Each complete loop that processing all data is an epochs.
  - **batch size**: For each batch of data, which means the model processing several data. If memory allows, larger data has more advantages in model accuracy and training speed. For GPU with dedicated RAM of 8GB, please use the default batch size 2. If you find that there's still lots of GPU memory available during training, you can safely increase the batch size to process more blocks at once.
  - **lr**: Learning rate cooperate with the optimizer control the speed and direction of the model learning (generally, the lower the learning rate, the more stable and easier to fall into the saddle point).
  - **Optimizer**: Optimizer, as above.
  - **lr scheduler**: Learning rate controller, the learning rate varies with the period to obtain better accuracy in some cases.
  - loss: The default loss function is cross entropy CrossEntropyLoss. FocalLoss generally has a better effect when the categories are not balanced, and may have worse effect when the label is wrong. So using CrossEntropyLoss is better.

> The preceding bold parameters are mandatory. Other parameters are optional.

Use training model for classification

Click *Classification >Classification by Deep Learning >Use training model for classification*





- **Model:** Select the Model to use. The name of the Model will be named after the training task name. The model will be automatically added into the Model management after training.

> Note: The model will be generated when the training task is completely stopped (the training task is completed, the training task is paused, and the training task exits when the memory exceeds).

Evaluate point cloud training results

Three indicators MIoU, Acc and MAcc are provided, which are mainly calculated based on confusion matrix

The True value is positive, the number that the model considers to be positive (True Positive=TP). The True value is positive, the number that the model considers to be negative (False Negative=FN) The true value is negative, the number that the model considers to be positive (False Positive=FP) The true value is negative, the number that the model considers to be negative (True Negative=TN)

		True category	
		1	0
Predicted category	1 Positive	True Positive True positive	False Positive False positive
	0 Negative	False Negative False negative	True Negative True negative

Such a table of four indicators becomes the confusion matrix

IoU (Intersection over Union)

The intersection of goals and predictions/the union of goals and predictions

$$IoU = \frac{target \cap prediction}{target \cup prediction} = \frac{TP}{TP + FP + FN}$$

MIoU(Mean Intersection over Union) MIoU

$$\frac{\sum_{i=0}^{n} IoU_i}{n}$$

Acc(Accuracy)

$$Acc = \frac{TP + TN}{TP + TN + FP + FN}$$

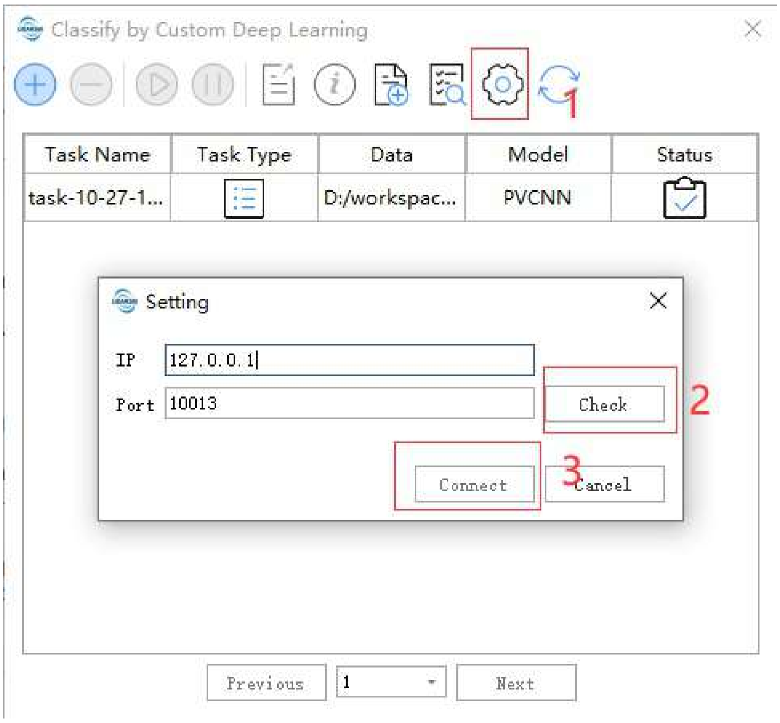
MAcc(Mean Accuracy)

$$mAcc = \frac{\sum_{i=0}^{n} Acc_i}{n}$$

In general, we only need to focus on the miou metric, which is the average intersection ratio

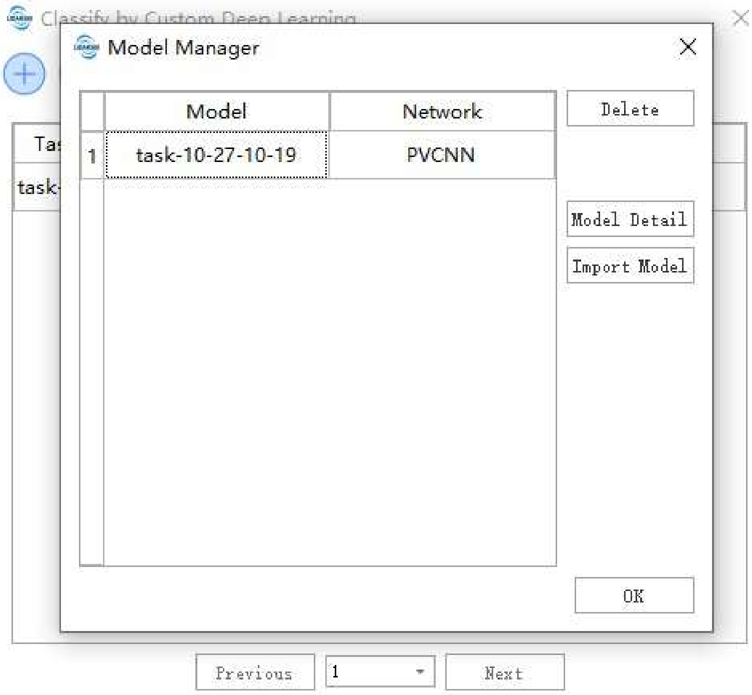
Set back-end

Click the following buttons in turn. Note that the third button will not be clicked when the back-end is not connected.



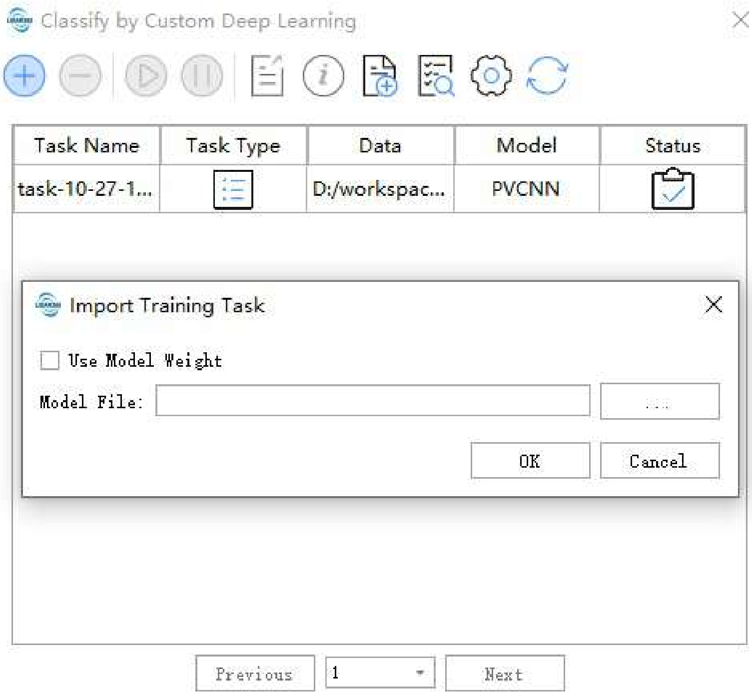
Model management

The model management interface manages available models and imports external models (lidar360 client output models only).



Import project

External models can be imported to obtain parameters/weights.



- **Model File**: External model file path (only supported by Lidar360 client output) - **Model Weight**: indicates the weight of the model. If it is checked, training continues. If it is not checked, only model training parameters are obtained

Deep learning model comparison

The following data are for the training stage. Use the voxel size of 0.5 for voxel subsampling, with a single data point of 20w and a batch of 2 statistics. The time complexity and space complexity are mainly affected by the degree of data density. It is difficult to control the point density after multiple subsampling of the model, and can only provide a general value for reference.

Hardware Configuration

cpu:i7-10700k(8t16c)

gpu:RTX3060

Algorithm	Speed(items/s)	Memory usage(GB)	Space complexity	Time complexity
PVCNN	1.82	8G	about O(logN)	about O(logN)
KPConv	0.04	11G	about O(logN)	about 0(n**2)
MinkUNet	1.05	8G	about O(n)	about O(logN)

KPConv generally has the highest level of precision and detail, and is recommended for small scenarios that require fine classification. The MinkUNet has a larger field of view with a decent speed and is recommended for large scenes. PVCNN has great detail performance and fastest speed, and is generally recommended.

Feature selection

rgb and intensity features can be added according to the specific data The rgb and intensity features can make the model fit earlier, with little difference in final accuracy (depending on the actual data), and negligible difference in memory consumption and computing speed.

Features	miou
rgb+intensity	99.1
intensity	98.8
rgb	99.0
None	98.7

Debugging guide

- **Insufficient memory**: Reduce the batch_size parameter and split transform to the split size.
- **Category imbalance**: Recommend that loss change to FocalLoss.
- **Loss jitter back and forth**: At the beginning of training, you can wait for more rounds. If in 10-20 epoch it is not stable, can try to make lr / 10 smaller. It is recommended to you to use CosineAnnealingWarmRestarts for lr_scheduler. If necessary, the large lr training can be used in the early stage, and the use of import model training function in the later stage can change to the small lr training.

> Note

1. The validation set uses data enhancement from the training set by default

2. If users didn't specify the input feature, we will input the basic feature as required by the model

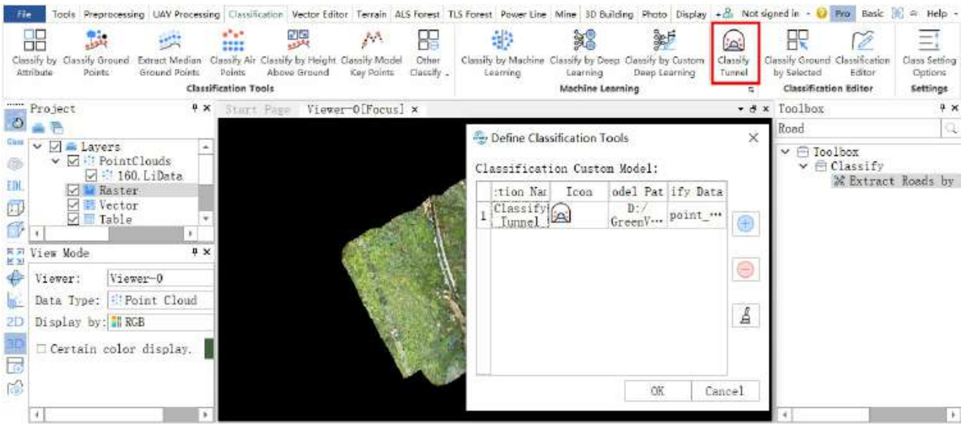
Custom Classification Settings

This feature uses a custom deep learning classification model to define classification functionality. Depending on the trained model type, this functionality can classify both point clouds and images.

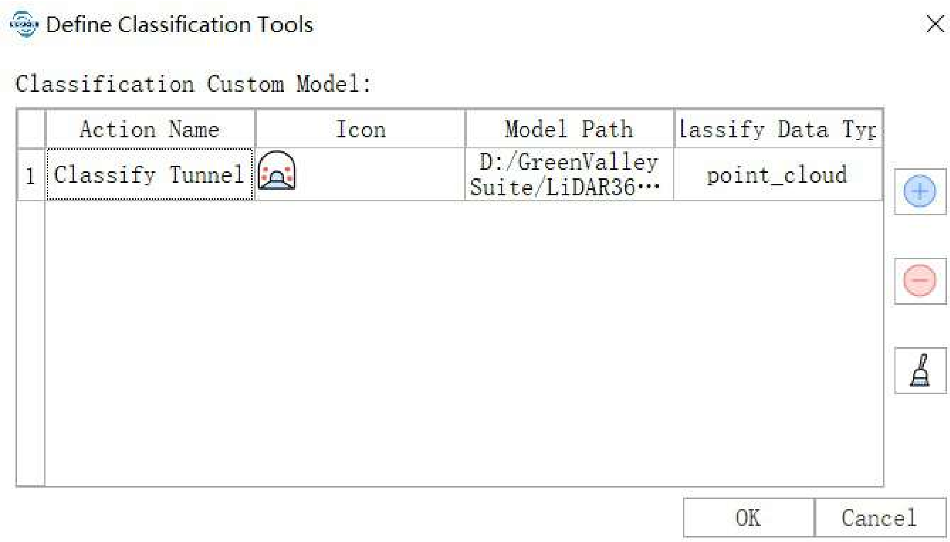
Ensure you have installed Deep Learning Service before using this feature.

Usage

1. Click on the settings function in the Custom Classification group on the Classification Page.



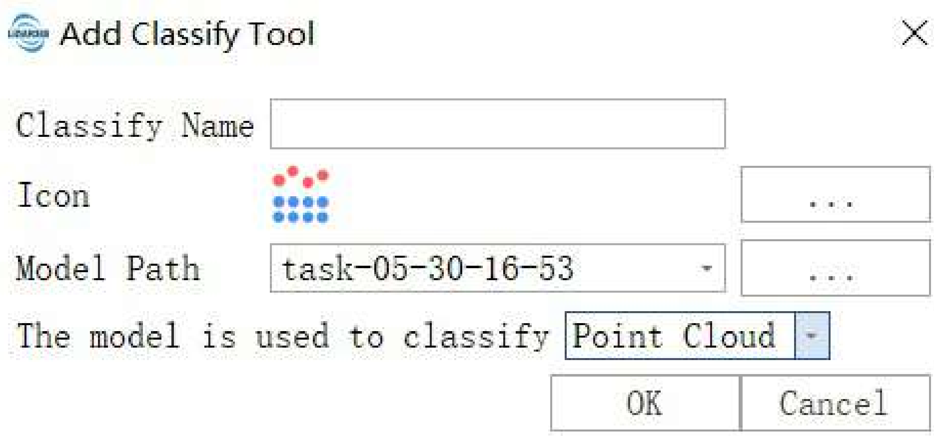
2. The following dialog will appear:



This list displays the custom classification functionalities that have already been added.

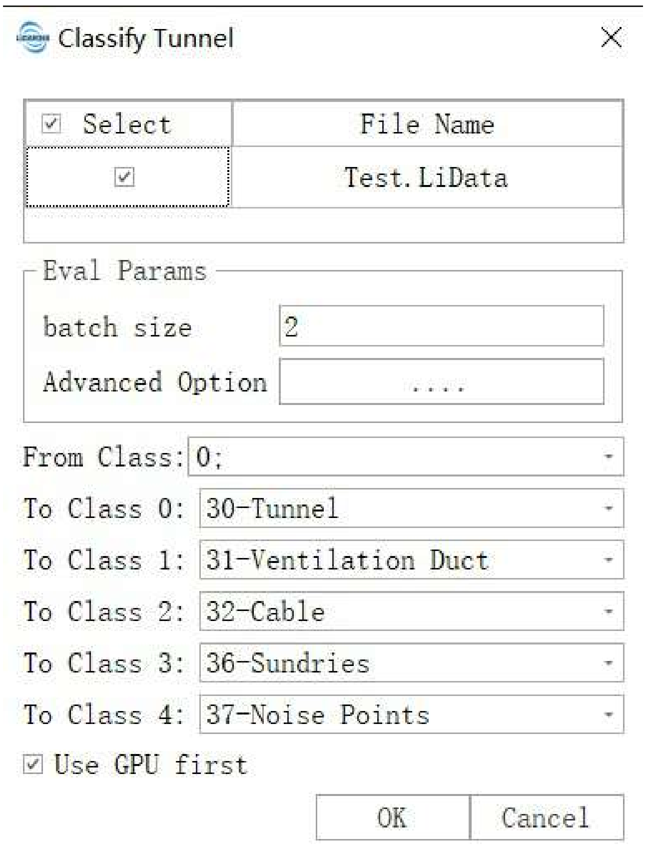
Adding a New Classification Function

 **Description**: Clicking this icon will open the new classification function interface.



In the interface shown above, define the classification name, select an icon, and choose the model file from the list of custom deep learning classification models. You can select a model from this list or import an external model file. Finally, set the classification data type for the model and click OK to define the classification functionality. After closing the settings interface, this functionality will be displayed on the Page.

For example, for "Classify Tunnel," clicking this function will display the following interface. Select the data to be classified, the initial category, and the deep learning classification parameters, then click OK to classify the selected data.



Deleting a Classification Function

Description: Deletes the selected classification function from the list. After closing the settings interface, the changes will be reflected on the Page.

interface, the changes will be reflected on the Page.

Clearing All Custom Classification Functions

Function Description: Clears all classification functions from the list. After closing the settings interface, all custom classification functions will be cleared from the Page.

Road Extraction

Function Overview

Automatically extract road lines from a 3-channel image file.

Usage

Click *Classification* > *Deep Learning Road Extraction*

Parameter Settings

- **Douglas Simplification**: Check to simplify the road contours.
- **Length Threshold**: The larger this value, the more significant the simplification effect.
- **Delete Length Less Than**: Delete road contour lines shorter than the specified value to filter out incorrectly identified roads.
- **Output Path**: Set the output path and format for the resulting vector file.

Profile Editing

Description：Users can draw a buffer on point cloud in 3D viewer and check correspongding profile in profile viewer. It can help with data check, measurement and classification.

- Block Editing Tool
- Draw Profile
- Draw Breaklines
- Breakline Tools
- Ground Points Estimation
- DEM Preview
- TIN Tools
- Select Tools
- Classify Methods Panel
- Profile Viewer Tools
- Linked Viewer

Profile and TIN Tools

TIN tool is to build triangular model in real time, you can use check editing effects.

> Note: You can use shortcut P on keyboard to adjust light, which can improve the display effect.

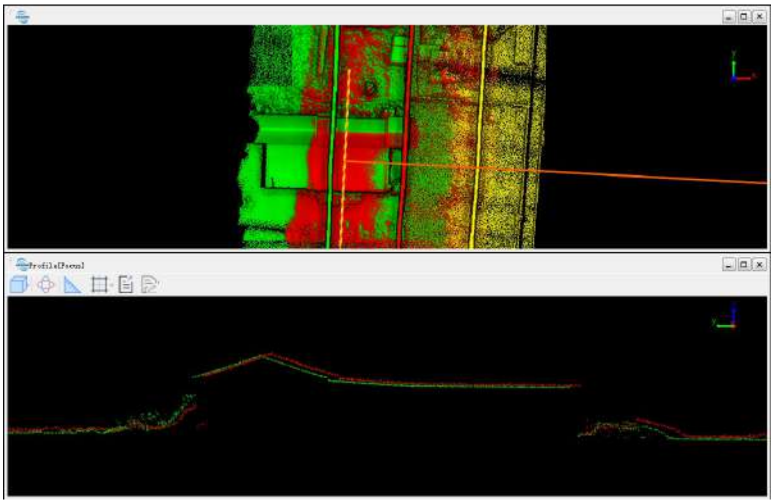
Please refer to TIN Tools for detailed information.

Profile and Measurement Tools

When the profile is turned on, the point cloud viewer and TIN viewer do not support measurement functions. The profile viewer supports single-point selection, multi-point selection, length measurement, angle measurement, slope measurement, and height measurement.

Profile and Boresight Correction

When using boresight correctio, we need to check mismatch issue between trajectory segments in profile viewer.



The image clearly shows the relative position of the point clouds before stitching. During the stitching process, you can also use the cross-section window to view the stitching effects in real-time with different parameters. Once confirmed, you can apply the current parameters to transform the point clouds and write the results directly to disk.

Besides, the measurement tools in profile viewer can help to evaluate Pith, Roll and Heading. Please refer to Strip Alignment for detailed information.

Profile and ALS/TLS Editor

When activate the ALS/TLS Editor, the main viewer will switch to 2D viewer. You can use profile viewer to check if seed points are correct in 3D view. Click Profile Tool to open profile viewer. Select a polygon area in point cloud viewer, and corresponding point cloud will appear in the profile viewer. Then, you can zoom or rotate in profile viewer to check seed points.

> Note：Profile in ALS/TLS Editor doesn't have the same profile tool bar as general profile tool.

Block Editing Tool

Block editing tool allows users to split the LiData into several tiles, and inspect and edit each block separately. After starting editing, software will determine the range of each memory data block according to the partitioning mode selected by the user and the related parameter Settings, and load all the point cloud data in the current data block for further editing. Users cancreate TIN modelfor the current memory point cloud to assist further classification work.

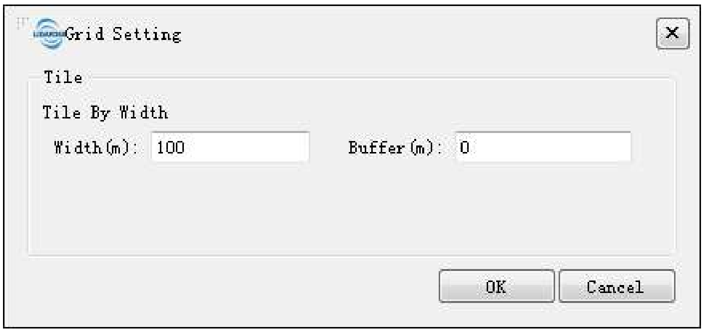
Icon	Description	Shortcut keys
	Start block editing, split the data into tiles.	
	Inspect and edit them separately	Ctrl+Z
	Redo	Ctrl+Y
	Save	Ctrl+Shift+S
	Settings	
	Shortcut keys Settings	
	Block Move Left	Alt+←
	Block Move Right	Alt+→
	Block Move Up	Alt+↑
	Block Move Down	Alt+↓
	Choose Block	
	Choose Blocks by Rectangle	
	Display/Hide the Text	

Edit Toolbar

Tile

 **Functional Description**: Clicking on the tile button, the window of "grid setting" will pop-up. Users can select one of the three blocking methods of "Block by Width", "Split by Width" and "Block by Polygon" according to the window prompts to set the relevant parameters. Click OK, and the software will display all the point cloud data in the grid block.

> Note: It is recommended that approximate 1500k points are in each block in average when setting the width of each grid.



Undo

Functional Description: This function allows users to redo the undone steps. This function will be disabled after saving the data. Support operations in big data mode and memory mode

> Note: In each profile editing, the undo and redo functions can only return results within 20 steps before the current result.

Redo

Description: This function allows users to redo the undone steps. This function will be disabled after saving the data. Currently, the big data mode is not supported.

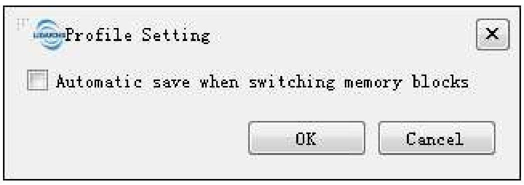
> Note: In each profile editing, the undo and redo functions can only return results within 20 steps before the current result.

Save

Functional Description: Save the modified data to the corresponding LiData file.

Settings

Functional Description: Profile editing supports automatic saving mode when users switch memory data blocks.



- **Save automatically when switching memory blocks**: This parameter is used for memory mode editing. If this option is checked, the software will automatically save the edit when users change the editing block to another one.

Block Change Toolbar

Block Change Toolbar is active only when the data is split into tiles.

Block Move Left

Functional Description: In the grid combination table displayed in the window, the data in the left grid of the current data block is displayed for data editing.

Block Move Right

Functional Description: In the grid combination table displayed in the window, the data in the right grid of the current data block is displayed for data editing.

Block Move Up

 **Functional Description**: In the grid combination table displayed in the window, the data in up grid of the current data block is displayed for data editing.

Block Move Down

 **Functional Description**: In the grid combination table displayed in the window, the data in down grid of the current data block is displayed for data editing.

Choose Block

 **Functional Description**: This tool is effective in memory editing mode. Click this button, users can click any grid in the main view window or TIN window for data display, viewing and edit.

Choose Blocks by Rectangle

 **Functional Description**: This tool is effective in the memory editing mode. Click this button, and users can select multiple grid blocks in the main view window or TIN window to view and edit the data by pulling a rectangle.

Display/Hide Blocks

 **Functional Description**: This tool is effective in memory editing mode, click the button, users can switch to Display/Hide the number corresponding to the blocks.

Breakline Tool

Description: The breakline tool supports users to draw, import and store point cloud breaklines. Based on the excellent display effect of point clouds and terrain models, it can clearly draw water bodies, steep slopes, rivers, roads and other breakline elements with high contrast. On the basis of drawing breaklines, using the drawn breaklines to assist [Tin Tool](TINTools.md) in adding breaklines for terrain editing and point cloud classification processing. After enabling memory point cloud editing, you can activate the edit tool for breaklines and correspondingly generate a broken line object loaded into the directory tree and participate in various windows for editing. The broken line editing tool includes sub-function modules such as broken line importation, broken line drawing, utility tools, broken line selection, entity capture and layer management.

Add Breakline

 Importing breaks lines from previously edited and stored files.

Draw Vectors

Supports drawing of multi-segment lines or polygons as breakpoints. For specific operations please refer to Vector Editor Entity Vectorization for multi-segment lines or polygon drawings.

Utility Tools

Supports deletion of breakpoints and attribute queries. For specific operations please refer to Vector Editor Basic Functions.

Snap

Supports capturing existing breakpoints or points clouds during breakpoint drawing process. For specific operations please refer to Vector Editor Entity Snapper.

Layer Management

Allows addition of layers or modification of layer attributes. For specific operations please refer to Vector Editor Layer Management.

Storage

When exiting the classification editing function, users will be prompted whether to save the modified breakpoint file. Click "Yes" to store and "No" to cancel storage.

Users can also use the layer management tool for breakline storage.

Breakline Tools

The section editing breakline tool is mainly for point cloud classification and section TIN editing, mainly to solve the problem of missing terrain features (such as ridges, rivers or other feature lines that want to be saved in TIN) caused by the direct construction of TIN from laser point clouds .

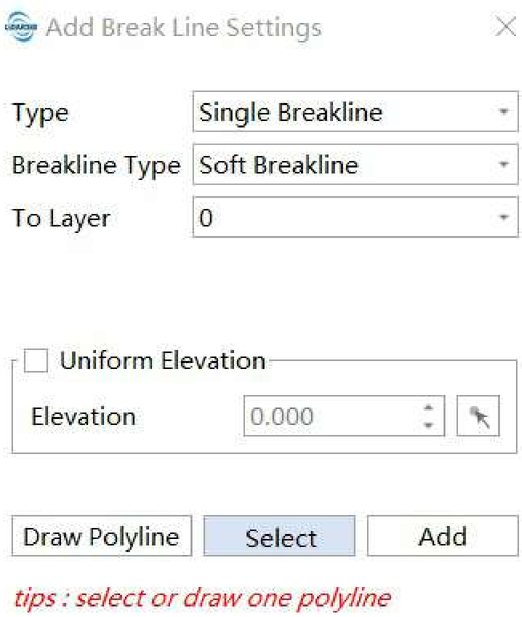
The breakline tool belongs to the section editing module. Activate the section editing module, click "Start Editing" on the menu bar, and execute the "TIN" to generate this function. The source of the breakline can be drawn by the user or imported externally.

Add Breakline

The user draws a breakline. Currently, there are mainly three ways to add breaklines: single breakline, closed area breakline (such as lake) and regional breakline between two lines (such as river).

Usage

(1) Click "Add BreakLine"in the menu bar , and a dialog box will pop up:



(2) Switch the type option to the type to be added;

(3) In the dialog box, click "Draw Polyline" or the toolbar to enable the drawing function, and draw a polyline on the triangle net (same as the vector editing module to draw a polyline, left click to add a point to the polyline, double click to end the drawing, right click to pop up a menu Choose to close and finish drawing);

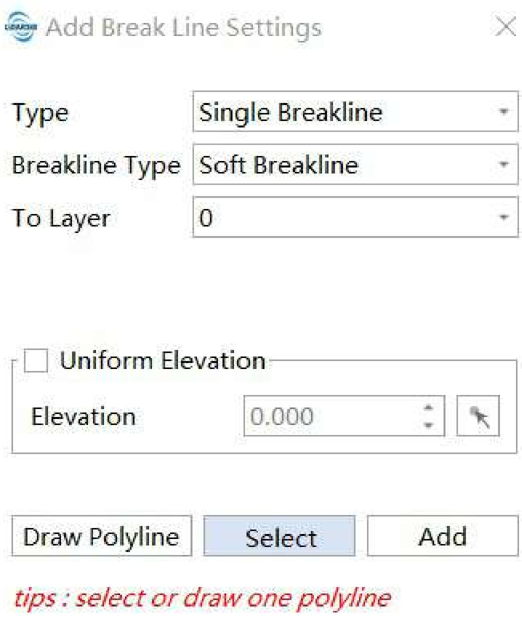
(4) Click the "Select" button in the dialog box or toolbar to enable the selection function, and select the polyline corresponding to the type in the scene;

(5) Set the corresponding parameters and click the "Add" button in the dialog box to complete the addition.

Single break line

Mainly deal with ridges, roadsides and other types of breaklines. Select a polyline in the scene (only one can be selected at a time);

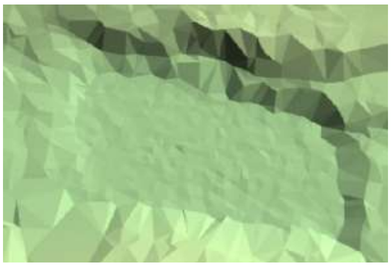
Parameters Settings

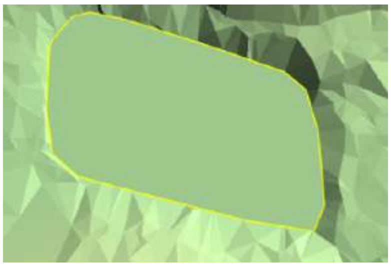


- **Breakline type**: Divided into two types: soft and hard. The result point of the soft breakline type is on the triangular network, and the result point of the hard breakline is on the polyline;
- **Add to layer**: The result of the breakline is saved to the corresponding layer;
- **Unified elevation**: Only effective below the hard break line. If unified elevation is activated, the result line will be projected to the specified elevation plane. The default is the elevation of the last point of the polyline, which can be set by manually inputting or clicking the "select point" button and or selecting the point model. Right-click to exit the point selection mode;

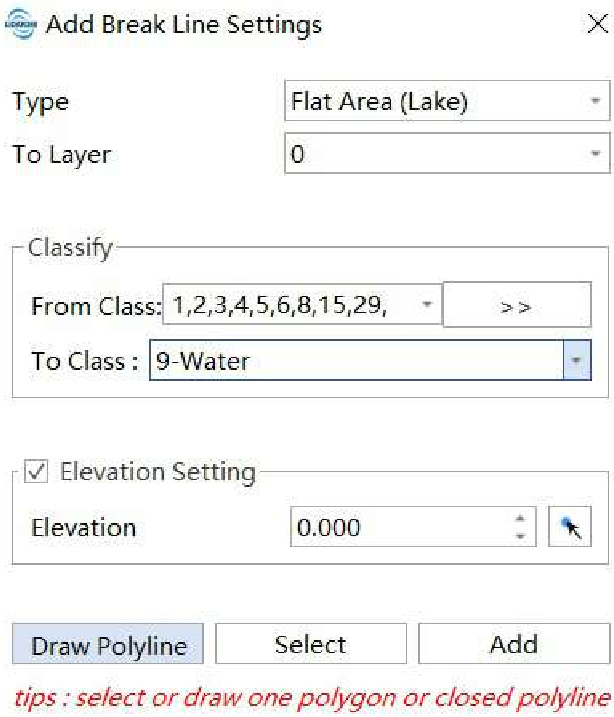
Closed area breakline

Such as the lake breakline. Select a closed polyline in the scene (only one can be selected at a time);





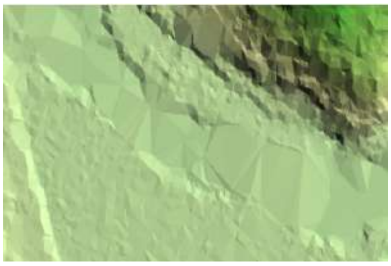
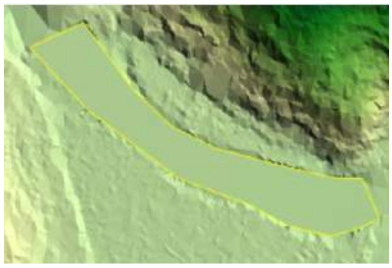
Parameters Settings



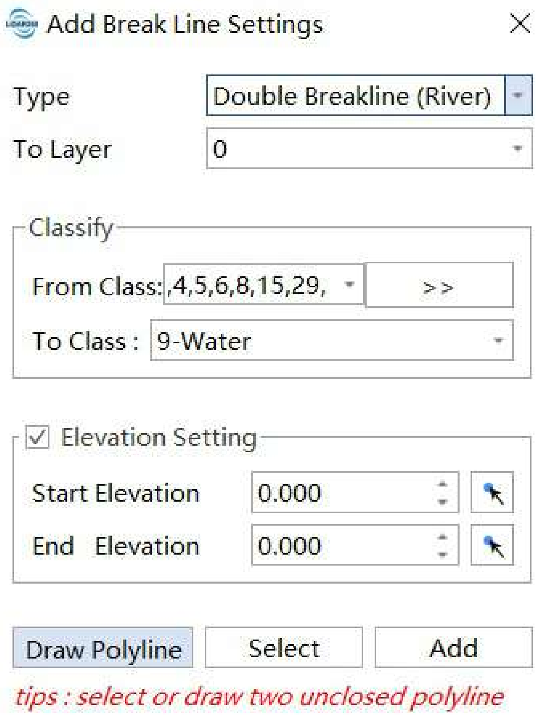
- **Add to layer**: The result of the breakline is saved to the corresponding layer;
- **Initial category**: The category of the point cloud in the area of the breakline to be processed, generally the network type or all selections;
- **Target category**: classify the point cloud within the selected breakline as the target category;
- **Elevation setting**: If the elevation setting is activated, the result line will be projected to the elevation plane, and the default is the elevation of the last point of the polyline. You can manually enter the setting or dick the "select point" button or select the point model to set it, right click can exit the point selection mode;

Two break line area

Such as the area between the two banks of a river. Select two non-closed polylines in the scene;

Parameters Settings



- **Add to Layer**: Set the layer where the breakline will be placed after adding.
- **Source category**: The source category of the point cloud corresponding to the breakline to be processed, generally the network type.
- **Target category**: Divide the point cloud within the range of the breakline into this setting category.
- **Elevation setting**: Whether to enable the elevation setting. If the elevation setting is enabled, the two polylines will be adjusted incrementally according to the start and end elevations. The default is the start elevation value of the first polyline drawn, which can be set manually or click the "select point" button or select the point model to set, right click to exit the point selection mode.
- ***Starting elevation value***: Starting elevation.
- ***End elevation value***: End elevation value.

Delete breakline

Delete the specified breakline in the scene. Click the "Select" button in the Add BreakLine dialog box or toolbar to enable the selection function, select a vector line in the scene, and click the "Delete" button in the toolbar to delete the vector line and related operations.

Import breaklines

Support for importing vector files. By activating the "Import Breakline" button, select the file to import, and select the polyline to add the breakline by adding the polyline function.

Simulate Terrain Point

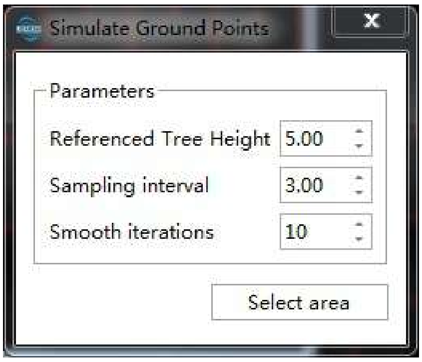
Function Description

The terrain point simulation tool mainly solves the phenomenon of missing terrain points in mountainous areas caused by dense vegetation occlusion, and simulates the real mountainous terrain triangular mesh by predicting the terrain of the missing points.

Usage

The terrain point estimation function belongs to the profile editing module. Activate the profile editing module -> click "Start Editing" -> execute "TIN" to generate this function. Click the terrain point estimation, and the parameter dialog box will pop up; select the profile tool, draw the profile of the simulated terrain point area, select the measurement tool in the profile window, and measure the height of the occluded vegetation; set the parameters in the parameter dialog box, click the select button (left mouse button Add a point, right-click to go back, double-click OK) to select the simulation area.

Parameter settings



By Reference Tree: Simulates based on the high points of the covered terrain.

- **Referenced Tree Height**: The approximate height of the obstructing vegetation measured from the profile.
- **Sampling Interval**: The average distance between simulated points.
- **Smoothing Iterations**: The number of iterations for smoothing the simulated ground points. More iterations result in a smoother simulated area.

By Neighboring Ground Points: Simulates based on the neighboring ground points of the area to be simulated. The selected range should cover the neighboring region.

- **Sampling Interval**: The average distance between simulated points.
- **Smoothing Iterations**: The number of iterations for smoothing the simulated ground points. More iterations result in a smoother simulated area.

Delete Simulated Points

Deletes the specified set of simulated terrain points in the scene.

Usage

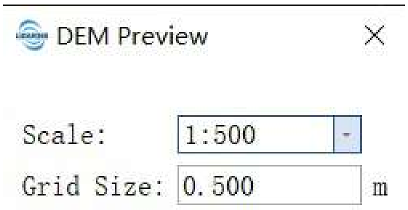
Click the "Select" button in the toolbar to enable the selection function, select the simulated terrain point set in the scene, and click "Delete" in the toolbar  button deletes the point set.

DEM Preview Tool

The DEM preview tool mainly checks the constructed TIN by constructing a temporary DEM of the local point cloud.Activate the section editing module, click "Start Editing" in the menu bar, and this function is available after "TIN" is generated.

Usage

(1) Click "DEM Preview" in the menu bar, a dialog box will pop up:



(2) Draw a polygon in the section window or TIN window, double-click the mouse button to end the drawing; right click to cancel and redraw. After double-clicking, the DEM will be displayed overlaid with the point cloud.

Parameter Settings

- **Scale (default 1:500)**: Supports four large scales from 1:500 to 1:5000.
- **Grid size (meters) (default "2")**: The resolution at which the DEM is generated.

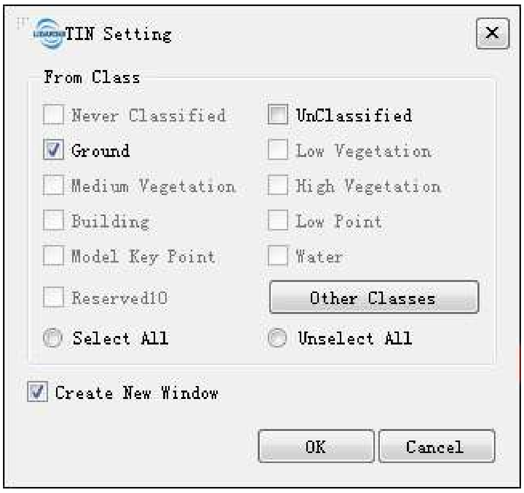
Scale	Grid Size
1:500	0.5
1:1000	1
1:2000	2

TIN Tools

TIN tool can be used to build a triangulation network model. When users edit point cloud categories, the triangulation network will change in real time, so as to assist users to view the editing effect. To speed up editing, TIN can be built in blocks with width values, and different blocks can be displayed by moving up, down, left, and right while editing, as well as selected blocks. In fact, it is to browse and edit the data in memoryLiTin,including adding and deleting points.

Create TIN Model

Click*TIN*and open setting window, as shown below:



Parameters setting：

- **Initial Class**： User needs to specify the initial category. For example, ground points can be selected to construct a triangulation model to help edit the classification of ground points.
- **Create a new window**： Whether to create a window.
  - **Yes**（**default**）： Create a new window to display TIN。
  - **No**： Display the TIN in main window.

After completing the TIN creation of the memory point cloud block,the TIN object will be added to the main object tree control on the left side of the software for unified management. At this moment, the TIN objects on the directory tree can be renamed, displayed by category, displayed by elevation, and deleted through the right-click menu.

Clear TIN Model

Clear TIN button is available after the generate TIN operation is performed. If you want to display TIN mode in a new window, click "Clear TIN" to close the new window. If you want to display TIN mode in the main window, click "Clear TIN" to clear TIN in the main window.

Classification Panel

The classification method panel supports the following filtering operations: seven ground point filtering methods, TIN Filter、Conicoid Filter、Slope Filter、Extract Median Ground Points、Air Points Filter、Building Filter、Steep Terrain Filter. The above seven filtering methods are only available when the memory mode is enabled.

Ground Point Filtering

The current version includes four ground point filtering methods: Tin filter , Conicoid filter , slope filter , median filter . After enabling the memory point cloud block editing state, users can click on the corresponding icon to open the corresponding filtering method.

Noise Filtering

Noise filtering is used to separate noise points from memory point clouds.

Building Filtering

Building Filtering is used to extract building points from memory point clouds.

Steep Terrain Filter

Steep terrain filter is used to extract ground points in steep terrain from memory point clouds.

Associated Windows

Function Overview

With the "Associated Windows" function, users can load various attribute data such as DOM and models supported by the platform into a new window, and link it with the point cloud window in the classification editor to assist in classification operations.

Window Settings

- **Associated Windows**: The associated windows list contains all windows opened after opening the classification editor. Select the window to be associated for assisted editing.

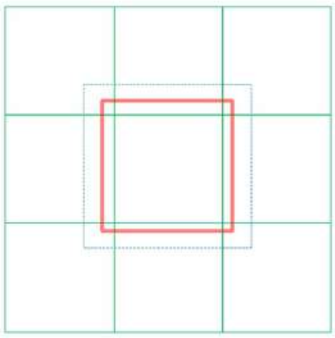
Terrain

The Terrain module contains a range of products required for terrain production, such as DEM, DSM, contour line and cross section, etc. of which the Digital Elevation Mode indicates that the bare earth surface (vegetation and other objects are removed), the Digital Surface Model represents the surface characteristics of terrain and other objects (e.g. tree canopy), and Canopy Height Model indicates the normalized height of vegetation and ground objects. Hillshade maps can be created by any digital model for visualization.

Interpolation Parameters Setting

When generating terrain products, you need to set some grid cell size parameters and determine the interpolation method used for grid cell calculations. The parameters settings are divided into conventional and standard.

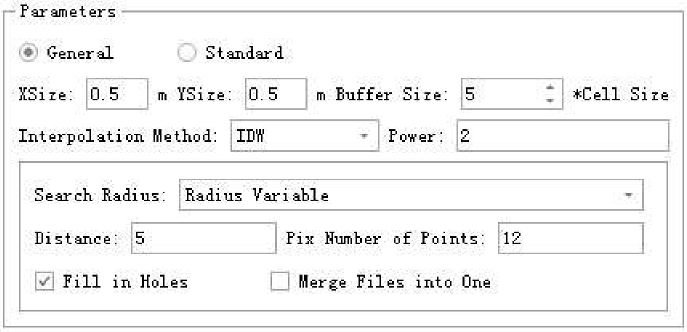
Under the standard parameters, the schematic diagram of the framing frame and each corresponding range is as follows:



Green box: standard framing box range;

The red double-line box: the result range box (the expansion range is related to the scale M, which is 0.01/M (m), for example, the scale is 1: 500, and the expansion range is 5m), and the range is calculated according to the internal scale;

Blue dotted frame: In the process of DEM generation, the size of the buffer area is expanded outside the result range frame to ensure the same elevation in the overlapping range of adjacent frames.



Parameter Type

- **General**: General DEM parameter configuration.
- **Standard**: Configured according to standard DEM production parameters.

Grid Size

Through XSize and YSize, users can set the size of sampling interval (resolution) in meters. For instance, if XSize and YSize values are set to 2, respectively, the grid unit size is 2 meters by 2 meters.

Output with Point Cloud Name (Only for Standard Mode)

Check this option to prefix the output file name with the name of the first input point cloud.

Scale bar (only set under standard mode parameters)

The scale of the framed frame.

Buffer size

- **Buffer Size (cell) (default is "5")**: The size of each block of raster data extending around, in unit of cell.

Interpolation Methods

LiDAR360 provides three kinds of raster cell interpolation methods: IDW (Inverse Distance Weigh) interpolation, kriging interpolation, and TIN (Triangulated Irregular Network) interpolation.

IDW

In the inverse distance weighting interpolation method, the value of a grid element is computed using its nearby points, and the weighted average value is judged by the distance of the center point of the grid unit. Users need to set the weight value. The search radius can be used to define the input points of each raster image meta value interpolation, which consists of variable radius and fixed radius.

- **Power(default value is "2")**: The power of the sampling point to the center distance of the pixel to control the degree of the influence of the sampling point elevation on the pixel center.
- **Radius Variable**: Use a "Variable" to find a specified number of sampling points for interpolation.
    - **Distance(Pix)(default value is "5")**: Restricts the distance to search for adjacent points, by default 5 pixels.
    - **Pix Number of Points(default value is "12")**: The number of nearest neighboring points used for interpolating, which is set 12 as default.
- **Radius Fixed**: Interpolation using all points in a fixed radius range.
    - **Distance(Pix)(default value is "5")**: The point within the range of the radius is used for interpolation, by default 5 pixels.
    - **PixNumber of Points(default value is "12")**: The smallest number of points used for interpolation, which is set 12 by default. If required points were not found within the specified radius, it will increase the search radius until find the minimum number of points.

Kriging

The Kriging interpolation method computes the optimized covariance and uses the Gaussian process interpolation grid value. The search radius can be used to define the input points of each raster image meta value interpolation, which consists of variable radius and fixed radius.

- **Radius Variable**: Use a "Variable" to find a specified number of sampling points for interpolation.

- **Distance(Pix)(default value is "5")**: Restricts the distance to search for adjacent points, by default 5 pixels.
- **Number of Points(default value is "12")**: The number of nearest neighboring points used for interpolating, which is set 12 as default.

* **Radius Fixed**: Interpolation using all points in a fixed radius range.

  - **Distance(Pix)(default value is "5")**: The point within the range of the radius is used for interpolation, by default 5 pixels.
  - **Number of Points(default value is "12")**: The smallest number of points used for interpolation, which is set 12 by default. If required points were not found within the specified radius, it will increase the search radius until find the minimum number of points.

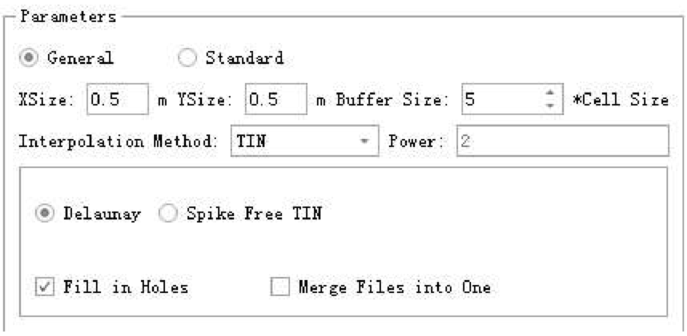
TIN

Extracts a grid cell value from a surface formed by a plurality of triangles consisting of the nearest adjacent point. This tool offers two options of interpolation.

- **Delaunay**: Create the Delaunay triangulation by traditional point-by-point insertion. All the points will be involved.

- **Spike Free TIN**: Remove all the points with abnormal altitude. It can lead to generating triangulated network without obvious spikes.

  - **Freeze Distance (default value is "1.0 meter")**: The shortest distance in xy plane of each side of triangle in the triangulated network. When inserting a new point and its Z value lower **Insertion Buffer**, freeze all the triangles whose three sides are all less than Freeze Distance. The frozen triangles will no longer change. The larger this value is, the fewer points will be involved in creating the network, the more smooth the network will be and the less details will be included. Conversely, more points will be involved in creating the network, the more details will be included and the more possible spikes will appear in the network.
  - **Insertion Buffer (default value is "0.5 meter")**: When trying to freeze a new triangle right after freezing another one, the difference in altitude value should be lager than this threshold. Decreasing this value will result in more triangles will be frozen too early, new point cannot be inserted, less spikes will appear, processing will be faster, and more details will be lost. Conversely. more details will be included, and more spikes will be included in the network.




```
    @inproceedings{
        author={ Khosravipour A, Skidmore A K, Isenburg M},
        title={Generating spike-free digital surface models using LiDAR raw point clouds: A new approach for forestry applications},
        booktitle={ International journal of applied earth observation and geoinformation, 52: 104-114},
```

```
    year={2016}
}
```

Merge files into one

If this option is not checked, each point cloud data will be processed separately, resulting in more than one grid file. Check this option to merge all generated raster files into one file.

Fill in holes

If there are no points near the grid unit, the grid cells may not have data values. When this option is checked, data values can be computed by analyzing adjacent grid units and using the selected interpolation method to fill in an area with no values.

> Note: The "Fill in holes" function is only for closed holes.

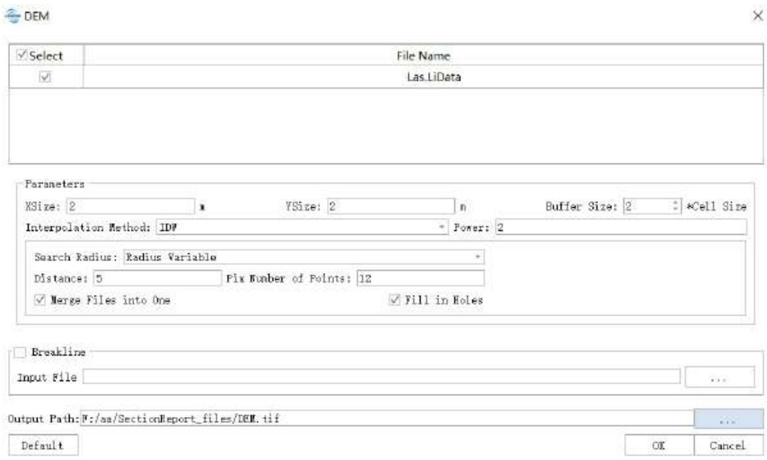
DEM

Summary

Digital Elevation Model (DEM), is the digitized simulation of terrain through limited topographic elevation data (i.e. the digitized representation of terrain surface). It represents the ground elevation with a set of ordered numerical array. It is a branch of the digital terrain model, which can be used to generate all other terrain feature values.

Usage

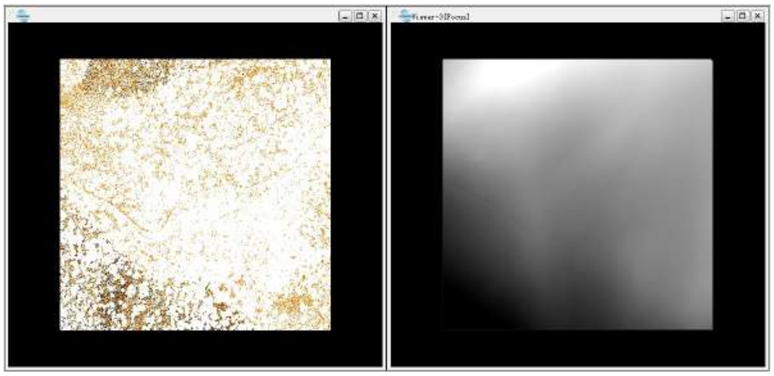
Before you can generate a digital elevation model, first **Classify Ground Points**. Click on the *Terrain* > *DEM*.



Settings

- **Input Data**: The input file can be a single point cloud data or a point cloud dataset, which must have classified ground points and opened in LiDAR360 software.
- **Parameter Settings**: See Grid Parameters Settings.
- **Import terrain features (optional)**: Import terrain features (breaklines, simulated terrain points) component files, breaklines and simulated terrain points will participate in DEM generation.
- **Output Path**: Save DEM file to a path.
- **Default Value**: Click this button to restore all parameter defaults.

The original point cloud (left below) and the DEM (right below).



Based on DEM, a myriad of products including Slope, Aspect, Roughness and Contour can be generated.

> Note: Point clouds that are used to generate DEM need contain ground points.

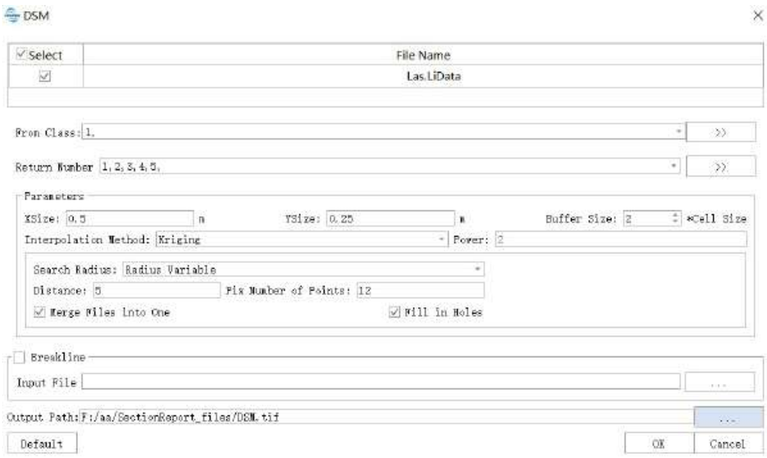
DSM

Summary

Digital Surface Model (DSM) refers to the digital representation of height of the surface including the buildings, bridges, trees etc. Compared to a DEM, a DSM contains more elevation information for buildings, bridges, forests and other surface object that don't exist in the DEM. DSM is based on DEM and further covers the elevation of surface information other than the ground.

Usage

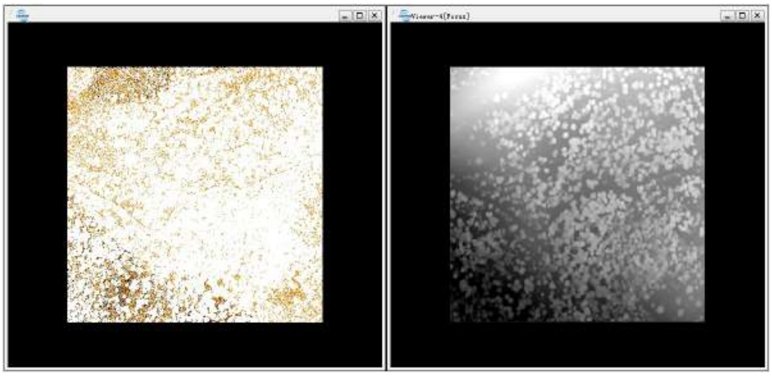
Click on the *Terrain* > *DSM*.



Settings

- **Input Data**: The input file can be a single point cloud data or a point cloud dataset, which must be already opened in LiDAR360 software.
- **Include Class**: Classifications of point cloud for DSM generation.
- **Parameter Settings**: See Interpolation Parameters Settings.
- **Output Path**: Save DSM file to a path.
- **Default Value**: Click this button to restore all parameter defaults.

The DSM generated from the point cloud whose initial category is ground and unclassified. Point cloud data is shown on the left, DSM is shown on the right.



CHM

Functional Overview

Canopy Height Model (CHM) can be obtained by subtracting DEM from DSM, and the following figure shows the relationship of DSM (Digital Surface Model), DEM (Digital Elevation Model) and CHM(Canopy Height Model).

LiDAR360 Terrain

Usage

Using this function requires a DEM and a DSM. Click on the Terrain > *CHM*.

Parameters Settings

- **Input DSM**: Input a DSM file, the generation method can be seen in DSM.
- **Input DEM**: Input a DEM file, the generation method can be seen in DEM.
- **Output CHM**: The path of CHM file to save.

The original point cloud (seen left below) and the generated CHM(see figure right below).

LiDAR360 Terrain

DOM

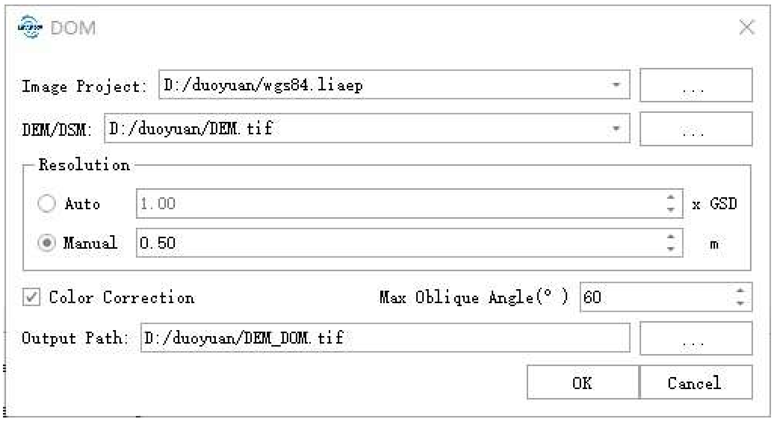
Functional Overview

DOM (Digital Orthophoto Map) refers to aerial photos that have been geometrically corrected and possess both map geometric accuracy and image features. This function uses the results of aerotriangulation project to generate DOM.

Usage

To generate digital orthophoto images, it is necessary to import the aerotriangulation project from various software and convert them into LiDAR360's internal image project files. Then, use the DEM or DSM function within the terrain module to interpolate the corresponding area's point cloud files and generate a DEM or DSM.

Click*Terrain > DOM*。



Parameters Settings

- **Image Project**: Select the imported image project file.
- **DEM/DSM**: Select the generated DEM or DSM file, supporting single channel TIFF files with 32-bit depth.
- **Resolution**: The resolution can be set in two modes. When selecting the automatic mode, the program will automatically calculate the ground sample distance (GSD), and the parameter set is a multiple of the GSD. When selecting the manual mode, the DOM resolution can be set directly.
- **Color Correction**: Whether to perform color correction on the photos to reduce color difference for some photos. This is only effective for image projects containing tie points.
- **Maximum Oblique Angle**: Only use photos with oblique angles less than this value for orthorectification.
- **Output Path**: Save the generated DOM file to the specified path.

  > In automatic mode, the calculated resolution will not be less than 0.05 meters.

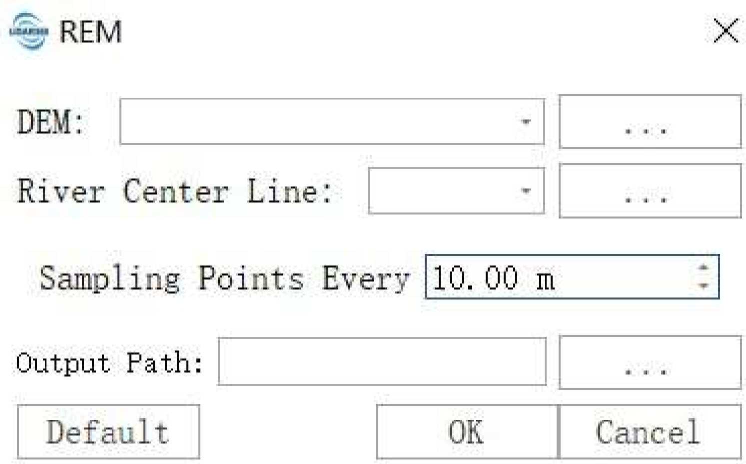
REM(Relative Elevation Model)

Function Overview

The REM (Relative Elevation Model) visualizes the geomorphic features of rivers by eliminating the natural slope of the terrain. The principle is to set the elevation at the riverbed to zero, and the elevation outside the river represents the height relative to the riverbed.

Usage

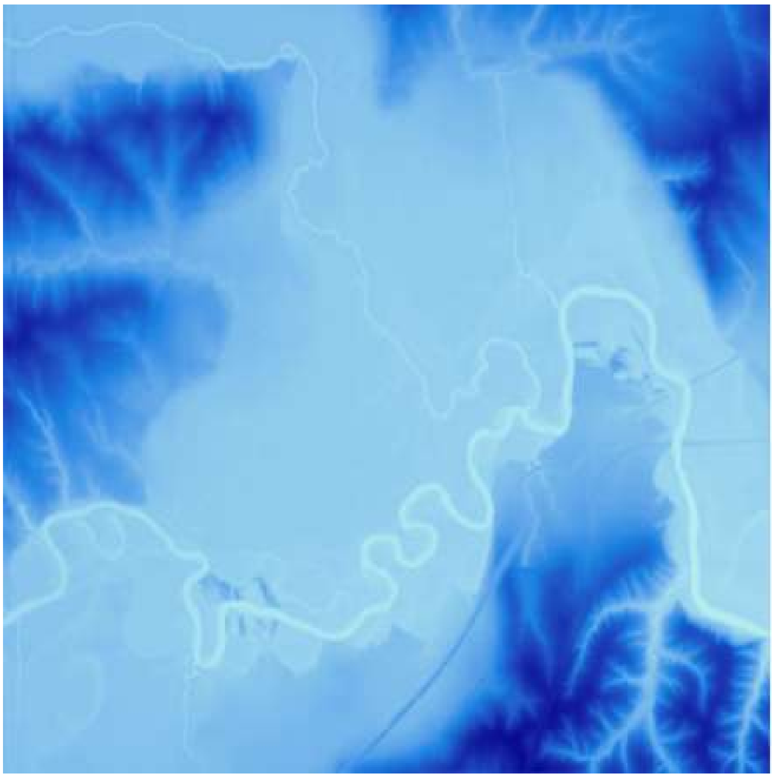
Click on *Terrain* > *REM*.

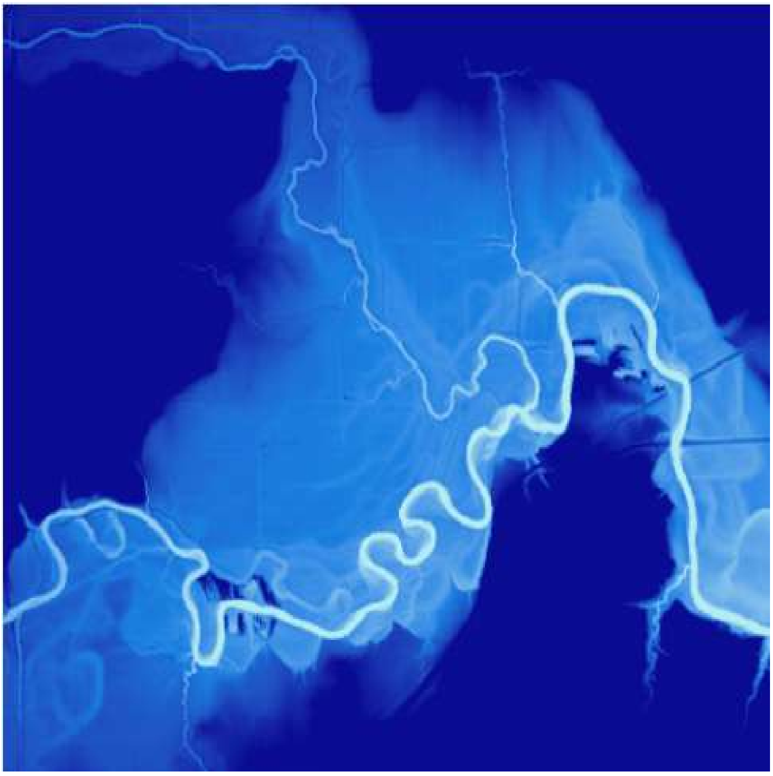


Parameter Settings

- **DEM File**: The original DEM used to generate the relative elevation model.
- **River Center Line Vector File**: The river's main centerline vector file, which can be drawn using the vector editing module.
- **Sampling Point Every**: The distance between adjacent sampling points on the centerline, typically set to the width of the river channel.
- **Output Path**: The path to save the generated REM file.

Comparison between the original DEM and the REM is shown below:





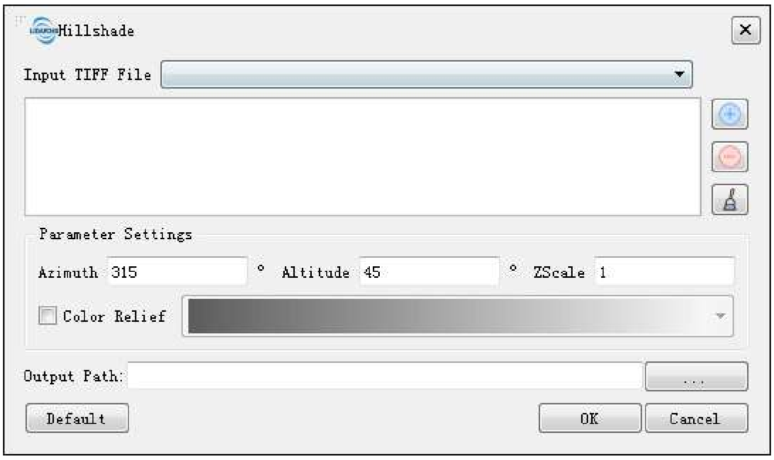
Hillshade

Summary

The Hillshade tool obtains the assumed illumination of the surface by determining the illumination for each image element in the grid. The assumed illumination can be obtained by setting the position of the light source and calculating the illumination value of each image element related to the adjacent pixels. In the analysis or graphic display, the Hillshade tool can greatly enhance the surface visualization especially when using transparency.

Usage

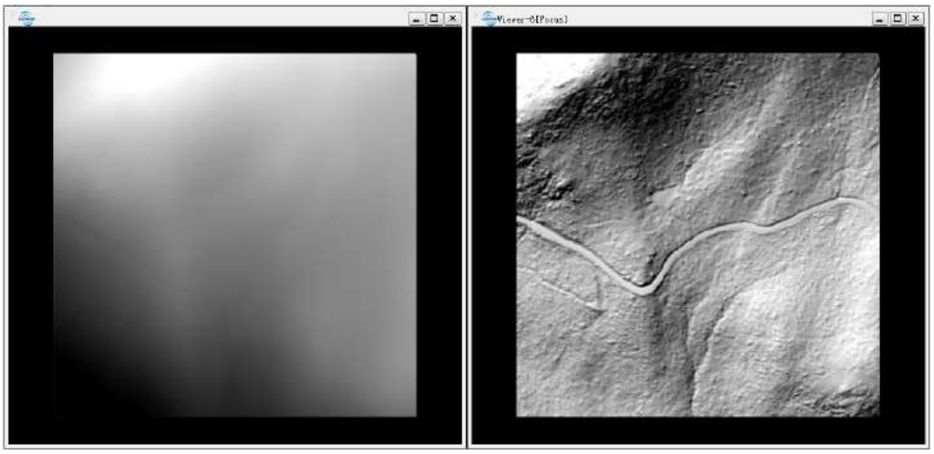
Click on the *Terrain* > *Hillshade*.



Settings

- **Input Tiff File**: Input DEM file format in Tiff. The drop-down box allows to select the Tiff data that has already been opened in the LiDAR360 software. Users can also import Tiff data by clicking .OR delete imported Tiff data by clicking .Or clear all the imported data by clicking .
- **Azimuth (°)(default value is "315")**: Using North as the benchmark, the degree range is 0~360 clockwise and the default is set to 315 degree(NW).
- **Altitude(°)(default value is "45")**: The angle between the incident direction of the light source and the ground plane, which ranges from 0 to 90 degrees and uses 45 degree as default.
- **ZScale(default value is "1")**: The stretch scale of the Z value.
- **Color Relief**: Color rendering of hillshade.
  - **Yes**: There are 11 types of optional color bar.
  - **No(default)**: Black and white color bar.
- **Output path**: The path where the generated hillshade map file will be saved.
- **Default Value**: Click this button to restore all default parameters.

DEM (left below) and Hillshade map (right below).



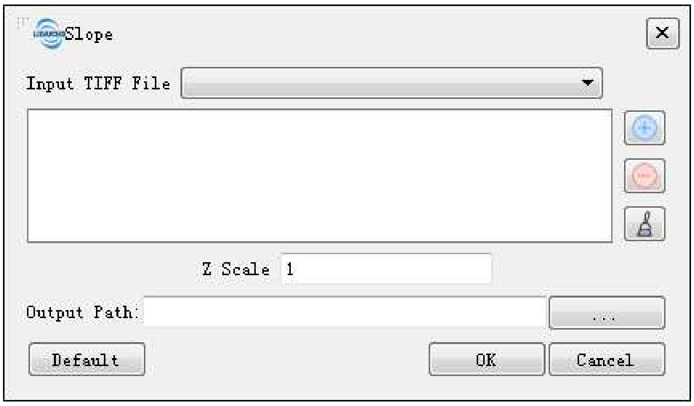
Slope

Functional Overview

Slope shows how deep is the terrain surface. This function analyzes terrain slope based on DEM and generates slope images.

Usage

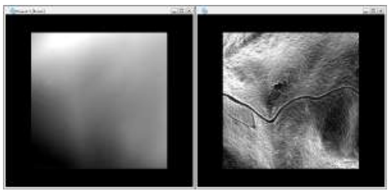
Click on *Terrain* > *Slope*.



Parameters Settings

- **Input Tiff File**: Input DEM file in Tiff format. The Drop-down box allows to select the Tiff data that has already been opened in the LiDAR360 software. User can also can click to import Tiff data.Delete the imported Tiff data by clicking .Clear all the imported data by clicking .
- **ZScale (default value is "1")**: The stretch scale of the Z value.
- **Output path**: The path where the generated slope map file to be saved.
- **Default Value**: Click this tool option to restore all default parameters.

DEM (left below) and slope map (right below).



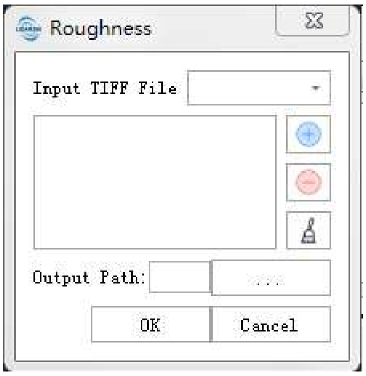
Roughness

Functional Overview

Indicators that reflect changes in surface fluctuations and degree of erosion. It is generally defined as the ratio of surface area to the projected area on the horizontal plane.

Usage

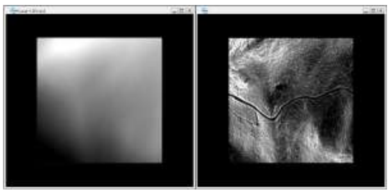
Click on *Terrain* > *Roughness*.



Parameters Settings

- **Input Tiff File**: Input DEM file in Tiff format. The Drop-down box allows to select the Tiff data that has already been opened in the LiDAR360 software. User can also can click to import Tiff data.Delete the imported Tiff data by clicking .Clear all the imported data by clicking .
- **Output Path:** The path where the generated roughness map file to be saved.

DEM (left below) and roughness map (right below).



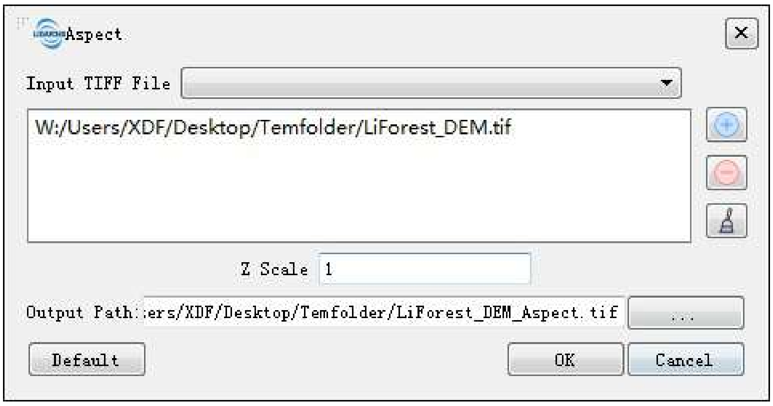
Aspect

Functional Overview

Aspect refers to the orientation of the terrain slope, defined as the direction of the projection of the slope normal on the horizontal plane. Slope direction is of great importance for mountain ecology. The slope is an angle, which is measured clockwise with a range between 0 (north) and 360 (north as well). This function can analyze terrain aspect based on DEM and generates aspect map.

Usage

Click on *Terrain* > *Aspect*.



Parameters Settings

- **Input TIff File**: Input DEM file in Tiff format. The Drop-down box allows to select the Tiff data that has already been opened in the LiDAR360 software. User can also can click to import Tiff data.Delete the imported Tiff data by clicking .Clear all the imported data by clicking .
- **Z Scale (default value is "1")**: The stretch scale of the Z value.
- **Output Path**: The path where the generated aspect file to be saved.
- **Default Value**: Click this tool option to restore all default parameters.

The DEM (left below) and the aspect map (right below).

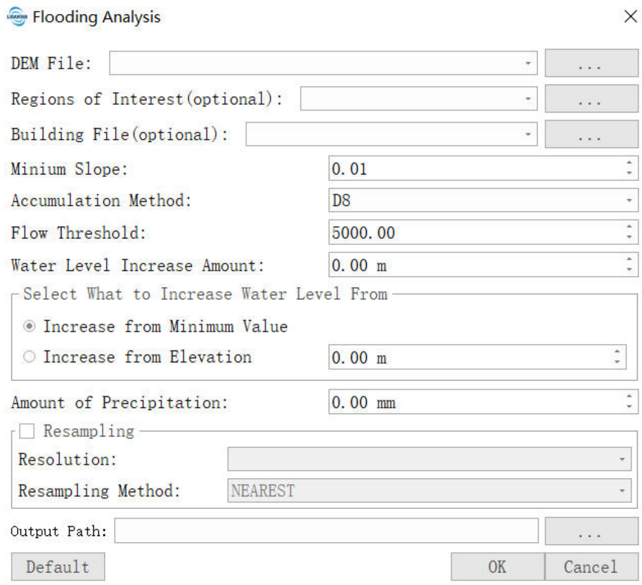
Flooding Analysis

Function Overview

Flooding analysis is a common hydrological GIS application used to simulate and predict areas that may be affected by flooding events.

Usage

Click on *Terrain* > *Flooding Analysis*.



Parameter Settings

- **Input DEM File**: Input DEM file. For generation methods, refer to DEM Generation.
- **Regions of Interest**: Input vector file, can be empty. If an area of interest file is provided, the analysis

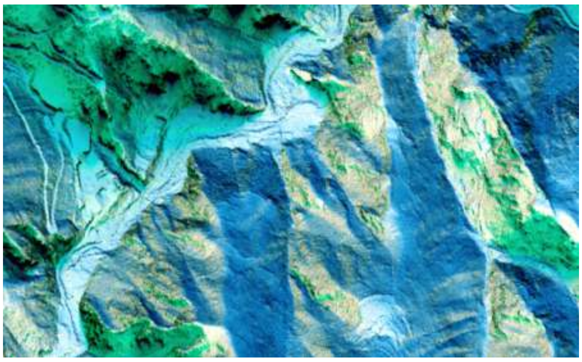
will be confined to that area.

- **Building File**: Input vector file containing extracted building data, can be empty. If provided, the analysis will include the impact on buildings.
- **Minimum Slope**: Used for the depression filling process in DEM data.
- **Accumulation Method**: Used for flow accumulation, currently supports D8, D-Infinity, and MFD methods.
- **Flow Threshold**: Used for river network extraction. Only areas with values above this threshold will form rivers.
- **Water Level Height**: The initial water level height
  - **Raster Minimum Height**: The initial water level height.The minimum height value of the current raster is used as the initial water level height. If the area of interest is not empty, the minimum raster value within the area of interest is used as the initial water level height.
  - **Current Water Level Elevation**: The currently specified value is used as the initial water level height.
- **Subsampling:**Resampling is performed based on the raster pyramid. Different pyramid levels and resolutions are determined by specific data, and consecutive layers are downsampled at a 2:1 ratio. If this option is not selected, the original data will be used directly for processing.
  - **Resolution**: The resolution of the raster. For raster data with fewer than 1,000 rows and columns, lower levels of resolution will not be displayed. The higher the selected resolution, the more accurate the generated results will be, but the process may fail due to insufficient hardware resources.
  - **Sampling Method:** The method of raster subsampling.
- **Amount of Precipitation**: Specify the precipitation amount in millimeters (mm). The precipitation will affect the water level in each watershed, ultimately influencing the flooded areas.
- **Output Path**: The algorithm will output the flooding analysis results and related reports.

Annual Insolation

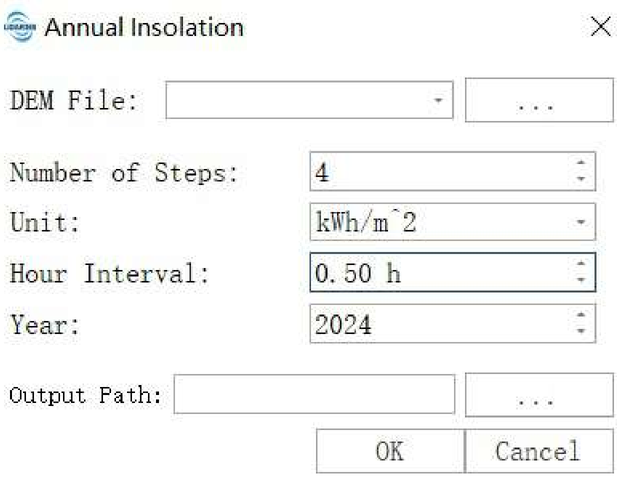
Function Overview

Annual Insolation is used to estimate the annual solar radiation within a specific area. It samples time points throughout a year according to specified parameters, calculates insolation for each sampling time (day), and outputs a multiband dataset recording insolation values for each time sample. This can be used to assess trends in annual solar radiation for the area.



Usage

Click on *Terrain* > *Annual Insolation*.



Parameter Settings

- **Input DEM File**: Input Digital Elevation Model (DEM) file. Refer to DEM Generation for methods of generating DEM.
- **Number of Steps**: Number of sampling days in the current year.
- **Units**: Specifies the unit of output insolation.
- **Hour Interval**: Time interval for calculating solar radiation on the sampling day.
- **Year**: Specifies the year for calculation.
- **Output Path**: Path to output the multiband TIF dataset.

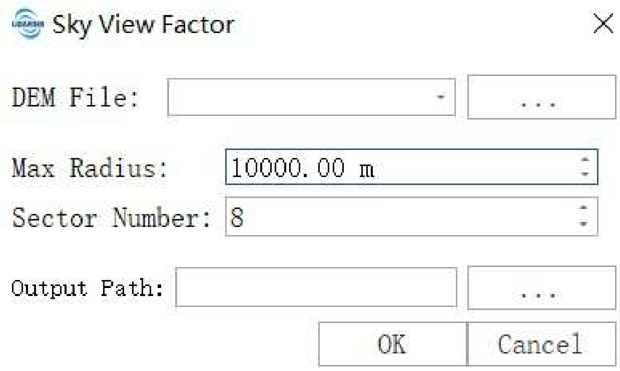
Sky View Factor

Function Overview

The Sky View Factor is commonly used to assess the degree of sky obstruction by buildings in complex urban environments. It is an important indicator in urban morphology.

Usage

Click on *Terrain* > *Sky View Factor*.



Parameter Settings

- **Input DEM File**: The input DEM file. For information on generating a DEM, see DEM Generation.
- **Max Radius**: The maximum search radius for the calculation process.
- **Sector Number**: The number of sectors used in the calculation process, which refers to the sampling number of visibility computations for each pixel.
- **Output Path**: Specify the output path for the results.

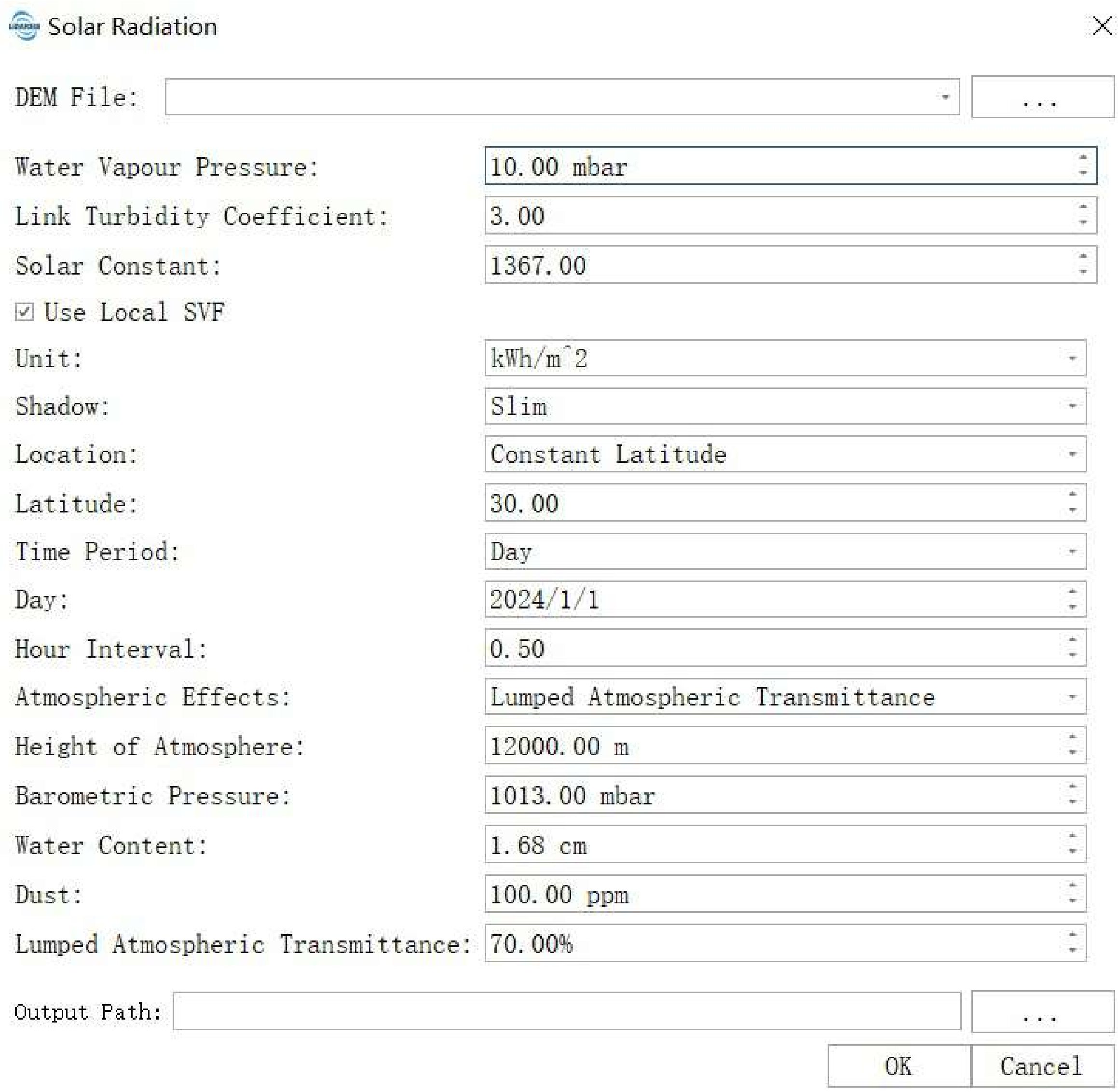
Solar Radiation

Function Overview

The Solar Radiation tool is used to calculate solar radiation for a specific time period, outputting direct, diffuse, and total solar radiation values.

Usage

Click on *Terrain > Incident Solar Radiation*.



Parameter Settings

- **Input DEM File**: The input DEM file. For information on generating a DEM, see DEM Generation.
- **Water Vapour Pressure**: Describes the extent of absorption of solar radiation.
- **Link Turbidity Coefficient**: The atmospheric turbidity coefficient, describing the extent to which

colloidal particles obstruct light transmission. Higher values generally indicate more severe air pollution.

- **Solar Constant**: The amount of solar radiation received per unit area per second at the top of the atmosphere, perpendicular to the solar rays, at an average distance (D=1.496e8km) from the sun.
- **Use Local SVF**: Whether to use the sky view factor for the local slope.
- **Unit**: The unit of radiation values in the output file.
- **Shadow**: Options for shadow tracking:
  - slim: Tracks shadows at grid nodes (faster but may produce artifacts).
  - fat: Tracks shadows for the entire grid.
  - none: Ignores shadow effects.
- **Location**: Describes the latitude of the area.
- **Latitude**: Used when the location is set to a fixed latitude.
- **Time Period**: Sets the time period for the calculation; options include moment, day, range of days.
- **Start Day**: The start time of the time period, valid when the time period option is set to range of days.
- **End Day**: The end time of the time period, valid when the time period option is set to range of days.
- **Day**: The specific date to be calculated, valid when the time period option is set to day.
- **Moment**: The specific moment to be calculated, valid when the time period option is set to moment.
- **Hour Interval (Hours)**: The hourly interval used for calculating solar radiation on a daily scale, valid when the time period options are day and range of days.
- **Day Interval (Days)**: The daily interval used for calculating solar radiation over a period of time, valid when the time period option is range of days.
- **Atmospheric Effects**: Describes the atmospheric influence model, including atmospheric height, water content, dust, and atmospheric transmittance:
  - Height of Atmosphere and Vapour Pressure
  - Air Pressure, Water and Dust Content
  - Lumped Atmospheric Transmittance
  - Hofierka and Suri
- **Height of Atmosphere**: The height of the atmosphere, default value is 12000 meters.
- **Barometric Pressure**: The atmospheric pressure, default value is 1013 mbar.
- **Water Content**: Describes the impact of atmospheric water content; the vertical atmospheric water content (cm) ranges from 1.5 to 1.7.
- **Dust**: Describes the impact of dust.
- **Lumped Atmospheric Transmittance**: Describes the impact of atmospheric transmittance.
- **Output Path**: The path for the output files, including direct, diffuse, and total solar radiation TIF files.

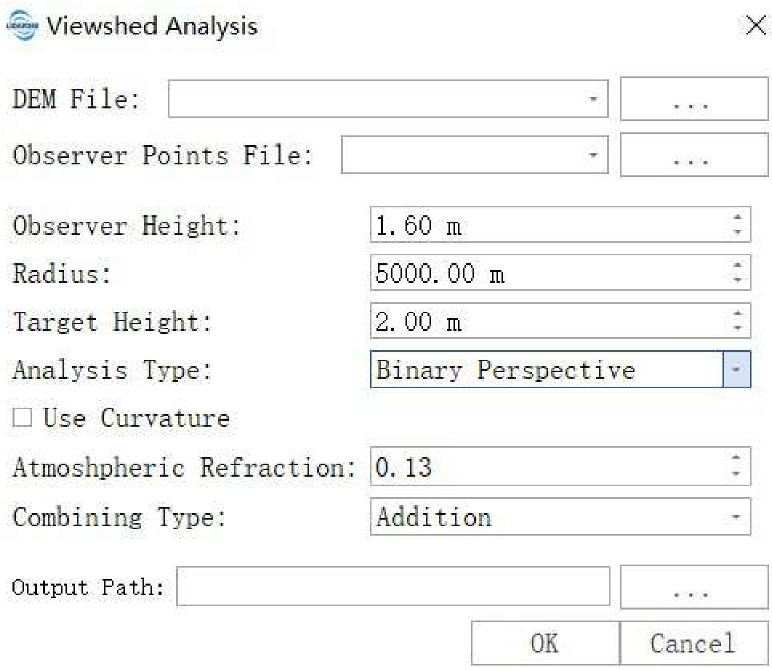
Viewshed Analysis

Function Overview

The Viewshed Analysis tool determines whether each pixel value in the terrain data is visible based on a series of viewpoint data.

Usage

Click on *Terrain* > *Viewshed Analysis*.



Parameter Settings

- **Input DEM File**: The input DEM file. For information on generating a DEM, see DEM Generation.
- **Observer Points File**: Input point-type vector file representing the positions of observation points.
- **Observer Height**: The height of the observer relative to the viewpoint.
- **Radius**: The radius for the viewshed analysis.
- **Target Height**: The height value of all visible terrain areas checked from the observation point.
- **Analysis Type**: Determines the output result:
  - 0: Binary Perspective; visible areas are 1, non-visible areas are 0.
  - 1: Depth Below the Horizon ; the height each location needs to reach to be visible.
  - 2: Horizon Boundary; the edges of the viewshed.
- **Use Curvature**: Whether to consider the curvature of the Earth.
- **Atmospheric Refraction**: Also known as the vertical refraction coefficient, describing the atmospheric refraction rate factor affecting visibility.
- **Combining Type**: When using multiple observation points, the outputs will be combined based on this parameter.

- **Output Path**: Specifies the path for the output results.

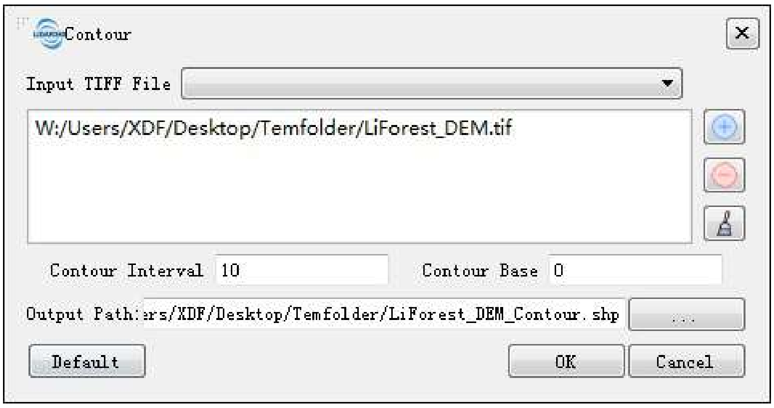
Raster to Contour

Functional Overview

The contour refers to the curve of the adjacent points with equal elevation on the topographic map. Raster to Contour is to connect the grid points that have the same elevation value, which can build the contour.

Usage

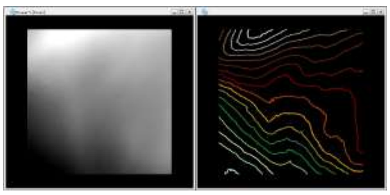
Click*Terrain > Raster to Contour*.



Parameters Settings

- **Input TIff File**: Input DEM file in Tiff format. The Drop-down box allows to select the Tiff data that has already been opened in the LiDAR360 software. User can also can click to import Tiff data.Delete the imported Tiff data by clicking .Clear all the imported data by clicking .
- **Contour Interval (m, default value is "10")**: The elevation difference between adjacent contours.
- **Contour Base (default is "0")**: The elevation of the generated contour is calculated from the reference elevation, that is, the elevation which differs from the reference by **interval** multiples of the interval is the contour elevation.For example, the Contour Base is 0, Contour Interval is 10, then the elevation distribution of the contour is: 0, -10, -20, -30... 10, 20, 30... .
- **Output path**: The path where the generated contour file to be saved.
- **Default Value**: Click this tool option to restore all default parameters.

DEM (left below) and contour (right below)



> Note: To smooth the contours, you can usethe elevation smoothing tool in LiModel Editing.

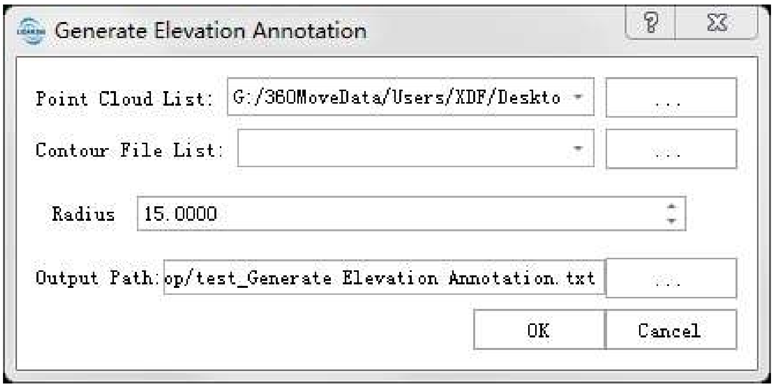
Generating elevation annotations

Summary

Using point cloud and contour lines to generate elevation points for topographic mapping.Notes Elevation Points File.

Usage

Click *Terrain > Generating elevation annotation*.



Settings

- **Input Tiff File**: Input DEM file format in LiData.
- **Input Contour File (.shp)**: Input Contour file format in shp，The file needs to contain one or more contours.
- **Radius (m)(default value is "15")**: A Note Elevation Point is generated within a region with a defined radius.
- **Output path**: The path of the generated contour lines file(s) to save.

Point Cloud to Contour

Functional Overview

The Point Cloud to Contour tool extracts elevation contours from input point cloud data., The specific process is as follows: Develop a triangular regular network (TIN) from ground class points. This is then used to TIN to Contour.

Usage

Click on *Terrain* > *Point Cloud to Contour*.

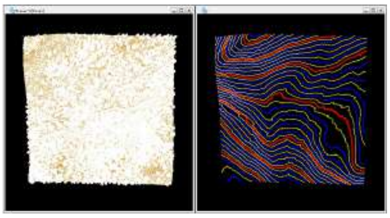
Parameters Settings

- **Input Data**: Ensure that each input point cloud data is the data that has been classified by ground points; the input file can be a single data file or a point cloud data set; the file(s) to be processed must be opened in the LiDAR360 software.

- **Scale**: There are 11 scales for generating the point cloud, which correspond to different height intervals.
  - **1: 500**: 1: 500 scale.
  - **1: 1000**: 1: 1000 scale.
  - **1: 2000**: 1: 2000 scale.
  - **1: 5000**: 1: 5000 scale.
  - **1: 10000(default)**: 1: 10000 scale.
  - ... ...
- **From Class**: Point cloud classes that participates in the construction of contour lines.
- **Contour Base (m)(default value is "0")**: The elevation of the generated contour is calculated from the base elevation. The difference between **elevation of the contour and base elevation is** the integral multiples of the contour interval. For example, the criterion is 0, **and the contour interval** is 10m, thus the distribution of contour is: 0, -10, -20, -30, 10, 20, 30...
- **Triangle's Maximum Length (m)(default value is "30")**: The triangulation network is constructed from elevation attributed ground points, that is used to generate contours. If the length of a triangle side, or arc, in the network is greater than the defined threshold, that edge will be excluded from the construction of elevation contours. If there is no interruption in the expected generated contour line, this threshold can be set to exceed the maximum size of the ground point cloud void.
- **Minor Contour (default state is checked)**: It is also known as minor contour. A contour line drawn at half contour when the basic contour does not show some local topography. If you do not need to generate this type of contour line, deselect it.
  - **Spacing (m)(default value is "2.5")**: Height interval, the absolute value of height difference between two adjacent minor contour lines.
  - **Color (default value is "yellow")**: The color of Minor Contours. It is modifiable.
  - **Line Weight (default value is "1")**: The weight of Minor Contours. It is modifiable.
- **Basic Contour (default state is checked)**: It is also known as basic contour. It allows users to include lines of equal elevation that represent the landscape. If you do not need to generate this type of contour line, deselect it.
  - **Spacing (m)(default value is "5")**: Height interval, the absolute value of height difference between two adjacent minor contour lines.
  - **Color (default value is "blue")**: The color of Basic Contours. It is modifiable.
  - **Line Weight (default value is "2")**: The weight of Basic Contours. It is modifiable.
- **Major Contour (default state is checked)**: It is also known as major contour. In order to facilitate the interpretation of contour elevation, contours are drawn from the elevation starting surface and at every 4th interval above and below the Base Contour. If you do not need to generate this type of contour line, deselect it.
  - **Spacing (m)(default value is "25")**: Height interval, the absolute value of height difference between two adjacent major contour lines.
  - **Color (default value is "red")**: The color of Major Contours. It is modifiable.
  - **Line Weight (default value is "3")**: The weight of Major Contours. It is modifiable.
- **Generate Notes Elevation Points (default state is checked)**: Generate notes elevation points in outputs. Notes Elevation Points formatrefer to the appendix.
  - **Radius (m)(default value is "15")**: A Note Elevation Point is generated within a region with a defined radius “15”.
- **Optimization**: Optimized settings for smoothing the generated contours.

  - **Mean Smooth (default state is checked)**: The Mean Smoothing contour optimization method replaces the measured or estimated ground elevation points along a contour line with mean contour points that represent weighted average coordinates of adjacent contour points found on a given line of equal elevation.
    - **3**: The number of neighboring contour points participating in smoothing at the current point is 3.
    - **5 (default)**: The number of neighboring contour points participating in smoothing at the current point is 5.
    - **7**: The number of neighboring contour points participating in smoothing at the current point is 7.
  - **Bezier Smooth (default value is "155")**: A Smoothing contour optimization method that fall within an angle threshold range from 0 to 180 degrees. The larger the threshold setting, the smoother the contour lines.
  - **Delete the contour when its area <(m2)(default value is "25")**: When the area of the closed contour is less than this threshold, it will be deleted.
  - **Delete the contour when its length (m)(default value is "5")**: Non-closed contours are deleted if their length is less than this threshold.
- **Generate Shp (default)**: Save Point Cloud to Contour tool outputs to a shapefile (shp) format with the following feature attributes: contour line type, contour line width, contour line color, and contour line elevation.
  - **Polyline (default)**: The linear information in the Shp file is 2-D.
  - **polyline25D**: The linear information in the Shp file is 2.5-D.
- **Generate DXF**: The generation of DXF format contour files,
- **Output path**: The path where the generated contour file to be saved.
- **Default Value**: Click this tool option to restore all default parameters.

Point cloud data (left below) and contour map (right below).



> Note: Before using this tool, ground points should be generated in advance.

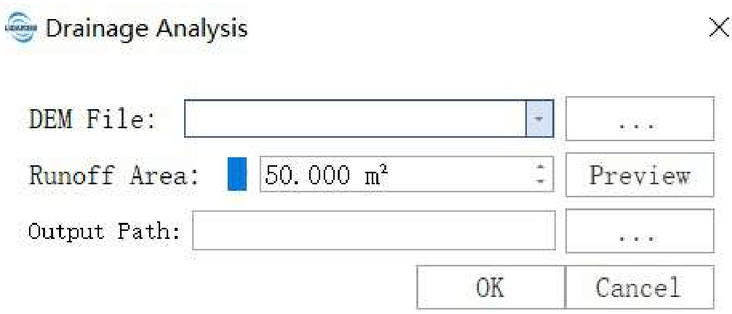
Drainage Analysis

Function Overview

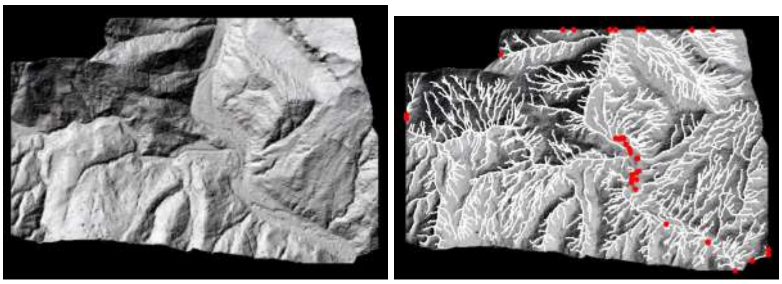
The Drainage Line Algorithm is a method used to calculate the flow paths of water in a DEM (Digital Elevation Model). This algorithm is commonly used in GIS (Geographic Information Systems) and hydrology to simulate and analyze the flow of surface water.

Usage

- **To use this function, a digital elevation model (DEM) must first be generated.** Click *Terrain* > *Drainage Analysis*.



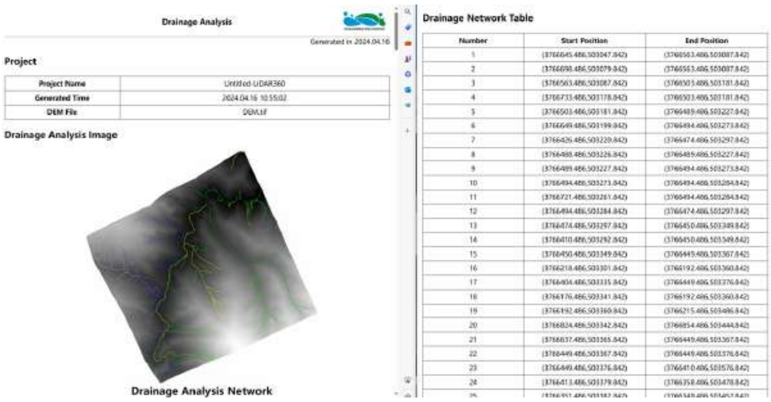
Parameter Settings ### - **DEM File**:The DEM file to be input.Refer to the [DEM](DEM.md) for details。 - **Runoff Area**:The total area of water flow passing through a given pixel during surface water flow. A smaller value will extract more tributary paths, while a larger value will retain more of the main flow paths. - **Output Path**: The algorithm will output the drainage line vector results and a related analysis report. ---------- DEM data (see the image on the left) and the drainage line analysis vector results (see the image on the right).



</div>

The white lines represent drainage lines, which simulate the flow paths of surface water. The red points indicate the positions of confluence points, where surface water flows converge to a single outlet point.

The analysis report is shown in the image below.



Drainage Analysis

Generated in 2024.04.16

Project

Project Name	Untitled-LIDAR360
Generated Time	2024.04.16 10:55:02
DEM File	DEM.tif

Drainage Analysis Image

Drainage Analysis Network

Drainage Network Table

Number	Start Position	End Position
1	(3766645.486,503047.842)	(3766563.486,503087.842)
2	(3766698.486,503079.842)	(3766563.486,503087.842)
3	(3766563.486,503087.842)	(3766503.486,503181.842)
4	(3766733.486,503178.842)	(3766503.486,503181.842)
5	(3766503.486,503181.842)	(3766489.486,503227.842)
6	(3766649.486,503199.842)	(3766494.486,503273.842)
7	(3766426.486,503220.842)	(3766474.486,503297.842)
8	(3766488.486,503226.842)	(3766489.486,503227.842)
9	(3766489.486,503227.842)	(3766494.486,503273.842)
10	(3766494.486,503273.842)	(3766494.486,503284.842)
11	(3766721.486,503261.842)	(3766494.486,503284.842)
12	(3766494.486,503284.842)	(3766474.486,503297.842)
13	(3766474.486,503297.842)	(3766450.486,503349.842)
14	(3766410.486,503292.842)	(3766450.486,503349.842)
15	(3766450.486,503349.842)	(3766449.486,503367.842)
16	(3766218.486,503301.842)	(3766192.486,503360.842)
17	(3766404.486,503335.842)	(3766449.486,503376.842)
18	(3766176.486,503341.842)	(3766192.486,503360.842)
19	(3766192.486,503360.842)	(3766215.486,503486.842)
20	(3766824.486,503342.842)	(3766854.486,503444.842)
21	(3766637.486,503365.842)	(3766449.486,503367.842)
22	(3766449.486,503367.842)	(3766449.486,503376.842)
23	(3766449.486,503376.842)	(3766410.486,503576.842)
24	(3766413.486,503379.842)	(3766358.486,503478.842)
25	[illegible]	[illegible]

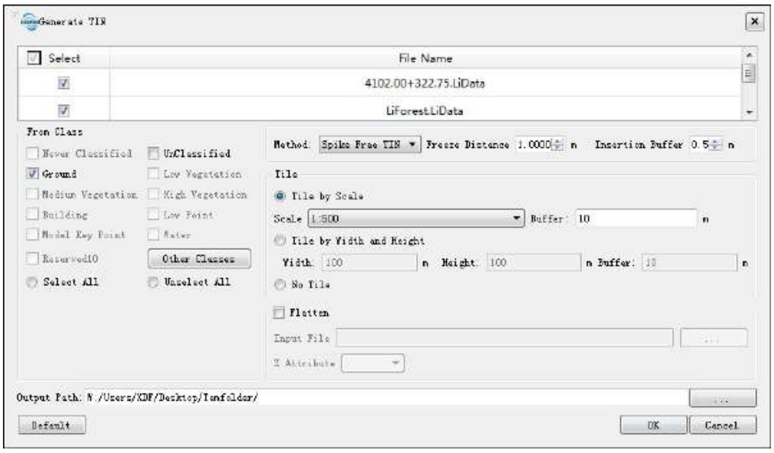
Generate TIN

Summary

Generate irregular triangulation model based on point cloud, LiDAR360 adopts custom triangulation model file format*.LiTin。

Usage

Click*Terrain* > *Generate TIN*。



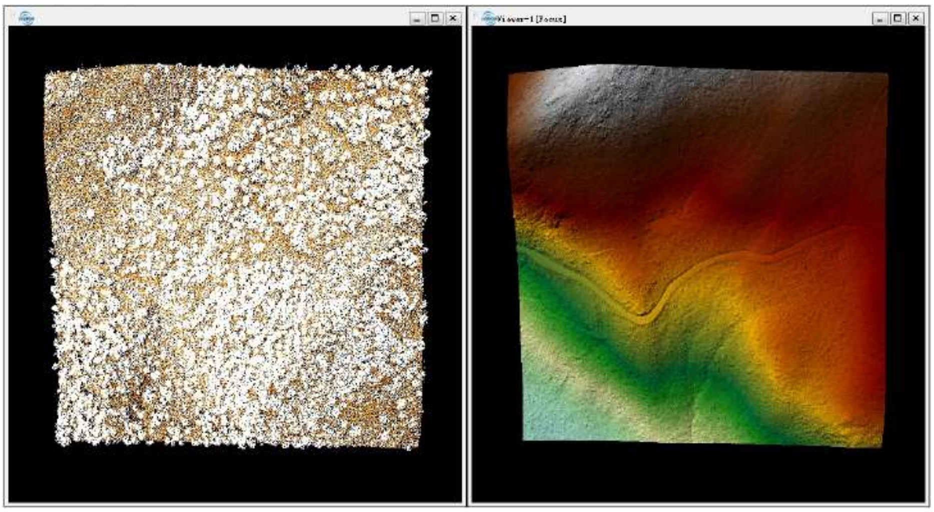
Parameters Setting

- **Input data**: The file can be a single point cloud data file or a point cloud data set; the data to be processed must be opened in the LiDAR360 software.
- **From Classes**: Source class(es).
- **Method**: Methods to generate TIN. Now there are two methods, Delaunay and Spike Free TIN.
    - **Delaunay**: Use point-by-point insertion to generate Delaunay TIN.
    - **Maximum Triangle Side Length(default value is "30 meter")**: Delete any triangle whose longest size is longer than this value.
    - **Spike Free TIN**: Remove the points with abnormal elevation, in order to generate TIN without obvious spikes.
        - **Freeze Distance (default value is "1.0 meter")** :The shortest distance in xy plane of each side of triangle in the triangulated network. When inserting a new point and its Z value lower Insertion Buffer, freeze all the triangles whose three sides are all less than Freeze Distance. The frozen triangles will no longer change. The larger this value is, the fewer points will be involved in creating the network, the more smooth the network will be and the less details will be included. Conversely, more points will be involved in creating the network, the more details will be included and the more possible spikes will appear in the network.
        - **Insertion Buffer (default value is "0.5 meter")** :When trying to freeze a new triangle right after freezing another one, the difference in altitude value should be larger than this threshold. Decreasing this value will result in more triangles will be frozen too early, new point cannot be inserted, less spikes will appear, processing will be faster, and more details

will be lost. Conversely, more details will be included, abd more spikes will be included in the network.

- **Tile**：The results consist of three types of tiles: "Tile by Scale","Tile by Width and Height" and "No Tile".
    - **Tile by Scale(default)**：Generate TIN by a certain scale.
        - **Scale**：Point cloud tile scale.
            - **1：500(default)**：1：500 scale.
            - **1：1000**：1：1000 scale.
            - **1：2000**：1：2000 scale.
            - **1：5000**：1：5000 scale.
        - **Buffer(m))(default value is "10")**:The buffer size of the overlap between two adjacent point cloud tiles.
    - **Tile by Width and Height**:Tile by a certain width and height.
        - **Width(m)(default value is "100")**:Width of point cloud tile.
        - **Height(m)(default value is "100")**:Height of point cloud tile.
        - **Buffer(m)(default value is "10")**:The buffer size of the overlap between two adjacent point cloud tiles.
    - **No Tile**：The point cloud generates a TIN as a whole and no tile.
- **Flatten(optional)**：The flatten tool is used to flatten the specified range based on user input files.
- **Input File**: The input file is the polygon-type shapefile. The irregular triangulation will be flattened based on the file range and the Z attribute.
- **Z Attribute**:The Z attribute of flattening.
- **Import terrain features (optional)**: Import terrain features (breaklines, simulated terrain points) component files, breaklines and simulated terrain points will participate in TIN generation.
- **Output path**:The path where the generated TIN file to be saved.

Point cloud data (left below) and TIN (right below).




```
    @inproceedings{
        author={ Khosravipour A, Skidmore A K, Isenburg M},
        title={Khosravipour A, Skidmore A K, Isenburg M. Generating spike-free digital surface models using L
iDAR raw point clouds: A new approach for forestry applications},
        booktitle={ International journal of applied earth observation and geoinformation, 52: 104-114},
```

```
    year={2016}
}
```

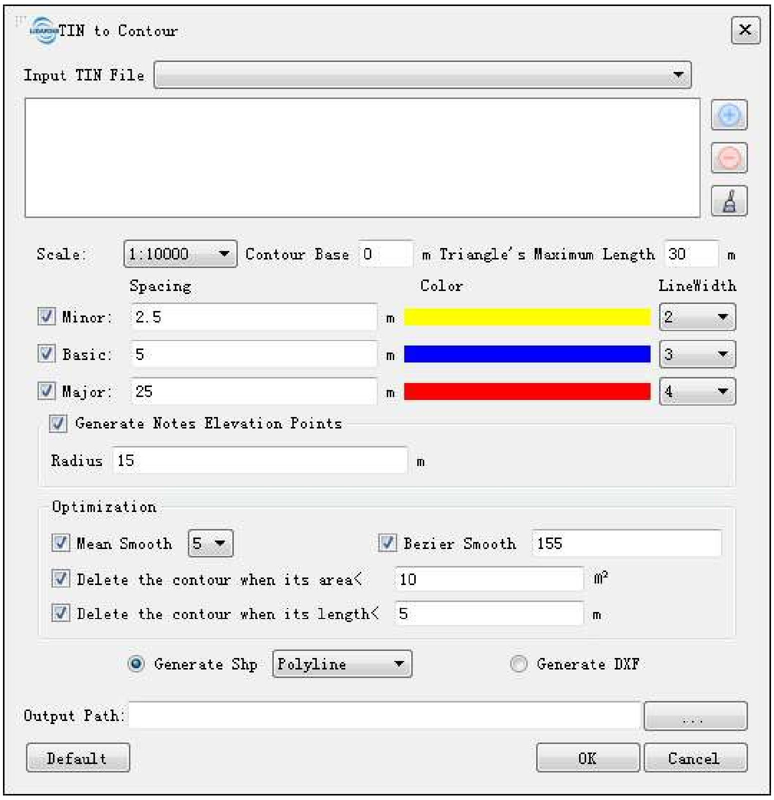
TIN to Contour

Summary

This function directly connects contour point positions of the same elevation on the TIN stored in the LiTin file to generate contour lines.

Usage

Click on the *Terrain* > *TIN to Contour*.



Settings

- **Input TIN File**: Input LiTin file(s). The drop-down box allows to select the LiTin data that has already been opened in the LiDAR360 software. User can also import LiTin data by clicking . Or delete imported LiTin data by clicking . Or clear all the imported data by clicking .
- **Default Value**: Click this button to restore all default parameters.
- **The other parameters settings of the interface are the same as Point Cloud to Contour.**

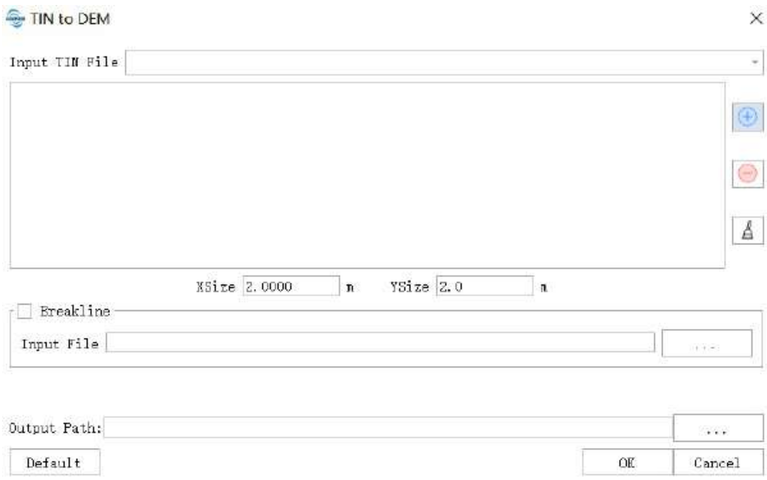
TIN to DEM

Summary

This function can generate DEM with the LiTin file.

Usage

Click *Terrain > TIN to DEM*。



Settings

- **Input TIN File**:Input LiTin file(s). The drop-down box allows to select the LiTin data that has already been opened in the LiDAR360 software. User can also import LiTin data by clicking . Or delete imported LiTin data by clicking . Or clear all the imported data by clicking .
- **XSize(m)(default value is "2.0")**:The XSize of the grid to generate the DEM.
- **YSize(m)(default value is "2.0")**:The YSize of the grid to generate the DEM.
- **Import Breakline(optional)**:Import the breakline file and add the multi-segment lines in the file as soft breakline to TIN.
- **Output path**:The path where the generated DEM file to be saved.
- **Default Value**: Click this button to restore all default parameters.

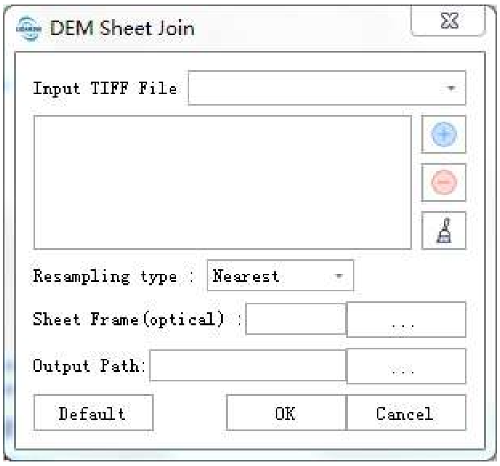
DEM Edge Alignment

Functional Overview

By processing the border area of the digital elevation model of adjacent frames, the same grid point elevation within the overlapping range is guaranteed to be consistent.

Usage

1. Click *Terrain* > *DEM Edge Alignment*, the DEM Edge Alignment interface will pop up:



1. Click  to add the DEM file and select the digital elevation model file of the edge to be connected (currently only .tif format is supported);

2. Set the parameters, and click OK to execute the DEM Edge Alignment.

Parameters Settings

- **Resampling**: The method used for resampling the overlapping area.
  - **Nearest Neighbor sampling (default)**: The nearest neighbor method, sampling from the nearest neighbor.
  - **Bilinear sampling**: Bilinear sampling (2 x 2 kernel).
  - **Cubic convolution approximation**: Cubic Convolution Approximation (4×4 kernel).
  - **Cubic B-spline approximation**: Cubic B-spline approximation (4 x 4 kernel).
  - **Lanczos window sinusoidal interpolation**: Lanczos Windowed Sine Interpolation (6 x 6 kernel). Lanczos can be used as a low-pass filter or to smoothly interpolate between its samples The value of the digital signal.
  - **Average sample**: Calculates the average of all non-null pixels.
  - **Statistical sample**: Select the value that occurs most frequently among all sampling points.
- **Tile frame (optional)**: Add a frame. If no framing frame is added, it will be saved in the original size of each DEM file; if a framing frame is set, the framing will be performed according to the framing frame.

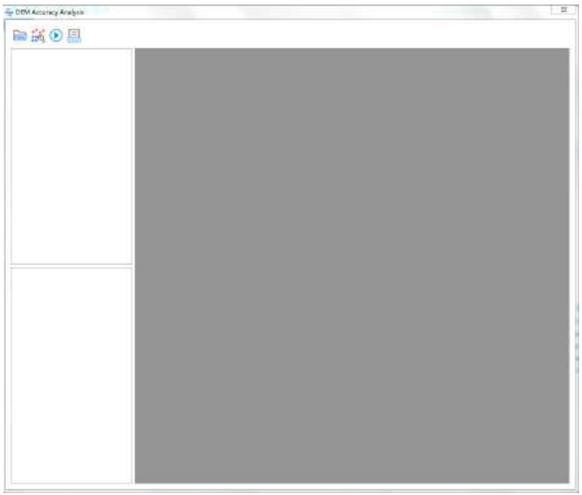
DEM Accuracy Analysis

Functional Overview

The elevation accuracy of the digital elevation model is analyzed by the checkpoint method.

Usage

1. Click*Terrain* > *DEM Accuracy Assessment*, the DEM Accuracy Assessment interface will pop up:



1. Click Add DEM file, select the digital elevation model file be analyzed (currently only .tif format);
2. Click Add Check point to add a checkpoint file (only .txt format is supported);；
3. Click Execute Analysis ,set parameters to execute precision assessment and assessment result statistics (corresponding to checkpoints) number, number of gross error points, gross error rate, average error, medium error, tolerance);
4. Click the export report , set the relevant information to export the precision assessment report.

Import DEM

Add the digital elevation model file to be analyzed, currently only .tif format files are supported.

Import checkpoint

Add checkpoints. Checkpoints can be derived from GNSS measurement, photogrammetry, existing results (topographic maps, digital elevation models, etc.) Reference System.

Analysis

To perform an accuracy analysis, it must be performed after adding the DEM file and adding the checkpoint file.

Analysis Settings

- **RMSE formula**: The formula for calculating the error in elevation.
  - ***High precision***:

$$(M = \pm\sqrt{\frac{\sum_{i=1}^{n}\Delta_i^2}{n}})$$

  - ***Equal precision***:

$$(M = \pm\sqrt{\frac{\sum_{i=1}^{n}\Delta_i^2}{2n}})$$

    In the formula, M is the error in the result, n is the number of detection points, and Δ_i is elevation difference.
- **Tolerance**: Error tolerance

Note: ①Usually, in the case of high-precision detection, use 2 times the allowable medium error as the tolerance, and use formula 1 to calculate the result medium error; in the same precision detection, use the $2\sqrt{2}$. The allowable middle error is used as the tolerance, and the middle error of the result is calculated by formula 2. Errors within the tolerance range participate in the precision statistics, and errors beyond the tolerance are regarded as gross errors. ② After the analysis is performed, the analysis status of each DEM file, indicates the digital elevation model error The gross error rate is less than 5%, indicates that there is no detection point within the range of the digital elevation model, means that the gross error rate of the digital elevation model is greater than 5%.

Export report

Export the error detection report in elevation.

Export information settings

- **Project Name**: The project name.
- **scale**: Scale bar.
- **Inspection Method**: The detection method.
- **Unit**: Unit.
- **Instrument Name, Model**: Instrument name, model.
- **Instrument Number**: The instrument number.
- **Output Path**: The report save path, currently only supports .html format.

Deviation Analysis

Description

To calculate the distance between two point cloud. The distance of each point of the compared point cloud from the reference point cloud will be saved as an additional attribute. This function can calculate the distance between any two point in 3D space, which can be used for deviation analysis.

Usage

Click *Terrain* > *Deviation Analysis*.

Settings

- **Reference Cloud**: Selected reference LiDAR data.
- **Compared Cloud**: Selected LiDAR data to be compared.
- **From Class**: Select point class involved in Deviation Analysis.
- **Voxel Size**: The smaller the voxel size, the more precise the deviation analysis.
- **Minimum Distance**: The additional attribute value of points smaller than this distance will be set to 0. Additional properties with distances greater than this threshold will be saved.
- **By Default**: Click this button to restore default settings.

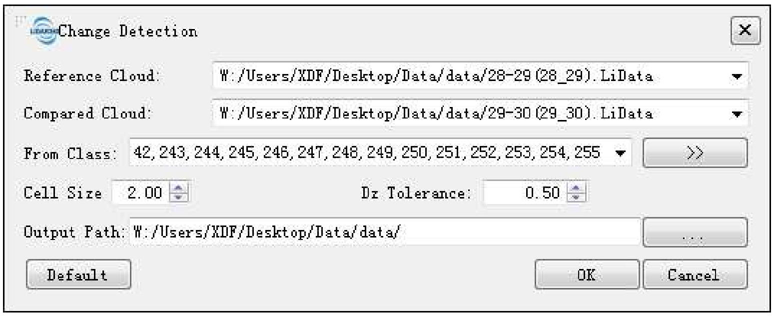
Change Detection

Functional Overview

Calculate the relative change in height between two point clouds and output it as a TIFF image and an HTML report. In the image, red represents an increase while green represents a decrease, with other areas displayed in grayscale according to their elevation values. The relative change in height between the two point clouds will be added as additional attributes to the corresponding LiData files (for example, the elevation change of a comparison point cloud relative to a reference point cloud will be written as an additional attribute to the comparison point cloud's LiData). This function can be used for disaster analysis, illegal construction comparison, earthwork volume change analysis, vegetation growth change analysis, etc.

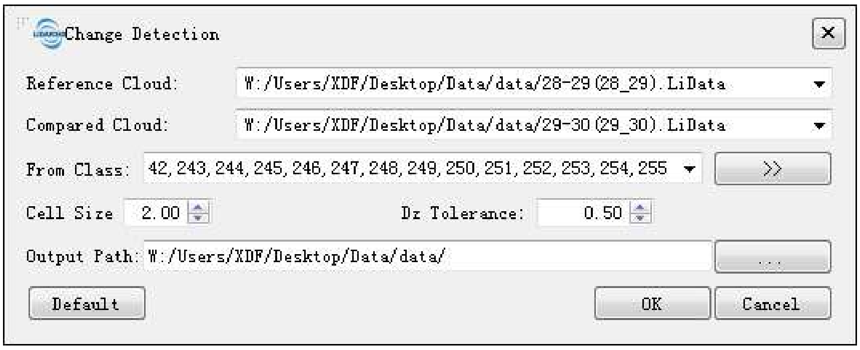
Usage

Click *Terrain* > *Change Detection*.



Parameters Settings

- **Reference Cloud**: Select the reference point cloud. The reference point cloud is used as a baseline for comparing against other clouds; its changes (currently only Z-values) are recorded as "DistanceReference" attribute and shown in green color on images.
- **Compared Cloud**: Select the comparison point cloud. Changes (currently only Z-values) of this compared with those of Reference Point Cloud are recorded as "DistanceCompare" attribute and shown in red color on images.
- **From Class**: Starting class of points that participate in change detection. Each category from both sets of points will be compared separately.
- **Cell Size**: Grid length when converting points into grids; smaller values result in more detailed comparisons but take longer time.
- **Dz Tolerance Min/Max**: Only when changes fall within these limits would they be assigned distance values; otherwise, they are considered as no change and the corresponding distance value is 0.
- **Output Path**: Output path for intermediate results and HTML report.
- **Default**: Click this button to restore all parameters to their default values.
- **Report Setting**: After the function finishes running, a pie chart of change statistics will be generated as shown below.



> Note: The LiData versions of both reference point cloud and comparison point cloud must be 2.0 or above. Lower version LiData can be converted to higher version using Convert LiDatatoLiData. Note: Additional attribute values can be displayed by right-clicking on them in the menu.

Construction Progress Analysis

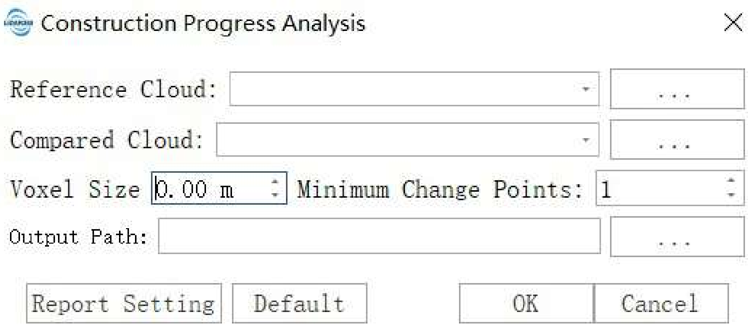
Function Overview

Calculates the volumetric changes between two phases of point cloud data and outputs them as TIFF format images and HTML reports. In the images, red represents increase in volume, green represents decrease, and other areas are shown in grayscale based on elevation values. The relative volume change between the two phases will be added as an additional attribute to the corresponding LiData files. Specifically:

- Areas where the volume has increased relative to the reference point cloud will have the attribute ConstructionProgressAnalysis set to "1".
- Areas where the volume has decreased relative to the reference point cloud will have the attribute ConstructionProgressAnalysis set to "-1".
- Areas where there is no change will have the attribute set to "0".

Usage

Click on *Terrain* > *Construction Progress Analysis*.



Parameter Settings

- **Reference Cloud**: Select the reference point cloud. This serves as the baseline for comparison. Areas where the volume has decreased relative to the comparison point cloud will have this attribute written into the reference point cloud and shown in green in the image.
- **Compared Cloud**: Select the comparison point cloud. Areas where the volume has increased relative to the reference point cloud will have this attribute written into the comparison point cloud and shown in red in the image.
- **Voxel Size**: Size of the grid used to rasterize the point clouds. Smaller values provide finer detection of changes between point clouds, while larger values provide a coarser detection.
- **Minimum Change Points**: After detecting changed points, regions smaller than this value will not be considered as having volumetric changes.
- **Report Setting**: Click this button to set the name and information for the report.
- **Default**: Click this button to reset all parameters to their default values.
- **Output Path**: Path for outputting intermediate results and HTML reports.

> Note: Both the reference and comparison LiData versions must be 2.0 or higher. Lower versions of LiData can be converted to a higher version using Convert LiData to LiData.
>
> Note: Attribute values can be displayed using the right-click menu based on additional attributes.

Section Analysis

Functional Overview

Section Analysis aims at road section and tunnel section, mainly solving problems such as design of reference section line based on point cloud and model data, extraction of real section, analysis of section over-cut and undercut, generation of report and so on. This function is based on the measured data to obtain the measured section of each position or the real topographic relief situation; The measurement section can be edited and managed; Finally output the information of each section and generate over-cut and undercut report It is recommended to use ground points for section generation to obtain accurate section data.

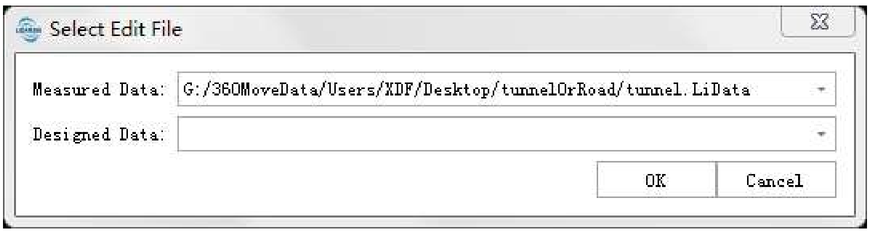
Usage

Load the point cloud or model data into the current activation window, click Terrain > Section Analysis, the section analysis tool bar will appear above the data display window.

From left to right: Editor, create guide lines, import guide lines, extract section point clouds, generate orthogonal guide line, save reference guide line, import design section, generate road section, generate tunnel section, hide\display vector tool, clear tool, and exit tool.

Click*Editor > Start edit*

Select the point cloud or model data that the window has loaded to start the section analysis. Users need to choose between measured data and design data. The measured data is used to generate the real section (red by default), while the design data is used to generate the design section. The measured data is required and the design data is optional. Only when the design data is selected can the over-cut analysis be carried out after the real section is generated.



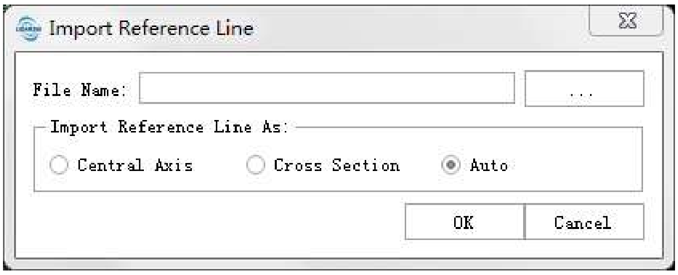
Draw the center line/longitudinal section

Functional Description: Use the mouse to pick up several points to create a reference section. You can draw multiple reference sections.

Import the reference section

Functional Description: Click this button to open the import guide interface, supporting text files (*.txt), graphics files (*.dxf), vector files (*.shp)

(*.txt), graphics files (*.dxf), vector files (*.shp)

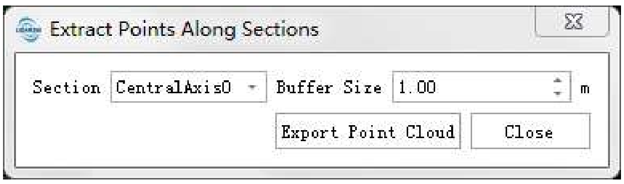


Import result format refer tothe section result file format.. When Import dxf, it will automatically match dxf's text as a reference line's name, the matching way is traverse the reference lines of the file,compare all the texts' positions with the beginning and end points of the reference line. Find the text closest to the starting or ending points as the name of this guide line. This text will not be used by other guides. Guides that do not match the text will be automatically named by the software according to the current naming mode. Refer to the "Settings" function in "Generate Road/Tunnel Sections" for the naming mode. Guides can be imported in the following types:

- **Central Axis**: All guides will be imported as centerlines.
- **Cross section**: Guides with only two points are imported as cross sections.
- **Auto**: Guides larger than two points will be imported as centerlines; Guides with only two points are imported as cross-sections, and the intersection with each centerline is automatically determined to obtain the appropriate name and mileage.

Extract the point cloud along the reference section line

Functional Description: Click this button to extract the point cloud data file around the reference center line. If the model file is loaded, all the vertexes of the triangulation gird within the model scope are extracted.

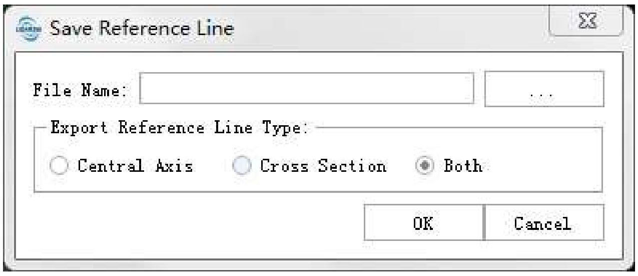


Parameters

- **Section**Select the reference center line from which the point cloud is to be extracted.
- **Buffer Size (m) (default: "1.00")**: The buffer distance perpendicular to the center line/vertical section. For 100 m, each end of the center line is buffered 50 m.

Save Reference Section

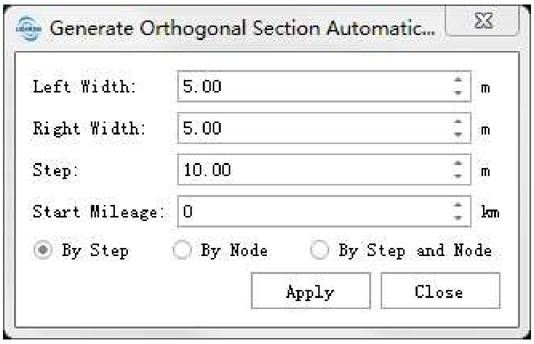
Functional Description: Click this button, the Save Guide dialog box will appear, you can save the created reference line data as txt, shp or dxf format file.



- **Center Axis**: Only the center line and its name are exported.
- **Cross section**: Only the cross section and its name are exported.
- **Both**: Export all guides and their names.

Generate orthogonal section automatically

Functional Description: This function is used to generate several cross sections perpendicular to the reference center line.



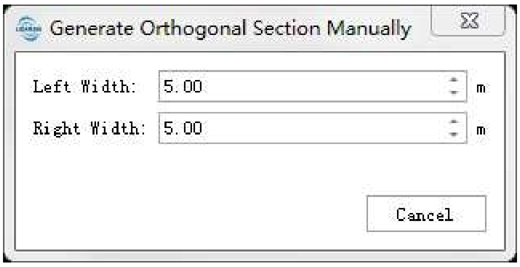
Parameters

- **Left Width (m) (default: 5)**: The length of the left side of the reference cross section along the line.
- **Right Width (m) (default: "5")**: The length of the right end of the reference cross section along the line.
- **Step (m) (default: "10")**: The step size of the reference cross section generated along the center line direction.
- **Starting Mileage (km) (default: "0")**Generates the starting mileage of the reference cross section, and the mileage value of the cross section is accumulated on this basis.
- **By Step**: Generate a reference cross section only by a specific step, regardless of nodes.
- **By Node**: Generate a reference cross section only by a specific node, regardless of divisions.
- **By Step and Node**: Generate a reference cross section by a specific step and process the nodes simultaneously.
- **By Slope**: When there is a height difference between the nodes of the longitudinal profile/centerline, the cross-section position can be calculated using the slope method. The slope is defined as the ratio of the horizontal distance and height difference between two points on a line segment, usually expressed as a percentage. In this mode, the larger the slope, the more cross-sections obtained.

- Slope (%) (default "5"): The program automatically calculates the height difference for each section of centerline, then uses input slope value to calculate horizontal step length and finally places cross-sections at these intervals.
- **Maximum Mileage Step Length (m) (default "10")**: Horizontal step length threshold used in conjunction with slope threshold. This threshold ensures that at every certain distance in gentle slopes there will be one cross-section when setting small values for slope thresholds which may result in too few cross sections.

Generate orthogonal section manually

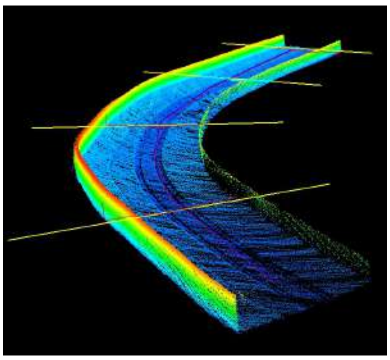
Functional Description: Click this button to enable the function of manually generating reference cross section. You can add a cross section at any position on the longitudinal section with the mouse.



Parameters

- **Left Width (m) (default: 5)**: The length of the left side of the reference cross section along the line.
- **Right Width (m) (default: "5")**: The length of the right end of the reference cross section along the line.

The cross-section shown in the figure.



Import design section

Functional Description: Click this button to import the design section into the software. Support text file (*.txt), read the section line will all be displayed as the design section line. For details about the section format, see "Import Reference Section”

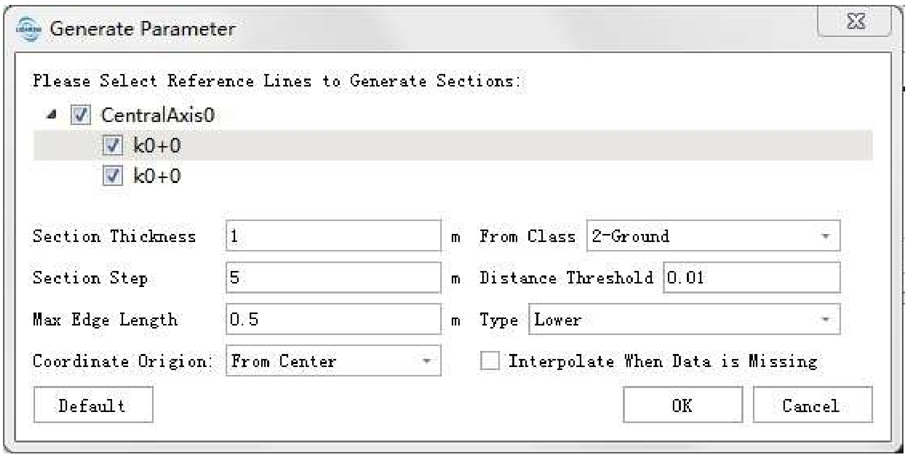
Generate road/tunnel sections

Cross section drawing refers to the section drawing perpendicular to the middle line of the road at the center of the pile. In traditional surveying and mapping, the section drawing takes the horizontal distance as the horizontal coordinate and the elevation as the vertical coordinate and is drawn on the millimeter square paper. The scale is generally 1: 100 or 1: 200. The aspect scale must be consistent. Earthwork quantity calculation and construction lofting, are based on this. For road sections, the measured section can be generated using points of any class (the default is ground points), and you can adjust **the distance threshold** and **the maximum side length** to control the refinement of the section

Functional Description: Generate road sectional view based on reference section line.

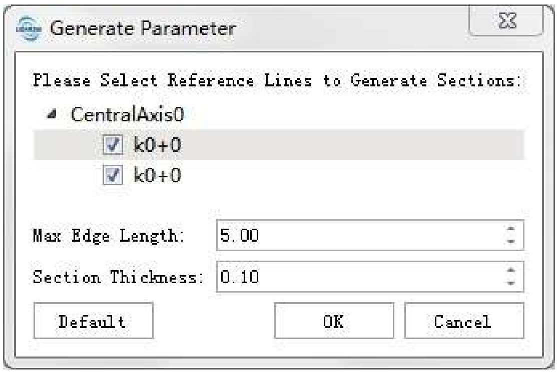
The right side of the section interface is the section manager. Double-click on the manager tree node to jump to the corresponding section. The attributes of this section will also be displayed in the attributes bar below. Click "Calculate" button to pop up the parameters settings dialog box of calculate section. Different parameters are used for road section and tunnel section.

Parameters



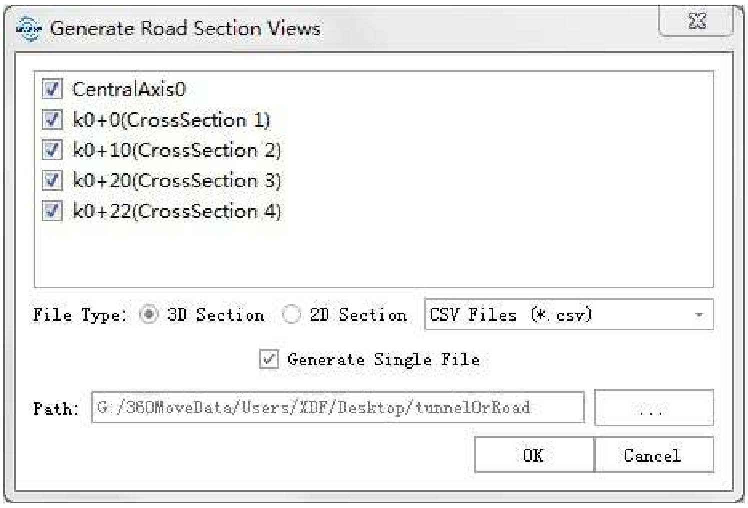
- **Road section parameters**:
  - **Select Section**: The user can select one or more guides from the list. By default, all guides are selected. Guides in the list have been grouped by the center line they belong to. For road sections, the centerline can generate a measured section. The measured cross section generated by the selected guides will override the old results.
  - **Section Thickness (m) (default: "1")**: The extent of the section point clouds extracted along the centerline/longitudinal section direction will be used to calculate the true section. When the point cloud is sparse, the value can be appropriately increased in order to better preserve the details of the measured section. This best value is set to 3 to 5 times the average point spacing.
  - **From Class**: Indicates the type of point cloud that participates in the generation of a cross-section.
  - **Section step (m) (default: "5")**: The section can be segmented based on the set step size **according to the distance threshold**, the section will use linear interpolation method to interpolate points at integer multiples of section step size. If this value is set to 0, you should simplify the whole section. The larger the value, the less the true section points are retained.

- **Distance Threshold (default: "0.01")**: The profile will be condensed using the Douglas algorithm according to this parameter. The smaller the value, the higher the refinement degree of the cross-section; otherwise, the higher the simplification degree. After simplification, it is possible to appear point clouds that exist below the sectional line.
- **Type**: Take the section data according to the elevation type to generate the section view.
  - **Lower (default)**: Take the lowest point of the section data to generate the section view.
    - **Upper**: Take the highest point of the section data to generate the section view.
- **Max Edge Length (m) (default: "0.5")**: Threshold of maximum line segment side length. If the distance between two points in the cross section is greater than this value, it will be broken into two sections.
- **Coordinate Origion**: Starting position of section line in x direction.
  - **From Center**: The x-direction starting position of the section line is the intersection point with the central axis.
  - **From the Beginning**: The x-direction starting position of the section line is the left end along the central axis.



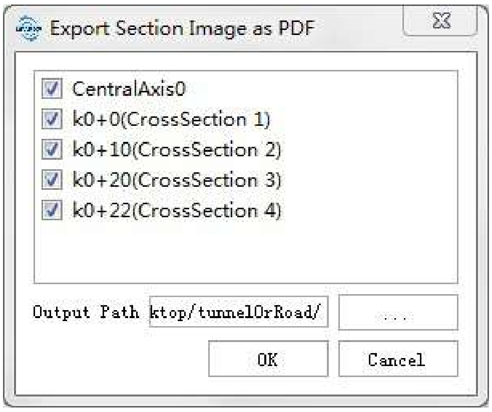
Functional Description: Generate tunnel section view according to section line.

- **Tunnel section parameters**:
  - **Select Guides**: The user can select one or more guides from the list. By default, all guides are selected. Guides in the list have been grouped by the center line they belong to. For the tunnel section, the center line does not generate the measured section. The measured cross section generated by the selected guides will override the old results.
  - **Maximum Side length (m) (default: "5")**: Threshold for maximum line segment side length. If the distance between two points in the cross-section is greater than this value, it will break into two segments. The larger the value, the less the true section points are retained.
  - **Section Thickness (m) (default: "0.1")**: Generate the sectional view using point cloud data along the section line width.



Click the calculate button to bring up the parameters settings dialog box. Click OK to generate the tunnel cross-section. The mouse can be used to translate the canvas, zoom and other operations.

- **Export Section**: Export the generated section lines and save them to one or multiple files. We support csv, HDM, HDM (Southern), shp, and dxf formats in Chinese, but hdm formats are not supported in English. Export result format refer tothe section result file format..
- **Export Image**: Save the current dialog cross-section as a image, can save the format as pdf, the user can set the resolution and size of the exported cross-section in the export interface.

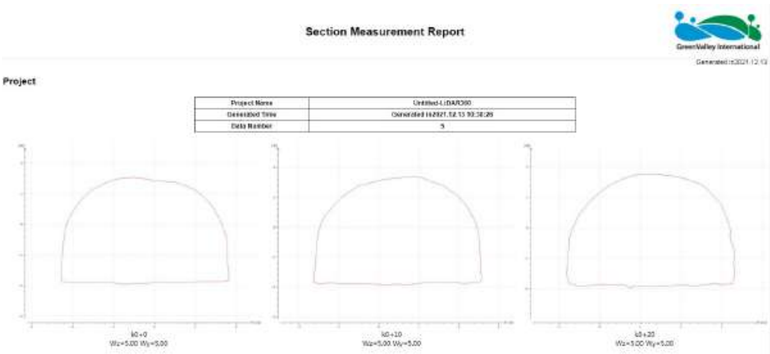


- **Edit section line node**: Edit the node of the tunnel section line. After editing is enabled, you can drag the node. It is used to edit incorrect tunnel boundaries or tunnel boundaries with noisy point.
- **Section comparison**: The generated tunnel reference section is compared with the tunnel design section, and the area of the filled excavation and the total amount of filled excavation (volume) are calculated. Fill area and dig area are shown in different colors (color can be set).
  - The method of calculating the total amount of fill excavation is as follows: The half distance of the previous mileage pile and the second mileage pile to this mileage pile multiply by the area of fill excavation of this mileage pile, and only the second half and the first half of the distance of starting mileage pile and ending mileage pile are calculated respectively. For example, if the mileage pile is K0+80, the excavated area is S, the previous mileage pile is K0+72, and the later mileage pile is K0+90, then the excavated amount of the mileage pile V = ((80-72)/2 + (90-80)/2) * S.The total filling and excavation volume can be obtained by summing up the volume of each mileage pile.

- **Export comparison report**: You can export the section and section information on the current canvas into a report, output to the specified folder.

- **Distance measurement**: Enabling the distance measurement, you can use the left mouse button to click on the canvas to pick up the required measurement points, it can also scale and shift the canvas at the same time. Double-click the left mouse button to complete the measurement, click the "Distance Measurement" button again to end the measurement.
- **Setting**: The Setting dialog box is displayed.
  - **Display**: You can set the color of the point cloud, the color of the section node, and the color of the filling and digging side.
  - **Section Name**: You can name the sections according to a specific naming pattern. It can be divided into prefix and suffix. The prefix can be entered arbitrarily. The suffix is mileage. They are mileage (km+m) and mileage respectively.
- **Show Section Attributes**: You can show/hide the current section attributes.

The measuring section report is shown in the figure below.



Show/hide Sections

 **Functional Description**: Click this button and choose to show or hide vectors according to your needs, including**drawn lines, orthogonal lines, sectional lines and all vectors**.

Remove Sections

 **Description**: Click this button and select whether to clear the vector as required, including**clear drawn lines, clear orthogonal guides, clear sectional lines and clear all**.

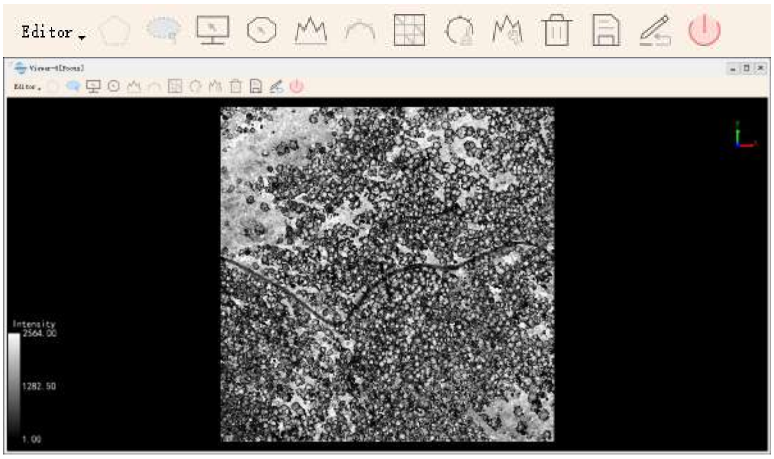
Exit

 **Functional Description**: Exit section analysis.

LiModel Editor

Functional Overview

Provides editing operations forLiModel. The user can select area of interest using Polygon Selection, Lasso Selection, Screen Selection, or shp Selection, which can be edited by various operations such as Flatten Height, Smooth Height, Repair No Data, Repair Height by Variance, Repair Height and Delete Height.

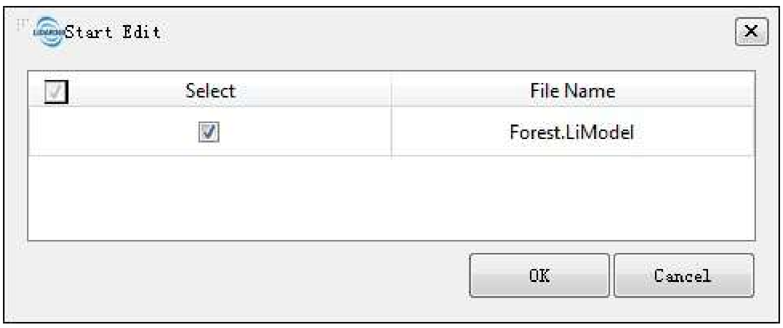


Usage

Click *Terrain* > *LiModel Editor*, The current activation window will show the LiModel Editor toolbar.

Data Selection

Click Start Edit in the drop-down menu for editing. You can select one or more data which has been opened in LiDAR360 for editing.



Polygonal Selection

Left-click to add the polygon vertex. double- click to finish the selection. Right-click to Back One Point or Clear Selection before finish. Unlike Polygon Selection tool, you cannot select an area that exceeds the data range.

Lasso Selection

Left-click to start the Lasso Selection, move to add one point, and double-click to finish the selection. Right-click to Back One Point or Clear Selection before finish. Unlike Lasso Selection tool, you cannot select an area that exceeds the data range.

Screen Selection

Left-click to add the polygon vertex. double- click to finish the selection. Right-click to Back One Point or Clear Selection before finish. You can select the area that exceeds the data range.

Breakline of Water Area (River、Lakes)

Draw closed areas interactively with the mouse, set parameters, and click the Add button to complete the addition. - **Elevation of Selected Points**: Project the selected area onto the same elevation plane. Users can manually input or activate the selection button to pick points in the scene for elevation settings. The default is the elevation of the last drawn point. - **Minimum Elevation**: Set the elevation of pixels within the range to the elevation corresponding to the lowest point. ### Linear Breakline ###

Draw a polyline interactively with the mouse, set parameters, and click the Add button to complete the addition.

- **Change Elevation**: Default is unchecked. Whether to change the elevation of the points along the line.
    - **Starting Elevation**: Define the starting elevation value. Users can manually input or activate the selection button to pick points in the scene for elevation settings.
    - **Ending Elevation**: Define the ending elevation value. Users can manually input or activate the selection button to pick points in the scene for elevation settings.

Breaklines of Gradual River

Draw two riverbanks interactively with the mouse, set parameters, and click the Add button to complete the addition.

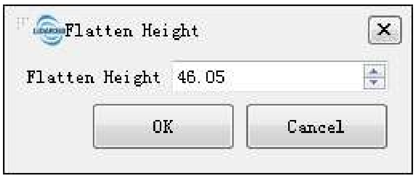
- **Starting Elevation**: Define the starting elevation value of the river. The default is the elevation of the starting point of the second riverbank. Users can manually input or activate the selection button to pick points in the scene for elevation settings.
- **Ending Elevation**: Define the ending elevation value of the river. The default is the elevation of the endpoint of the second riverbank. Users can manually input or activate the selection button to pick points in the scene for elevation settings.

Shp Selection

For a specific range, the existing 2D SHP format vector file can be imported, and the polygon is identified by the software as the editing area.

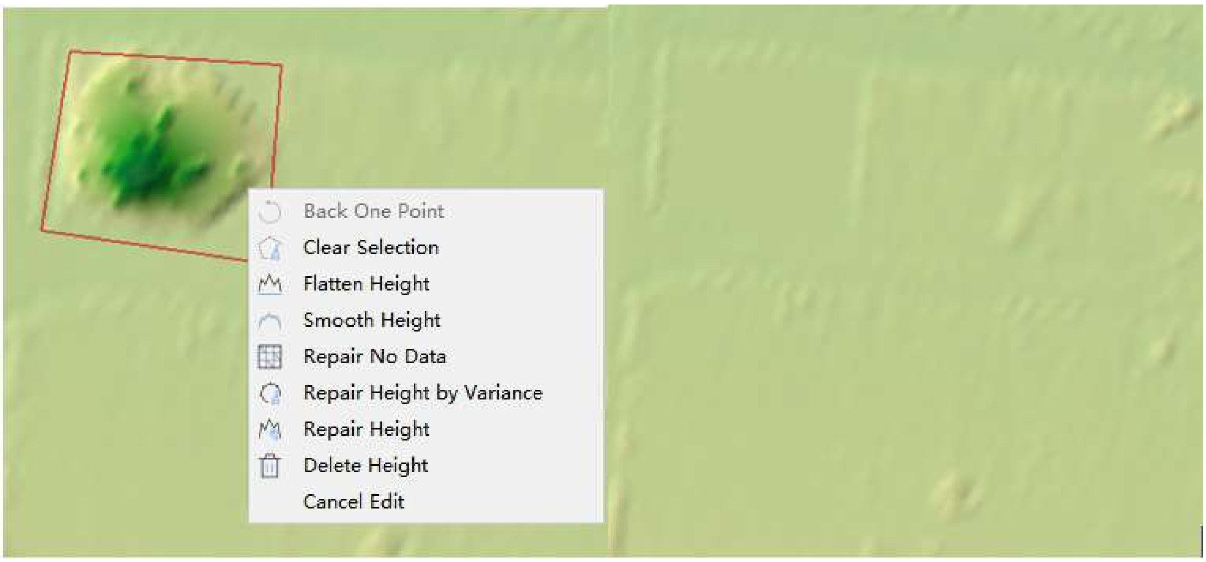
Flatten Height

Click this button, or right-click to select Flatten Height. You can set elevations for selected area range models to specified elevation values for rivers or waters that have the same altitude.



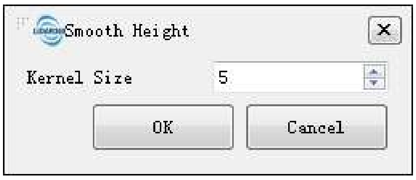
- **Flatten Height(meter)**: Set the elevation value. The default value is the average of the elevation values of the vertices of the selected region.

The schematic diagram is as follows, the left image is the original data, and the right image is the edited version.



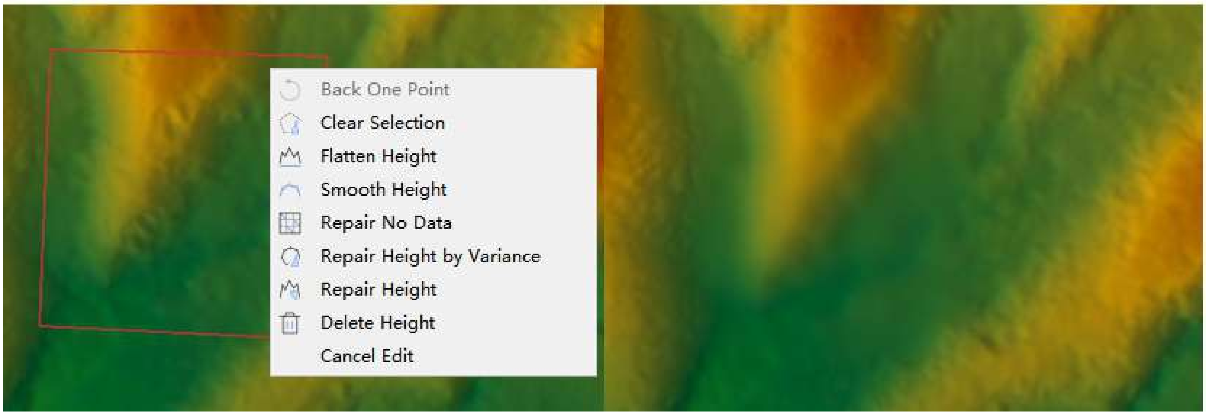
Smooth Height

Click this button, or right-click to select Smooth Height, which can smooth the selected area range model. Using image mean smoothing to process the model in the selected region point by point, it is suitable for smoothing DEM to generate smooth contour lines.



- **Kernel Size (default value is "5")**: The size of the mean filter kernel, it can only be odd.

The schematic diagram is shown below. The left figure shows the original data, and the right figure shows the effect after editing.



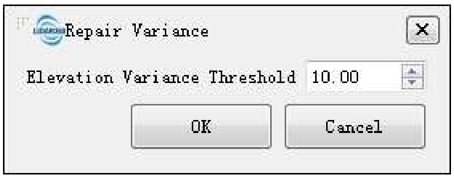
Repair No Data

Click this button, or right-click to select Repair No Data. Use bilinear interpolation to calculate the elevation value of NO Data values in the area.

The schematic diagram is shown below. The left figure shows the original data, and the right figure shows the effect after editing.

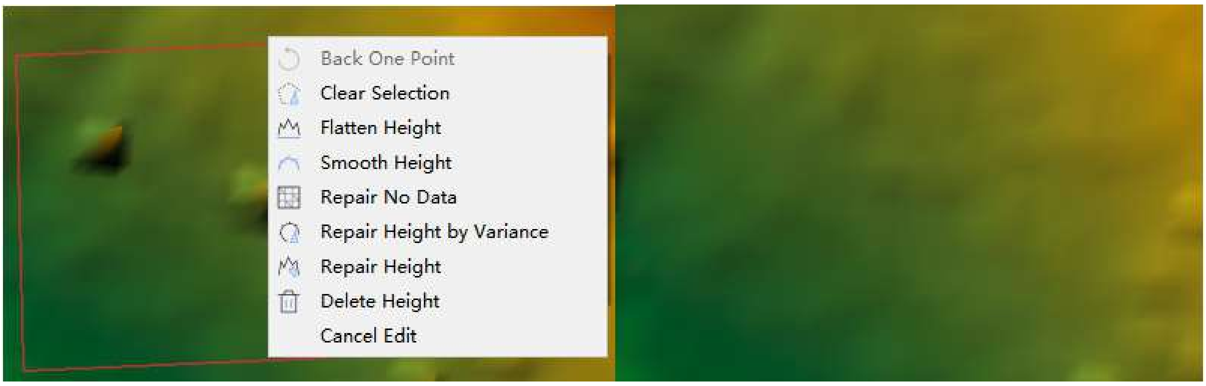
Repair Height by Variance

Click this button, or right-click to select Repair Height by Variance. It is used to repair the noise caused by noise, according to the variance of its elevation value with the neighborhood point to determine whether it is a noise. Bilinear interpolation is used to calculate the elevation of the noise point.



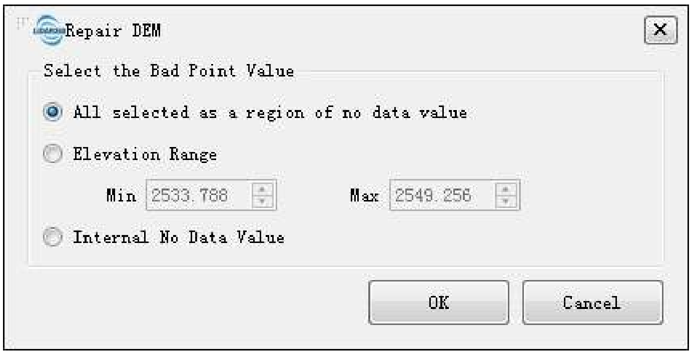
- **Elevation Variance Threshold(the default value is 10)**: The elevation variance threshold is used to determine whether it is a noise.

The schematic diagram is shown below. The left figure shows the original data, and the right figure shows the effect after editing.



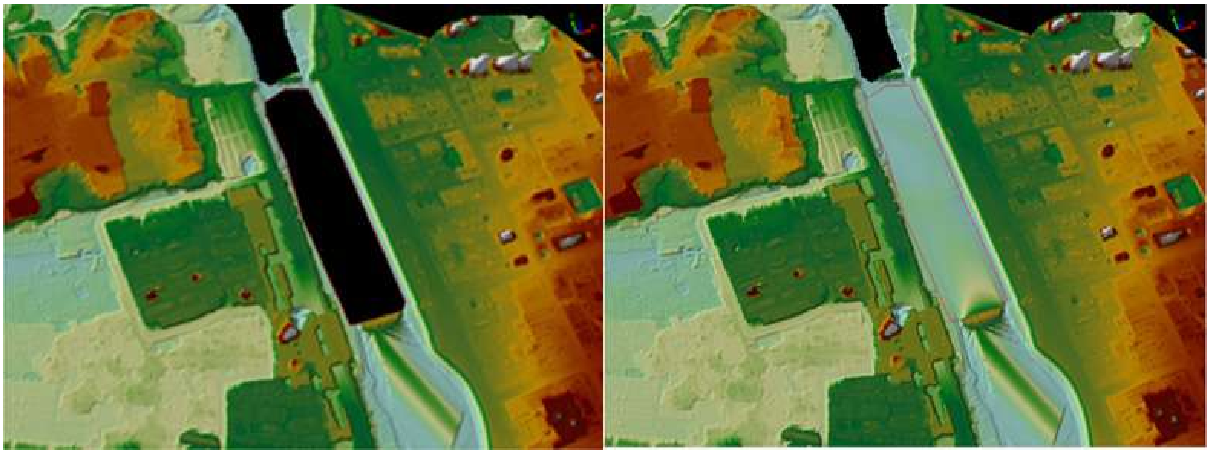
Repair Height

Click this button, or right-click to select Repair Height. It is used to repair the elevation in the specified range. The bilinear interpolation method is used to calculate the elevation of the parameter conditions.



- **Grid point type**: This parameter defines the range of grid point to be processed and may constrain the processing scope.
  - **All grids(default)**: All selected areas will be repaired.
  - **Elevation Range**: Only grids with elevation values within the minimum and maximum values are repaired. The default values for the maximum and minimum values are the maximum and minimum elevation values for the vertices of the selected area.
  - **Internal No Data Value**: Only the non-valued areas are repaired.

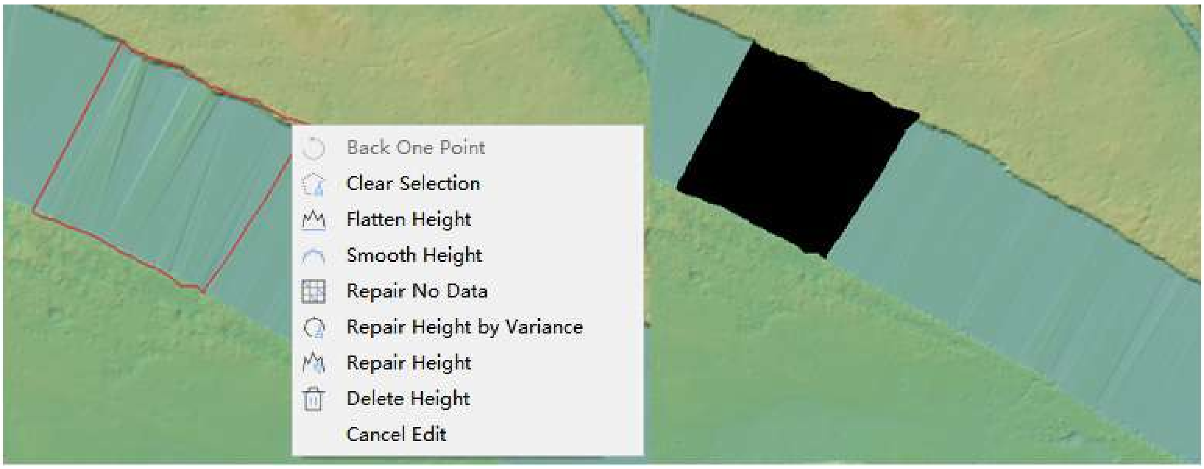
The schematic diagram is shown below. The left figure shows the original data, and the right figure shows the effect after editing.



Delete Height

Click this button, or right-click to select Delete Height. Delete the selected point elevation point.

The schematic diagram is shown below. The left figure shows the original data, and the right figure shows the effect after editing.



Save DEM

Save the edited result to LiModel file.

Cancel DEM

Cancel editing, discard edit result, reload original file to display.

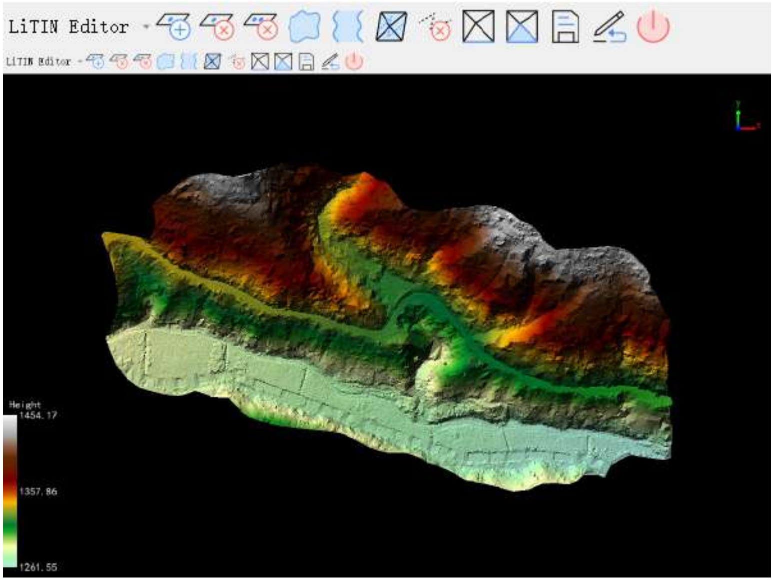
Exit

Click the exit button in the toolbar to exit LiModel editor. If the editing result is not saved, and click the Exit button or click the Edit button to finish the editing of the menu, the software will pop up the Exit button or click the Edit button to finish the editing of the menu, the software will pop up the prompt window as shown below. Click Yes to save the editing result and exit. Click No to exit without saving the editing result. Click Cancel returns.

LiTIN Editor

Description

Provides irregular grid model LiTin for editing operations,provides adding single point, deleting single point, deleting multiple points, adding single fracture line, adding closed area (lake) Break-lines, Add Double Region (River) breaklines, Delete breaklines, Select Edges, Select Triangles.



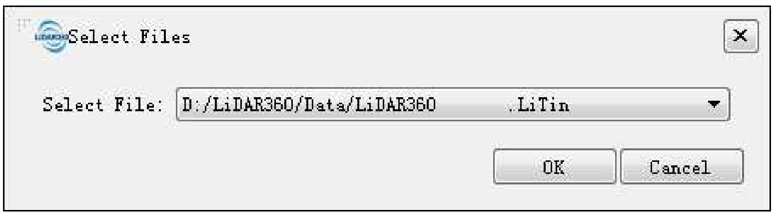
> Note: Using the model display tool as shown in the figure below, you can display LiTin by model, triangle or point respectively (you can also use the letter key W on the keyboard to switch between different display modes). For easy viewing of editing effects, it is recommended to use the disclosure triangle to display LiTin.
>
> 

Usage

Click *Terrain* > *LiTin Editor*, the LiTin Editing bar will appear in the current activation window.

Select Edit File

Click Start Editing in the Edit drop-down menu and select the data to be edited.The data to be edited must have been opened in the LiDAR360 software, and only one data can be selected for editing at a time.



Add Single Point

Click the left mouse button to add a vertex and insert a single point into LiTin. The XY coordinates of the point are determined by the mouse click. The sources of elevation values include triangulation surface, maximum surface elevation, minimum surface elevation and user input. This function is suitable for local area editing, intervening in the contour trend of this area.

- **Elevation Source**: This parameter defines the elevation value of the added point.
  - **From Surface (default)**: The elevation of the insertion point is determined by triangular interpolation.
  - **Max Z From Surface**: The maximum value of the range of elevation values within this model.
  - **Min Z From Surface**: The minimum value of the range of elevation values within this model.
  - **As Specified**: User input custom elevation value.

Delete Single Point

Left mouse click to delete a vertex, delete a single point from the LiTin model, and reconstruct the triangulation network with adjacent points. This function is suitable for local area editing, removing noise points or misclassified points, and intervening the contour trend of this region.

Delete Multi Points

Select an area interactively with the mouse, and the triangulation network points in the area will be deleted in batches, causing local triangulation network reconstruction. It is suitable for local area editing. It can delete noise points or misclassified points in batches according to the polygon range, and intervening the contour trend of this region.

Breakline of Water Area

Draw a closed area through mouse interaction, set parameters, and click the Add button to complete the addition.

- **Set Elevation**: Project the break line point to the same elevation plane, users can manually input or activate the selection button to pick up the points in the scene for elevation setting. The default is the elevation of the last drawn point.
- **Minimum Elevation**: Project the break line point to the elevation plane corresponding to the lowest point in the area.

Breakline of Gradual River

Draw two riverbank lines interactively with the mouse (they need to be drawn in the same direction), set the parameters, and click the add button to complete the addition.

- **Start Elevation Value**: This parameter defines the starting elevation value of the river. The default is the elevation of the starting point of the second river bank, and the user can manually input or activate the selection button to select the midpoint of the scene to set the elevation.
- **End Elevation Value**: This parameter defines the end elevation value of the river. The default is the elevation of the end point of the second river bank. Users can manually input or activate the selection button to select the midpoint of the scene to set the elevation.

Liner Breakline

Draw a PL interactively with the mouse, set the parameters and click the add button to complete the addition.

- **Breakline Type**: This parameter defines the break line type.
  - **Soft Breakline (default)***: Soft break line.
  - **Hard Breakline***: Hard break line.
  - **Change Elevation**: Unchecked by default, available under hard fracture, whether to change the elevation of the fracture line point.
  - **Start Elevation Value***: This parameter defines the starting elevation value of the hard fracture. Users can manually enter or activate the selection button to select the midpoint of the scene for elevation setting.
  - **End Elevation Value***: This parameter defines the end elevation value of the hard fracture. Users can manually enter or activate the selection button to select the midpoint of the scene for elevation setting.

Delete breakline

Delete the selected break line by mouse interaction.

Select Edges

Select the sides of the triangle interactively with the mouse. The selected sides are displayed in red.

Select Triangle

Select the triangle interactively with the mouse, the selected triangle is displayed in red.

Save TIN Edit

Save editing results to LiTin file.

Cancel TIN Edit

Cancel editing, discard edit result, reload original file to display.

Undo

Undo (Shortcut: **Ctrl+Z**), reverts the last modification. The number of undos is limited to 20 times.

Redo

Redo (Shortcut: **Ctrl+Y**), re-applies the last undone modification. The number of redos is limited to 20 times.

Exit

Click the Exit button on the tool bar to exit LiTin editing. If the editing results are not saved, click the Exit button or click the Edit button to pop up the menu to end editing, the software will pop up the prompt window as shown in the figure below, click Yes to save the editing results and exit, click No to exit directly without saving the editing results, click Cancel returns.

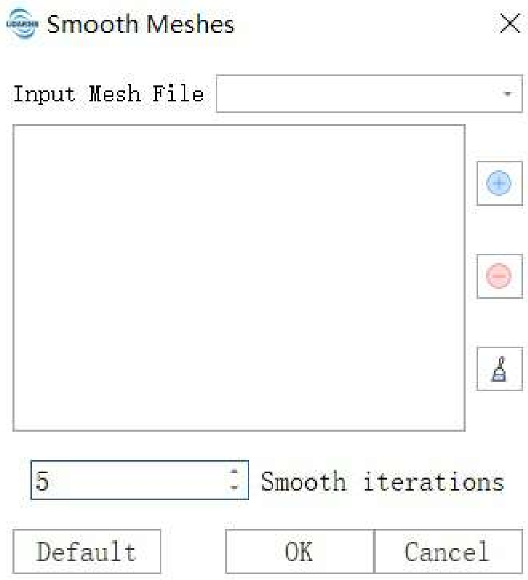
Smooth Meshes

Functional Overview

Smooth the triangulation model

Usage

Click*Terrain* > *Smooth Meshes*, the Parameters Settings interface pops up, select the data setting parameter to execute.



Parameters Settings

- **Input data**: Select from the current project directory or select the model data file to smooth from the folder.
- **Smooth Iterations**: The number of smoothing iterations to be performed, the larger the model, the smoother the model and the longer it takes.
- **Default Value**: Click this tool option to restore all default parameters.

Contour Editing

The contour line editing functionality provides capabilities for generating, trimming, merging, filtering, and detecting contour lines. Based on vector editing, the features have been adjusted to align with the characteristics of contour line tasks, with additional tools to support various business operations.

Open the *Contour Line Editing* toolbar, which includes the following functional modules:

- Editing
- Utility Tools
- Draw Feature
- Contour Process Tools
- Select Entity
- Snap Entity
- Layer Management

Editor

Control the initiation and termination of contour line editing. Contour lines are essentially vector objects; during editing, the system will not recognize whether entities in the file are contour lines, but operations will only produce the expected results when applied to contour line entities. This module supports editing only one contour vector object at a time and includes functions for selecting files for editing, creating new files, importing files, saving, and ending editing.

Start Editor

Click Edit -> Start Editor

- If there are vector object data in the editing window, a dialog box will appear to select the vector object data. The vector object data already loaded into the editing window will appear in the selection list. Choose the data to be edited.
- If there are no vector object data in the editing window, a dialog box will appear to select vector object data from disk for editing.

Once editing is initiated, other functions on the vector editing toolbar will be activated.

During editing, the vector data being edited cannot be removed from the editing window.

New File Editor

Click Edit -> Create New File for Editing

- A file selection dialog will appear. Generate the corresponding vector data file based on the user-selected path and load it into the editing window to begin editing that vector data.

Once editing is initiated, other functions on the contour line editing toolbar will be activated.

During editing, the vector data being edited cannot be removed from the editing window.

Import File

Once editing is started, you can *click Edit -> Import File*

- A file selection dialog will appear. Load the user-selected vector data file into the currently edited vector object.

Save

Once editing is started, you can *click Edit -> Save* to save the current edited data to the original file.

End Edit

Click Edit -> End Editing or click the End button 

- Ending the editing session will prompt a dialog to save the editing results. Choose as needed and exit the contour line editing module.

> If editing is not started, you can also directly click the End button to exit the editing module.

Utility Tools

Basic functions include Undo, Redo, Delete Entities, and Attribute Query.

Undo

Undo the last operation.

Redo

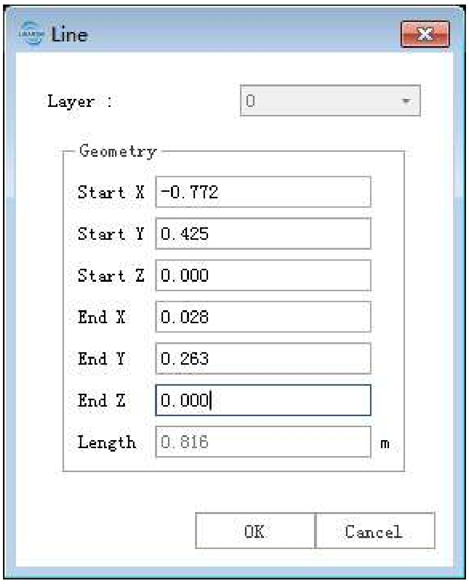
Redo the last undone operation.

Delete Selected

Delete multiple selected entities within the scene.

Parameter Dialog

Query the geometric properties of an entity. Left-click to select an entity, and a dialog will pop up displaying the geometric properties of the entity. The image below shows the geometric properties of a line.



Draw Feature

The editing tools are divided into three sections: Editing Contours, Editing Vectors, and Drawing Features.

Draw Feature

Surface Feature

Interact with the mouse to break contours in surface features.

1. Click the left mouse button to select a polygon or closed polyline.
2. Continue clicking the left mouse button to select additional polygons or closed polylines, or right-click to trim surface features.

Bilinear Feature

Interact with the mouse to break contours in linear features.

1. Click the left mouse button to select the first line.
2. Click the left mouse button to select the second line to trim linear features.

Join Contour

Interact with the mouse to connect contours within a selected area.

1. Click the left mouse button to select the first point of the break area, continue clicking until the area is selected.
2. During the selection, you can right-click to undo a point, and double-click to end the box selection and start connecting contours within the area.
3. Repeat step 1 to connect the next break area.

Parameters

- **Straight Connection**: Connect endpoints of contours with the same elevation directly within the area.
- **Curve Connection**: Smooth the connected contours within the area using Bezier curves based on the direct connection.
  - **Sampling Interval**: Specify the average distance between adjacent points after smoothing.

Join Two Contour

Interact with the mouse to connect selected contour lines.

1. Click the left mouse button to select the first contour line.
2. Continue to select the second contour line. If the elevations are the same, connect the two closest endpoints of the contour lines.

Editing Vectors

Basic vector editing tools needed for contour editing.

Break at Point

 Interact with the mouse to break a polygon at a point.

1. Click the left mouse button to select the polyline, B-spline, or Bezier curve to edit, or right-click to exit editing.
2. Click the left mouse button to select an intersection point on the polyline, or right-click to end the current edit and start a new one.
3. Right-click to exit break at point.

Break in Line

 Interact with the mouse to break a polygon at a line.

1. Click the left mouse button to select the polyline, B-spline, or Bezier curve to edit, or right-click to exit editing.
2. Click the left mouse button to select the first intersection point on the polyline, or right-click to end the current edit and start a new one.
3. Click the left mouse button to select the second intersection point to complete the break, or right-click to undo and reselect the first intersection point.
4. Right-click to exit break at line.

Trim

 Interact with the mouse to trim polygons.

1. Click the left mouse button to select a polyline, rectangle, or closed polyline as a baseline, or right-click to exit editing.
2. After selecting a baseline, click the left mouse button to continue selecting baselines, or right-click to prepare to select vectors to be trimmed.
3. In the trimming selection mode, click the left mouse button to select the vector to be trimmed, which will delete the vector based on the intersection with the baseline.
4. Right-click to revert to reselecting the baseline, and right-click again to exit trimming.

> Note: Trimming is based on intersection points in the xy plane, regardless of the elevation of the vector.

Join Polylines

Interact with the mouse to merge polylines.

1. Click the left mouse button to select the starting or ending point of the first vector line, and right-click to exit.
2. Click the left mouse button to select the starting or ending point of the second vector line to complete the merge.

Merge Polylines

Interact with the mouse to combine multiple polylines.

1. Click the left mouse button to select multiple non-closed vector lines (polylines, B-splines, or Bezier curves), then use the combine function to merge the selected vectors.
2. After activating the combine function, click the left mouse button to select the vectors to be combined, and right-click to merge the selected vectors.
3. Click the left mouse button to select one vector, then right-click to prepare to select another vector. Click the left mouse button to select the second vector and complete the merge.
4. Right-click to exit combining.

Intersect Polylines

Interact with the mouse to intersect polylines.

1. Click the left mouse button to select the first vector line to intersect, or right-click to exit editing.
2. Click the left mouse button to select the second vector line to intersect, or right-click to revert to step 1.
3. Choose the intersection points to complete the intersect operation.

Add Node

Interact with the mouse to add nodes to polygons.

1. Click the left mouse button to select the polyline or polygon entity to edit, or right-click to exit editing.
2. Click the left mouse button to add a node at the selected position on the polyline or polygon entity, or right-click to end the current edit and start a new one.
3. Right-click to exit adding nodes.

Append Node

Interact with the mouse to append nodes to the start or end of polylines, B-splines, or Bezier

Interact with the mouse to append nodes to the start or end of polylines, B-splines, or Bezier curves.

1. Click the left mouse button to select the polyline entity to edit, or right-click to exit editing.
2. Click the left mouse button to select a point to determine the insertion position based on distance. If closer to the start, add the point before the start; otherwise, add it after the end. Right-click to end the polyline editing and start a new one.
3. In the property panel on the right, select the type of segment to append (polyline, B-spline, or Bezier curve), defaulting to polyline. Click to switch between the three types.
4. Right-click to exit appending nodes.

Here is the translation of the provided text into English:

Editing Tools

Delete Node

Edit polygons to delete nodes through mouse interaction.

1. Left-click to select the polyline or polygon entity you want to edit, or right-click to exit editing.
2. Left-click on the polygon entity's node to delete it, or right-click to end the current edit and start a new one.
3. Right-click to exit node deletion.

Smooth and Simplify

Perform smoothing or simplification operations on polylines.

1. Click the line smoothing and simplification button to open the parameter settings window.
2. Select the vector line object and execute the operation.

Parameter Settings:

- **Smooth**: Whether to apply smoothing.
- **Filter Distance**: Effective when smoothing is enabled; specifies the average distance between adjacent points after smoothing.
- **Simplify**: Whether to apply simplification.
- **Filter Distance**: Specifies the maximum deviation distance from the line to retain points. As the distance decreases, more points are retained and fewer are removed from the line.

Point

Draw points to mark point features using mouse left-click. The steps are as follows:

1. Left-click to select the location for the point entity.

2. Repeat step one to draw additional points.
3. Right-click to finish drawing point entities.

Two-Point Polyline

Draw line segments to mark linear features using mouse left-click to draw the starting point. The steps are as follows:

1. Left-click to select the starting point of the line segment, or right-click to finish drawing the line segment.
2. Left-click to select the ending point of the line segment, or right-click to revert to step one.
3. Right-click to finish drawing the line segment.

Draw Polyline

Draw polylines through mouse interaction to mark linear features.

1. Left-click to select points, or right-click to exit polyline drawing.
2. Repeat step one to add more points, or right-click to finish the current polyline drawing. To close the polyline, click the close button, and repeat step one to start a new polyline drawing.

Line Parallel

Draw multiple parallel polylines through mouse interaction to mark linear features.

1. Configure the parameters on the right to control the spacing and repetition quantity, as well as the repetition direction.
2. Left-click to select points, or right-click to exit polyline drawing.
3. Repeat step one to add points, or right-click to finish the current polyline drawing. Repeat step two to start a new polyline drawing, and adjust parameters during the drawing process.

Polygon

Draw polygons through mouse interaction to mark area features, with functions for adding and deleting nodes.

1. Left-click to select points, or right-click to exit polygon drawing.
2. Repeat step one to add more points, or right-click to finish the current polygon drawing. Repeat step one to start a new polygon drawing.
3. Right-click to exit polygon drawing.

Feel free to let me know if there’s anything else you need!

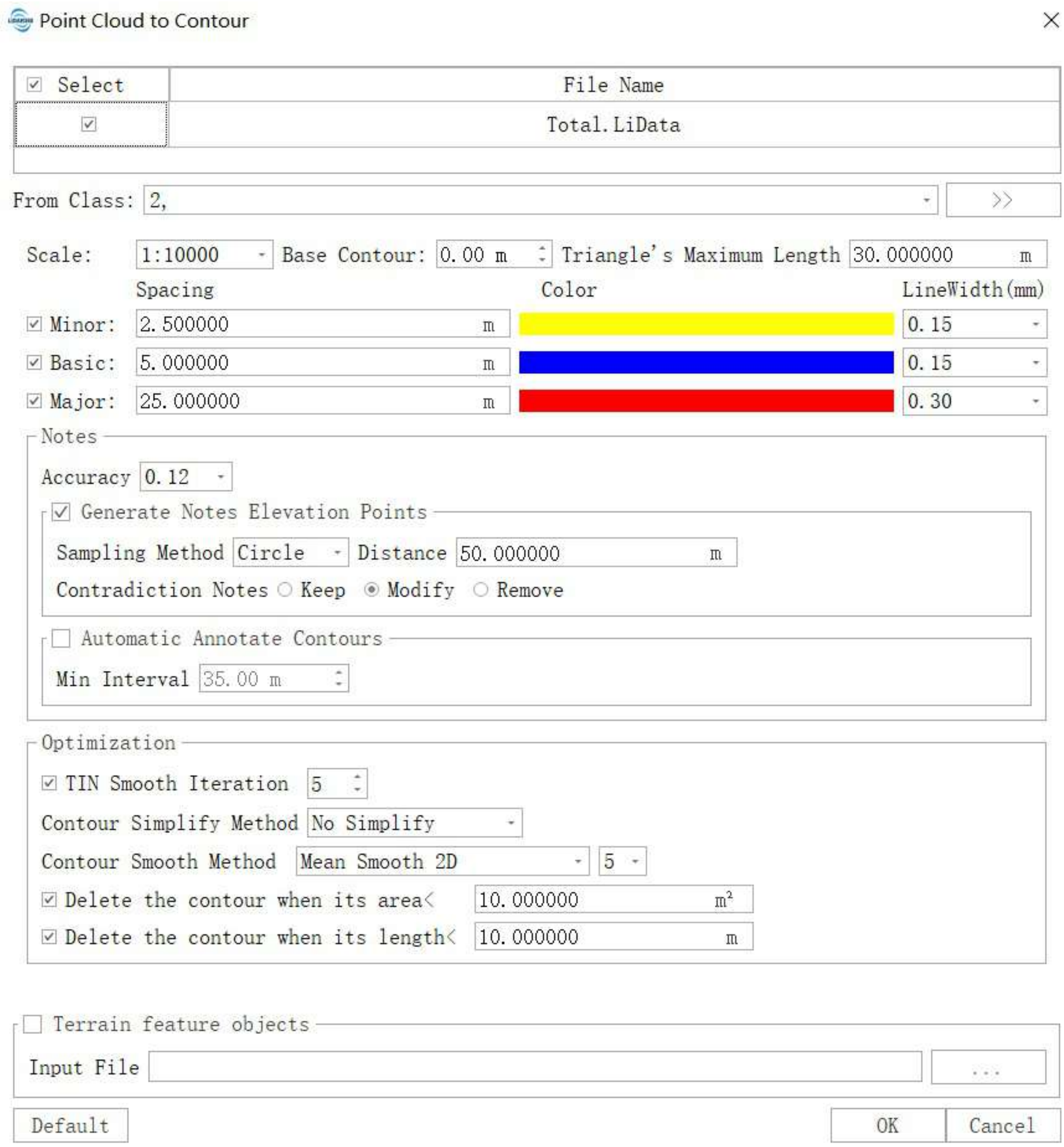
Contour Processing Tools

The contour processing module can generate contours, create elevation annotation points, detect contour conflicts, and filter by minimum area based on the point cloud data in the scene.

Point Cloud to Contour

Generate contour lines within a selected region of the point cloud data and add them to the current editing object.

1. Click the left mouse button to select a region, double-click to finish.
2. A parameter configuration interface will appear. Set the parameters as follows:



- **Select Data**: Ensure that each input point cloud data file has been classified. Input files can be either a single point cloud file or a dataset. The data must be opened in LiDAR360 software.

- **Scale**: The scale for contour generation from the point cloud, with 11 different scales corresponding to different contour intervals.
  - **1:500**: 1:500 scale.
  - **1:1000**: 1:1000 scale.
  - **1:2000**: 1:2000 scale.
  - **1:5000**: 1:5000 scale.
  - **1:10000 (default)**: 1:10000 scale.
  - ... ...
- **From Class**: The starting point cloud category used to construct the contours.
- **Reference Height (meters) (default "0")**: The height from which contours are calculated, i.e., the height difference relative to the reference that corresponds to integer multiples of the **contour interval**. For example, with a reference of 0 and a contour interval of 10, the contour heights would be: 0, -10, -20, -30..., 10, 20, 30...
- **Maximum Triangle Edge Length (meters) (default "30")**: In the triangular network constructed from ground points, triangles with edge lengths greater than this threshold will not participate in contour generation. This will result in discontinuities in the contours where point cloud gaps exceed this threshold. To avoid such gaps, set this threshold larger than the maximum size of point cloud gaps.
- **Intermediate Contours (default selected)**: Also known as half-distance contours. When the primary contours do not display certain local features, intermediate contours are drawn at half the contour interval. If not needed, uncheck this option.
  - **Contour Interval (meters) (default "2.5")**: The height difference between adjacent intermediate contours.
  - **Color (default yellow)**: The color of the intermediate contours, which can be modified.
  - **Line Width (default "1")**: The line width of the intermediate contours, which can be modified.
- **Primary Contours (default selected)**: Also known as basic contours. These are the main contours representing the topography, drawn according to the basic contour interval. If not needed, uncheck this option.
  - **Contour Interval (meters) (default "5")**: The height difference between adjacent primary contours.
  - **Color (default blue)**: The color of the primary contours, which can be modified.
  - **Line Width (default "2")**: The line width of the primary contours, which can be modified.
- **Index Contours (default selected)**: Also known as bold contours. To facilitate height reading, index contours are drawn every four primary contours. If not needed, uncheck this option.
  - **Contour Interval (meters) (default "25")**: The height difference between adjacent index contours.
  - **Color (default red)**: The color of the index contours, which can be modified.
  - **Line Width (default "3")**: The line width of the index contours, which can be modified.
- **Generate Elevation Annotation Points (default selected)**: Generate elevation points for terrain map output.
  - **Radius (meters) (default "15")**: Generates an elevation annotation point for a 15-meter radius area.
- **Optimization Options**: Optimization settings for smoothing the generated contours.
  - **Mean Smoothing (default selected)**: Smooths contours by replacing the current point with a weighted average of neighboring contour points.

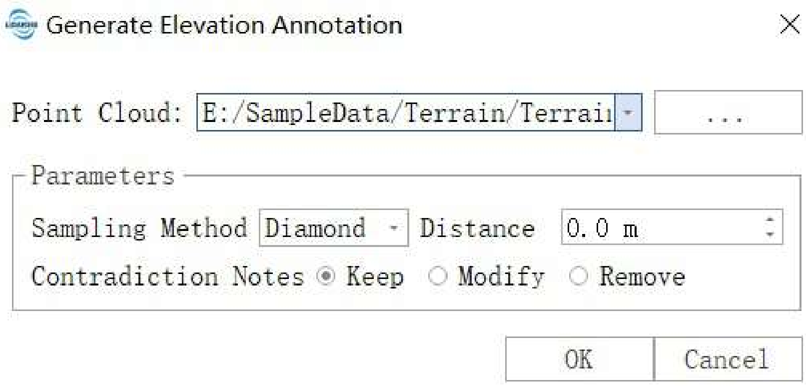
                - **3**: Number of neighboring contour points for smoothing.
                    - **5 (default)**: Number of neighboring contour points for smoothing.
                - **7**: Number of neighboring contour points for smoothing.
            - **Bezier Smoothing (default "155")**: A curve smoothing method with an angle threshold range of (0,180) degrees; a larger threshold results in smoother contours.
            - **Delete Contours When Area < (m²) (default "25")**: Delete closed contours with an area smaller than this value.
            - **Delete Contours When Length < (meters) (default "5")**: Delete open contours shorter than this value.
    - **Default Values**: Click this button to restore all parameters to default values.
3. After configuring the parameters, click OK to generate the contours and add them to the scene. To save to a file, click Save.

Generate Elevation Annotation Points

Generate elevation points for terrain map output based on the point cloud and contours.

1. Click the left mouse button to select a region, double-click to finish.

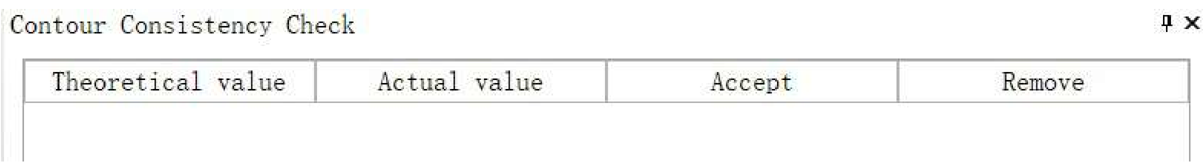
2. A parameter interface will appear. Set the point cloud and elevation point radius.



    - **Input Point Cloud Data**: Input file is a single point cloud data file.
    - **Radius (meters) (default "15")**: Generates an elevation annotation point for a 15-meter radius area.
3. Click OK to generate the elevation annotation points and add them to the scene. To save to a file, click Save.

Check Contour Consistency

If the current contour objects contain contour data and elevation points, contour conflicts can be detected. The results will be marked in the scene (red circles) and displayed in the right panel with detailed conflict information:



- **Theoretical Value**: Theoretical height values of conflicting points based on point cloud calculations.

- **Actual Value**: The actual height values of the conflicting points, i.e., the z-values of the current points (vector entities).
- **Accept**: Click to remove the circular marking in the scene.
- **Delete**: Delete the selected elevation annotation points.

Click on a row in the table to highlight the corresponding point in the scene, and double-click to center the point in the scene.

> Accepting or deleting will not modify the file. To save changes, click Save. The circular or highlight effects will not be saved to the file.

Filter by Minimum Area

Filter out contour entities with an area smaller than a given value, marking them as selected in the scene.

- **Layer**: The layer to be used for filtering; layers not selected will not be processed.
- **Minimum Area (m²)**: Filter out data with an area smaller than this value.

> Only closed contours are considered; open contours are not processed.

Topological Check

Check for intersection errors in contour points in the scene and mark them as conflicts in both the table and the scene.

Entity Selection

Select vectorized entity features, including options for selection, select all, deselect all, select entities, window selection, select intersecting entities, select layer, and invert selection.

Select

Default selection mode allows you to select and modify entities.

- **(Optional) Single-click to Select Entity**: Left-click to select an entity and clear previous selection state. Holding the left Shift key allows you to add to the selection.
- **(Optional) Rectangle Window Selection**: Left-click to interactively click the top-left and bottom-right corners of a rectangle. Entities completely within this rectangle will be selected. Holding the left Ctrl key will deselect entities within this rectangle. By default, this will clear previous selections; holding the left Shift key allows you to add to the selection.
- **(Optional) Rectangle Window Selection**: Left-click to interactively click the bottom-right and top-left corners of a rectangle. Entities partially or fully within this rectangle will be selected. Holding the left Ctrl key will deselect entities within this rectangle. By default, this will clear previous selections; holding the left Shift key allows you to add to the selection.
- **(Optional) Modify Entity Vertex**: Click on a vertex of the selected object and drag to move and modify the position of that point. Only the X and Y coordinates of the point can be modified.
- **(Optional) Modify Entities**: Press the M key, left-click to select the first reference point, and drag to move and modify the position of all selected entities. Only the X and Y coordinates of the objects can be modified.

Select All

Select all entities in the scene.

Deselect All

Deselect all entities in the scene.

Select Layer

Left-click on an entity to select all entities in the layer that the entity belongs to.

Invert Selection

 Invert the current selection state: selected entities become deselected, and deselected entities become selected.

Entity Snap

Snapping features include endpoint snapping, midpoint snapping, nearest point snapping, intersection snapping, center (or circle center) snapping, and orthogonal direction movement only.

Endpoint Snap

Endpoint snapping. Endpoints include the starting point of a line, the top, bottom, left, and right endpoints of a circle, etc.

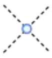
Midpoint Snap

Midpoint snapping of entities. For example, snapping to the midpoint of a line segment.

Nearest Snap

Snapping to the nearest point on an entity relative to the mouse position. This allows movement along the entity.

Intersection Snap



Snapping to the intersection points of multiple entities, such as the intersection of two line segments.

Center Snap

Snapping to the center point of an entity (or the center of a circle).

Restrict Orthogonal

Entity drawing operation. After selecting a current point, subsequent points can only move in the orthogonal direction (vertical or horizontal) from the current point.

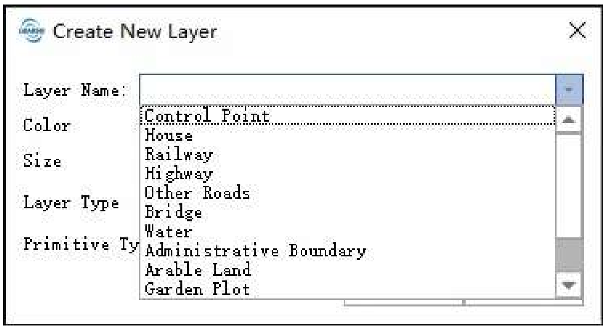
Layer Management

Description: Vector editing is managed by layers. The right-click menu for layers includes options to add a layer, save all layers, and remove all layers. The right-click menu for a single layer includes options to save, remove, show, and hide.

Add Layer

Click *Layer* > *Right-click Menu* > *Add Layer* to open the “Create New Layer” dialog box.

1. Select from the dropdown list of layer names, which include: Control Points, Buildings, Railways, Roads, Other Roads, Bridges, Water Bodies, Administrative Boundaries, Arable Land, Orchards, Forests, Grasslands, Other Vegetation, or enter a layer name directly, as shown below:



2. Click on the layer color to choose the layer color.

3. Set the size of points and the width of lines, within a range of 1-10 pixels.

Save Layer

Click “Save All Layers” or use the right-click menu on a single layer and select “Save” to save the selected layer to the current file.

Show/Hide

Show or hide the graphics drawn on the current layer.

ALS Forest

This chapter describes how to deal with aerial (both UAV and airborne) LiDAR data for forestry applications. This chapter include: Statistical Parameter Extraction, Regression Analysis, Individual Tree Segmentation, Batch Processing, Clear Tree ID, Extract Point Clouds by Tree ID and ALS Editor.

Theory of Elevation Metrics

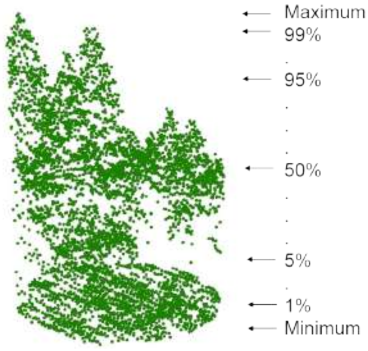
Elevation Metrics are statistical parameters related to point cloud elevation. They are frequently used in regression analysis as independent variables. In this model, 46 statistical parameters related to elevation and 10 parameters related to point cloud density can be calculated,

Principle Description

- **Average Absolute Deviation**: formula: $V = \frac{\sum_{i=1}^{n}(|Z_i - \bar{Z}|)}{n}$, where Z_i represents the elevation of ith point within a statistical unit, $\bar{Z}$ represents the average elevation of all points within a statistical unit, and n is the number of points in a statistical unit.
- **Canopy relief ratio**: formula: $V = \frac{mean - min}{max - min}$, where mean represents the average elevation of a statistical unit, min represents the minimum elevation of a statistical unit, and max represent the maximum elevation of a statistical unit.
- **AIH (15)**: Within a statistical unit, all normalized lidar point clouds are sorted according to the elevation and the cumulative heights of all points are calculated. The cumulative height of X% points in each statistical unit is the statistical unit's AIH. In LiDAR360, 15 AIH can be calculated, including 1%, 5%, 10%, 20%, 25%, 30%, 40%, 50%, 60%, 70%, 75%, 80%, 90%, 95% and 99%.
- **AIH Interquartile Distance**: formula: $V = \mathrm{AIH75\%} - \mathrm{AIH25\%}$, where AIH75% represents the 75% AIH statistical layer, and AIH25% represents the 25% AIH statistical layer.
- **Coefficient of Variation**: It is the coefficient of variation of Z values for all points in a statistical unit. formula: $V = \frac{Z_{std}}{Z_{mean}} \times 100\%$, where, Z_{std} represents the standard deviation of elevation within a statistical unit, and, Z_{mean} represents the average elevation within a statistical unit.
- **Density Metrics (10)**: The point cloud data is divided into ten slices with the same height interval from low to high, and the proportion of returns in each height interval is the corresponding density metrics.
- **Kurtosis**: The kurtosis of the Z value of all points in a statistical unit. formula: $\mathrm{Kurtosis} = \frac{\frac{1}{n-1}\sum_{i=1}^{N}(Z_i - \bar{Z})^4}{\sigma^4} = \frac{\sum_{i=1}^{N} Z_i^4 + 6\bar{Z}^2\sum_{i=1}^{N} Z_i^2 - 4\bar{Z}\sum_{i=1}^{N} Z_i^3 - 4\bar{Z}^3\sum_{i=1}^{N} Z_i + n\bar{Z}^4}{(n-1)\sigma^4}$, where Z_i represents the elevation of ith point within a statistical unit, $\bar{Z}$ represents the average elevation of all points within a statistical unit, n is the number of points in a statistical unit, and σ is the standard deviation of point cloud height distribution within a statistical unit.
- **MADMedian**: The median of median absolute deviation.
- **Maximum**: The maximum value of Z of all points in a statistical unit.
- **Minimum**: The minimum value of Z of all points in a statistical unit.
- **Mean**: The mean value of Z of all points in a statistical unit.
- **Median**: The median of Z of all points in a statistical unit.
- **Generalized means for the 2nd power**: formula: $V = \sqrt[2]{\frac{\sum_{i=1}^{n} Z_i^2}{n}}$, where Z_i is the Z value of the ith point n in a statistical unit, and n is the number of points in a statistical unit.
- **Generalized means for the 3rd power**: formula: $V = \sqrt[3]{\frac{\sum_{i=1}^{n} Z_i^3}{n}}$ where, Z_i represents the elevation of

ith point within a statistical unit, and n is the number of points in a statistical unit.

- **Elevation Percentile (15)**: Within a statistical unit, all normalized lidar point clouds are sorted by elevation, and then the elevation at which X% of points in each statistical unit are located is the elevation percentile of this statistical unit. In LiDAR360, 15 elevation percentiles are calculated, including 1%, 5%, 10%, 20%, 25%, 30%, 40%, 50%, 60%, 70%, 75%, 80%, 90%, 95% and 99%.
- **Elevation Percentile Interquartile Distance**: formula: $V = Elev75\% - Elev25\%$, where Ele75% represents the 75% elevation statistical layer, and Ele25% represents the 25% elevation statistical layer.
- **Skewness**: This value shows the symmetry of Z values of all the points in each statistical unit. formula: $V = \frac{\sum_{i=1}^{n}(Z_i - \bar{Z})^2}{n}$, where Z_i represents the elevation of ith point within a statistical unit, $\bar{Z}$ represents the average elevation of all points within a statistical unit, n is the number of points in a statistical unit, and σ is the standard deviation of point cloud height distribution within a statistical unit.
- **Standard Deviation**: The standard deviation of Z values of all points in a statistical unit.
- **Variance**: The variance of Z of all points in a statistical unit.



Theory of Intensity Metrics

Summary

The intensity metrics are similar to the elevation metrics with the exception that point intensity is used rather than point elevation. Therefore, this function can be used only if the point cloud data contains intensity information. Overall, 42 statistical parameters related to intensity can be calculated, and the resultant product is a table in CSV format or a set of TIFF files.

Principle

- **Average Absolute Deviation**: Computed using the following equation: $\mathrm{V}=\frac{\sum_{i=1}^{n}(|I_i-\bar{I}|)}{n}$, where I_i represents the intensity of i^{th} point within a statistical unit, $\bar{I}$ represents the average intensity of all points within a statistical unit, and n is the number of points in a statistical unit.
- **All (15)**: Within a statistical unit, all normalized lidar point clouds are sorted according to their intensity values and the cumulative intensities of all points are calculated. The cumulative intensity of X% points in each statistical unit is the statistical unit's AIH. In LiDAR360, 15 AIH can be calculated, including 1%, 5%, 10%, 20%, 25%, 30%, 40%, 50%, 60%, 70%, 75%, 80%, 90%, 95% and 99%.
- **Coefficient of Variation**: Computed using the following equation: $\mathrm{V}=\frac{I_{std}}{I_{mean}}\times 100\%$, where I_{std} represents the standard deviation of intensity within a statistical unit, and I_{mean} represents the average intensity within a statistical unit.
- **Kurtosis**：The kurtosis of the intensity values of all points in a statistical unit. The calculation formula is $\mathrm{Kurtosis}=\frac{\frac{1}{n-1}\sum_{i=1}^{n}(I_i-\bar{I})^4}{\sigma^4}=\frac{\sum_{i=1}^{n}I_i^4+6\bar{I}^2\sum_{i=1}^{n}I_i^2-4\bar{I}\sum_{i=1}^{n}I_i^3-4\bar{I}^3\sum_{i=1}^{n}I_i+n\bar{I}^4}{(n-1)\sigma^4}$, in which Z_i is the intensity value of the i-th point in each statistical unit, $\bar{Z}$ is the average intensity of all points in each statistical unit, n is the point number in each statistical unit, and σ is the standard deviation of point cloud intensity distribution within a statistical unit.
- **MADMedian**：The median of median absolute deviation of the intensity values of all points in a statistical unit.
- **Maximum**：The maximum of the intensity values of all points in a statistical unit.
- **Minimum**：The minimum of the intensity values of all points in a statistical unit.
- **Mean**：The mean of the intensity values of all points in a statistical unit.
- **Median**：The median of the intensity values of all points in a statistical unit.
- **Skewness**: This value shows the symmetry of intensity values of all the points in each statistical unit. The calculation formula is $Skewness=\frac{\frac{1}{n-1}\sum_{i=1}^{n}(I_i-\bar{I})^3}{\sigma^3}=\frac{\sum_{i=1}^{n}I_i^3-3\bar{I}\sum_{i=1}^{n}I_i^2+3\bar{I}^2\sum_{i=1}^{n}I_i-n\bar{I}^3}{(n-1)\sigma^3}$, in which Z_i is the intensity value of the i-th point in each statistical unit, $\bar{Z}$ is the average intensity of all points in each statistical unit, n is the point number in each statistical unit, and σ is the standard deviation of point cloud intensity distribution within a statistical unit.
- **Standard Deviation**：The standard deviation of the intensity values of all points in a statistical unit.
- **Variance**: The variance of the intensity values of all points in a statistical unit.
- **Intensity Percentile (15)**：Within a statistical unit, all normalized lidar point clouds are sorted by intensity, and then the intensity at which X% of points in each statistical unit is located is the intensity percentile of this statistical unit. In LiDAR360, 15 intensity percentiles are calculated, including 1%,

5%, 10%, 20%, 25%, 30%, 40%, 50%, 60%, 70%, 75%, 80%, 90%, 95% and 99%.

- **Intensity Percentile Interquartile Distance**: Computed using the following equation: $V = Int75\% - Int25\%$, where Int75% represents the 75% intensity statistical layer, and Int25% represents the 25% intensity statistical layer.

Theory of Canopy Cover

Canopy Cover is the percentage of vertical projection of forest canopy to forest land area (Jennings et al.,1999). It is an important parameter in forest management, and it is also an essential factor for estimating forest volume.

Principle Description

LiDAR360 adopts two different algorithms for calculating canopy cover for point cloud with and without return number information respectively. If the point cloud has return number information, it will be first divided into different grids according to a user-defined xsize and ysize.

$$\mathrm{CC} = \frac{n_{vegfirst}}{n_{first}}$$

Where CC is canopy cover, $n_{vegfirst}$ is the number of vegetation first returns, n_{first} is the number of all first returns.

If the point cloud hasn't return number information. For a certain statistical unit, canopy cover can be calculated as the ratio of vegetation points to total points. Similar to the calculation of Theory of Gap Fraction, points higher than the height threshold in the calculation process are considered as vegetation points.

$$\mathrm{CC} = \frac{n_{veg}}{n_{total}}$$

Where CC is canopy cover, n_{veg} is vegetation return, n_{total} is the total number of returns.

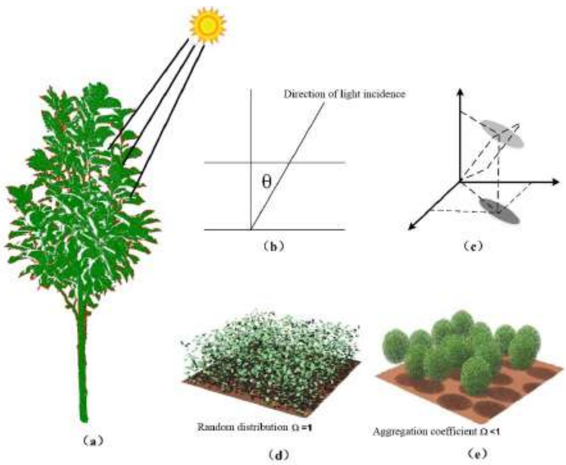
According to (Ma et al., 2017), the canopy cover generated from these two methods have no significant difference. The resultant product is a raster file in TIFF format, the value ranges from 0 (no canopy cover/completely open) to 1 (completely covered by canopy).

```
    @inproceedings{
        author={Jennings S B, Brown N D and Sheil D},
        title={Assessing forest canopies and understorey illumination: canopy closure, canopy cover and other
measures},
        booktitle={Forestry,72(1): 59-73},
        year={1999}
    }
```

```
@inproceedings{
    author={Ma Q, Su Y J and Guo Q H},
    title={Comparison of Canopy Cover Estimations From Airborne LiDAR, Aerial Imagery, and Satellite Imag
ery.},
    booktitle={IEEE Journal of Selected Topics in Applied Earth Observations and Remote Sensing,10(9): 42
25-4236},
    year={2017}
}
```

Theory of Leaf Area Index

The Leaf Area Index (LAI) is one of the most basic parameters that characterizes the forest canopy structure. It is defined as half of the surface area of all leaves projected on the surface area of a unit (Chen et al., 1991), LAI can be calculated from the normalized LiDAR vegetation points.



Schematic diagram of LAI theory principle (a) Direction of light incidence, (b) the zenith angle refers to the angle between the incident ray and the vertical direction, (c) the projection of the leaf in the vertical direction, (d) describes the extinction coefficient, and (e) describes the aggregation coefficient.

Principle Description

LAI value is calculated using the following equation:

$$LAI = -\frac{cos(ang) \times ln(GF)}{k}$$

where, ang is the average scan angle, GF is the gap fraction, and k is the extinction coefficient, which is closely related to the leaf angle distribution (Richardson et al.,2009).

The average scan angle is calculated from LiDAR points using the following equation:

$$ang = \frac{\sum_{i=1}^{n} angle_i}{n}$$

where, ang is the average scan angle, n is the number of LiDAR points and $angle_i$ is the scan Angle of the ith LiDAR point.

The Gap Fraction (GF) is calculated using the following equation:

$$GF = \frac{n_{ground}}{n}$$

where, n_{ground} is the number of ground points and n is the total number of LiDAR points.

```
@inproceedings{
    author={Chen J M and Black T A},
    title={Measuring leaf area index of plant canopies with branch architecture},
    booktitle={Agricultural and Forest Meteorology,57(1-3): 1-12},
    year={1991}
}
```

```
@inproceedings{
    author={Richardson J J},
    title={Modeling approaches to estimate effective leaf area index from aerial discrete-return LiDAR},
    booktitle={Agricultural and Forest Meteorology, 149 (6): 1152-1160},
    year={2009}
}
```

Theory of Gap Fraction

Summary

Gap Fraction is a crucial variable that governs interactions between light and vegetation and requires accurate modelling to predict light climate in the canopy, photosynthetic activity or canopy reflectance.

Principle

The Gap Fraction (GF) is calculated using the following equation:

$$GF = \frac{n_{ground}}{n}$$

where n_{ground} is the number of ground points and n is the total number of vegetation and ground points. Note that all LiDAR points with a normalized height lower than the user-defined height break (2m is commonly used) are treated as ground points in gap fraction calculation (Richardson et al.,2009).

```
@inproceedings{
    author={Richardson J J, Moskal L M and Kim S H},
    title={Modeling approaches to estimate effective leaf area index from aerial discrete-return LIDAR},
    booktitle={Agricultural and Forest Meteorology,149: 1152-1160},
    year={2009}
}
```

Forest Metrics

The Forest Metrics menu contains Elevation Metrics, Intensity Metrics, Canopy Cover, Leaf Area Index, and Gap Fraction generated from point cloud data.

- Calculate Forest Metrics by Grid
- Calculate Forest Metrics by Polygon
- Calculate Forest Metrics by Forest Stands

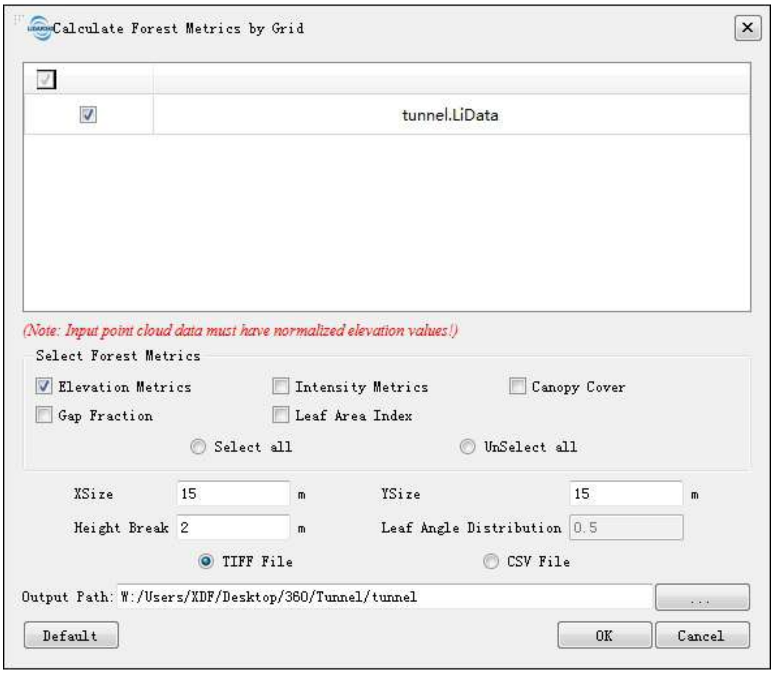
Calculate Forest Metrics by Grid

Functional Overview

Calculate the forest metrics based on the point cloud data in grids. Split the point cloud data with multiple grids, and then calculate the forest metrics in each grid. The forest metrics include elevation, intensity, canopy cover, Leaf Area Index and gap fraction. First, this function will divide the point cloud into different grids with a certain size in the horizontal direction. And then Calculate the forest metrics based on users selection, and, for each grid, generate a CSV file or TIFF file to store the result.

Usage

Click *ALS Forest* > *Forest Metrics* > *Calculate Forest Metrics by Grid* to generate the selected forest metrics.



Parameters Settings

- **Input data**: Ensure that each input point cloud data is normalized; the input file can be a single data file or a point cloud data set; the file(s) to be processed must be opened in the LiDAR360 software.
- **XSize and YSize (meter) (default value is "15")**: This value should be larger than the crown size of the trees. For the data acquired from most of the forest, this value should be larger than 15.
- **Height Threshold (meter) (default value is "2")**: The threshold to divide the data in vertical direction. The point cloud above this height will be used to calculate the forest metrics. The default value is 2m.
- **Leaf Angle Distribution (default "0.5")**: A mathematical expression of leaf probability distribution in three-dimensional space, which is related to vegetation type, leaf angle, and beam direction. According to the empirical formula, the user can determine the value of leaf angle distribution according to the actual situation of the forest. Studies have shown that the elliptic distribution of leaf

angle, which has a leaf angle distribution value of 0.5, may be applicable to the actual situation.

- **Output Path**: Output path. After running, each input point cloud data file will generate a corresponding CSV file or a set of TIFF files, which can be used as independent variables in regression analysis.
- **Default**: Restore the setting parameters to the default values.

> Note: Only when the point cloud data is loaded in the software can you use the Elevation Metrics function; otherwise, the message "There is no point cloud data meeting the conditions of calculation!" will pop up. If the maximum Z value of the point cloud is greater than 200 or the maximum Z minus the minimum Z is greater than 200, the data is considered to be unnormalized, and the software will pop up the prompt message shown in the figure below, click "YES", this type of data Still participating in the calculation, click "NO", this type of data will not participate in the calculation, and the user can re-select the data that meets the conditions.

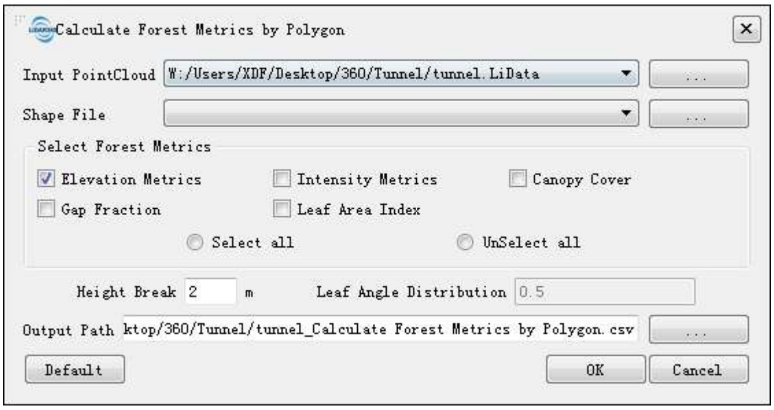
Calculate Forest Metrics by Polygon

Functional Overview

Calculate the forest metrics based on the point cloud data in the given polygon file. The forest metrics include elevation, intensity, canopy cover, Leaf Area Index and gap fraction. Read the position information of the polygon(s) stored in the input SHP file. Calculate the forest metrics and generate a result record for each polygon.

Usage

Click *ALS Forest* > *Forest Metrics* > *Calculate Forest Metrics by Polygon* to generate the selected forest metrics.



Parameters Settings

- **Input Data**: Make sure that all the input files are normalized. The input file(s) can be a single file or a file set. The file(s) to be processed must be opened in LiDAR360 before using this function.
- **SHP file**: The vector data that contains the polygon(s) in which the users need to generate the forest metrics. The SHP file can be added to LiDAR360 or manually input by the users.
- **Height Threshold (meter) (default value is "2")**: The threshold to divide the data in vertical direction. The point cloud above this height will be used to calculate the forest metrics. The default value is 2m.
- **Leaf Angle Distribution (default "0.5")**: A mathematical expression of leaf probability distribution in three-dimensional space, which is related to vegetation type, leaf angle, and beam direction. According to the empirical formula, the user can determine the value of leaf angle distribution according to the actual situation of the forest. Studies have shown that the elliptic distribution of leaf angle, which has a leaf angle distribution value of 0.5, may be applicable to the actual situation.
- **Output Path**: Path of the output file. After the output path is run, each calculation will generate a corresponding CSV file.
- **Default**: Restore the height threshold and leaf inclination angle distribution to the default values.

Note: Only when the point cloud data is loaded in the software can you use the Elevation Metrics function; otherwise, the message "There is no point cloud data meeting the conditions of calculation!" will pop up. If the maximum Z value of the point cloud is greater than 200 or the maximum Z minus the minimum Z is greater than 200, the data is considered to be unnormalized, and the software will pop up the prompt message shown in the figure below, click "YES", this type of data Still participating in the calculation, click "NO", this type of data will not participate in the calculation, and the user can re-select the data that meets the conditions.

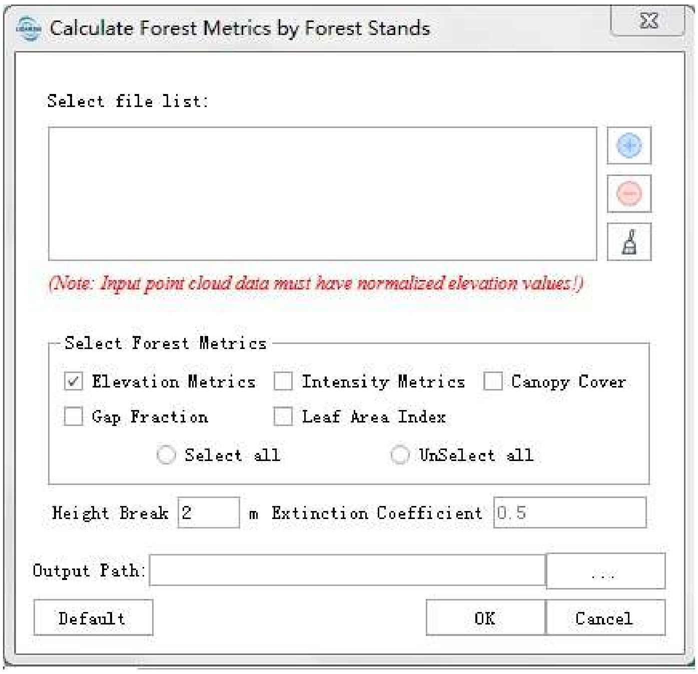
Calculate Forest stands by Forest Stands

Functional Overview

Calculate forest stands by forest stands for each point cloud data,can calculate many kinds of forest metrics.The metrics include Elevation Metrics, Intensity Metrics, Canopy Cover, Leaf Area Index, and Gap Fraction. For each point cloud data, a record will be generated and stored in a CSV file.

Usage

Click *ALS Forest* > *Forest Metrics* > *Calculate Forest Metrics by Forest Stands* to generate the selected forest metrics.



Parameters Settings

- **Input data**: Ensure that each input point cloud data is normalized; the input file can be a single data file or a point cloud data set; the file(s) to be processed must be opened in the LiDAR360 software.
- **Height Break (meter) (default value is "2")**: The threshold to divide the data in vertical direction. The point cloud above this height will be used to calculate the forest metrics. The default value is 2m.
- **Extinction Coefficient (default "0.5")**: A mathematical expression of leaf probability distribution in three-dimensional space, which is related to vegetation type, leaf angle, and beam direction. According to the empirical formula, the user can determine the value of leaf angle distribution according to the actual situation of the forest. Studies have shown that the elliptic distribution of leaf angle, which has a leaf angle distribution value of 0.5, may be applicable to the actual situation.
- **Output Path**: Output path. After running, each input point cloud data file will generate a corresponding CSV file or a set of TIFF files, which can be used as independent variables in regression analysis.
- **Default**: Restore the height threshold and leaf inclination angle distribution to the default values.

> Note: Only when the point cloud data is loaded in the software can you use the Elevation Metrics function; otherwise, the message "There is no point cloud data meeting the conditions of calculation!" will pop up. If the maximum Z value of the point cloud is greater than 200 or the maximum Z minus the minimum Z is greater than 200, the data is considered to be unnormalized, and the software will pop up the prompt message shown in the figure below, click "YES", this type of data Still participating in the calculation, click "NO", this type of data will not participate in the calculation, and the user can re-select the data that meets the conditions.

Regression Analysis

The basic idea of the regression analysis is: firstly, establish a regression model between in-situ forest parameter measurements and LiDAR statistics at the plot level, and then use the obtained model to predict forest parameters from LiDAR statistics. Studies have shown that the regression methods can estimate forest parameters that cannot directly derive from LiDAR point cloud, such as aboveground biomass, and achieve a relative high accuracy without saturation (Popescu et al., 2004). However, building regression models requires a number of field measurement data. Please refer to Forestry FAQ for the selection of field measurements.

- Sample Data and Independent Variables
- Linear Regression
- Support Vector Machine
- Fast Artificial Neural Network
- Random Forest Regression
- Run Existing Regression Model

```
    @inproceedings{
        author={Popescu S C and Wynne R H},
        title={Seeing the trees in the forest: Using lidar and multispectral data fusion with local filtering
and variable window size for estimating tree height},
        booktitle={Photogrammetric Engineering and Remote Sensing, 70(5): 589-604},
        year={2004}
    }
```

Sample Data and Independent Variables

Sample Data

All regression methods require sample data derived from field measurements to train the regression model. The sample data can be imported using the “Import Training Data” dialog of each regression model. The imported file should be in text format (.txt extension) and must contain a header to label each column. The first two columns should be the X and Y coordinates of each sample, followed by any dependent values.Note that multiple dependent values can be given, but only one can be used for each regression execution. If each sample corresponds to a tree, the value of the dependent variable can be the height of the tree. SeeSample Data File in the File Formatssection of the Appendix for information about sample data file.

 then be viewed in the Import Training Data section.

- **Dependent Variable**: This parameter defines the dependent variable used in the regression analysis. Only one dependent variable from the sample data file can be selected each time.
- **Plot Type**: This parameter defines the plot type. It should be selected based on the plot surveying methodology.
- **Square (default)**: The plot is square.
- **Circle**: The plot is circle.
- **Length (meter)**: When the plot type is square, set the length of the plot.
- **Radius (meter)**: When the plot type is circle, set the radius of the plot.
- **Local Error Buffer**: When the plot type is circle, set the radius of the plot.
- **X**: X coordinate of the plot center.
- **Y**: Y coordinate of the plot center.

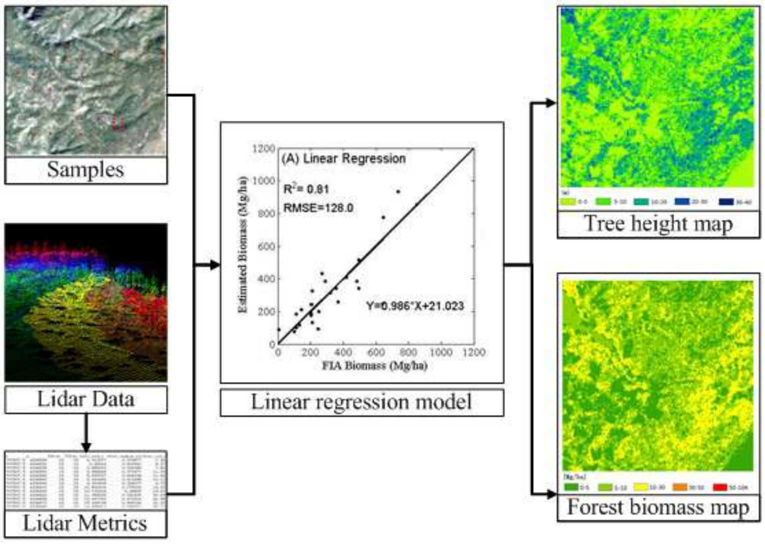
Independent Variables

Users can also import data in .csv or .tif format. While importing a.csv table, only one can be selected at a time, and must contain at least four fields, including X, Y, X size, and Y size (each column would be added to the list as an independent variable). However, multiple .tif images can be added at once. Every time a .tif image is added successfully, an independent variable named after the file name of .tif will be added to the list. n the selection of the independent variables, select the independent variable used in regression analysis (for more information about each independent variable, refer to elevation, intensity, canopy cover, leaf area index and gap fraction).

Linear Regression

Description

This tool use Python scikit-learn and NumPy to build linear regression model.



Usage

Click *ALS Forest* > *Regression* > *Linear Regression*.

Parameters

- **Sample data**: Refer to Sample data and dependent variable.
- **Dependent variable**: Refer to Sample data and dependent variable.
- **Plot type**: Square or circle.
- **Length**: Length or radius of the plot.
- **Optimize by considering location uncertainty**: If check, the value in the location uncertainty represents the accuracy value of the range query. The model will query all sample points that meet

the conditions according to the range (if the number of sample points exceeds 50, the first 50 points will be selected as samples), and then the model will be selected based on these sample points The best point is used as analysis data. If uncheck, the model will select the closest point as the analysis data according to the sample point.

- **Accuracy assessment**：Using the K-Fold cross evaluation model. According to the input K-Fold parameters, the samples are divided into K class, one of them is taken as the test data in turn, the others are used as the training data for model training, the test data is used for testing, and the smallest error is selected The model of is used as the best model. Note: The K-Fold value must be greater than or equal to 2.
- **Save regression Model**：If check, the model will be generated in output path.
- **Save regression dataset**：If check, the csv will be generated in output path.
- **Output path**：To select output path. The corresponding model report (linear regression.html) file will be generated, which records the error and related values of the model; the corresponding result file (linear regression.tif) will be generated, which is calculated according to the model and the imported tif or csv file The value of the independent variable predicts the result of the corresponding dependent variable; and according to the check situation, choose to generate the regression model and data set.

Linear Regression Summary

Regress Type	LINEAR				
Linear Regression Coffs	**a1**	**a2**	**a3**	**a4**	**a5**
	0.42594355501	-0.920513309986	1.08612214327	0.515717171997	-3.89992128691
	a6	**a7**	**a8**	**a9**	**a10**
	5.32208922164	-2.03571577184	2.55419044528	-4.50899390082	-3.43392046593
	a11	**a12**	**a13**		
	5.46055222409	1.08123669727	-0.575347246215		
K-Fold	10				
R	0.944891274743				
R Square	0.892819521086				
RMSE	0.00853442112782				
Probability Value	0.0				
The Result of K-fold Test Insignificant	Yes				

Dependent and Independent Variable

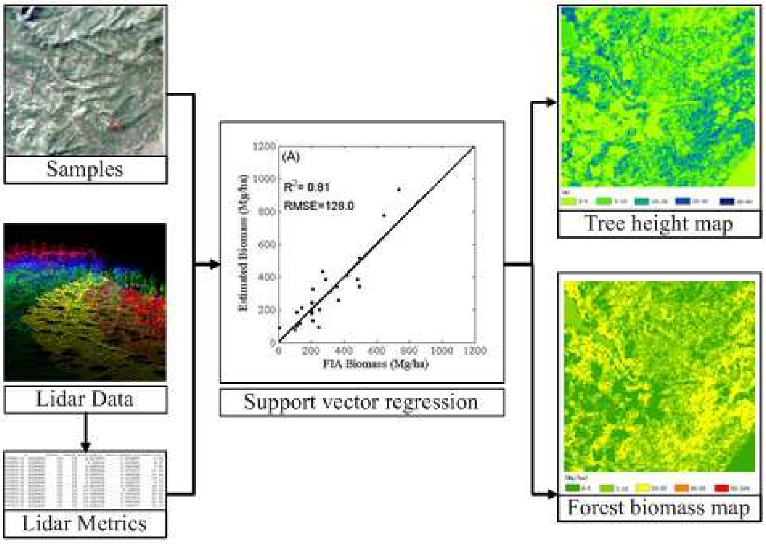
Dependent Variable	Biomass	
Independent Variable	elev_percentile_5th	elev_percentile_10th
	elev_percentile_20th	elev_percentile_25th
	elev_percentile_30th	elev_percentile_40th
	elev_percentile_50th	elev_percentile_60th
	elev_percentile_70th	elev_percentile_75th
	elev_percentile_80th	elev_percentile_90th
	elev_percentile_95th	

> Note: The imported sample data must be included in the imported independent variable data range. Independent variables can be added or deleted according to the actual situation. The final result file is generated based on the imported independent variable information.

Support Vector Machine

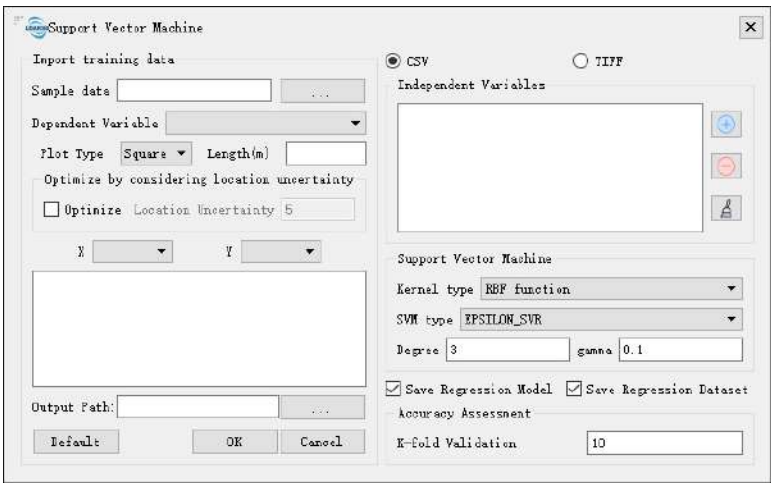
Summary

This tool use Python Package scikit-learn and NumPy to build up the Random Forest model.



Usage

Navigate to *ALS Forest > Regression Analysis > Support Vector Machine*.



Settings

- **Import Training Data**: Refer to Sample Data and Independent Variables.
- **Independent Variables**: Refer to Sample Data and Independent Variables.
- **Kernel Type**: Users can select the type of kernel function here including RBF function, Linear, Polynomial, and Sigmoid.
  - **RBF Function (default)**: $\exp(-\gamma|x-x'|^2)$, where γ > 0.
  - **Linear**: $<x,x'>$.

- **Polynomial**: $(\gamma < x, x' > +r)^{degree}$, where γ > 0.
- **Sigmoid**: $\tanh(\gamma < x, x' > +r)$.

- **SVM Type**: Two types of SVM method are provided.
  - **EPSILON_SVR (default)**: EPSILON SVR(ϵSVR).
  - **NU_SVR**: NU SVR(νSVR).
- **Degree (default value is "3")**: Kernel function parameter.
- **Gamma (default value is "0.1")**: Kernel function parameter.
- **Accuracy Assessment**: Based on the K-Fold cross validation model, a sample would be partitioned into k subsets according to input K-Fold value (no less than 2). Take one of subsets as a validation dataset and the remaining subsets as training datasets to form a model, then run this model and test the fitting of validation set to training sets. Repeat this process until every subset is treated as a validation set at least once and select out the model with the least MSE (mean square error) as the optimal model.
- **Save Regression Model**: Tick the checkbox to save the SVM model (Support Vector Machine.model) under the output path.
- **Save Regression Dataset**: Tick the checkbox to save the training dataset (Support Vector Machine.csv) in .csv format under the output path.
- **Output Path**: Choose an output directory. A support vector machine regression model report (Support Vector Machine.html), recording the model's parameters and accuracy (R-square, RMSE), would be generated under this directory. A prediction result file (Support Vector Machine.tif), based on the support vector machine regression model and input variables from a .tif or .csv file, would also be generated under this output directory.

Support Vector Regression Summary

Degree	3
Gama	0.10000000149
K-Fold	10
R	0.801855560156
R Square	0.642972339353
RMSE	0.0284289120661
Probability Value	0.0
The Result of K-fold Test Insignificant	No

Dependent and Independent Variable

Dependent Variable	Biomass	
Independent Variable	elev_percentile_1st	elev_percentile_5th
	elev_percentile_10th	elev_percentile_20th
	elev_percentile_25th	elev_percentile_30th
	elev_percentile_40th	elev_percentile_50th
	elev_percentile_60th	elev_percentile_70th
	elev_percentile_75th	elev_percentile_80th
	elev_percentile_90th	elev_percentile_95th
	elev_percentile_99th	

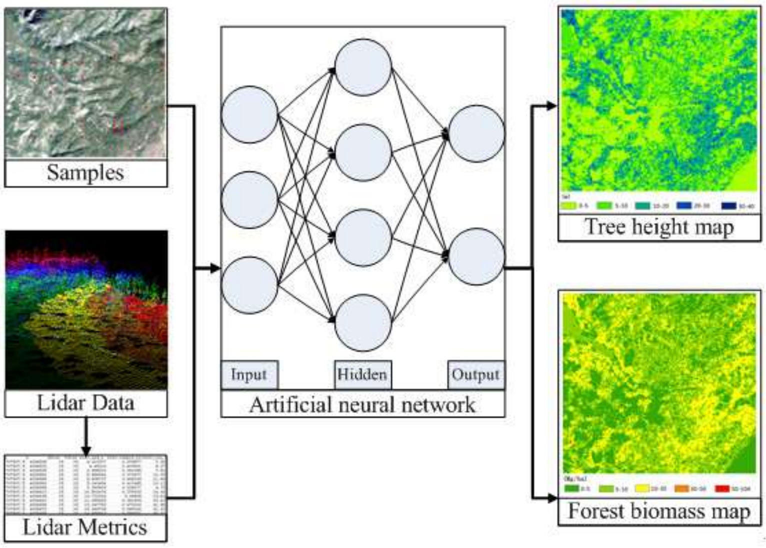
> Note: The dimension of imported sample/training data must be within the scope of independent variables, which may be adjusted accordingly. The model/result is based on the passed-in variables.

```
@inproceedings{
    author={Chang C C and Lin C J},
    title={LIBSVM: A Library for Support Vector Machines},
    booktitle={ACM,2(3):1-27},
    year={2011}
}
```

Fast Artificial Neural Network

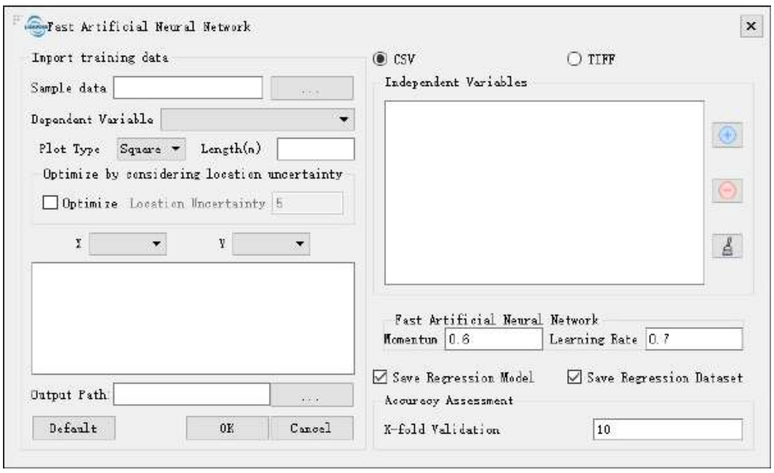
Summary

This tool uses the Python packages scikit-learn and NumPy to establish an artificial neural network regression model.



Usage

Navigate to and click on *ALS Forest* > *Regression Analysis* > *Fast Artificial Neural Network*.



Settings

- **Import Training Data**: Refer to Sample Data and Independent Variables.
- **Independent Variables**: Refer to Sample Data and Independent Variables.
- **Momentum (default value is "0.6")**: Set the momentum parameter in ANN regression analysis for selecting the optimized path.
- **Learning Rate (default value is "0.7")**: The global learning rate for training the network.

- **Accuracy Assessment**: Based on the K-Fold cross validation model, a sample would be partitioned into k subsets according to input K-Fold value (no less than 2). Take one of the subsets as a validation dataset and the remaining subsets as training datasets to form a model, then run this model and test the fitting of the validation set to training sets. Repeat this process until every subset is treated as a validation set at least once then select out the model with the least MSE (mean square error) as the optimal model.
- **Save Regression Model**: Tick the checkbox to save the ANN model (Fast Artificial Neural Network.model) under the output path.
- **Save Regression Dataset**: Tick the checkbox to save the ANN training dataset (Fast Artificial Neural Network.csv) in .csv format under the output path.
- **Output Path**: Choose an output directory. A fast artificial neural network regression model report (Fast Artificial Neural Network.html), recording the model's parameters and accuracy (R-square, RMSE), would be generated under this directory. A prediction result file (Fast Artificial Neural Network.tif), based on the fast artificial neural network regression model and input variables from a .tif or .csv file, would also be generated under this output directory.

Artificial Neural Network Regression Summary

Learning Rate	0.699999988079
Momentum Rate	0.600000023842
K-Fold	10
R	0.89832207752
R Square	0.80698255496
RMSE	0.0153693301027
Probability Value	0.0
The Result of K-fold Test Insignificant	Yes

Dependent and Independent Variable

Dependent Variable		
Independent Variable	elev_max_z	elev_min_z
	elev_mean_z	elev_median_z
	elev_percentile_1st	elev_percentile_5th
	elev_percentile_10th	

> Note: The dimension of imported sample/training data must be within the scope of independent variables, which may be adjusted accordingly. The model/result is based on the passed-in variables.

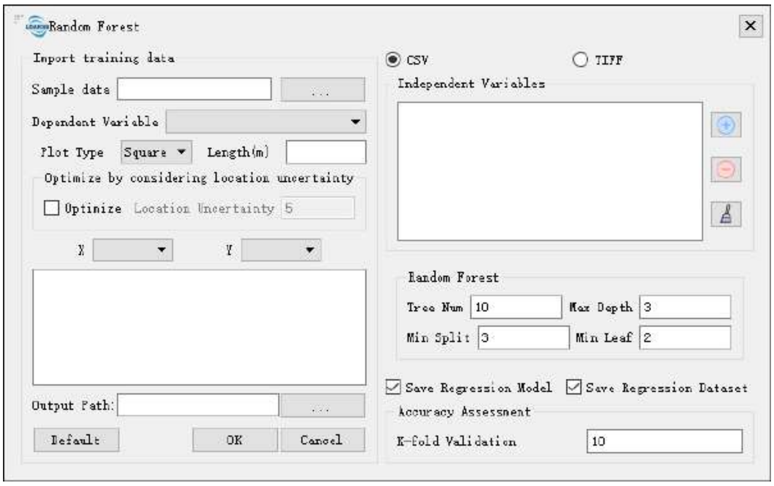
Random Forest Regression

Description

This tool use Python Package scikit-learn and NumPy to build up the Random Forest model.

Usage

Click *ALS Forest* > *Regression Analysis* > *Random Forest Regression*.



Parameters

- **Sample Data**: Please refer to Sample Data and Independent Variables。
- **Independent Variables**: Please refer to Sample Data and Independent Variables。
- **Random Forest**: These values define Random Forest's parameters.
  - **Tree Num**： Tree number in the Random Forest model.
  - **Max Depth**： The maximum depth of Random Forest model.
  - **Min Split**： The minimum split of Random Forest model.
  - **Min Leaf**： The minimum leaf number in Random Forest model.
- **Accuracy Assessment**： Use K-Fold cross-validation model. According to the inserted K-Fold parameters, divide the sample into K groups. Each group will be taken as testing data by training the model using other remianing samples. Note that K-Fold value should be larger than 1 (don't include 1).
- **Save Regression Model**: If the box is checked, a model named (Random Forest.model) will be generated in the output path, after the program being successfully run.
- **Save Regression Dataset**： If the box is checked, a training data model named (Random Forest.csv) will be generated in the output path, after the program being successfully run.
- **Output Path**: The path for the output files. The software will generate a model report (Random Forest.html) with the residuals and related values of the model, a result file (Random Forest.tif), and a regression model file (optional).
- **Default Value**: Restore all the default values for all parameters.

Random Forest Regression Summary

Tree Num	10
Max Depth	3
Min Split	3
Min Leaf	2
K-Fold	10
R	0.965381896766
R Square	0.931962206603
RMSE	0.0054176207024
Probability Value	0.0
The Result of K-fold Test Insignificant	Yes

Dependent and Independent Variable

Dependent Variable	Biomass	
Independent Variable	elev_percentile_1st	elev_percentile_5th
	elev_percentile_10th	elev_percentile_20th
	elev_percentile_25th	elev_percentile_30th
	elev_percentile_40th	elev_percentile_50th
	elev_percentile_60th	elev_percentile_70th
	elev_percentile_75th	elev_percentile_80th
	elev_percentile_90th	elev_percentile_95th
	elev_percentile_99th	

> Note: The inserted sample data must be included in the range of inserted independent variables. The number of independent variables can be changed based on the users' situation. The final result is generated according to the inserted independent variables. Max Depth and Tree Num should be greater than 0.

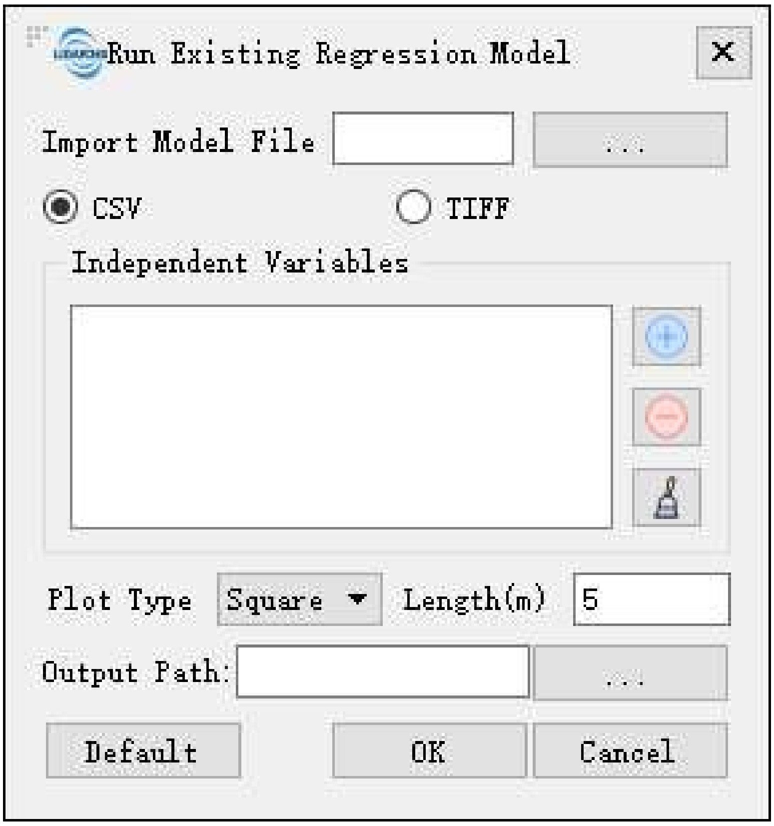
Run Existing Regression Model

Summary

This tool estimates forest metrics using any available regression models (Linear Regression, Support Vector Machine, Fast Artificial Neural Network and Random Forest Regression).

Usage

Navigate to *ALS Forest > Regression Analysis > Run Existing Regression Model*.



Insert Model File

Select a regression model file (*.model) generated by the regression analysis tools, including linear regression, support vector machines, fast artificial neural network, or random forest, etc.

Insert Independent Variables

Inserted file should be in CSV or TIF format. CSV files must have attributes of X, Y, XSize, and YSize. Each attribute will be added into the list as a independent variable. Only one CSV file can be added; while several TIF files can be added. Each TIF will be added to the list as a independent variable with its file name. The number of independent variables must be the same as the number of independent variables in the model selected. (To learn more about each independent variable, please refer to Elevation Metrics, Intensity Metrics, Canopy Cover, Leaf Area Index, and Gap Fraction)

Setting

- **Plot Type**: This parameter defines the plot type. Users can choose that according to the survey in the sample data.
  - **Square (default)**: The plot is in square shape.
  - **Circle**: The plot is in circle shape.
- **Length (meters)**: When the plot type is square, set the length of the plot.
- **Radius (meters)**：When the plot type is circle, set the radius of the plot.

- **Output Path**: Select the path for the output files. The software will generate two files, "Regression Predict.tif" file and "Regression Predict.html" report, after succeeding to predict. The TIF file is the result of prediction.

> Note: the number of inserted independent variable must be the same as the number of independent variables in the inserted model. Otherwise, it will lead to failing to predict or large residuals.

Segmentation

Segmentation provides a series of functions for segmenting individual trees and therefore acquire individual tree attributes from ALS data.

CHM Segmentation

Summary

CHM segmentation utilizes the watershed segmentation (Chen et al., 2006) technique to identify and delineate individual trees, and therefore obtain individual tree information, such as tree location, tree height, crown diameter, crown area and tree boundaries.

Principle

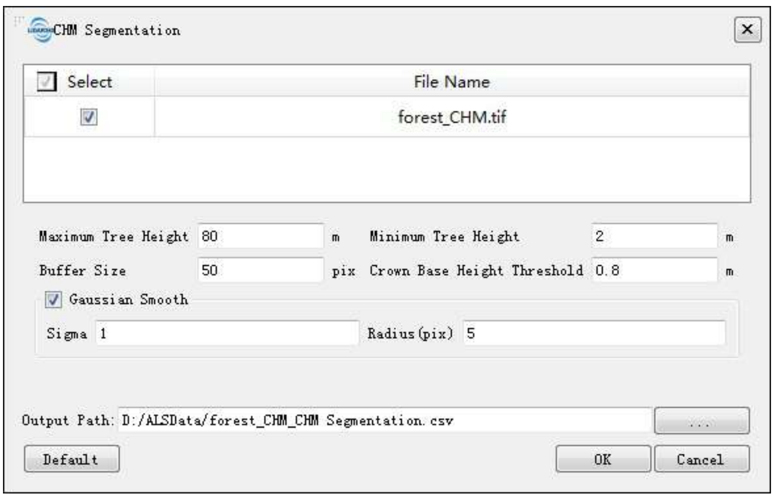
The basic principle of watershed segmentation algorithm is if placing a water source in each regional lowest point in the CHM and flooding the entire relief, barriers can be generated when different water sources meet. The resulting set of barriers can build a watershed by flooding, which is the segmentation result. Note that the CHM segmentation result can be affected by the CHM resolution. It is recommended that the CHM should have a spatial resolution higher than 1 m to ensure a sufficient segmentation result. Moreover, the CHM segmentation result can also be largely influenced by tree density and tree species. If the algorithm does not work well in certain study area, users can try to use other segmentation algorithms to get the best segmentation result.

(a) (b) (c)

(a) CHM; (b) Watershed Segmentation Algorithm; (c) CHM Segmentation Result

Usage

Navigate to *ALS Forest* > *Segmentation* > *CHM Segmentation*.



Settings

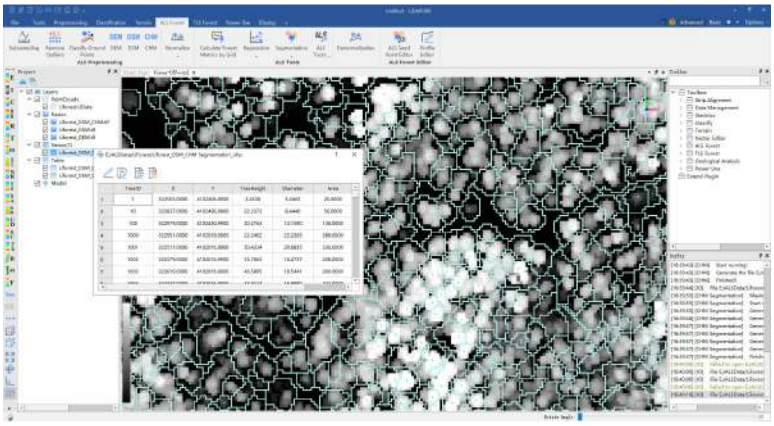
- **Input Data**: The input file can be a single CHM file or multiple CHM files. The data should be opened in LiDAR360 before being processed.
- **Maximum Tree Height (meter)(default value is "80")**: The threshold defining the maximum tree height in a study areas. CHM pixels above this threshold will not be considered in the segmentation.
- **Minimum Tree Height (meter)(default value is "2")**: The threshold defining the minimum tree height in a study areas. CHM pixels below this threshold will not be considered in the segmentation.
- **Buffer Size (pix)(default value is "50")**: A threshold controls the block size for performing CHM segmentation, which can avoid physical memory overflow. This value should be larger than the maximum crown area and smaller than 1500. If the CHM is larger than 1500 by 1500 pixels, the CHM will be forced to divided into a number of blocks with a size of 1500 by 1500 pixels.
- **Crown Base Height Threshold (meter)(default value is "0.8")**: The starting height of the crown range. A reasonable base height value can help to improve the accuracy of the boundary and area of crown. After setting the base height, the pixels higher than this height will be included to generate the vector boundary of crown, while the pixels lower than this height will be excluded. Users should set this value according to the species and growing situation of the trees in order to get the best result.
- **Gaussian Smooth (optional)**: This parameter is to control whether to perform Gaussian smoothing. In general, it is recommended to check the Gaussian smoothing option to remove noise effects.
- **Sigma (default value is "1")**: Gaussian smoothing factor. The greater the value is, the smoother the results are. The degree of smoothness can affect the number of trees being segmented. In the case of under-segmentation, it is recommended to reduce this value (e.g. 0.5); and in the case of over-segmentation, it is recommended to increase the value (e.g. 1.5).
- **Radius (pix)(default value is "5")**: The window size used by Gaussian smoothing, which should be an odd number. Generally, it can be set to the average crown diameter.
- **Output Path**: Path of the output file. The resultant product is a comma-separated table in “csv” format and a polygon file in "shp" format. The CSV table contains the ID, location (x and y coordinates), height, crown diameter, and crown area of each segmented tree. The shp file contains the boundary of each tree, and its corresponding attribute table contains the ID, location (x and y coordinates), height, crown diameter, and crown area of each segmented tree. Refer to the CHM Segmentation Results for segmentation result examples.
- **DefaultValue**: Restore the default parameters.

> Note: The CHM Segmentation function can only be used when the raster data is loaded in the software. Otherwise, the message “There is no raster data!” will pop up.

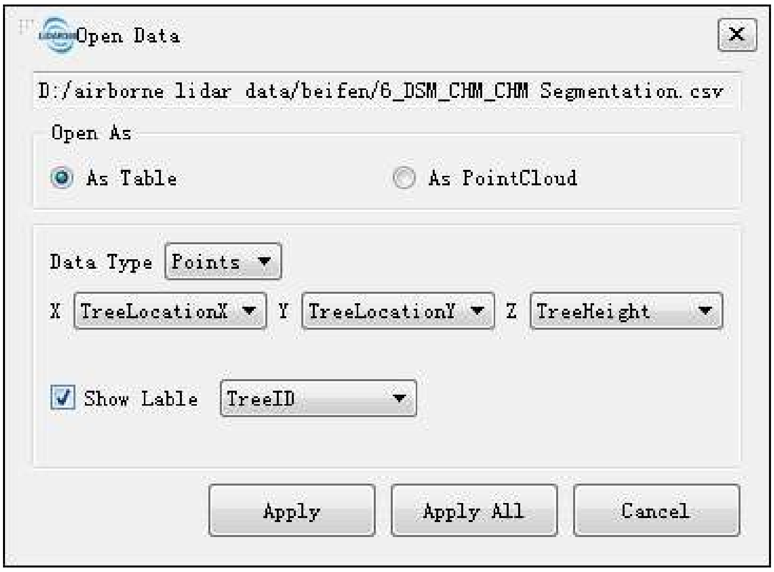
```
@inproceedings{
    author={Chen Q, Baldocchi D, Gong P and Kelly M},
    title={ Isolating Individual Trees in a Savanna Woodland Using Small Footprint Lidar Data},
    booktitle={Photogrammetric Engineering and Remote Sensing, 72(8): 923-932},
    year={2006},
}
```

View the CHM Segmentation Results

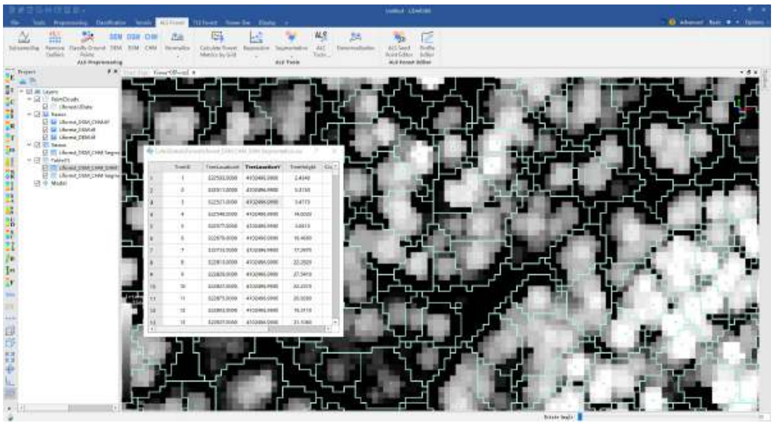
After the CHM segmentation is completed, each input CHM data will generate a corresponding CSV file and SHP file, and the CHM can be superimposed and displayed with the CSV file and the shp file. Load the CHM, csv and shp files into the software respectively. The following figure shows the superimposed display effect of CHM and shp data, and the attribute table of csv data.



Open the csv file as a table, as shown below, and select the X, Y, Z corresponding to the TreeLocationX, TreeLocationY and TreeHeight, respectively. Check the "Show Label" option (if the label has too much text and block other information, one can remove the data and reopen it without checking the "Show Label" option). Click “Apply” to load the csv file in the software.



On the csv file name, click the mouse right button, select Table Attribute, and the property information will be prompted. Double-click on the mouse left button anywhere in each line, the viewer will jump to the corresponding individual tree location.



CHM Pits Filling

LiDAR canopy height models (CHM) may display unnatural holes or pits, where pixel values are significantly lower than their neighboring pixels. These pits can result from various factors during data acquisition and post-processing. These pits not only affect the visual quality of the CHM images but also impact semi-automated canopy extraction and lead to significant errors in biomass estimation. Therefore, it is necessary to effectively fill these pits in the CHM images.

Principle Description

CHM pit filling uses the algorithm by Ben-Arie et al., 2009, to identify and fill pits in CHM images. While filling the pits, the values of non-pit areas remain unchanged.

1. Apply a Laplacian filter to the original CHM image to obtain the Laplacian edge detection image.
2. Apply a median filter to the original CHM image.
3. Construct a cumulative histogram of the Laplacian edge detection image and apply algorithm parameters to segment the histogram. Use the threshold value from the segmentation to binarize the Laplacian image. Values below the threshold are set to 0, and values above the threshold are identified as pits.
4. For all pit locations, extract pixel values from the median filtered image to fill the pits.

The threshold parameter for cumulative histogram segmentation generally ranges from 1% to 30%. The final threshold parameter can be determined by adjusting the parameter and previewing the pit filling effect.

Usage

In the toolbox, click *Airborne Forestry* > *Individual Tree Segmentation* > *CHM Pit Filling*.

Parameter Settings

- **CHM File**: When there is CHM data in the view, it will be automatically added. It can also be added from the dialog box. Refer to Canopy Height Model for the method of generating CHM.
- **Threshold**: The parameter for segmenting the cumulative histogram of the Laplacian edge detection image. The pixel value corresponding to this threshold is used to binarize the Laplacian edge detection image. This value typically ranges from 1% to 30%, with a default value of 3%.
- **Output Path**: The path for the pit-filled CHM image. The output image format is TIF.
- **Default**: Click this button to restore the default value of 3%.

Preview

Due to the varying distribution and size of pits in different CHM images, the local preview function can be used to check the applicability of the current threshold parameter. During the preview, adjust the parameter and click different areas in the view to check the pit filling effect. If the threshold parameter shows good filling effects across different areas, it can be used as the final threshold parameter to perform pit filling on the entire CHM image.

Click the *Preview* button in the dialog box to start the preview function. In the view, click on the image to be pit-filled, and the dialog box will display a 512x512 pixel area around the clicked location with the pit-filling effect. The preview image can be moved and zoomed to check the pit-filling effect.

> Note:
>
> 1. The input CHM image should be a single-channel image.
> 2. CHM pit filling can be performed repeatedly. Pit-filled images can be filled again if needed.

Ben-Arie J R, Hay G J, Powers R P, et al. Development of a pit filling algorithm for LiDAR canopy height models[J]. Computers & Geosciences, 2009(9). DOI:10.1016/J.CAGEO.2009.02.003.he applicability of the current threshold parameter. During preview, adjust the parameter and click different areas in the view to check the pit filling effect. If the threshold parameter works well for different areas, it can be used as the final threshold parameter to perform pit filling on the entire CHM image.

Click the *Preview* button in the dialog box to start the preview function. In the view, click on the image to be filled with pits, and the dialog box will display the pit-filled effect image within a 512*512 pixel range around the clicked area. You can move and zoom the preview image to check the pit filling effect.

> Note:
>
> 1. The input CHM image should be a single-channel image.
> 2. CHM pit filling can be performed repeatedly. You can fill pits again on the pit-filled image.

```
Ben-Arie, J. R., Hay, G. J., Powers, R. P., et al. (2009). Development of a pit filling algorithm for LiDAR canopy height models. *Computers & Geosciences*. DOI: [10.1016/J.CAGEO.2009.02.003](https://doi.org/10.1016/J.CAGEO.2009.02.003).
```

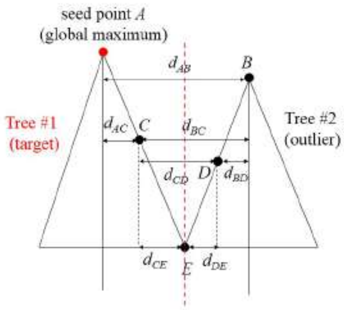
Point Cloud Segmentation

Summary

The point cloud segmentation algorithm determines the position, tree height, crown diameter, crown area, and crown volume of individual tree by analyzing the elevation values of points and their distances from other points.

Principle Description

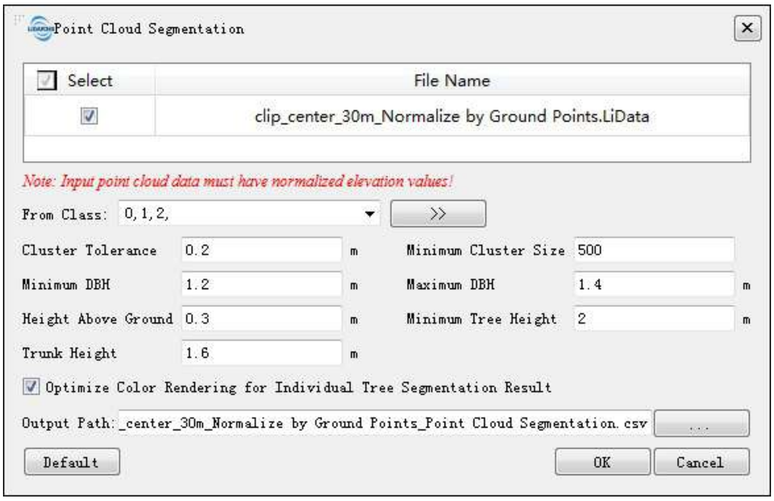
Li et(Li et al.,2012) developed a point cloud segmentation algorithm that separates individual trees from the point cloud one by one. The algorithm works as follows:



Starting with seed point A (global maximum), this algorithm estimates lower points based on critical spacing and minimum spacing rules to develop a tree cluster from seed point A. For example, if point A is the highest point, it is considered as the top (target) of Tree 1. Then classify all points below A successively. First, Point B is classified as Tree 2 because dAB > a set critical value (this parameter is determined by users). Then we set Point C whose distance dAC is less than the critical value. By comparing with Points A and B, Point C's category was set to Tree 1 because dAC < dBC. Comparing with Points B and C classifies Point D into Tree 2; comparing with Points C and D classifies Point E into Tree 2. The critical value should be equal to the canopy radius. When the critical value is too large or too small, under-segmentation or over-segmentation may occur.

Usage

Navigate to and click on *TLS Forest* > *Point Cloud Segmentation*.



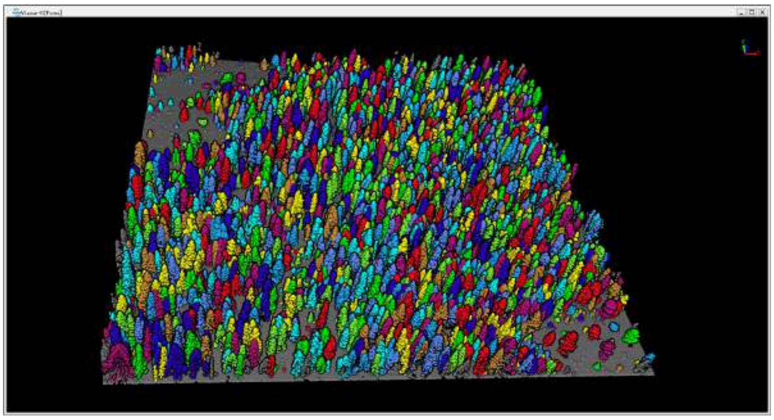
Settings

- **Input Data**: Ensure that each input point cloud data is Normalize by DEM or Normalize by Ground Points.
- **From Class**: Class involved in point cloud segmentation, with default selection of all categories contained in the point cloud data.
- **Grid Size (m) (default: "0.6")**: Grid resolution parameter used to identify tree positions, generally set to one-fifth of the average crown diameter; this parameter usually ranges from 0.3~0.6 meters.
- **Buffer Size (pix) (default: "50")**: When the length and width range of the segmented data exceeds 1500***grid size**, block processing will be performed, and this value is the buffer threshold for blocking, measured in pixels. It can be set as maximum crown diameter divided by raster resolution.
- **Minimum Tree Height (m) (default: "0")**: Minimum tree height threshold for segmenting individual trees; those below this value will be segmented into individual trees.
- **Height Above Ground (m)(Default:"2")**: Points lower than this threshold are considered not part of a tree and are ignored during segmentation process.
- **Gaussian Smoothing**: Whether to perform Gaussian smoothing; it is recommended to select Gaussian smoothing option to remove noise interference.
- **Sigma(Default:"1")**: Gaussian smoothing factor; higher values result in smoother results while lower values lead to less smoothness. Smoothness affects number of trees segmented out; if under-segmentation occurs, reduce Sigma value(e.g., 0.5), otherwise increase Sigma value(e.g.,1.5); in addition to the Gaussian smoothing factor, CHM segmentation results are also affected by CHM resolution. To adjust CHM resolution, DEM and DSM resolutions need to be adjusted.
- **Radius (pix) (default: "5")**: Window size used for Gaussian smoothing; this value is odd and can generally be set to the average crown diameter.
- **Optimize Color Rendering for Individual Tree Segmentation Result(Default Selected)**: By rearranging ID information after tree segmentation, it can greatly solve the problem of adjacent trees being assigned with same color.
- **Output File Type**: Select output file type including CSV table files and tree attribute files.
- **Output Path**: Output path where each point cloud data will generate corresponding segmentation result; the result is a comma-separated CSV table or tree attribute file containing attributes such as Tree ID, x,y coordinates, tree height,crown diameter,crown area,and crown volume; refer to Point Cloud Segmentation Result File Format in Appendix for details on format.
- **DefaultValue**: Reset each parameter to the default value.

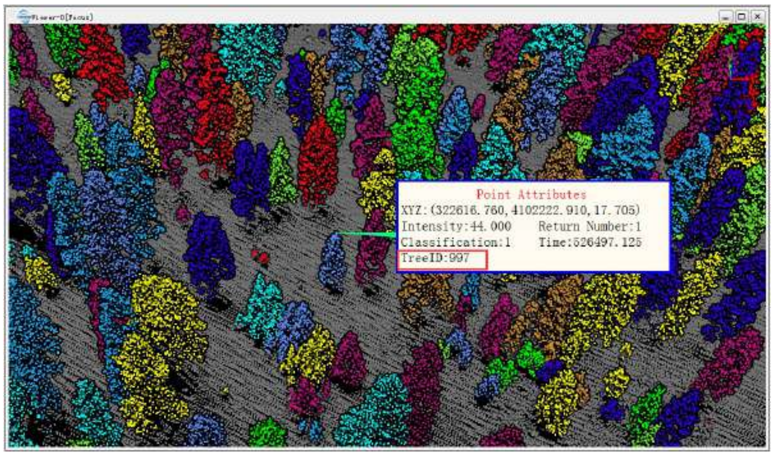
```
    @inproceedings{
        author={Tao S L, Wu F F, Guo Q H, Wang Y C, Li W K, Xue B L, Hu X Y, Li P, Tian D, Li C,Yao H, Li Y M
, Xu G C and Fang J Y},
        title={Segmentation tree crowns from terrestrial and mobile LiDAR data by exploring ecological theori
es},
        booktitle={ISPRS Journal of Photogrammetry and Remote Sensing,110:66-76},
        year={2015}
    }
```

View the Point Cloud Segmentation Results

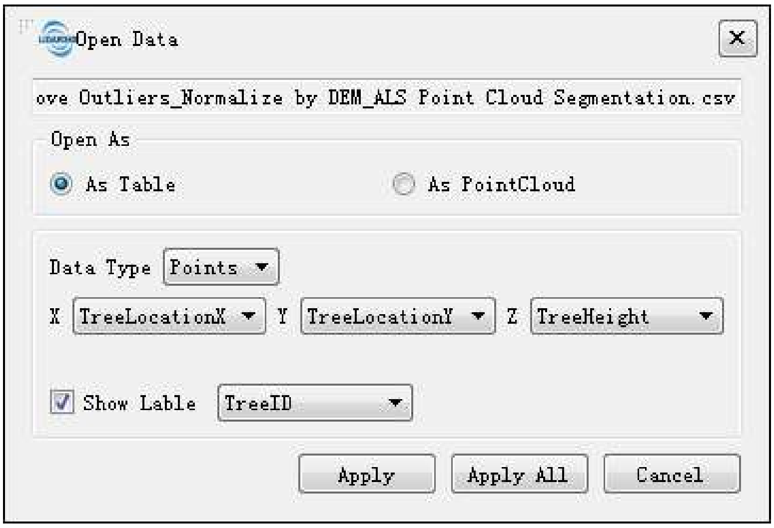
After segmenting the trees, Tree IDs are saved to each point in the point clouds used in the segmentation, the results can be viewed within a window viewer. Load the point cloud used for the tree segmentation into a 3D viewer. Ensure the viewer is active and press the Display by Tree ID button in the color tools toolbar. The following is an example of a point cloud colored by individual tree. The ALS Editor tool can be used to check the single tree segmentation results. At the same time, it can be used for editing operations such as add, delete seed points etc., and segment the point cloud data based on the edited seed points.



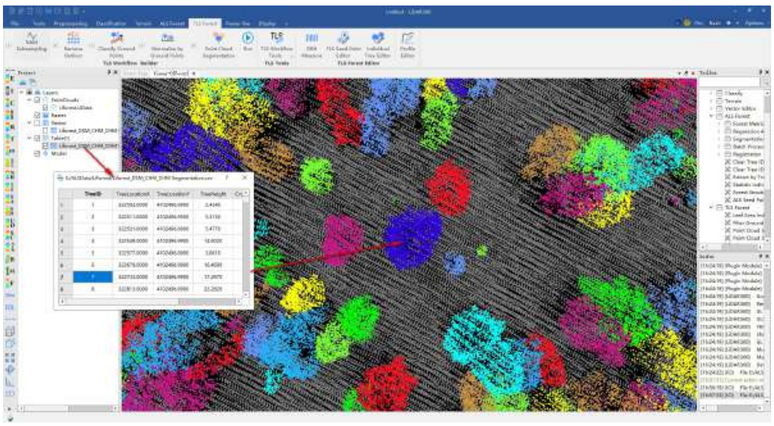
The tree ID attribute of each point can be queried by the Pick Point（ ）tool on the toolbar.



CSV file generated by segmentation can be superimposed with the point cloud, and the CSV file can be opened as a table. Select X, Y and Z as the TreeLocationX, TreeLocationY and TreeHeight respectively in the CSV file as shown below, and check the Show Label (if the label has too much text and blocking other data, affect the display effect, you can remove the data and open again, do not check the Show Label), click “Apply” to load the CSV file in the software.



Click the right mouse button on the CSV file name, select Table Attribute, you can view the property information, double-click the left mouse button anywhere in each line, you can jump to the corresponding location. The following figure shows the effect of superimposed display of point cloud data and CSV file. Open the attribute table of the CSV file and double-click to jump to the selected row.



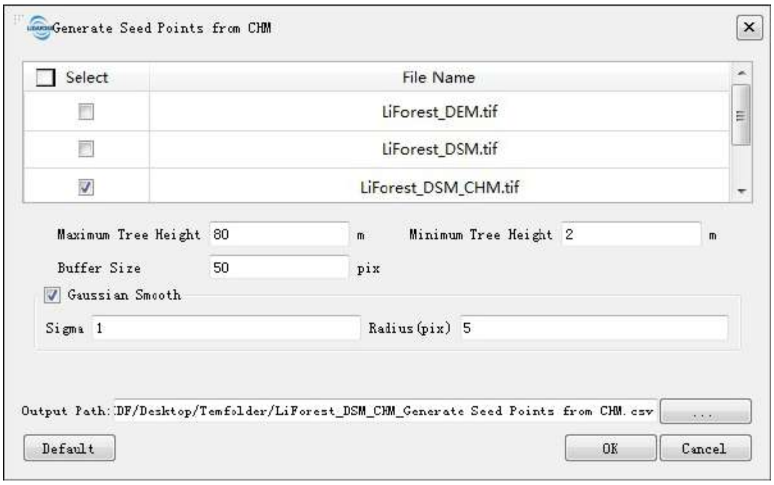
Generate Seed Points by CHM

Summary

This tool aims to generate individual tree seed points from CHM so that they can be used for Point Cloud Segmentation based on Seeds. The ALS Editor tool can be used to edit the seed points and therefore improve the individual tree segmentation accuracy.

Usage

Navigate to *ALS Forest* > *Segmentation* > *Generate Seed Points from CHM*.



Settings

- **Input Data**: The input file can be a single CHM file or multiple CHM files. The data should be opened in LiDAR360 before being processed.
- **Maximum Tree Height (meter)(default value is "80")**: The threshold defining the maximum tree height in a study areas. CHM pixels above this threshold will not be considered in the segmentation.
- **Minimum Tree Height (meter)(default value is "2")**: he threshold defining the minimum tree height in a study areas. CHM pixels below this threshold will not be considered in the segmentation.
- **Buffer Size (pix)(default value is "50")**: A threshold controls the block size for performing CHM segmentation, which can avoid physical memory overflow. This value should be larger than the maximum crown area and smaller than 1500. If the CHM is larger than 1500 by 1500 pixels, the CHM will be forced to divided into a number of blocks with a size of 1500 by 1500 pixels.
- **Gaussian Smooth (optional)**: This parameter is to control whether to perform Gaussian smoothing. In general, it is recommended to check the Gaussian smoothing option to remove noise effects.
- **Sigma (default value is "1")**: Gaussian smoothing factor. The greater the value is, the smoother the results are. The degree of smoothness can affect the number of trees being segmented. In the case of under-segmentation, it is recommended to reduce this value (e.g. 0.5); and in the case of over-segmentation, it is recommended to increase the value (e.g. 1.5).
- **Radius (pix)(default value is "5")**: The window size used by Gaussian smoothing, which should be an odd number. Generally, it can be set to the average crown diameter.

- **Output Path**: Path of the output file. The resultant product is a comma-separated table in “csv” format which contains ID and x, y, z coordinates of each seed points.
- **DefaultValue**: Restore the default parameters.

> Note: Generate Seed by CHM Segmentation function can only be used when the raster data is loaded in the software. Otherwise, the message “There is no raster data!” will pop up. The interface and parameters setting of Generate Seed by CHM are the same as those of CHM Segmentation. The difference between these two tool is that CHM Segmentation will generate a csv file that includes attributes such as tree height, crown diameter and crown area, and a shp file for tree boundary, while Generate Seed by CHM will not generate a shp file. For the seed point file format, refer to Seed Points File.

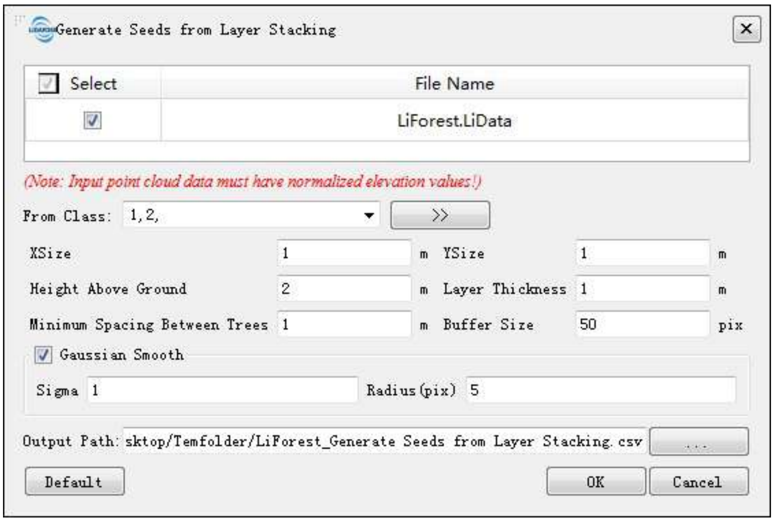
Generate Seeds from Layer Stacking

Summary

This tool aims to generate individual tree seed points using an layer stacking algorithm so that they can be used for Point Cloud Segmentation based on Seeds. The ALS Editor tool can be used to edit the seed points and therefore improve the individual tree segmentation accuracy.

Usage

Navigate to *ALS Forest* > *Segmentation* > *Generate Seeds from Layer Stacking*.



Settings

- **Input Data**: Ensure that each input point cloud data is Normalize by DEM or Normalize by Ground Points. The input file can be a single file or multiple data files. Point cloud data should be opened in LiDAR360 before being processed.
- **XSize (meter)(default value is "1")**: Grid x-axis resolution. It is recommended to set this parameter between 0.3m and 2m.
- **YSize (meter)(default value is "1")**: Grid y-axis resolution, It is recommended to set this parameter between 0.3m and 2m.
- **Height Above Ground (meter)(default value is "2")**: Usually, it is desirable to ignore points below a particular height to avoid influence low vegetations (e.g., grass and shrub). Points below this threshold will not be considered in the segmentation. A value of 2 m is commonly used.
- **Layer Thickness (meter)(default value is "1")**: The layer thickness is used to cut the layer for stacking. It is recommended to set this parameter between 0.5m and 2m.
- **Minimum Spacing Between Trees (meter)(default value is "1")**: This parameter should be set as the minimum spacing of the tree. This parameter can be adjusted to avoid over- or under-segmentation.
- **Buffer Size (pix)(default value is "50")**: A threshold controls the block size for performing segmentation, which can avoid physical memory overflow. This value should be larger than the

maximum crown area and smaller than 1500. If the image is larger than 1500 by 1500 pixels, it will be forced to divided into a number of blocks with a size of 1500 by 1500 pixels.

- **Gaussian Smooth (optional)**: This parameter is to control whether to perform Gaussian smoothing. In general, it is recommended to check the Gaussian smoothing option to remove noise effects.
- **Sigma (default value is "1")**: Gaussian smoothing factor. The greater the value is, the smoother the results are. The degree of smoothness can affect the number of trees being segmented. In the case of under-segmentation, it is recommended to reduce this value (e.g. 0.5); and in the case of over-segmentation, it is recommended to increase the value (e.g. 1.5).
- **Radius (pix)(default value is "5")**: The window size used by Gaussian smoothing, which should be an odd number. Generally, it can be set to the average crown diameter.
- **Output Path**: Path of the output file. The resultant product is a comma-separated table in “csv” format which contains the ID and x, y, z coordinates of each seed points.
- **DefaultValue**: Restore the default parameters.

> Note: Only when the point cloud data is loaded in the software can you use the Generate Seed by Layer Stacking function; otherwise, the message "There is no point cloud data meeting the conditions of calculation!" will pop up. If the maximum Z value of the point cloud is greater than 200 m or the maximum Z minus the minimum Z is greater than 200 m, the data is not considered to have been normalized, and the prompt information shown in the figure below will pop up. Click “YES” to keep using this type of data in the operation; otherwise, click "NO" and reselect the input data file. The seed point file obtained by layer stacking is the same as the seed point file generated by CHM, please refer to Seed Points File.

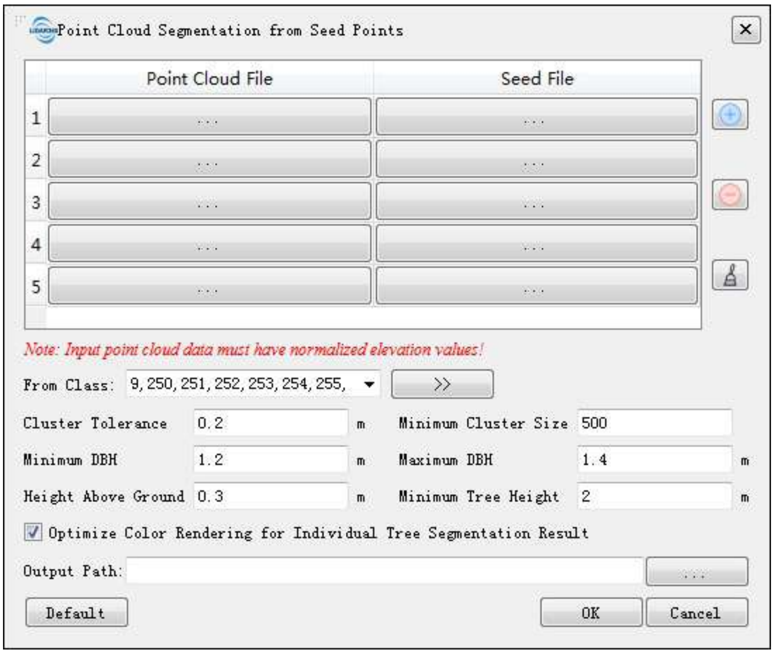
ALS Point Cloud Segmentation from Seed Points

Summary

This function supports batch processing of multiple files. The input data consists of normalized point cloud data and the corresponding seed point files. For seed point file generation, please refer to Generate Seed by CHM and Generate Seed by Layer Stacking。

Usage

Navigate to and click on *TLS Forest* > *Point Cloud Segmentation from Seed Points*.



Parameters

- **From Class**: Classes which participate in the PCS with seeds (all classes by default).
- **Point Cloud File**: Click ... to select the point cloud data to be processed.
- **Seed File**: Click ... to select the seed file.
- : Five datasets can be batch processed per tool run. Click to add files to be processed.
- : Delete the selected point cloud and seed points file.
- : Clear the file list.
- **Height Above Ground (meter)(default value is "2")**: Only the points above this hight will be involved in individual tree segmentation. This parameter is used to decrease the influence of ground points and weeds to the segmentation. It will influence the accuracy of the detection of trunk, if this value is too large.
- **Optimize color rendering for individual tree segmentation result (checked by default)**: By reorganizing the tree ID generated after the individual tree segmentation, it can greatly solve the problem of rendering the same color to the trees next to each other. Note: if choosing to optimize the color rendering, the tree IDs in new csv file for individual tree segmentation are not one-to-one

correspond to those in the input seeds files.

- **Output File Format**:Select the format of output file, choice including .csv attribute table and tree attribute file.
- **Output Path**: Path of the output file, which is a comma-separated database table in the .csv format containing the ID of each individual tree identified during the segmentation process, the x, y coordinate of each individual tree, individual tree heights, DBHs, crown diameters, crown areas, and crown volumes.
- **DefaultValue**: Reset each parameter to the default value.

> Note：For methods to view segmentation results, please refer to Point Cloud Segmentation Results.

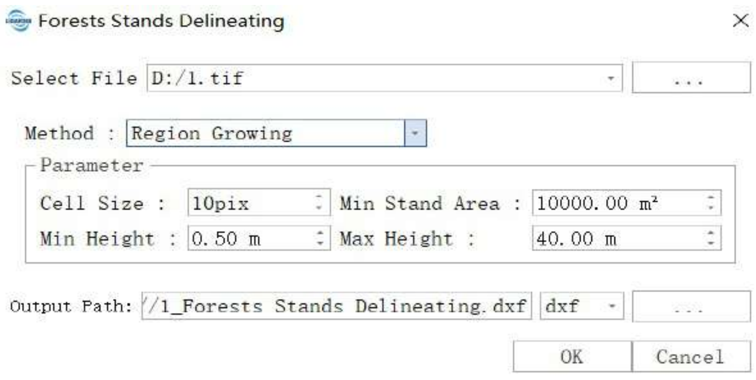
Forest Stands Delineating

Overview

Automatically delineate forest stand regions based on tree similarity.

Usage

Click *ALS Forest* > *Forest Stands Delineating* to enable the feature.



Parameter Settings

- **Select File**: Canopy Height Model file generated using CHM.
- **Method**: Choose between the Region Growing method and the cell automation method. The latter results in more subdivisions than the former.
- **Cell Size**: The pixel size of the initial cell unit. Generally, larger cell units result in fewer subdivisions.
- **Min Stand Area**: The minimum area of a forest stand. Areas that do not meet this condition will be filtered out.
- **Min Height**: The minimum tree height within the region.
- **Max Height**: The maximum tree height within the region.
- **Initial Stand Area**: Applicable only to the cellular automata method. Larger values result in fewer subdivisions.

Batch Process

The batch processing menu contains the automated workflows for the extraction of forest metrics, which includes Forest Metrics, Point Cloud Segmentation, Canopy Height Model(CHM) Segmentation, and all prerequisites for these operations as follows:

1. Remove Outliers
2. Classify Ground Points (Filter)
3. DEM
4. Normalize by DEM
5. Forest Metrics, Point Cloud Segmentation or Canopy Height Model(CHM) Segmentation processing

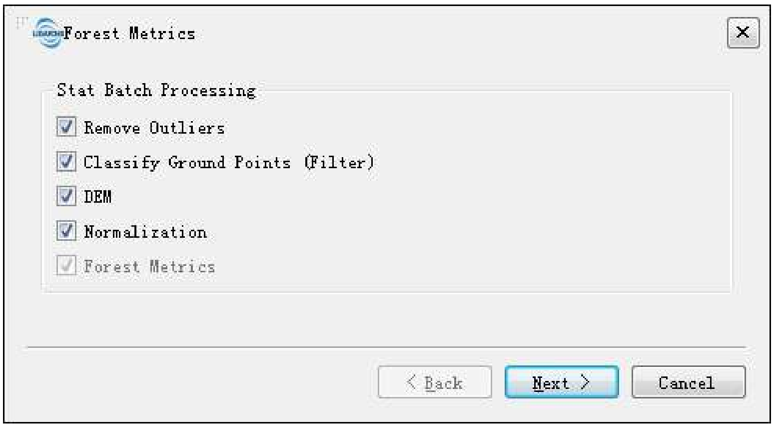
- Forest Metrics
- Point Cloud Segmentation
- Canopy Height Model(CHM) Segmentation

Forest Metrics

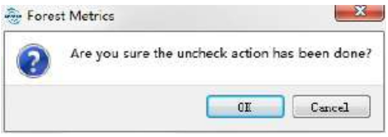
Usage

Navigate to *ALS Forest* > *Batch Process* > *Forest Metrics*. The For Forest Metrics tool can extract all metrics found in the *ALS Forest* > *Forest Metrics* automatically.

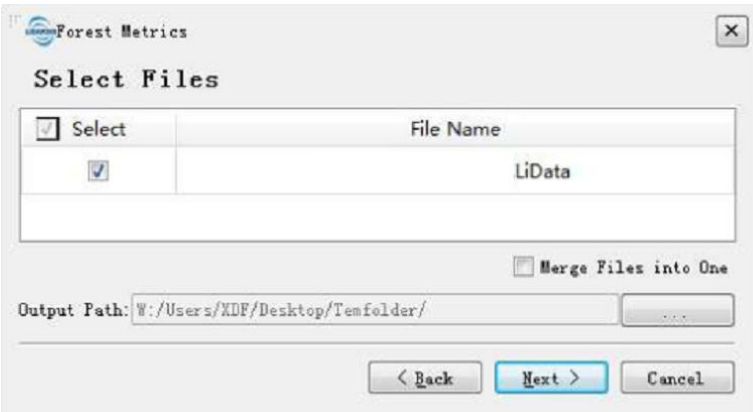
The first interface that appears in the dialog box is a series of steps to be followed in the automatic extraction of statistical parameters.



If any process has been completed, it can be ignored in the batch. Clicking “Next” will prompt a dialog shown as follows to ask whether the uncheck step has been done? If yes, click "OK" to the next step to select the file for batch processing.

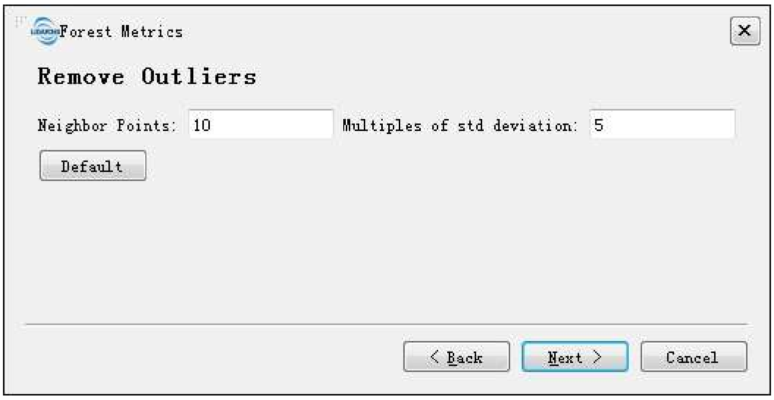


All the point cloud data opened in the software is showed in the list of unprocessed file. Users can determine whether the data is involved in the operation use the check box in front of the file name. By default, each point cloud data will be processed separately. If you check "Merge files into one", the software will merge the selected point cloud data into one file before subsequent processing.

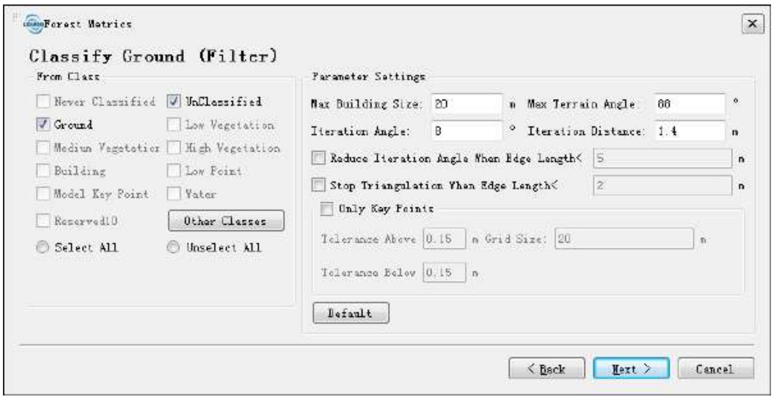


Click “Next” and set appropriate parameters for all steps involved in the batch processing.

For parameters setting of "Outlier Removal", please refer to Remove Outliers.

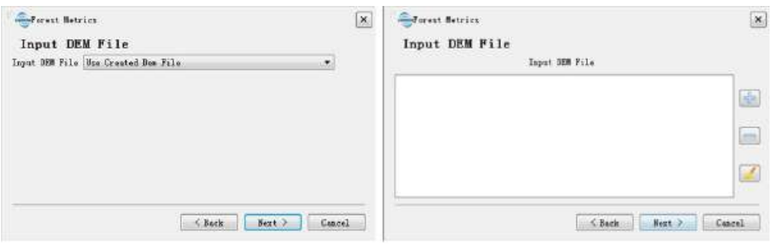


For parameters setting of "Classify Ground Points", please refer to Classify Ground Points.



For parameters setting of "DEM", please refer to DEM.

At the step of "Input DEM File", users can choose the DEM generated by the last step or an existing one already loaded into the project (the speed of batch processing will increase dramatically if an existing DEM is used rather than creating it at every batch).



Click “Next” and choose the "Forest Metrics Type" which need to be extracted. Depending on the choice, a set of parameters will need to be given, please refer to Forest Metrics.

Click “Finish” to begin the batch process.

Point Cloud Segmentation

Usage

Navigate to *ALS Forest > Batch Process > Point Cloud Segmentation*. The Point Cloud Segmentation tool is used for automatic segmentation and processing of airborne LiDAR data.

The steps of this process are similar to the Forest Metrics. For more information about the point cloud segmentation, please refer to the Point Cloud Segmentation.

Canopy Height Model(CHM) Segmentation

Usage

Navigate to *ALS Forest > Batch Process > Canopy Height Model(CHM) Segmentation*. The CHM Segmentation tool is used for automatic segmentation and processing of CHM derived from LiDAR data.

The steps of this process are similar to the Forest Metrics. For more information about the CHM segmentation, please refer to the CHM Segmentation.

Registration

Registration method for forestry data.

- Tree Based Point Cloud Registration

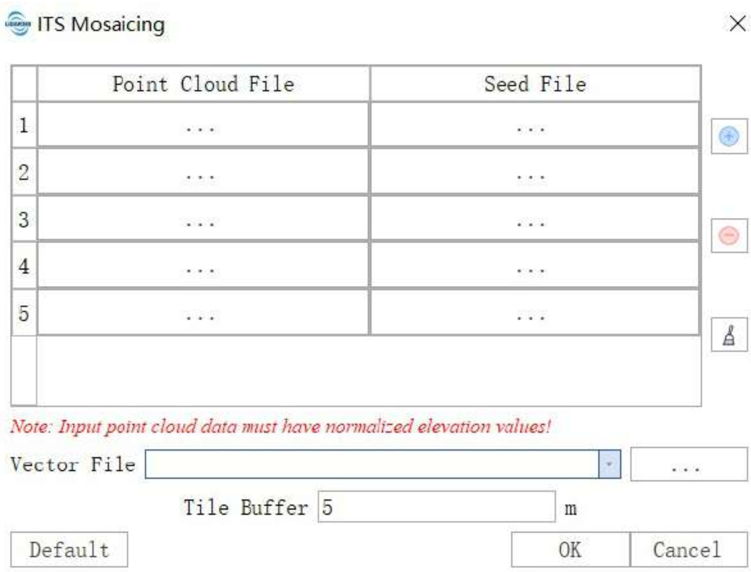
Auto Registration by Tree Locations

Summary

This function register two point cloud data sets based on the location of the trees (Li J , Zhao P , Hu Q , et al., 2020) (Guan H , Su Y , Hu T , et al., 2019). The location information of the trees is stored in a CSV file, and the file should at least contain two attributes, the TreeLocationX and TreeLocationY. Please refer to Individual Tree Segmentation Result File Format for the detailed format information. The data to be registered must be two flat surface with trees, which means the registration will only translate the data and rotate the data around the Z-axis. The calculation of the transform matrix consists of two steps, namely the calculation of the horizontal rotation matrix and the calculation of the vertical translation. For horizontal transform, this function first creates feature descriptors for each individual tree position; then votes based on the similarity of the feature descriptors. The individual tree position with the higher score will be used as the matching point, and finally calculate the two-dimensional rotation matrix according to the matching points. For registration in the vertical direction, this function first obtains the point cloud in the neighborhood of each individual tree, and then uses the maximum or minimum value in the point cloud for registration in the Z direction. This process uses the 3σ principle to make the confidence level of the difference in Z direction highest. The success rate of registration depends on the matching degree between the point cloud to be registered and the corresponding position of the registration point cloud. The higher the matching degree, the higher the algorithm success rate. The matching degree of individual tree depends on the degree of data overlap. the accuracy of position, and so on.

Usage

Click *ALS Forest* > *Registration* > *Auto Registration by Tree Locations*



Settings

- **Reference Point Cloud**: The reference point cloud data. After the registration, the point cloud to be registered will be transform to the coordiante system of the reference point cloud.

- **Point Cloud to be Registered**: The point cloud data to be registered. After the registration, the transform matrix will be applied to this point cloud data and generate a new file.
- **Number of Neighborhood Points (default value is 8)**: The size of the neighborhood used to generate the feature descriptors. The increase of the size of the neighborhood will increase the time complexity of the algorithm, while it will increase the robustness of the feature descriptors at the same time. The decrease of this value will improve the algorithm efficiency, while it will increase the number so that the matching point outliers as well. This value should be set to 8 ~ 12.
- **Minimum Votes (default value is 6)**: The individual tree positions below this number of votes will be considered as outliers and will not participate in the match. The smaller this value, the more matching points will be obtained, but the error of matching point pairs will also be greater. The larger this value is, the fewer matching points are obtained. If the value is too large, there will not be enough matching points for transform matrix calculation. When the number of neighborhood points increases, this value can be increased accordingly. This value should be set to 6 ~ 9.
- **Homography Matrix Backprojection Error (default value is 3.0)**: Backprojection error of homography matrix. Candidate points selection is performed by using homography matrix combined with Ransac algorithm. The smaller the backprojection error, the fewer the matching points of the selected single wood and the higher the matching degree. The larger this value is, the looser the filter conditions for matching points are, and the greater the error of matching point pairs. While this value is too small, it may lead to not enough matching points participating in the calculation. This value should be set to 2~4 most suitable.
- **Searching Radius （meter) (default value is 2.0)**: When performing Z-direction registration, the search radius used for neighborhood search. When registering in the Z direction, users need to find the real point cloud of each individual tree in a certain neighborhood on the XY plane, and then use the coordinates of the real point cloud for registration. If the value is too large or too small, the point cloud will not be able to describe the real terrain relief. This value should be set to 1.0~3.0, if it is too large, it will reduce the efficiency of the algorithm.
- **Z Translation ("Minimum" by default)**: There are two registration methods in the Z direction, namely "maximum value" and "minimum value". This value describes that after performing a radius search on the position of the individual tree, the "maximum/smallest value" of the point cloud in the neighborhood is taken as the true matching point, which is used to calculate the final translation in the Z direction. When using the "maximum value", it is necessary to ensure that there is no missing or distorted treetop point cloud near the individual tree, otherwise it will be impossible to extract accurate matching points from the point cloud. When using the "minimum value", it is necessary to ensure that there are enough ground points near the individual wood, otherwise the Z direction matching will fail.

```
@inproceedings{
    author={Li J , Zhao P , Hu Q , et al.},
    title={ Robust point cloud registration based on topological graph and Cauchy weighted lq-norm},
    booktitle={Isprs Journal of Photogrammetry & Remote Sensing, 160:244-259},
    year={2020},
}
```

```
@inproceedings{
    author={Guan H , Su Y , Hu T , et al.},
    title={ A Novel Framework to Automatically Fuse Multiplatform LiDAR Data in Forest Environments Based on Tree Locations},
    booktitle={IEEE Transactions on Geoence and Remote Sensing},
    year={2019},
```

```
    }
```

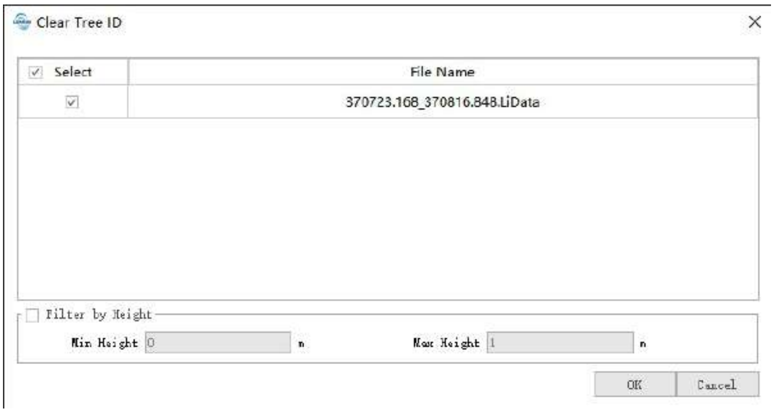
Clear tree ID

Functional Overview

After the point cloud is segmented, the tree ID information will be saved in the LiData file. If the point cloud needs to be segmented again, the tree ID information needs to be cleared first.

Usage

Click on *Airborne Forestry* > *Clear Tree IDs*.



Select the point cloud data to be processed and click OK.

Parameter Settings

- **Filter by height**: Whether to filter by height, if this option is checked, the tree ID information of the points between the minimum height and the maximum height will be cleared, if this option is not checked, the tree ID information of all points will be cleared Tree ID information.
- **Minimum (default "0")**: Clears the minimum elevation of the tree ID.
- **Max (default "1")**: Clears the maximum elevation of the tree ID.

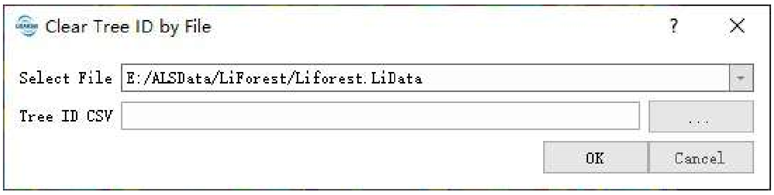
Clear Tree ID based on File

Summary

After the point cloud is segmented, there may be a wrongly segmented tree. The ID corresponding to the wrongly segmented tree can be recorded in a CSV file, and then the CSV file can be used to clear the wrongly segmented tree in the point cloud.

Usage

Click *ALS Forest*> *Clear Tree ID by CSV File*.



Select the point cloud data to be processed and click OK.

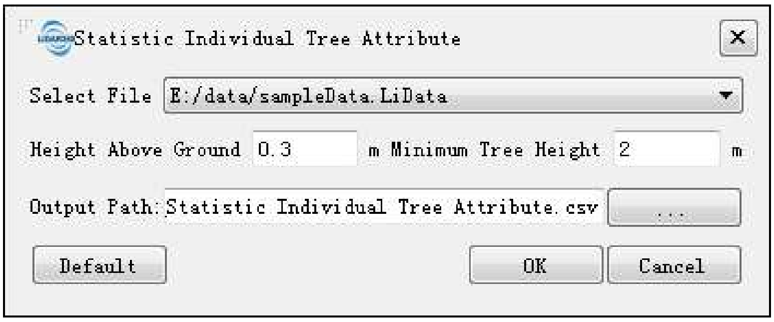
Statistic Individual Tree Attributes

Summary

This function is mainly used for attribute statistics and attribute extension on point clouds that have already been normalized and segmented into individual trees. It generates single tree attribute files, either by creating new files or updating existing ones.

Usage

Click *ALS Forest* > *Statistic Individual Tree Attributes*.



Parameters

The parameters are divided into two pages. The first page is for selecting the target files and the attributes to be counted, while the second page is for setting specific statistical parameters.

- **First Page**
- **New File**: Generates a new single tree attribute file. For the specific file format, refer to Individual Tree Segmentation Result File Format.
- **Update File**: Updates an existing single tree attribute file. This is mutually exclusive with "New File." If the attribute file does not contain the selected attributes, they will be added.
- **Attribute Selection**: At least one single tree attribute must be selected for statistics.

- **Second Page**

- **Input Point Cloud**: Select the point cloud to be counted, which must have been segmented into individual trees.

- **Height Above Ground**: Only counts point clouds higher than this height.
- **Minimum Tree Height**: Only counts individual trees with a height greater than this value.
- **Input DEM (optional)**: This option appears when "Elevation" is selected as a statistical item.
- **Input Slope (optional)**: This option appears when "Slope" is selected. The slope file is in tiff format. If a slope file is input, the slope at the tree location is obtained from the file and added as an extended attribute to the single tree attribute file.
- **Input Reference Point File**: The reference point file is a comma-separated text file with two rows and three fields: X, Y, Radius. X and Y are the reference point positions, and Radius is the radius. If a reference point file is input, the azimuth and distance of trees within a certain radius from the

reference point are calculated and added as extended attributes to the single tree attribute file.

The following table is an example of a reference point file:

```
X,Y,Radius
322716.24,4102494.69,15.0
```

The following table is an example of a single tree attribute file:

```
TreeID, TreeLocationX, TreeLocationY
1, 322716.24, 4102494.69
2, 322751.21, 4102499.9
3, 322519.35, 4102499.3
```

The following table is an example of a single tree attribute file with statistical data after inputting a reference point file (in CSV format):

```
TreeID, TreeLocationX, TreeLocationY, Azimuth, Distance
1, 322716.24, 4102494.69, 36.165, 8.982
2, 322751.21, 4102499.9, 41.282, 4.491
3, 322519.35, 4102499.3, 32.008, 4.708
```

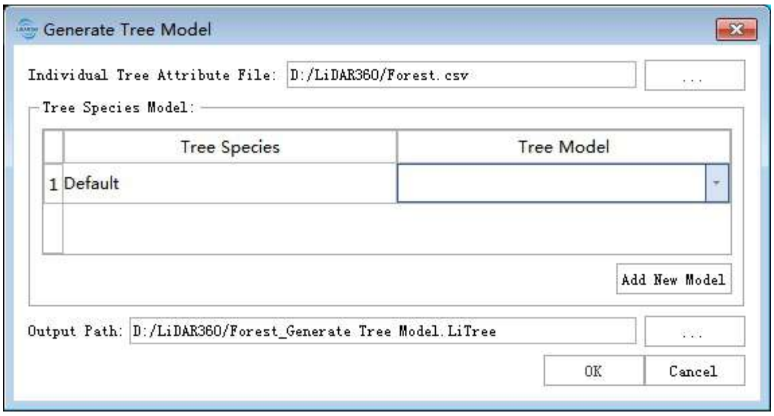
Generate Tree Model

Functional Overview

The real tree model is constructed based on the results of segmentation or attribute editing. It can provide a more intuitive and real model display form for the attribute information, and quickly check and browse the segmentation results. At the same time, the high-precision and high-fidelity tree model enhances the sense of reality and immersion for the digital city, woodland and other scenes.

Usage

Click *ALS/TLS Forest* > *Generate Tree Model*.



Parameters Settings

- **Individual Tree Attribute File**: The input file is the point cloud individual tree attribute file. The individual tree attribute file format can refer to the individual tree segmentation result file; The individual tree attributes file can also come from the individual tree attributes file with the tree species information after the Tree Species Marker.
- **Tree Species Model**: According to the tree species information contained in the selection file, select the tree model that shows the tree species in the tree model drop-down box.
- **Add New Model**: Add the tree model as required, set the model corresponding to the perspective display picture and close shot model object, tree model management reference option Settings in the forest Settings page of the tree model settings
- **Output Path**: Output path, set the storage path corresponding to the generated tree modeling file, and generate a folder with the same name in this path to save the resource file used by the tree modeling file. When we are copying the tree model file, copy the resource folder with the same name.

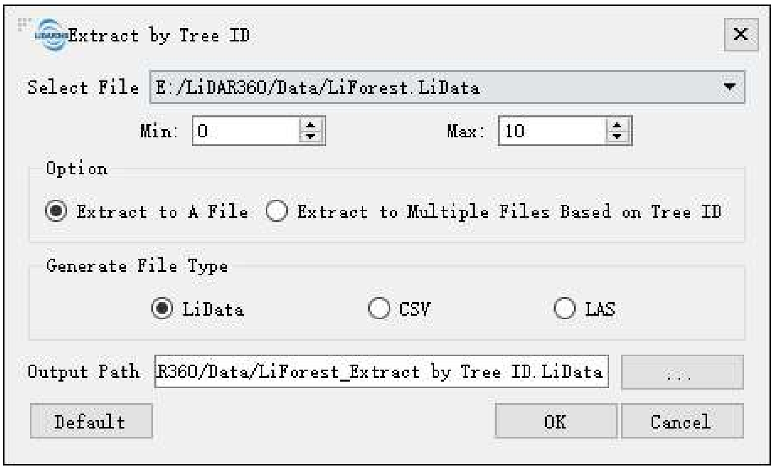
Extract by Tree ID

Summary

This function is used to extract part or all point clouds from the segmented point cloud based on Tree ID. When exporting the data into one file, the optional formats include LiData, CSV, and LAS. When exporting the data into multiple data, the optional format will be CSV file only.

Usage

Navigate to *ALS Forest* > *Extract Point Cloud by Tree ID*.



Settings

- **Select File**: Select the point cloud data with segmented tree ID information from the drop-down list. Only one file can be selected at a time, and it has to be opened in LiDAR360 already.
- **Min (default value is "0")**: The minimum value of tree ID to be extracted. The default value is zero.
- **Max**: The maximum value of tree ID to be extracted. The default value is the number of trees in the point cloud. If the point cloud has not been segmented, the minimum and maximum values of the tree ID are both zero.
- **Extract to A File (default)**: The point cloud in the selected range is extracted into a CSV, LiData, or LAS file that contains the X, Y, Z coordinates and tree ID information.
- **Extract to Multiple Files Based on Tree ID**: Extract the point cloud data based on tree ID and save them as different CSV files for each tree. The stored information are X, Y, Z coordinates and tree ID information for each tree.
- **Output Path**: Output path for the results.

Forest Structure Change Detection

Summary

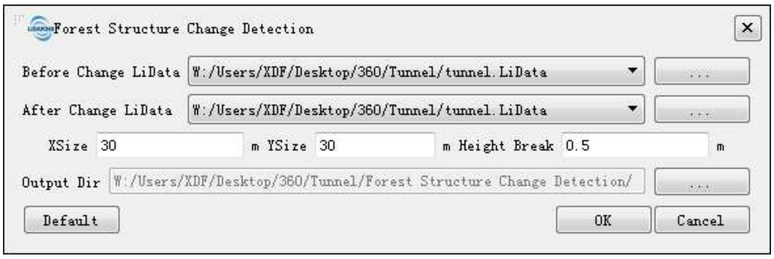
The LiDAR PAC (profile area change) metric, as a simple and integrated method, demonstrated promising potential in characterizing fine-grained changes in forest structure. The method can be beneficial for forest managers in evaluating fire-induced environmental and economic losses, and provide useful information for forest restoration design.

Principle Description

First, this function divides the data into different grids according to the XSize and YSize set by the users. For each grid, normalize the height value of all the points to the range of 0 to 1. And then, sort the points by height and calculate its cumulative height percentile. Calculate the height value for each integer cumulative percentile from 0 to 99% (100 values in total). Then fit the 100 cumulative height percentiles to the profile curve, and calculate the profile area (PA) delineated by the percentile profile curve and the X axis. After calculating the PA value of the curve area before and after the change, use the PA after the change minus the PA value before the change to get the PAC value.

Usage

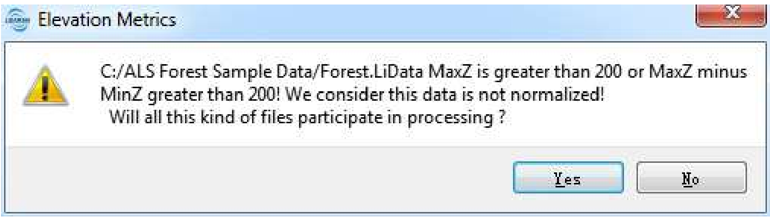
Click *ALS Forest > Forest Structure Change Detection*



Settings

- **Input**: Make sure that all the input files are normalized. The input file(s) can be a single file or a file set. The file(s) to be processed must be opened in LiDAR360 before using this function.
- **XSize (meter) (default value is "30")**: The grid size in X direction. This value should be larger than the crown size of the trees. For the data acquired from most of the forest, this value should be larger than 15.
- **YSize (meter) (default value is "30")**: The grid size in Y direction. This value should be larger than the crown size of the trees. For the data acquired from most of the forest, this value should be larger than 15.
- **Height Threshold (meter) (default value is "2")**: The threshold to divide the data in vertical direction. The point cloud above this height will be used to calculate the forest metrics. The default value is 2.
- **Output Path**: Path of the output file. Two PA images (before and after the change), a PAC image, and a report will be generated.

- **DefaultValue**: Restore the default parameters.

> Note: The selected point cloud data must be normalized ALS data.
>
> 
>

```
    @inproceedings{
        author={Tianyu Hu,Qin Ma,Yanjun Su,John J.Battles,Brandon M.Collins,Scott L.Stephens,Maggi Kelly,Qinghua Guo},
        title={A simple and integrated approach for fire severity assessment using bitemporal airborne LiDAR data},
        booktitle={Int J Appl Earth Obs Geoinformation,78(2019): 25-38},
        year={2019}
    }
```

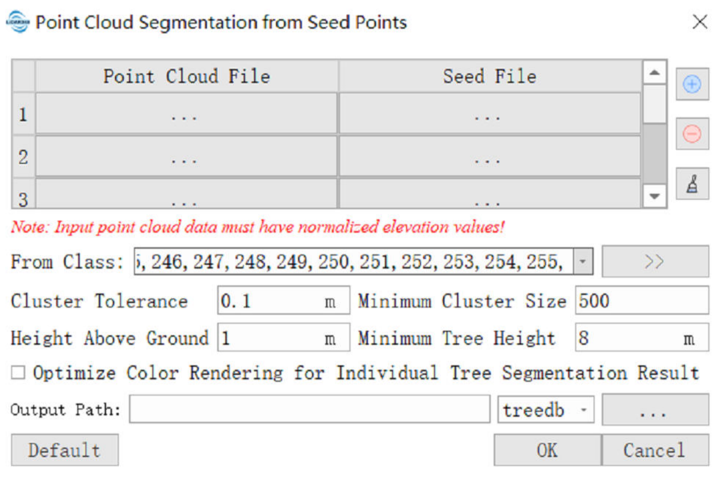
Individual Tree Segmentation and Matching

Functional Overview

When point cloud segmentation is required or the data area requiring fine tree segmentation is large, the point cloud needs to be divided into multiple blocks and distributed to multiple data processing personnel for collaboration. When usingthe block by rangeorblock by polygontool to block point cloud data,it is inevitable that the same tree in the block boundary area will be divided into two or more parts. In order to improve the accuracy of the data after the overall segmentation, buffer parameters are set according to the actual situation of the data during the partitioning. After each piece of data is segmented. You can remove the same tree in the data buffer area of adjacent point clouds.with this function.

Usage

Click*ALS Forest > Individual Tree Segmentation and Matching*, organize the results after point cloud segmentation.



Parameters Settings

- **Buffer**: This parameter is used to find the most matched polygon information in vector data for each point cloud data. It needs to ensure that the value is correct; otherwise, the correctness of the result sorting will be affected. This value needsto be the same as the buffer parametersetin the block-by-rangeor block-by-polygon function.
- **Input Data**: Please ensure that each input point cloud data isis the data after point cloud segmentationortree segmentation based on seed points.
- **Point cloud file**: Click ... to select the point cloud data to be processed.
- **Seed point file**: Click ... to select the seed point file corresponding to the point cloud data.
- : Five data can be processed by default. Click to increase the number of files to be processed.
- : Deletes the selected point cloud and corresponding seed point files.
- : Deletes the file list.

ALS Editor

Description

The ALS seed point editing tool is used to check the results of individual tree segmentation. At the same time, it allows for manual interactive editing such as adding or deleting seed points. Based on the edited seed points, the point cloud can be segmented again to improve the accuracy of individual tree segmentation. Since version 3.2, right-click shortcut menus are supported.

ALS Seed Point Editing Shortcut Keys

|Shortcut Key|Description|

|:----:|:--:|

|Shift + Mouse Left Button|Section translation|

|↑ |Move section up |

|↓ |Move section down |

|→ |Move section right |

|← |Move section left |

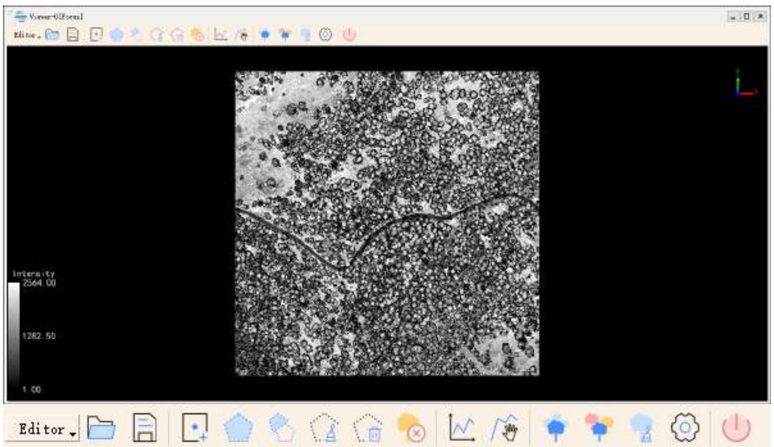
|Ctrl + z |Undo seed point editing|

|Ctrl + y |Redo seed point editing

Usage

In an active window, load the ALS point cloud data to be edited.

Navigate to and click on *ALS Forest* > *ALS Editor*.



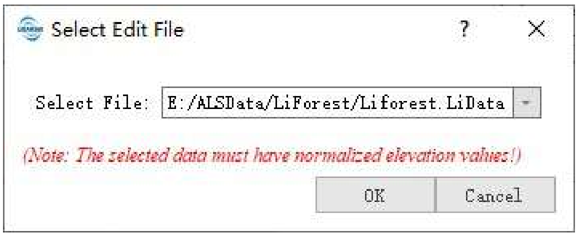
The ALS Editor toolbar options, from left to right, are: Start/End Edit, Open Seed Point File, Save Seed Point File, Add Seed Points, Select Seed Points, Deselect Seed Points, Cancel Selection, Delete Selected Seed Points, Clear All Seed Points, Profile, Filter Trees, Point Cloud Segmentation Based on Seed, Clear

TreeID, Seed Setting, and Exit.

Start/End Edit

Click *Editor* > *Start Edit*.

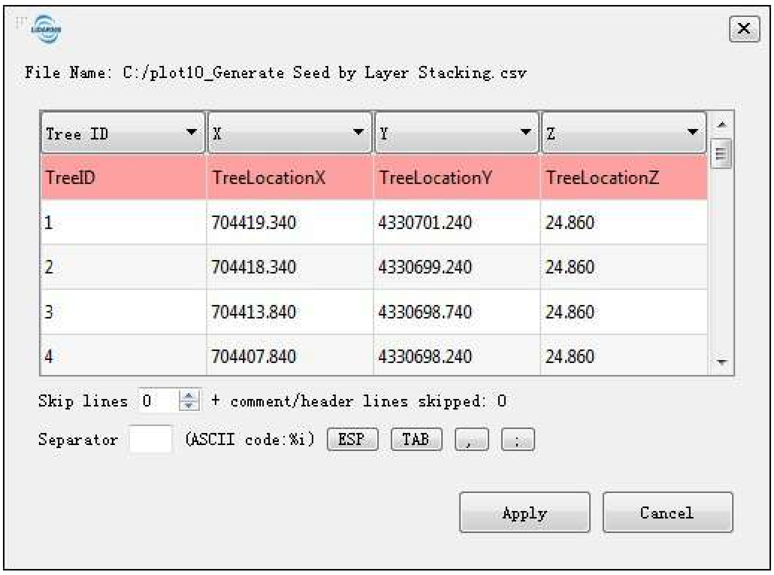
Select the data to be edited (this should be a normalized point cloud and only one dataset can be edited at a time). Click "OK", and then other functions on the ALS Editor toolbar will be available.

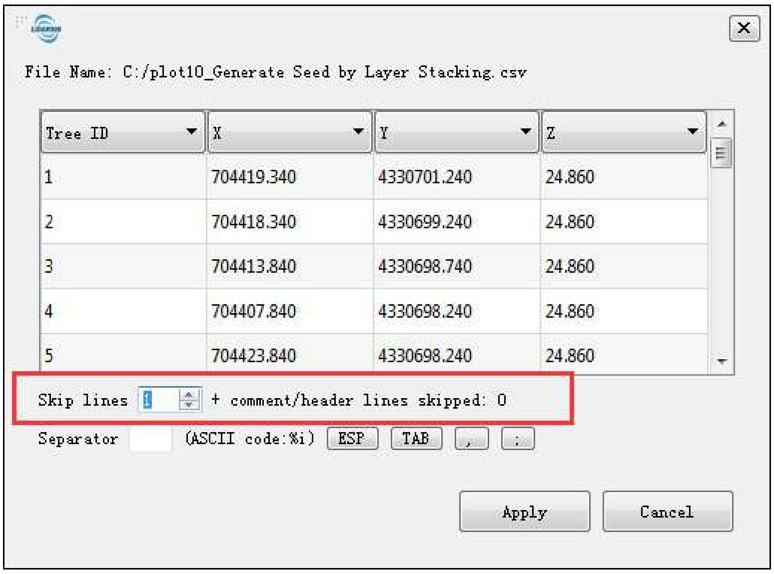


During the editing process the selected data cannot be removed from the window. When editing is completed, click "End Edit". Other toolbar functions will be unavailable at this point.

Open Seed Points File

Brief: Select a seed point file (please refer to Seed Points File in the appendix for the correct format) and the window below will pop up. Please note that there is a header line in the seed point file, therefore the first line should be ignored when the file is opened.





Save Seed Points File

Brief:When the editing is completed, the seed points can be saved as a new CSV file without overwriting the original file.

Add Seed Points

Brief: Add seed points manually in under-segmented areas. Users may be prompted to select the peak - or the point close to the peak - of a tree as the seed point.

Select Seed Points

Brief: Select seed points for editing.

Delete Selected Seed Points

Brief: This tool can be used to deselect incorrect seed points which may have been selected in error.

Cancel Selection

Brief: Cancel the selection of seed points.

Delete Selected Seed Points

Brief: For areas that are over-segmented, incorrect seed points can be selected and deleted by

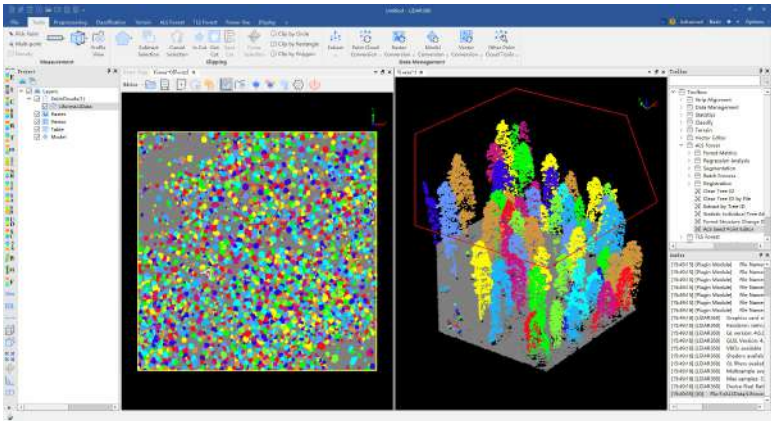
Brief: For areas that are over-segmented, incorrect seed points can be selected and deleted by using the **Select** tool.

Clear All Seed Points

Brief: Remove all seed points in the window.

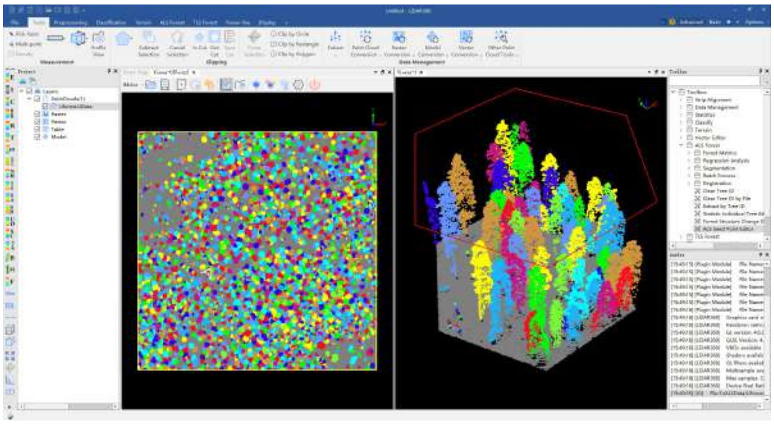
Profile

Brief: When the ALS Editor toolbar is opened, the point cloud window will change to 2D display mode. The profile tool can show if the seed points are accurate in 3D. Click the Profile Tool button and a new window will be created. Click the mouse left button to select a polygon and all points within the polygon will be displayed in the new window in 3D.



Tree Filter

Description：After point cloud data is segmented, attributes such as tree height and crown area can be obtained. The tree filter tool can be used to view (highlight), hide, delete, and export single-tree results within a specified attribute range. For example, setting the view of trees within a certain height range or viewing trees with larger or smaller crown areas to check the results of single-tree segmentation (larger crown areas may have under-segmentation while smaller ones are generally caused by dead trees or over-segmentation). To use this function, the segmented point cloud and segmentation result table must be loaded into the software. As shown in the figure below: filter point clouds by tree height, set minimum and maximum values to 2.01 and 10 meters respectively, and point clouds within this range will be highlighted.



Point Cloud Segmentation Based on Seed

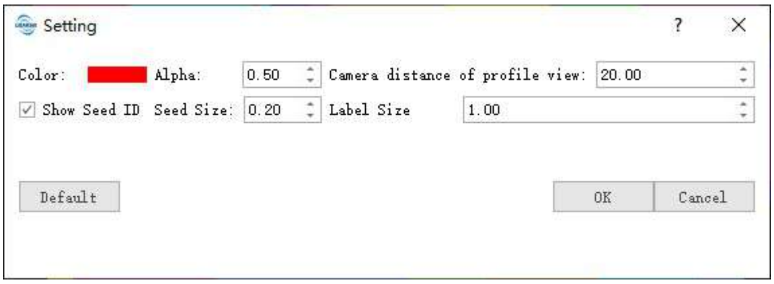
Brief: Segment the point cloud data based on the edited seed points. Refer to PCS with Seeds.

Clear Tree ID

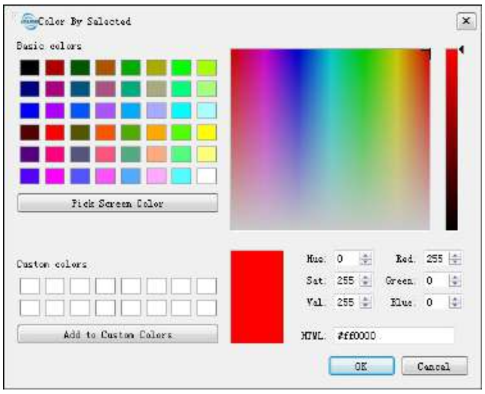
Brief: If the point cloud data has been previously segmented, the Tree ID should be removed prior to performing individual tree segmentation again. Refer to Clear Tree ID.

Seed Setting

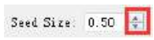
Brief: Users can set the color, alpha and size of the seed points, and may choose to show the seed point ID and its label size.



- **Color (default color is "Red")**: Click , and then the dialog for selecting colors will pop up as shown below. Any color may be selected as the seed point color from this window.



- **Alpha (default value is "0.5")**: Transparency of the seed point, ranging from 0 to 1, with 0 being completely transparent and 1 being opaque. The default value is 0.5. Click  and the value will be incremented or decremented by 0.1, or the user can directly enter a specific value.

- **Show Seed ID (optional)**: Use the check box to determine if the seed point ID is displayed in the window.
- **Seed Size (default value is "0.5")**: The seed size, ranging from [0,100). Click  and the value will be incremented or decremented by 1, or the user can directly enter a specific value.
- **Label Size (default value is "1")**: The label size, ranging from [0, 100). Click  and the value will be incremented or decremented by 1, or the user can directly enter a specific value.

Exit

Brief: To exit ALS Editor, click the exit button and a prompt window below will pop up. Click “Yes” to close the ALS Editor toolbar or click “No” to return to the editor window.

> Note: If the point cloud has been segmented, you need to Clear Tree ID prior to segmenting it again. If segmentation has not been performed, the message "All files have been segmented, please clear tree ID first!" will be displayed.

Undo/Redo

Description: During the seed point editing process, if there is an editing error or misoperation, press [Ctrl+Z] to undo the operation, and press [Ctrl+Y] to redo the operation.

> Note: The undo and redo functions are limited to 20 times each in a single seed point editing session.

Individual Tree Editor

Functional Overview

Individual Tree Editor can extract and edit individual trees in large data points cloud, and can quickly locate and filter out user-concerned individual trees. It is mainly used to solve over-segmentation, under-segmentation and missing segmentation and other cases, and can also repair fine of under-story trunk.

Usage

Load the point cloud data needs to be edited in the window and make sure to activate that window.

Click *TLS Forest* > *Individual Tree Editor* or *ALS Forest* >*Individual Tree Editor* to activate the editing bar in current window. Some functions of different modules maybe different.

The buttons in the individual tree editor tool bar, from left to right, are: start/end editing, profile, open the individual tree attributes file, individual tree filter, settings and exit.

Please note that if there is no special need, you should normalize the point clouds which need to be edited. Please refer to Normalization. After editing is complete, the original point cloud can be restored by reverse denormalization. The profile window takes the plane with Z value of 0 as the datum, and for the normalized point cloud, the datum is the ground. For the unnormalized point cloud, the point cloud is far from the datum due to the influence of elevation, which may cause the point cloud and the cut polygon may not exist in the profile view. Therefore, normalized point cloud data can be better edited. In addition, functions such as removing support brackets and re-segmentation under this module need normalized point clouds to get correct results.

Start/End Editing

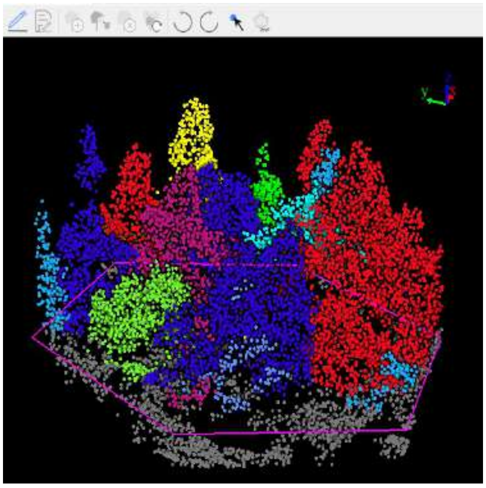
Click *Edit* > *Start Editing*

Select the data to be edited and click "OK" to render the point cloud as displayed by tree id. Other functions of the individual tree point cloud editing tool bar will become available.

The selected data cannot be removed from the window during editing. After editing, click "End Edit" to end the editing.

Profile

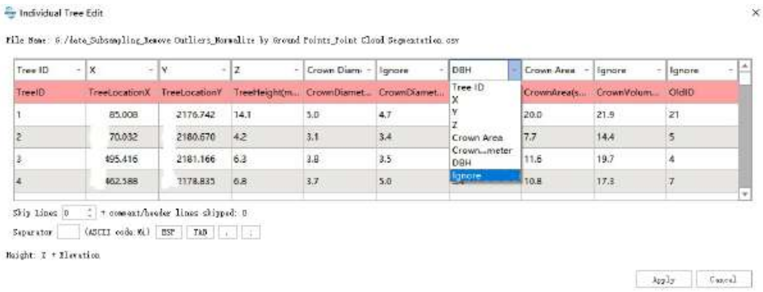
 **Functional Description**: Click the "Profile" button to open the profile window and draw a hexagonal area in the window where the point cloud is located. This area is the individual tree point cloud area to be edited. The individual tree editor toolbar appears above the profile window.From left to right: Load/remove edit area, Save, Create individual tree, Merge individual tree, Delete individual tree, Re-segment/remove support bracket, Back up, Redo, Select point, Hide tree.



> Note: 1. Please pay attention to the individual tree boundary when picking up the hexagonal area, because the incomplete part may affect the result, or try to avoid modifying the boundary when editing. None of the editing functions in this module will have an effect on hidden trees, so you can hide trees you don't want to edit to avoid affecting the results.
>
> 1. Since version 5.0, the method of drawing the profile area is modified to: Click the left mouse button, drag, and release the mouse to complete the drawing of the profile area.

Open Individual Tree Attributes File

Functional Description: Individual tree attributes file is a CSV file generated after individual tree segmentation. Please note that this function recognizes and skips several rows of the table header by default. If necessary, it can be adjusted by itself. Before clicking apply, ensure that the table header field is consistent with the corresponding column in the CSV file.

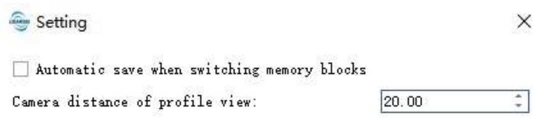


Individual Tree Filter

Functional Description: Before performing individual tree filtering operations, open the individual tree attributes file. This function can be set according to the parameter range, quickly screen interested individual tree. When the filter is enabled, the editor's main window sets the filtered individual tree color in the scene to transparent, and the profile window automatically applies the filter to hide the corresponding individual tree in the edit area. Currently, there are two filtering modes. Trees that match the filtering range are kept display in display mode, while other trees will be hidden. In hidden mode, the effect is reversed.

Settings

Functional Description: This includes profile distance, auto save and other settings.



- **Profile Distance**: When drawing the profile, please observe the distance between the point and the display area in the profile window for the users to view the editing area or editing effect according to the distance conveniently. For example, for ALS forest, the section distance can be set to hundreds of meters. And for TLS Forest, it can be set to tens of meters.
- **Auto Save**: It is used to auto-save the edited area when switching the profile areas.

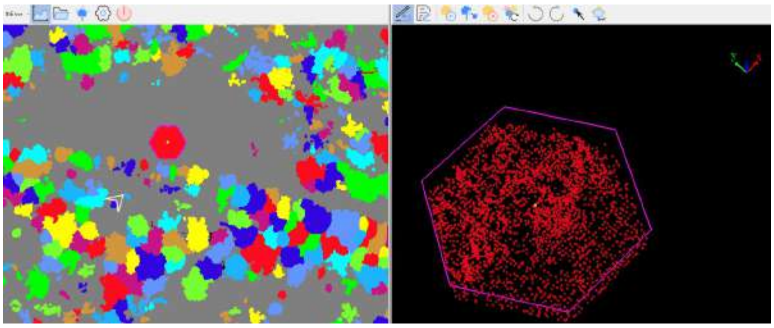
Load/Remove Editing Area

Functional Description: Click *Load Editing Area* button to set the area in the hexagon as the area to be edited. The data for editing the individual tree point cloud is the data in this area. All editing tools are available in the profile window. Enter the editing state at this time, selects the section area in the main window again, it will automatically choose the data in the selected area to be the new editing data.

Click “Remove Editing Area” button, and this action will exit the editing directly if no operation is done to the point cloud file; Otherwise the system will prompt you whether to save the point cloud data. Choose yes to save the point cloud data to a file, and you can see that the large data point cloud in the main window has been modified. Or you can choose no to exit the individual tree point cloud that is being edited without saving the data.

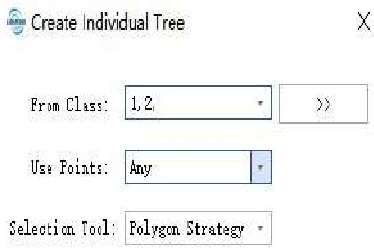
Save

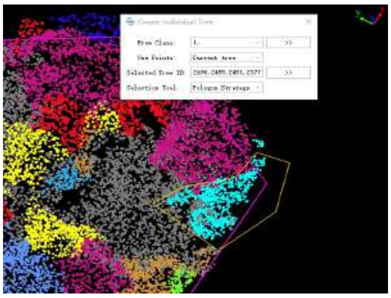
Functional Description: Click the "Save" button to save the modified individual tree point cloud data to the file. At the same time, you can see that the big data point cloud in the main window has been modified accordingly. Back up data if necessary, because it cannot be restored after being saved.



Create Tree

Functional Description: This function can effectively extract the selected point cloud and set it as the new tree ID. Click the "Create Tree" button and will appears the window shown as below. Select the category to create the tree point in the category drop-down list box, choose the category to participate in the point cloud segmentation from the filter point drop-down list box,select the whole category by default, select the scene job tool from the selection tool, select the polygon by default, then select the area in the profile window, double-click the end, you can see the color of the selected area changed, which means the tree is created. This function is often used in conjunction with other functions, such as creating a new tree ID for a point below a certain height and deleting it to remove the ground portion, or making an individual tree in a selection to be a new individual tree and segmenting it again.

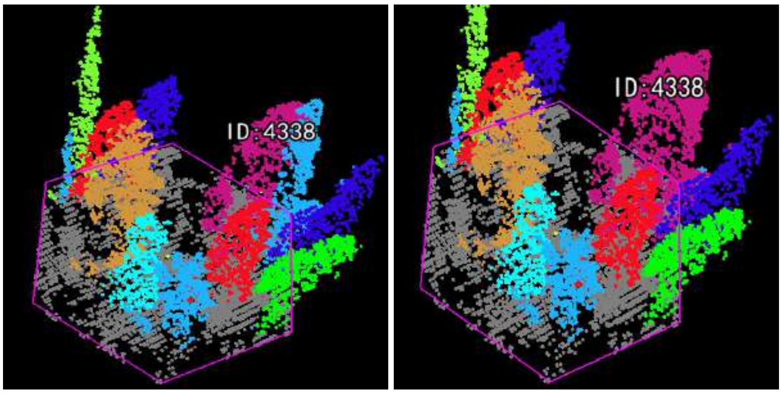




- **From Class**: Only specific classes of points are used to filter . Please note that the tree ID has nothing to do with the category, and it does not have to be classified as a tree to have a tree ID.
- **Use Points**: Only the points with specific tree IDs are used, avoiding making changes to other trees. There are three options:
  - **Any**: Applicable to all visible points.
  - **Current Tree**: Applicable only to the point cloud corresponding to the selected tree ID in the list. When selecting this option, pick the tree ID in the window to the input list by activating the pick tool. After picking, select "Pick seed point" to switch back to this function.
  - **No Tree ID**: Applicable only to the point with tree ID of 0.
- **Selection Tool**: Select the points within the area by different selection tools.

Merge Tree

Functional Description: It is used to manually solve the over-segmentation problem. Click "Merge Tree" button, use the mouse to pick up tree in the profile window, if the picked tree ID is not 0, mark the tree ID at the mouse position.Then pick up other trees, and they will be combined into the first picked individual trees. You can see that in the profile window, the color of the picked trees will change to the color of the first picked individual trees, right-click the mouse to end the pickup.

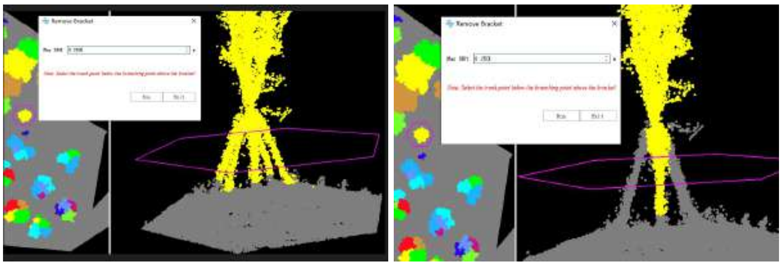


Delete Tree

Functional Description: Click "Delete Tree" button and use mouse to pick up trees in the profile window. If the picked tree’s ID isn’t 0, you can mark it as 0 and delete the tree.

Remove Bracket

Functional Description: This function only exist in the individual tree editor of TLS Forest, and can remove the support brackets of the tree trunk by semi-automatic interaction. Click the left mouse button to select a point on the trunk below the branch point above the support bracket as the reference point, process the point cloud below the height of this point, and set the tree ID of the point cloud corresponding to the support bracket to 0. The principle of the algorithm is the trunk and branches grow at the same time according to the height, and remove the growing branches. Therefore, when the lowest height of the support bracket is the same as the lowest height of the trunk, the result will be better.

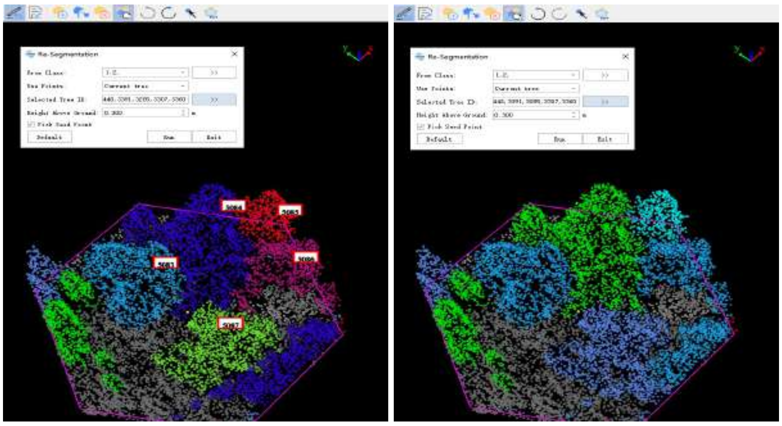


- **Max DBH**: Maximum DBH of the extracted trunk. Appropriate DBH settings can better extract the trunk. If the value is smaller, the extracted trunk will be finer, and if it is larger, more support bracket point clouds will be retained. When you open the individual tree attributes file and click the reference point, the DBH displayed on the input box will automatically update to a slightly larger DBH value for the corresponding tree in the file.

Re-segmentation

Functional Description: This function can only solve the problem of over-segmentation and under-segmentation of trees through semi-automatic interaction in the editing of the individual tree editor area under the ALS Forest. The algorithm process spatial search to the point clouds based on the seed points picked up in the view, and one seed point generates at most one individual tree. For over-segmentation, a point with obvious spatial characteristics (have a certain distance from other trees and easy to distinguish, generally the top of the tree) can be picked as the seed point, and a point closely related to this point in space will be classified as a new individual tree. For under-segmentation, multiple seed points need to be picked up and multiple single trees are generated by the same algorithm at each seed point position.

Click “Re-segmentation” button, and the function dialog box will pop up, click the left mouse button to pick up points as the seed points in the profile after setting the corresponding parameters, click "Run" to re-segment the point clouds in the editing area. The segmented tree IDs contain the biggest tree ID of the big data point cloud and the IDs above it, the tree segmented by this function will not be saved to the individual tree attributes file. Before using this function, you can hide the trees that don’t need to be divided, set the usage category and set of points to obtain higher precision segmentation effect.



- **From Class**: Only specific classes of points are used to filter. Please note that the tree ID has nothing to do with the category, and it does not have to be classified as a tree to have a tree ID.
- **Use Points**: Make targeted editing based on the point set of the selected tree ID. It is usually used for under-segmentation of an individual tree or under-segmentation of multiple individual trees. There are three options to use set of points:
    - **Any**: Applicable to all visible points.
    - **No Tree ID**: Applicable only to the point with tree ID of 0.
    - **Current Tree**: Applicable only to the point cloud corresponding to the selected tree ID in the list. When selecting this option, pick the tree ID in the window to the input list by activating the pick tool. After picking, select "Pick seed point" to switch back to this function.
- **Minimum Height**: Filter points below the minimum height to prevent dividing points on the ground. When the individual tree point cloud and the low grass bush on the ground are jagged and interleaved, the tree generated by the algorithm will grow downward and outward, resulting in

segmentation errors. This phenomenon can be effectively avoided when the parameter is set above the height of the ground low grass bush.

- **Pick Seed Point**: You can pick up seed points under the checked state. If you switch to tools such as "Pick Point" and "Measurement", the check box will be automatically unchecked. At this time, you can use these tools without quitting the function. If you manually check it, you can switch back to the mode of picking seed points.

Undo

Functional Description: Click *Undo* button to undo the last option. The maximum times of undoing is 20. Any operation before the last 20 steps cannot be undone.

Redo

Functional Description: Click *Redo* button to redo the last option. The maximum times of undoing is 20. Any operation before the last 20 steps cannot be undone.

Pick Point

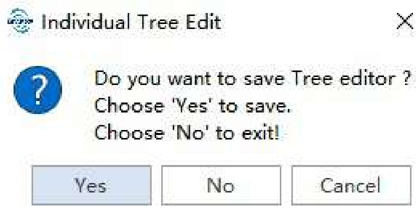
Functional Description: Click the "Pick Point" button and use the mouse to pick up the point cloud in the profile window. The attribute information of the point will be displayed.

Hide Tree

Functional Description: Click "Hide tree" button, use the mouse to pick up the point cloud in the profile window, and hide the tree with the corresponding tree ID of the point in the scene. The hidden tree can avoid being modified.

Exit

Functional Description: Click the "Exit" button to close the profile window and exit from individual tree editor directly if no editing operation has been done; Otherwise the system will prompt you whether to save the data. If you select Yes to save the data in the edit area to the file, you can see that the big data point cloud in the main window has been modified. Or you can select No to exit the editing area directly without saving the data. At the same time, close the profile window and exit from individual tree editor .


Individual Tree Edit
Do you want to save Tree editor ?
Choose 'Yes' to save.
Choose 'No' to exit!
Yes
No
Cancel

TLS Forest

The TLS Forest modules are used for terrestrial or backpack LiDAR point cloud data processing. The main functions include: leaf area index, ground points filters, point cloud segmentation, single tree segmentation based on seed points, clear tree ID, statistics and increase individual tree attributes, seed point editing, individual tree point cloud editing, DBH measure, tree species marker, etc.

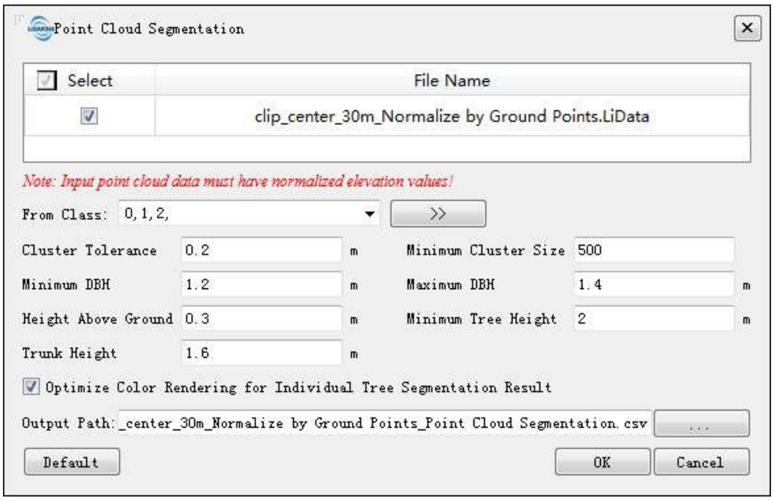
Point Cloud Segmentation

Function Overview

Unlike the top-down segmentation method for airborne LiDAR point clouds, terrestrial LiDAR point clouds are typically obtained from below the canopy. This allows for clear identification of tree trunks and segmentation of individual trees, as described by Tao et al. (2015). In this case, the diameter at breast height (DBH) of individual trees can be measured.

Usage

Click *TLS Forestry > Point Cloud Segmentation*.



Parameters

- **Input Data**: The input data must be normalized point cloud data. For normalization methods, refer to Normalization or Normalization by Ground Points. Input files can be a single point cloud data file or a point cloud dataset; the data to be processed must be opened in the LiDAR360 software.
- **From Class**: The starting class for point cloud segmentation. By default, all classes are selected.
- **Clustering Threshold (meters) (default: "0.2")**: This parameter allows users to control the efficiency and accuracy of individual tree segmentation. It controls the detection of individual trees and the growth of the tree canopy point cloud. A larger clustering threshold increases segmentation efficiency but may negatively affect segmentation results if set too high.
- **Minimum Clustering Size (default: "500")**: This value mainly affects the growth of the tree canopy point cloud. Smaller values improve segmentation results but slow down the process; larger values have the opposite effect.
- **Height Above Ground Points (meters) (default: "0.3")**: Only point clouds above this value are included in individual tree segmentation. This parameter helps reduce the impact of ground point cloud thickness or vegetation on segmentation results. Setting this value too high may affect the accuracy of detecting tree diameters.
- **Minimum Tree Height (meters) (default: "2")**: This parameter can be adjusted based on local tree growth conditions to filter out small trees.

- **Trunk Height (meters) (default: "1.6")**: The algorithm will segment point clouds within the range from **Height Above Ground Points** to **Tree Trunk Height** to detect the tree trunk, serving as the starting point for tree growth. It is recommended to set this parameter at the branch-free height.
- **Optimize Coloring Rendering for Segmentation Results (default selected)**: By rearranging the ID information after individual tree segmentation, this option helps resolve issues with adjacent trees being assigned the same color.
- **Output File Type**: Choose the type of output file. Options include CSV table files and tree attribute files.
- **Output Path**: After running, each point cloud data will generate corresponding segmentation results. The results are either comma-separated CSV files or tree attribute files, containing tree ID, x, y coordinates, tree height, DBH, crown diameter, crown area, and crown volume attributes. For the file format, refer to Ground-based Point Cloud Individual Tree Segmentation Result File Format. For viewing segmentation results, refer to Viewing Point Cloud Segmentation Results.
- **Default**: Reset parameters to default values.

@inproceedings{ author={Tao S L, Wu F F, Guo Q H, Wang Y C, Li W K, Xue B L, Hu X Y, Li P, Tian D, Li C, Yao H, Li Y M, Xu G C and Fang J Y}, title={Segmentation tree crowns from terrestrial and mobile LiDAR data by exploring ecological theories}, booktitle={ISPRS Journal of Photogrammetry and Remote Sensing, 110:66-76}, year={2015} }

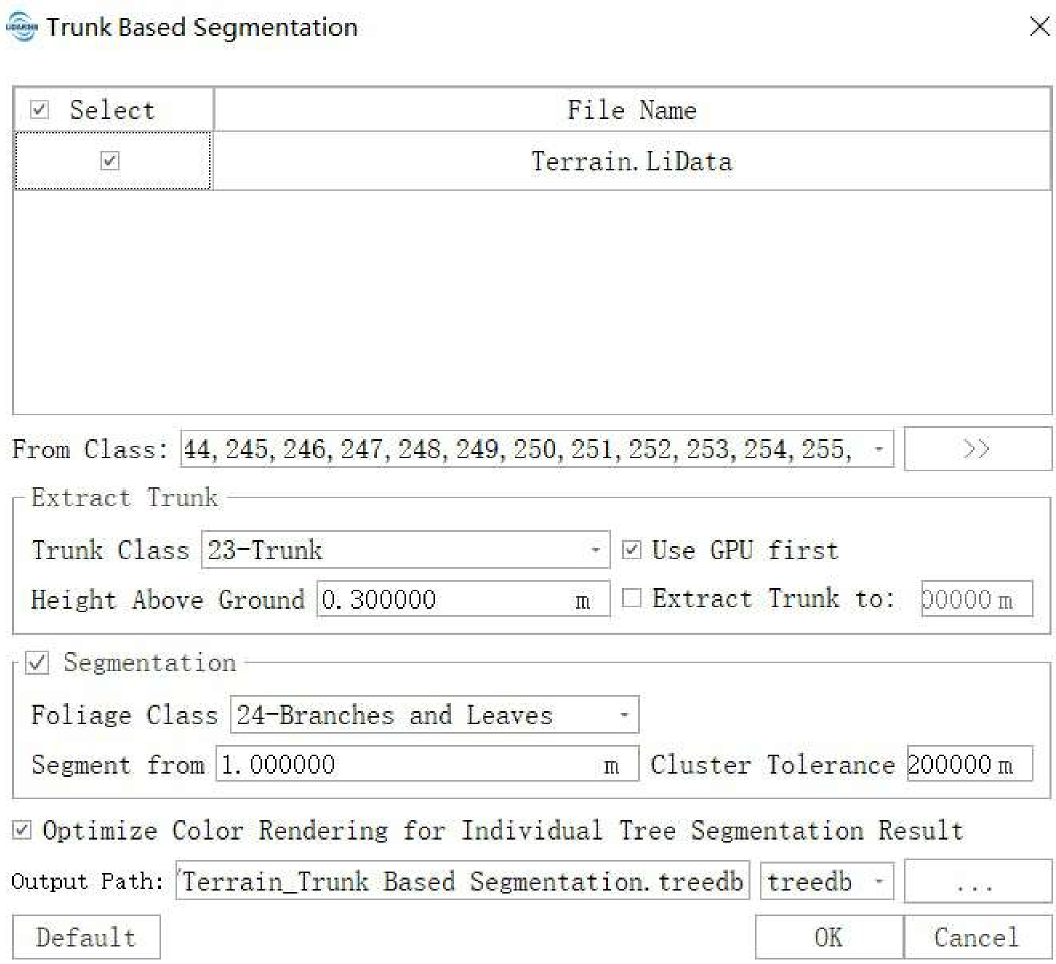
Trunk Based Segmentation

Function Overview

Unlike Point Cloud Segmentation, this function divides point cloud segmentation into two separate steps: "Stem Extraction" and "Point Cloud Segmentation". The first step uses deep learning to extract the tree stems (López Serrano et al., 2022). The second step grows the rest of the canopy from these stems. This method is less affected by underbrush compared to TLS Point Cloud Segmentation and provides more accurate stem counts, especially for plantations with DBH greater than 10 cm.

Usage

Click *Ground Forestry > Trunk Based Point Cloud Segmentation*.

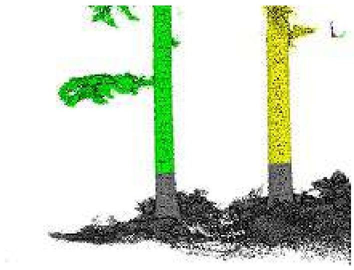
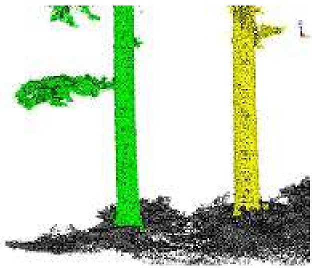
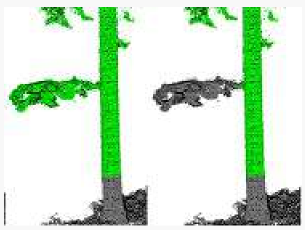


Parameter Settings

- **Select Data**: The input data must be normalized point cloud data. For the normalization method, refer to Normalization or Normalization by Ground Class. The input file can be a single point cloud data file or a point cloud dataset; the data to be processed must be open in the LiDAR360 software.
- **From Class**: The initial class for point cloud segmentation, with the default set to all classes.

- **Extract Trunk**: The necessary step for stem extraction.
    - **Trunk Class (default "23")**: The segmented stems will be marked with this class.
    - **Height Above Ground (meters) (default "0.3")**: Only point clouds higher than this value will be used for stem extraction. This parameter reduces the impact of ground point cloud thickness or weeds on individual tree segmentation. Setting this value too high may affect the accuracy of detecting tree diameters.
- **Segmentation (default "Checked")**: When checked, stems will only be segmented up to 5.0m.
    - **Foliage Class (default "23")**: The segmented canopy will be marked with this class.
    - **Cluster Threshold (meters) (default "0.2")**: Adjusting this parameter controls the efficiency and accuracy of individual tree segmentation. The value controls the detection of individual trees and the growth of canopy point clouds. A larger threshold increases segmentation efficiency but may negatively affect segmentation results.
    - **Segment From(meters) (default "1.0")**: Only point clouds higher than this value will be used for canopy segmentation. Points below this height remain part of the stem. This parameter further reduces the impact of ground point cloud thickness or weeds on individual tree segmentation. Setting this value too high may exclude low canopies from segmentation, affecting the determination of branch height. Setting it too low will fail to effectively reduce the impact of underbrush on segmentation results.
- **Optimize Display Colors for Individual Tree Segmentation Results (default checked)**: Reorganizes the ID information of segmented trees to significantly reduce the assignment of the same color to adjacent trees.
- **Output Path**: Each point cloud data file will generate corresponding segmentation results. The results are in a comma-separated CSV file, including tree ID, x and y coordinates, tree height, DBH, canopy diameter, canopy area, and canopy volume attributes. For more details, refer to the appendix on Individual Tree Segmentation Result File Format. To view the segmentation results, see View Point Cloud Segmentation Results.
- **Default Values**: Resets the parameter settings to their default values.

Impact of Parameters on Results:

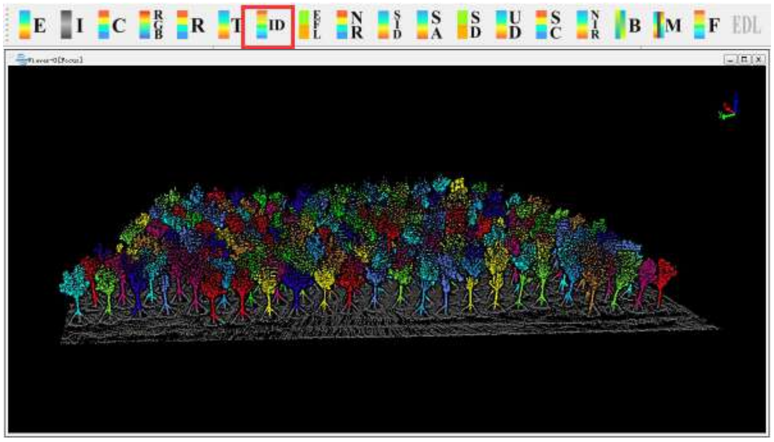
Parameter	Effect
Height Above Ground	 
Initial Segmentation Height	

```
@inproceedings{
```

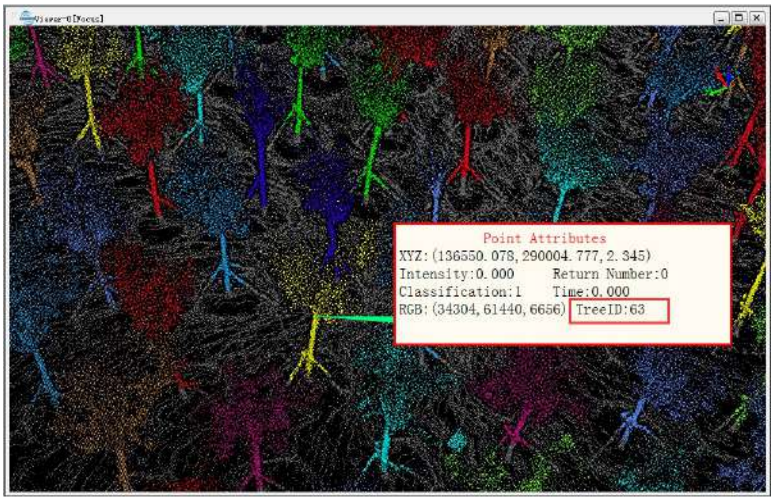
```
        author={López Serrano et al., 2022 F.R. López Serrano, E. Rubio, F.A. García Morote, M. Andrés Abellá
n, M.I. Picazo Córdoba, F. García Saucedo, E. Martínez García, J.M. Sánchez García, J. Serena Innerarity, L.
Carrasco Lucas, O. García González, J.C. García González},
        title={Artificial intelligence-based software (AID-FOREST) for tree detection: A new framework for fa
st and accurate forest inventorying using LiDAR point clouds},
        booktitle={Int J Appl Earth Obs Geoinf., 113 (2022), Article 103014},
        year={2022}
    }
```

View the Point Cloud Segmentation Results

After segmenting the trees, Tree IDs are saved as an attribute to each point in the point cloud data. The results can be viewed within a window viewer. Load the point cloud used for the tree segmentation into a 3D viewer. Ensure the viewer is active and press the Display by Tree ID button in the toolbar. The following is an example of a point cloud colored by individual tree. TLS Editor tools are used for checking the TLS individual tree segmentation results. Moreover, it can be used to edit operations such as add, delete seed points, and segment the point cloud data based on the edited seed points.

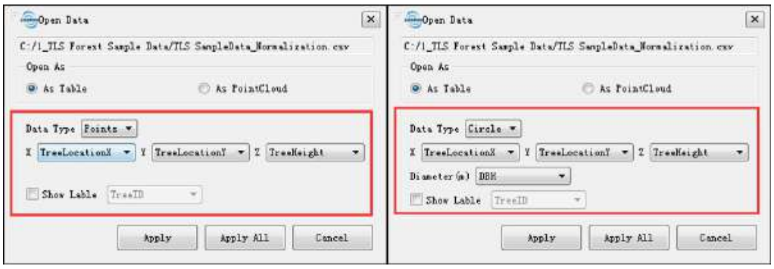


The tree ID attribute of each point can be queried by the Pick Point ( ) tool on the toolbar.

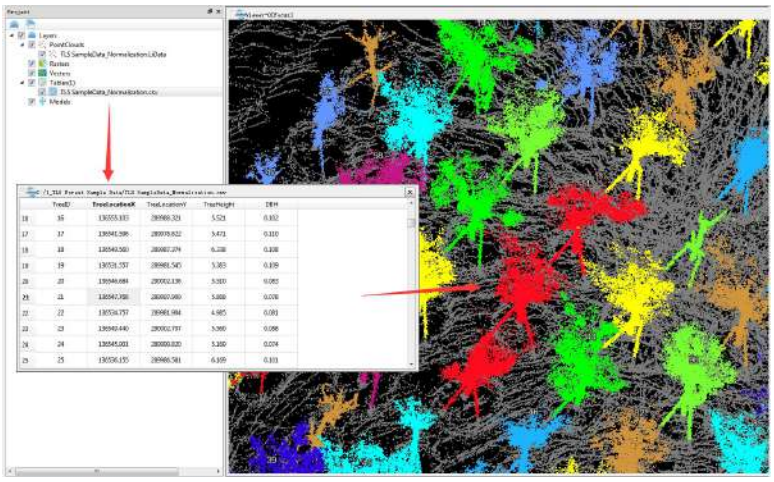


CSV file generated by segmentation can be overlaid with the point cloud, and the CSV file can be opened as a table. Select X, Y and Z as the TreeLocationX, TreeLocationY and TreeHeight respectively in the CSV file as shown below, and check the Show Label (if the label has too much text and blocking other data, one can remove the data and reopen it without checking the Show Label). Click “Apply” to load the CSV file in the software.

To view the DBH, CSV can also be selected to be displayed by circle and select DBH as the diameter.



Click the mouse right button on the CSV file and select Table Attribute. The property information will be displayed. Double-click the mouse left button anywhere in each line, and it will jump to the corresponding location. The following figure shows the effect of superimposed display of point cloud data and CSV file.



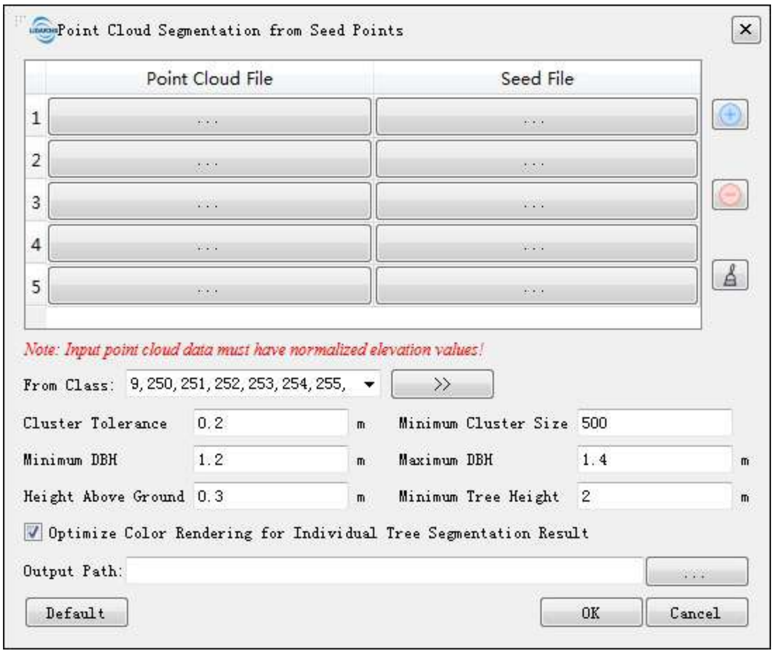
Point Cloud Segmentation from Seed Points

Functional Overview

The TLS PCS with Seeds function supports batch processing for multiple files. The input data includes normalized point cloud data and the corresponding seed point file. For TLS point cloud normalization, please refer to the Normalize by DEM or Normalize by Ground Points tool section of the LiDAR360 User Guide.

Usage

Navigate to and click on *TLS Forest* > *Point Cloud Segmentation from Seed Points*.



Parameters Settings

- **From Class**: Classes which participate in the PCS with seeds (all classes by default).
- **Point Cloud File**: Click ... to select the point cloud data to be processed.
- **Seed File**: Click ... to select the seed file.
- : Five datasets can be batch processed per tool run. Click to add files to be processed.
- : Delete the selected point cloud and seed points file.
- : Clear the file list.
- **Cluster Tolerance (m) (default is "0.2"):** Users can control the efficiency and accuracy of single tree segmentation by adjusting this parameter. The larger the cluster threshold, the higher the efficiency of single tree segmentation, but if it is too large, it will affect the segmentation effect.
- **Minimum Cluster Size (default is "500"):** This value mainly affects the growth of single tree crown point cloud. The smaller the number of points set, the better the segmentation effect but slower speed and vice versa.
- **Height Above Ground (meter)(default value is "2")**: Only the points above this hight will be

involved in individual tree segmentation. This parameter is used to decrease the influence of ground points and weeds to the segmentation. It will influence the accuracy of the detection of trunk, if this value is too large.

- **Minimum Tree Height (meters) (default: "2")**: This parameter can be adjusted based on local tree growth conditions and is used to filter out smaller trees.
- **Optimize Color Rendering for Individual Tree Segmentation Result (checked by default)**: By reorganizing the tree ID generated after the individual tree segmentation, it can greatly solve the problem of rendering the same color to the trees next to each other. Note: if choosing to optimize the color rendering, the tree IDs in new csv file for individual tree segmentation are not one-to-one correspond to those in the input seeds files.
- **Output File Format**:Select the format of output file, choice including .csv attribute table and tree attribute file.
- **Output Path**: Path of the output file, which is a comma-separated database table in the .csv format containing the ID of each individual tree identified during the segmentation process, the x, y coordinate of each individual tree, individual tree heights, DBHs, crown diameters, crown areas, and crown volumes.
- **DefaultValue**: Reset each parameter to the default value.

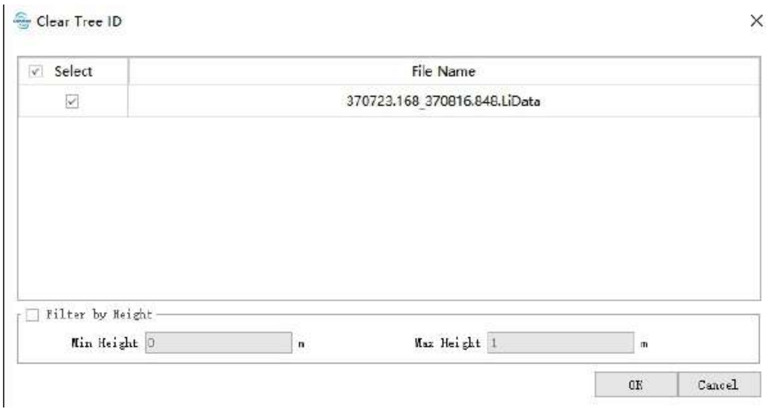
Clear Tree ID

Summary

After the point cloud has been segmented, the tree ID information is stored in a LiData file. If the segmented point cloud data needs to be segmented again, users need to clear the tree ID first.

Usage

Navigate to *TLS Forest* > *Clear Tree ID*.



Select the point cloud data to be processed, and then click "OK".

Parameters

- **Filter by Height**: Determines whether to filter by height. If this option is checked, it will clear the tree ID information for points between the minimum and maximum heights. If unchecked, it will clear the tree ID information for all points.
- **Min Height (default "0")**: The minimum elevation for clearing tree IDs.
- **Max Height (default "1")**: The maximum elevation for clearing tree IDs.

> Note: Checking the height filter option does not recalculate the tree ID information for the point cloud; it only removes the tree ID attribute for points within the specified height range. To re-separate the point cloud, you need to uncheck the filter by height option.

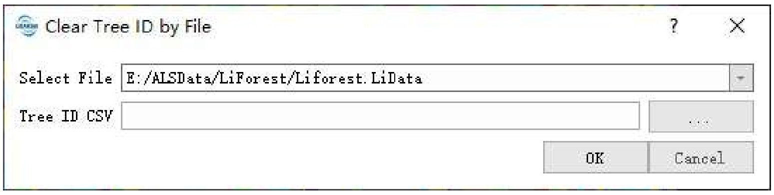
Clear Tree ID by File

Summary

After point cloud segmentation, there may be incorrectly segmented trees. The IDs of these incorrectly segmented trees can be recorded using a CSV file, which can then be used to clear the incorrect tree IDs from the point cloud.

Usage

Click *ALS Forestry* > *Clear Tree ID by File*.



Select the point cloud data to be processed and click **OK**.

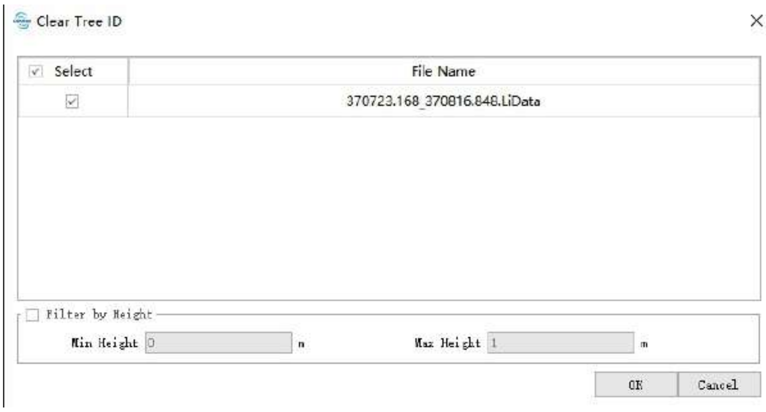
Clear Tree ID

Summary

After the point cloud has been segmented, the tree ID information is stored in a LiData file. If the segmented point cloud data needs to be segmented again, users need to clear the tree ID first.

Usage

Navigate to *TLS Forest* > *Clear Tree ID*.



Select the point cloud data to be processed, and then click "OK".

Parameters

- **Filter by Height**: Determines whether to filter by height. If this option is checked, it will clear the tree ID information for points between the minimum and maximum heights. If unchecked, it will clear the tree ID information for all points.
- **Min Height (default "0")**: The minimum elevation for clearing tree IDs.
- **Max Height (default "1")**: The maximum elevation for clearing tree ID.

Extract Points by Range/CSV File

- **Extract by Range/CSV File**: Extract point clouds within the selected treeID range. The range for treeID can be entered manually or read from a single-tree attribute file.
- **Extract to One/Multiple Files**: Export the point cloud to a single file, with file formats including CSV, LAS, and LiData. The exported information includes X, Y, Z coordinates, tree ID, and other attribute information.
- **Output Path**: The storage path for the point cloud extracted by tree ID.

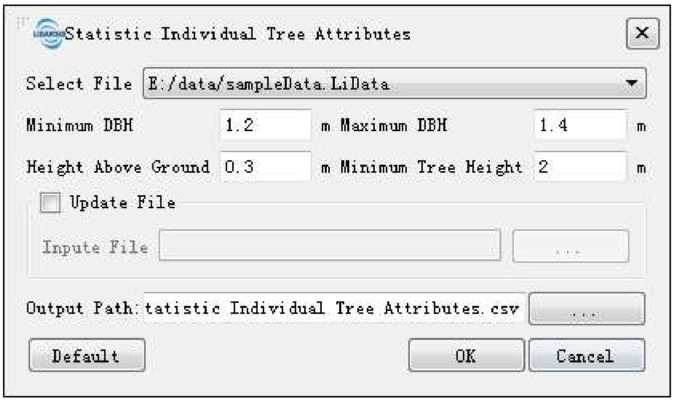
Statistic Individual Tree Attributes

Description

This function is mainly used to perform attribute statistics and attribute expansion on point clouds that have already been normalized and segmented into individual trees, generating individual tree attribute files. It can create new files or update existing ones.

Usage

Click *Ground Forestry > Individual Tree Attribute Statistics*.



Parameters

The parameters are divided into two pages. The first page allows you to select the target file and the attributes to be calculated, and the second page allows you to set specific statistical parameters.

- **First Page**
    - **New File**: Generates a new individual tree attribute file. For the specific file format, refer to Individual Tree Segmentation Result File Format.
    - **Update File**: Updates an existing individual tree attribute file, mutually exclusive with "New File". If the attribute file does not contain the selected attributes, they will be added.
    - **Attribute Selection**: At least one individual tree attribute needs to be selected for statistics.
- **Second Page**
    - **Input Point Cloud**: Select the point cloud to be analyzed. The point cloud needs to have undergone individual tree segmentation.
    - **Height Above Ground (meters) (default is "0.3")**: Only point clouds above this height will be analyzed.
    - **Minimum Tree Height**: Only individual trees taller than this value will be analyzed.
    - **Curvature and Trunk Volume**: This option is displayed only when stem curvature, cutting analysis, or stem volume is selected. Refer to Trunk Curvature Calculation for parameters.
    - **Input DEM (optional)**: This option appears when "Elevation" is selected for statistics.
    - **Reference Point File**: The reference point file is a comma-separated text file with two rows, containing three fields: X, Y, and Radius. X and Y are the coordinates of the reference point, and Radius is the radius. If a reference point file is provided, the azimuth and distance of trees within

the radius centered on the reference point will be calculated and added as extended attributes to the individual tree attribute file.

- **Slope File**: This option appears when "Slope" is selected. The slope file is in TIFF format. If a slope file is provided, the slope at the tree location will be extracted from the file and added as an extended attribute to the individual tree attribute file.
- **DEM File**: This option appears when "Elevation" is selected. The DEM file is in TIFF format. If a DEM file is provided, the ground height at the tree location will be extracted from the file and added as an extended attribute to the individual tree attribute file.

Below is an example of a reference point file:

```
X,Y,Radius
322716.24,4102494.69,15.0
```

Below is an example of an individual tree attribute file:

```
TreeID, TreeLocationX, TreeLocationY
1, 322716.24, 4102494.69
2, 322751.21, 4102499.9
3, 322519.35, 4102499.3
```

Below is an example of an individual tree attribute file after importing the "Reference Point File" and performing statistics (in CSV format):

```
TreeID, TreeLocationX, TreeLocationY, Azimuth, Distance
1, 322716.24, 4102494.69, 36.165, 8.982
2, 322751.21, 4102499.9, 41.282, 4.491
3, 322519.35, 4102499.3, 32.008, 4.708
```

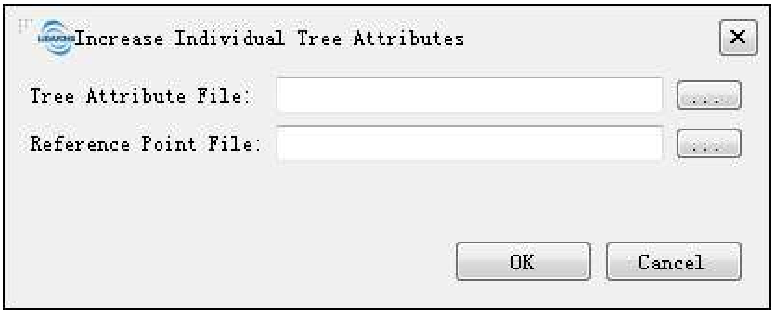
Increase Individual Tree Attributes

Summary

This function is mainly used to expand the attribute field of individual tree attribute file.

Usage

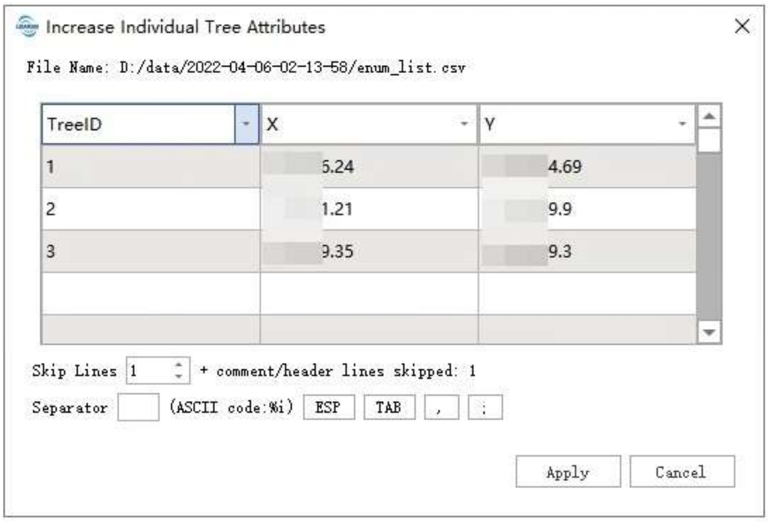
Click *TLS Forest* > *Increase Individual Tree Attributes*.



Settings

- **Individual Tree Attributes File**: The input data is a individual tree attribute file, which is a comma-separated CSV table, and must contain at least two fields: TreeLocationX, TreeLocationY, the detailed file format can refer to Individual Tree Segmentation Result File Format。
- **Reference File**: The reference point file is a comma-separated text file with two lines and three fields: X, Y, and Radius; X and Y are reference point positions, and Radius is radius. If the reference point file is input, the azimuth and distance of the trees within a certain radius with the reference point as the center are calculated, and then added to the individual tree property file as extended attributes.
- **Slope File**: The slope file is in TIFF format. If the slope file is imported, the slope of the tree position is obtained and added to the single tree attribute file as an extended attribute.
- **DEM File**: The DEM file is in TIFF format. If the DEM file is imported, the slope of the tree position is obtained and added to the single tree attribute file as an extended attribute.

After clicking OK, you need to set the fields of the reference point file. The three fields of X, Y, and Radius need to be set correctly, and the other fields are set to Ignore. For the setting method, please refer to the section on selecting reference Open file to add TXT data.



The following is an example of reference point files:

```
X,Y,Radius
322716.24,4102494.69,15.0
```

The following is an example of individual tree property file:

```
TreeID, TreeLocationX, TreeLocationY
1, 322716.24, 4102494.69
2, 322751.21, 4102499.9
3, 322519.35, 4102499.3
```

The following is an example of the extended individual tree property file:

```
TreeID, TreeLocationX, TreeLocationY, Azimuth, Distance
1, 322716.24, 4102494.69, 36.165, 8.982
2, 322751.21, 4102499.9, 41.282, 4.491
3, 322519.35, 4102499.3, 32.008, 4.708
```

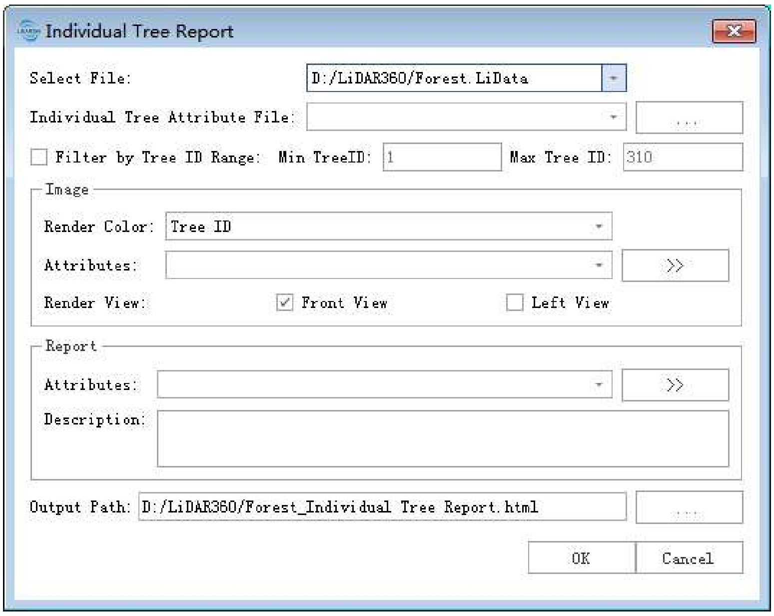
Individual Tree Report

Functional Overview

This function is based on the result file of individual tree segmentation, extracts the image of individual tree point cloud from the point cloud with individual tree segmentation, and displays the image of individual tree point cloud and individual tree attribute information to the user in the form of html report, which is convenient for users to quickly check and archive the result of individual tree segmentation.

Usage

Click *TLS Forest* > *Individual Tree Report*.



Parameters Settings

- **Select File**: Select the point cloud data to generate the individual tree report from the drop-down list of selected files. Only one file can be selected at a time. The file must be opened in the LiDAR360 software.
- **Individual Tree Attributes Files**: Select the point cloud individual tree attribute file. For the format of individual tree attribute file, please refer to the format of individual tree segmentation result file;
- **Extract by Tree ID**: If this parameter is selected, the tree ids for the report range from the minimum number to the largest tree ID. The tree ids can be queried in the selected CSV file.
- **Minimum Tree ID**: The default value of the minimum value of the tree ID that generates the report is 1.
- **Maximum Tree ID**: The default value is the maximum value of the tree ID in the point cloud.
- **Color Rendering**: Select the rendering method to generate the individual tree point cloud image. You can select three rendering methods: point cloud RGB rendering (optional when point cloud has RGB attributes), tree ID rendering and elevation rendering.
- **Image Drawing attributes**: Select the attribute information drawn on the individual tree point cloud

image. The attribute information is derived from the selected individual tree segmentation result file.
- **Render viewing**: Select a view mode for generating images. The front view, left view, and two views can be displayed at the same time.
- **Report attributes**: Select the property information displayed in the html report from the selected individual tree segmentation result file.
- **Description information**: User-defined description displayed by the user in the html report.
- **Output Path**: The path for storing the generated html report.

Note

This function is only applicable to the TLS point cloud data after individual tree segmentation. The ALS point cloud may be sparse, so the picture of individual tree point cloud in the report cannot directly show the individual tree shape.

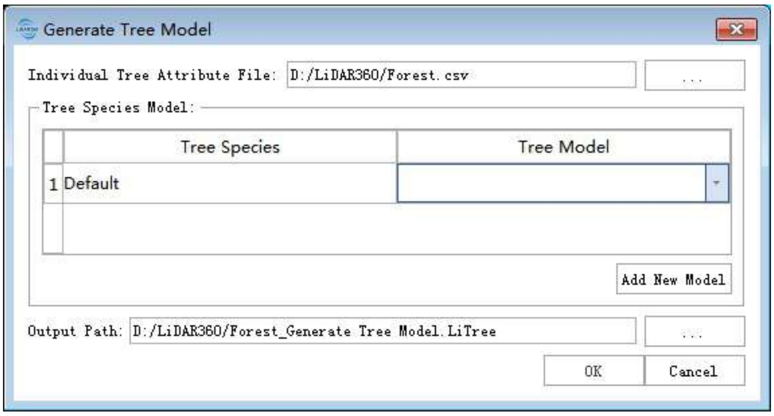
Generate Tree Model

Functional Overview

The real tree model is constructed based on the results of segmentation or attribute editing. It can provide a more intuitive and real model display form for the attribute information, and quickly check and browse the segmentation results. At the same time, the high-precision and high-fidelity tree model enhances the sense of reality and immersion for the digital city, woodland and other scenes.

Usage

Click *ALS/TLS Forest* > *Generate Tree Model*.



Parameters Settings

- **Individual Tree Attribute File**: The input file is the point cloud individual tree attribute file. The individual tree attribute file format can refer to the individual tree segmentation result file; The individual tree attributes file can also come from the individual tree attributes file with the tree species information after the Tree Species Marker.
- **Tree Species Model**: According to the tree species information contained in the selection file, select the tree model that shows the tree species in the tree model drop-down box.
- **Add New Model**: Add the tree model as required, set the model corresponding to the perspective display picture and close shot model object, tree model management reference option Settings in the forest Settings page of the tree model settings
- **Output Path**: Output path, set the storage path corresponding to the generated tree modeling file, and generate a folder with the same name in this path to save the resource file used by the tree modeling file. When we are copying the tree model file, copy the resource folder with the same name.

Tree Species Identification

Function Description

The tree species identification superimposes the point cloud and seed points according to the panoramic photos, determines the tree type and records it in the corresponding seed point file.

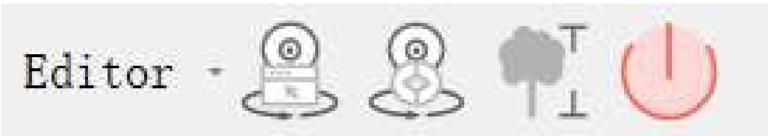
> Note: The point cloud must be the original point cloud and cannot be the normalized point cloud, and the elevation of the seed point file cannot be the normalized elevation.

Usage

Load the point cloud data to be edited into the window, and use this window as the active window, then open the panorama window, and the panorama photo will be superimposed to display the point cloud and seed points, and determine the tree species corresponding to each seed point.

Click *Foundation Forestry>Tree Species Identification*, and the tree species editing tool bar will appear in the current active window.

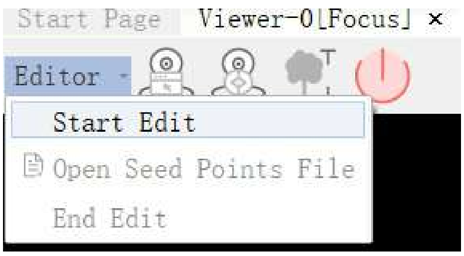
Tree species ID editing tool bar, from left to right: start/end editing, panorama, selection frame, single tree attribute window switch, exit.



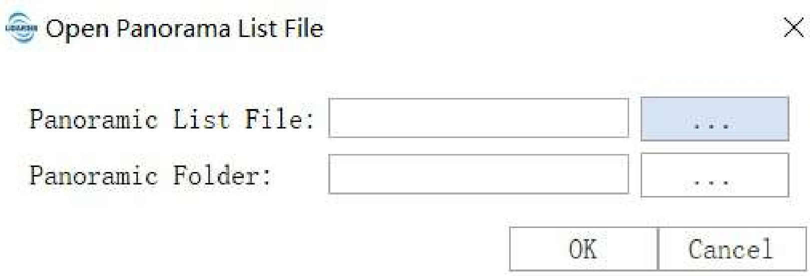
Start/End Editing

Click *Edit* > *Start Editing*

Select the data to be edited, click "OK", other functions of the tree species identification editing tool bar will become available, a single tree property window will pop up at the bottom of the window, and a dialog box for loading panorama will pop up at the same time.



During editing, selected data cannot be removed from the window. After editing, click End Editing to end the editing.



Load seed point file

Click *Edit* > *Load Seed Point File*

Select the seed point file data to be loaded, the seed point file loading window will pop up, determine the attributes of each column of the seed point, click Apply, the seed point data will be displayed in the singular attribute window, and will be added to the point cloud window at the same time.

During editing, selected data cannot be removed from the window. After editing, click End Editing to end the editing.

Panoramic window

Function description

Click the *panorama window* button to open the panorama window, and a dialog box for loading panorama files will pop up. By default, the point cloud of the main window will be added and displayed, and the panorama toolbar will appear above the panorama window. From left to right, they are: previous frame, next frame Frame, panorama roaming, point cloud display range, point cloud transparency.

select frame ### **Function Description**

Click the *Select Frame* button, use the mouse to pick up the point cloud in the point cloud window, and the panorama will automatically jump to the panorama photo closest to the pick-up position.

Single tree property window switch

Function description

Click the *Single Tree Property Window Switch* button to control whether the single wood property window is displayed.

> For the description of individual tree property measurement, please refer to Single Wood Property Measurement。

Exit

Function description

Click the *Exit* button to exit the tree species identification module.

Previous frame

Function Description

Click the *Previous Frame* button, and the panorama jumps to the previous frame.

Next Frame

Functional Description

Click the *next frame* button, and the panorama jumps to the next frame.

Automatic Roaming

Functional Description

Click the *Auto Roam* button, the panorama will browse all the next frames from the current frame according to a certain time interval until the last one Frame stop, or click the *Auto Roam* button again to stop roaming.

Only display selected seed points

Functional Description

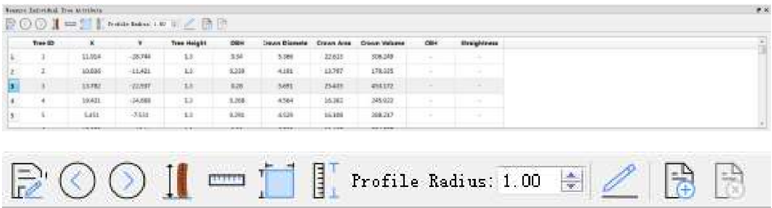
Click *Only display selected seed points* button, the panorama window will only display the selected seed points in the single tree attribute measurement window , the other seed points are hidden.

- **Panorama Point Cloud Conical Display Radius**: Adjust the radius of the conical range for point cloud and seed point identification in the panorama window.
- **Point cloud transparency**: Adjust the scroll bar for point cloud transparency, the point cloud becomes transparent to the left until it is invisible, and the point cloud becomes more opaque to the right until the point cloud is clearest.

Individual Tree Attributes

Functional Overview

Function description: Click this button to open the profile window to measure the properties of a single tree, including tree height, branches Attributes such as bottom height and tree straightness can be saved, and the edited single-tree attribute information file can be saved. When the single tree attribute editing state is turned on, when the seed point of the window changes, the single tree attribute table information will not be updated. If it needs to be updated, it should be closed and reopened.

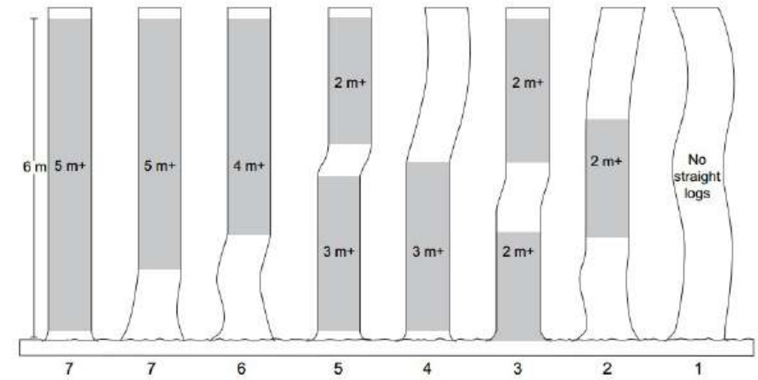


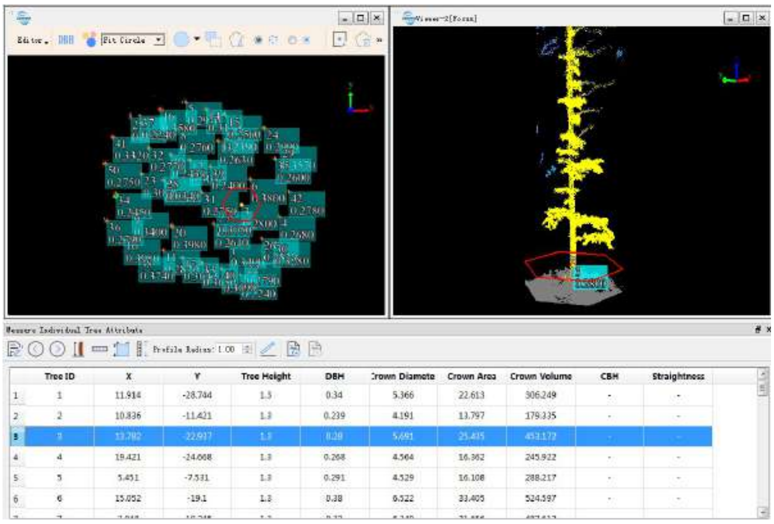
: Click this button to change the edited Single wood attribute information is saved as a csv file.

: This tool needs to be used in section window mode, jumps to the previous double-clicked single wood information item Single wood information, the section window also displays the jumped single wood point cloud, you can also use the shortcut key ← to jump forward.

: This tool needs to be used in section window mode, jumping to the next step of the single-wood information item currently selected by double-clicking Single wood information, the section window also displays the jumped single wood point cloud, and you can also use the → shortcut key to jump backwards.

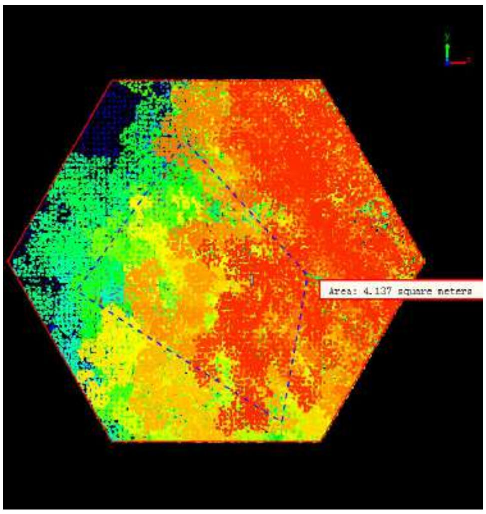
: Calculates tree straightness. It is necessary to use the selection tool in the section window to select the point cloud whose straightness needs to be calculated. According to the relevant standards for tree straightness calculation proposed by Macdonald E et al., the height of the selected point cloud must be greater than 6m, if the calculated straightness information is not refreshed, you can select the cell to right-click to refresh.





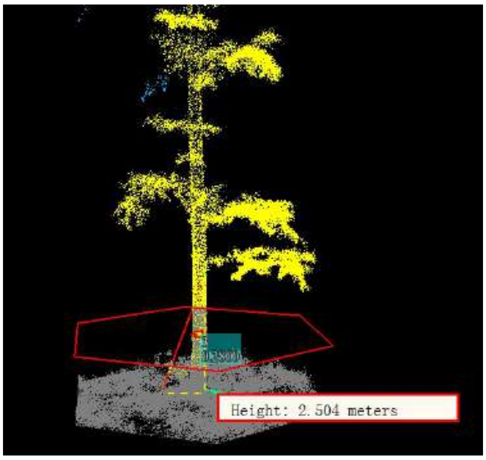
: Length measurement tool, you can measure tree height related information in the section window.

: Area measurement tool, can measure the crown area, crown diameter and other related information in the section window. The crown diameter can be obtained by first measuring the crown area, and then using the area measurement formula S=πr² to obtain the crown diameter 2r.



: Height measurement tool, in the section window, you can measure the tree height, the height of the branches and other related information.

the branches and other related information.



: start editing, you can edit the added fields after editing.

: add fields, add extended fields in the property sheet, support integer, real number, text, date, enumeration, etc. type.

: delete the field and delete the added extension field.

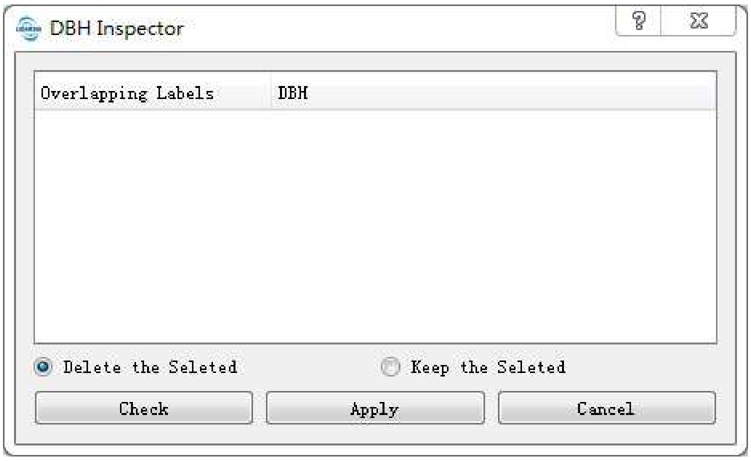
- **Section Radius**: Adjust the hexagon radius in the section window.
- **Display Attribute**: Adjust the displayed seed point identification, you can display any column of information in the singular attribute measurement in the scene, and control whether the seed point identification is displayed.
- **Jump Tree ID**: Jump to the seed point position corresponding to the tree ID. If you open a section, the section point cloud will jump to the vicinity of the corresponding seed point. When you open the panorama window, jump to the corresponding panoramic photo.

> Note: Before measuring the properties of a single tree, it is necessary to ensure that the seed points are included in the window. In addition, there is no refresh phenomenon for table-related data. You can use the right-click of the selected cell or use the F5 shortcut key to refresh the relevant table data.

DBH Check

Function description: The DBH check tool is used to check whether there is intersection or overlap of DBH. Users can double-click or right-click to view the checked DBH, and delete the wrong DBH by right-clicking or selecting.

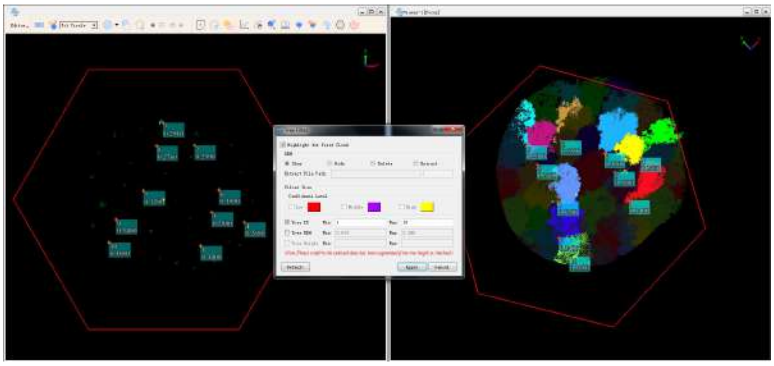
After the function is turned on, the following figure is shown:



Single Tree Screening

Function description: The single-tree screening tool is used to check and edit the DBH fitting results or single-tree segmentation results. According to the screening range, the DBH fitting results can be displayed, hidden, deleted and extracted, and the point cloud after single tree segmentation can be highlighted. Filtering operations include: filtering by confidence, filtering by tree ID range, tree DBH filtering, and filtering by tree height range. Among them, filtering by confidence is only applicable after batch fitting DBH; filtering by tree height is only applicable after single-tree segmentation operation.

As shown in the figure below: After the point cloud data is segmented for a single tree, filter the point cloud by tree ID, set the minimum and maximum values to 1 and 10 respectively, and the point clouds within this range will be highlighted.



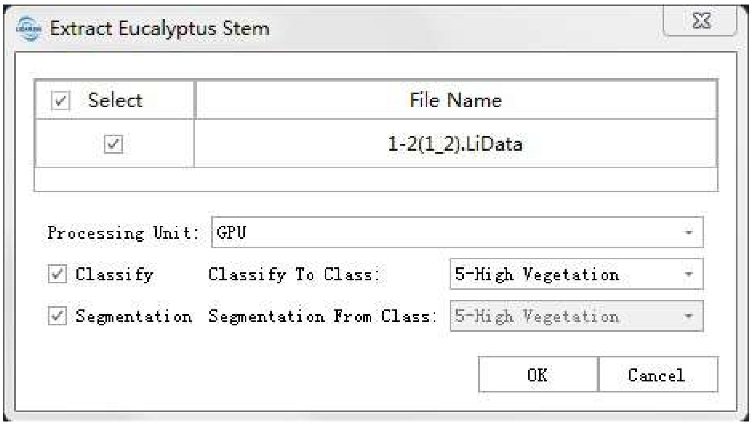
Extract Eucalyptus Stem

Functional Overview

This function will identify tree stem from Eucalyptus TLS forest point cloud data and divide them into target categories.

Usage

Click *TLS Forest* > *Extract Eucalyptus Stem*.



Parameters Settings

- **Input data**: The input file can be a single data file or a point cloud data set; the file(s) to be processed must be opened in the LiDAR360 software.
- **Computing Unit**: You can choose GPU or CPU.
- **Classification Target Class**: The user must select at least one function in classification and segmentation. After checking the classification and running, the stem will be classified into the target class.
- **Segmentation Source Class**: When the user selects segmentation, the source class trees will be segmented into individual trees. When the classification is checked, the target class of the classification will be set as the source class of the segmentation. Otherwise, the user needs to specify the segmentation source class.

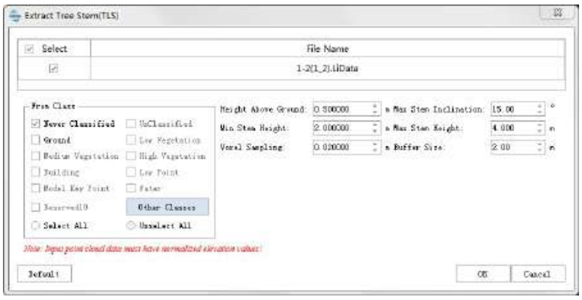
Extract Tree Stem(TLS)

Functional Overview

This function is used to extract stems from terrestrial laser scanning point cloud data. Point clouds belonging to the same stem will be given a unique tree ID. This function is mainly aimed at the point cloud data obtained by high-precision station scanning to solve the complex problem of forest land vegetation, and the DBH of the extracted stem can reach a minimum of 0.1m. This algorithm is not suitable for point clouds acquired by airborne (ALS), backpacks and other acquisition methods, because the normal consistency of the stem surface of these point clouds were not strong.

Usage

Click *TLS Forest> Extract Tree Stem(TLS)*.



Parameters Settings

- **Input Data**: The input data must be normalized point cloud data. For normalization method, please refer to normalization or normalization according to ground points. The input file can be a single data file or a point cloud data set; the file(s) to be processed must be opened the in LiDAR360 software.
- **Height Above Ground (m) (default value: "0.5")**: The point with z value lower than this will be regarded as the low understory vegetation and will not participate in the stem extraction. This parameter is used to reduce the influence of low understory vegetation on the effect of individual tree segmentation. 0.5-0.8m is appropriate for most TLS data.
- **Max Stem Height (m) (default value: "4.0")**: Maximum height of a stem. Points above this value will not be involved in the stem extraction. If this value is set too large, it will affect the accuracy of stem extraction and reduce the calculation efficiency. Taking rainforest data as an example, the height difference between trees may reach 20 to 30 meters, so only point clouds of common part height (less than 4.0 meters) are taken for stem extraction, and DBH and other forest parameters are calculated.
- **Min Stem Height (m) (default value: "1.5")**: Minimum height of a stem. The height of the shortest tree in the forest land. This height mainly affects the detection results of low stems. This value must be greater than "Height above ground point" and less than "Maximum stem height". Low stems with a height of 0.8 times this value will be detected.
- **Buffer Size (m) (default value: "2.0")**: Buffer area for getting data is used to solve the edge connection problem of adjacent data blocks in the algorithm. This value needs to be set to be slightly larger than the maximum diameter at breast height (DBH) of the trees in the scene. If this value is set

too large, the calculation efficiency will be reduced, and if it is set too small, it will result in over-segmentation of part of the stems. 2.0 is suitable for most TLS data.

- **Max Stem Inclination (degree) (default value: "15")**: The stem forms the maximum inclination with the vertical direction. Stem points with the inclination angle less than this value will participate in subsequent calculation as candidate points. The larger this value is set, the more stem points are extracted, but the accuracy is reduced. On the contrary, the smaller this value is set, the fewer the number of tree stem points are and the higher the precision is. 10~15° is suitable for most data.
- **Voxel Sampling (m) (default value: "0.02")**: The grid size of Voxel thinning. Raw TLS data needs voxel thinning to reduce the computational burden. When this threshold is greater than the average point density of the point cloud (calculated automatically within the algorithm), grid thinning will be performed according to this threshold, otherwise grid thinning will not be performed. If the data has been subject to resampling, this value can be set as the default value.

> @inproceedings{ author={Shengli Tao, Nicolas Labrière, Kim Calders ,Fabian Jörg Fischer, E-Ping Rau,Laetitia Plaisance, Jérôme Chave}, title={Mapping forest disturbances across the Southwestern Amazon: tradeoffs between open-source, Landsat-based algorithms}, booktitle={Environmental Research Communications, 3(9):091001(13pp)}, year={2021} }

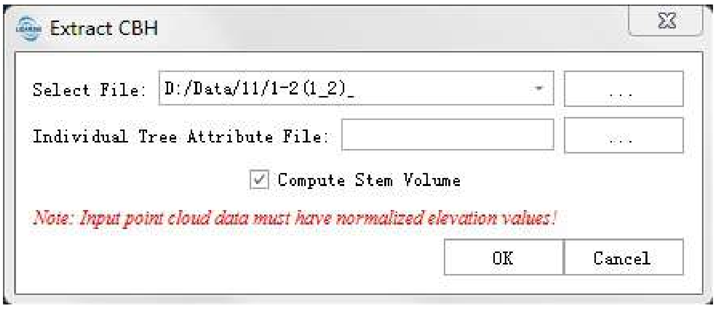
Extract CBH

Functional Overview

CBH (Crown base height) refers to the vertical height of a tree from the ground surface to the lowest branch point of the tree crown. CBH, also known as crown under height, is the height below the first level of stem, and in practice often refers to the height below the branches of the first main branch that forms the crown of the standing tree. Because of the difference of the genetic characteristics of each tree species, CBH is different. This function takes the individual tree attribute file and point cloud as input to calculate CBH and volume of the tree, and adds the calculation results to the end column of the individual tree attribute file.

Usage

Click *TLS Forest> Extract CBH*.



Parameters Settings

- **Input data**: The input data must be normalized point cloud data. For normalization method, please refer to normalization or normalization according to ground points. The input file can be a single data file or a point cloud data set.
- **Individual Tree Attribute File**: The input data is the single-wood attribute file of the corresponding point cloud. It is a comma-separated CSV table and must contain at least two fields: TreeLocationX and TreeLocationY. For details about the file format, see the format of TLS individual tree segmentation result file.
- **Calculate stem volume from Bottom to**: If selected, calculate the volume of the tree trunk.
  - **CBH**: Calculate stem volume to a point below the first branch.
  - **Tree top**: Calculate stem volume to the top of the tree.
- **Calculate stem volume**: When this option is selected, the calculation result of StemVolume will be added to the end column. If this column already exists, it will be overwritten.

> Note: After running, CBH calculation results will be added to the end column. If this column already exists, it will be overwritten.

Stem Curvature

Overview

This function calculates the curvature of individual tree stems. The stem shape is simplified into a B-spline curve according to canopy height, then segmented according to inflection points or specified lengths (cutting analysis). The curvature, volume, and individual segments are calculated and output. The calculation principle of curvature for a single tree segment is shown in the figure below. The curvature is the ratio of the maximum rise (h) to the DBH (d) at the top of the segment. The bottom end is the cutting position near the tree base, and the top end is the cutting position near the tree tip.

Parameter Settings

- **Input Data**: The input data must be normalized and segmented individual tree point cloud data. Refer to Normalization or Normalization by Ground Points for normalization methods. The input file can be a single point cloud data file or a point cloud dataset; the data must be opened in LiDAR360 software.
- **Individual Tree Attribute File**: The input data is the individual tree attribute file corresponding to the point cloud, supporting treedb or csv. The csv file is a comma-separated CSV table and must contain at least two fields: TreeLocationX, TreeLocationY. Refer to Individual Tree Segmentation Result File Format for details.
- **Tree Trunk Category (default "23")**: The category of the tree trunk contained in the point cloud. The algorithm will first extract the tree trunk according to this category.
- **Height Above Ground Points (meters) (default "0.3")**: Only point clouds above this height will be counted.
- **Layer Height**: Segment the trunk according to the layer height and simplify the trunk shape into a B-spline curve.
- **Minimum Rise (meters) (default "0.008")**: Rises below this value will be considered noise. This threshold is used to simplify the shape of the B-spline curve formed by the trunk.
- **Segmentation Method**:
  - **Automatic Segmentation**: Automatically segment the fitted trunk curve at inflection points.
  - **Segmentation by Length**: Segment the fitted trunk curve according to the specified length.

> Note: After the operation, the calculated curvature (CBH), volume, and individual segment results will be added to the end column. If this column already exists, it will be overwritten.

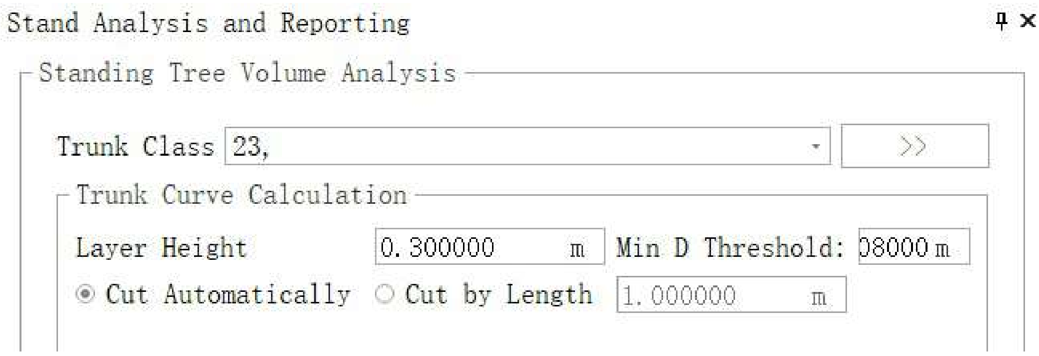
Standing Tree Volume Analysis

Function Overview

This function is used to calculate the trunk volume of point clouds that have already undergone normalization and individual tree segmentation.

Usage

Click *TLS Forestry* > *Stand Analysis and Report*.



Parameters

- **Input Point Cloud**: Select the point cloud to be processed. This point cloud must have undergone individual tree segmentation.
- **Individual Tree Attribute File**: The input data is an individual tree attribute file, which is a comma-separated CSV table that must include at least two fields: TreeLocationX and TreeLocationY. Refer to the Individual Tree Segmentation Result File Format for the specific file format.
- **Min D Threshold**: Minimum Distance for Detecting Trunk Points.
- **Trunk Fitting Method**: The method for fitting the trunk for each layer of the point cloud. Currently supports Convex Hull, Circle Fitting, Minimum Area Circle, and Concave Hull.
- **Layer Height (default 0.3)**: The height of each trunk segment. It can be set to about 5 times the smallest point distance in the point cloud. For straighter and denser trunks, it can be set to 5 to 20 times the smallest point distance in the point cloud.

TLS Seed Point Editor

Functional Overview

The TLS Seed Point Editor toolbar contains functions for checking the results of individual tree segmentation routines run on Terrestrial Laser Scan (TLS) input data. The TLS Editor is also used to extract Diameter at Breast Height (DBH) for individual trees, to add or delete seed points, to execute point cloud segmentation operations that include seed points, and to measure physical attributes of individual trees found in the source dataset. Since v3.2, the right-click shortcut menu is supported.

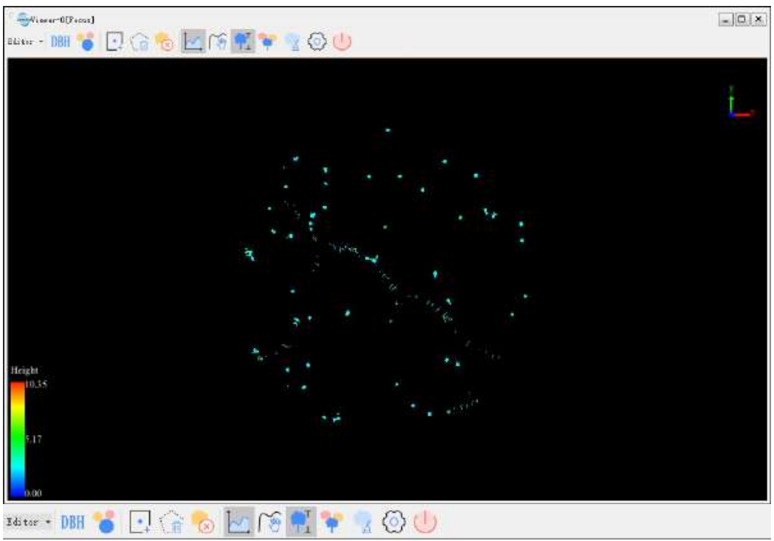
TLS Seed Point Editing Shortcuts

Shortcut	Description
shift + mouse left	Profile translation
↑	Translate the profile up
↓	Translate the profile down
→	Pan profile to the right / to the next tree in single tree attribute table
←	Pan profile to the left / to the previous tree in single tree attribute table
Ctrl + z	Undo seed point editing
Ctrl + y	Redo seed point editing

Usage

Load the point could data to be edited into the viewer and activate the view window.

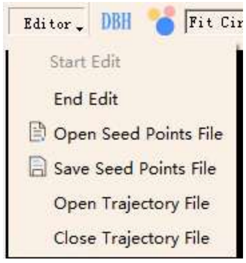
Navigate to and click on *TLS Forest* > *TLS Editor* to load the TLS Seed Point Editor toolbar in the active view window.



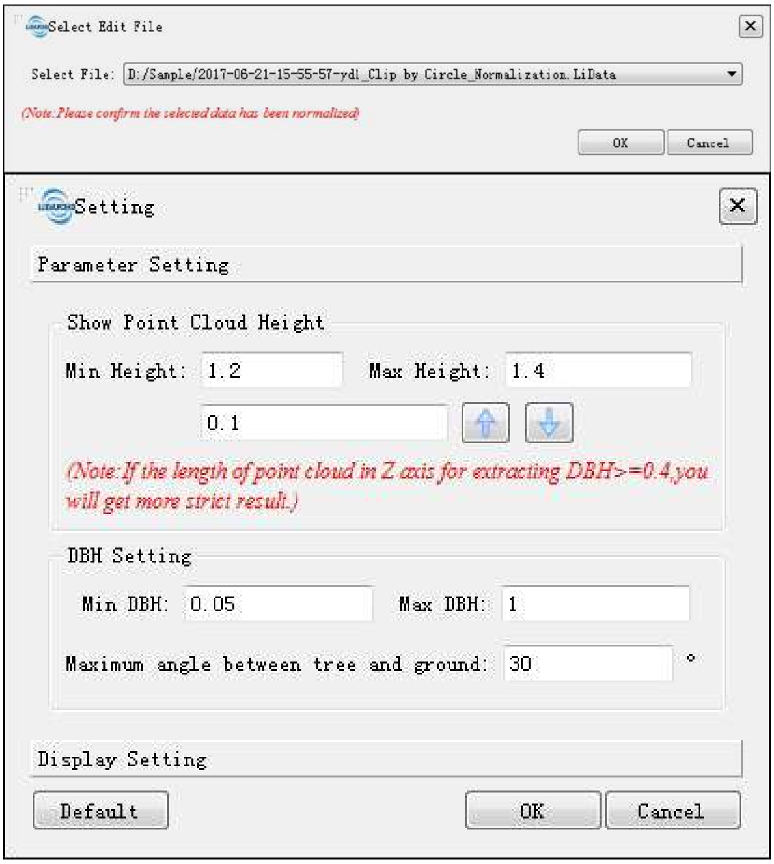
TLS Seed Point Editor toolbar functions (from left to right): Start/End Edit, Open Seed Points File, Open/Close Trajectory File, Fit DBH, Batch Extraction of DBH, Add Seed Points, Delete Seed Points, Clear All Seeds, Profile, Pan Profile, Measure Individual Tree Attribute, Point Cloud Segmentation from Seed Points, Clear Tree ID, Filter by Elevation, Setting, and Exit.

Start/End Edit

To begin editing session navigate to and click on *Editor* > *Start Edit*.



Select the data to be edited, click "OK", a setting window will pop up, the user can set the displayed point cloud height, the default point cloud is 1.2-1.4m, the user can set whether to display the mouse position synchronously in the profile view. Other functions of the TLS seed point editor toolbar will become available.

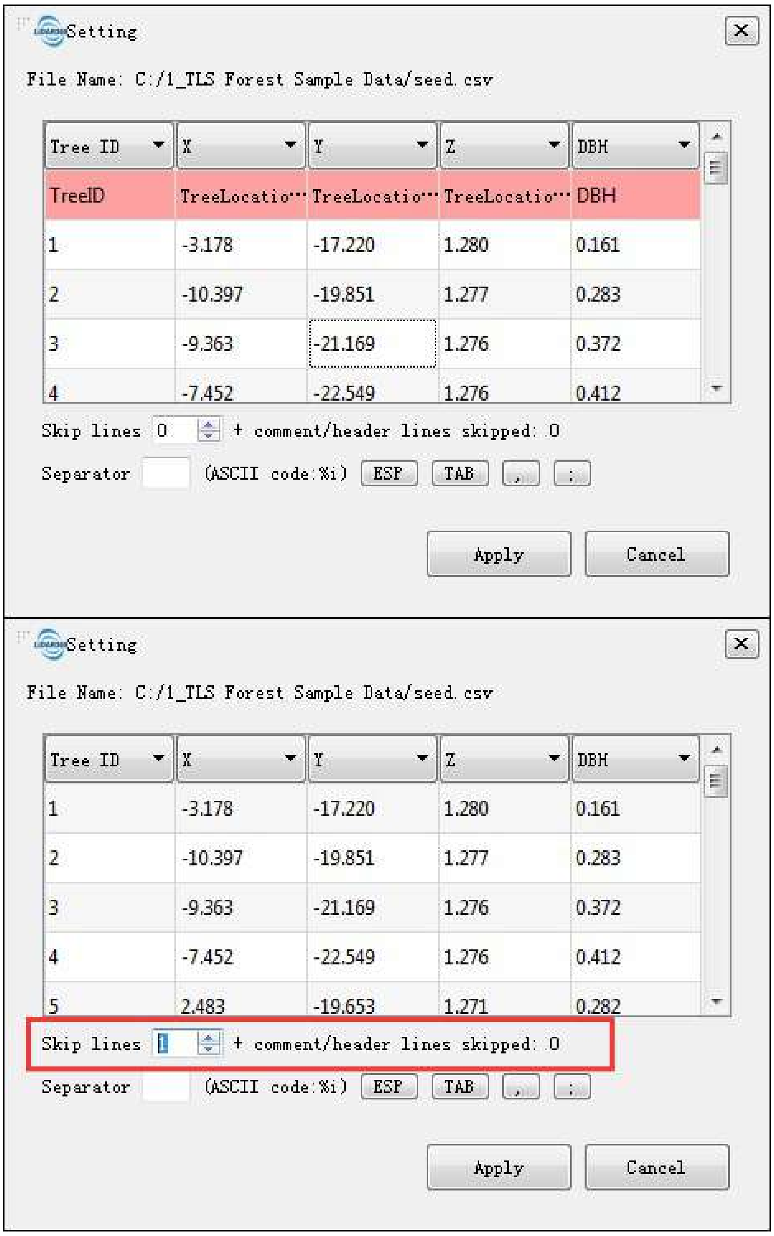


In the process of editing, the selected data cannot be removed from the window. Click “End Edit” to complete the editing, and other functions in the toolbar will then be unavailable.

Open Seed Points File

Function Description: Select a seed point file (please refer to Seed Points File in the appendix)

Function Description: Select a seed point file (please refer to Seed Points File in the appendix) for the format of seed point file, and the window below will pop up. Please note that there is a header line in the seed point file, therefore, the first line should be ignored when the file is opened.



Save Seed Points File

Function Description:When editing is completed, the seed points can be saved as a new .csv file without overwriting the original file.

Open Trajectory File

Function Description: Load trajectory file, support the output trajectory file format (*.xyz) of LiBackpack series products.

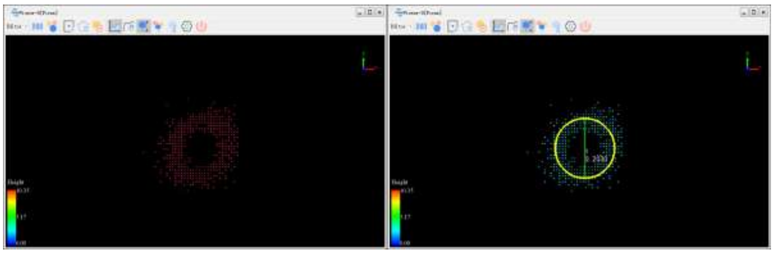
Close Trajectory File

Function Description: Close the loaded trajectory file.

Fit DBH

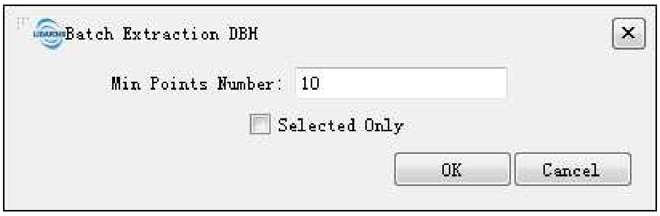
Function Description: Select the desired individual tree for DBH fitting.

The picture below (left) is the result of selecting the point cloud data of an individual tree. The picture below (right) is the result of DBH fitting: 1 indicates ID number, 0.2030 is the fitted DBH value in meters.

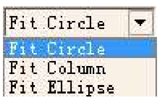


Batch Extraction of DBH


Function Description: Select the point cloud data of multiple trees and then perform batch fitting of tree DBH (the parameter settings are shown as below). By default, the function uses the entire point cloud in the window for batch DBH fitting. Users can also select out a part of point cloud for fitting.



- **Min Points Number (default value is "10")**: Minimum threshold for the number of trees of each category. If the number is less than this value, then no DBH fitting will be conducted.
- **Methods of Fitting DBH**: Users can flexibly select the point cloud in a 2D window by setting a height or using the profile window for DBH fitting. Methods of fitting DBH include Fitting by Circle, Fitting by Column and Fitting by Ellipse. By default, the method of Fitting by Circle is adopted. If the tree is slanting, the Fitting by Cylinder method can be used.

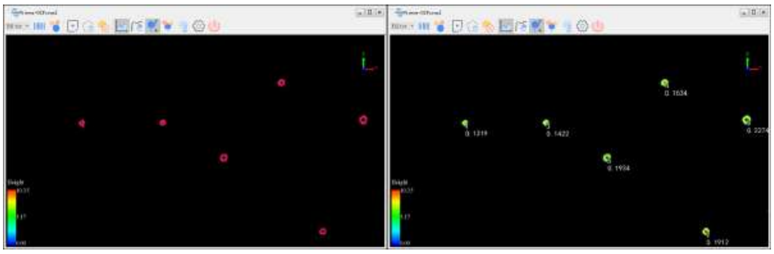


- **Fit Circle (Default)**: Use the least squares method to the fit a circle from the x-y coordinates of input points.
- **Fit Column**: Use the least squares method to fit a three-dimensional cylinder from input points where

the diameter is the DBH of the tree.

- **Fit Ellipse**: Use the least squares method to fit an ellipse from the x-y coordinates of input points, and the DBH will be calculated as 2 *sqrt*（*major axis* minor axis). This method is for trees with an oval-shaped trunk.

The picture below (left) is the result of selecting the point cloud data of multiple trees. The picture below (right) is the result of DBH fitting: 1, 2, 3, 4, 5 and 6 indicate ID number, 0.1319, 0.1912, 0.1422, 0.1634, 0.1934, and 0.2274 are the fitted DBH value in meters.



Add Seed Points

Function Description: Interactively add seed points manually, support adding seed points in the edit window and profile window.

Delete Seed Points

Function Description: Delete the selected seed point data. Click this button, the selection tool dialog box will pop up, the user can select the appropriate selection tool (polygon selection, circle selection, rectangle selection tool) according to the needs, and pick a suitable area in the scene to delete the seed points in the area.

> Note: Since version 5.0, the previous process of deleting seed points has been modified to simplify selecting and deleting seed points into one operation. After selecting a seed point, the selected seed point will be deleted immediately.

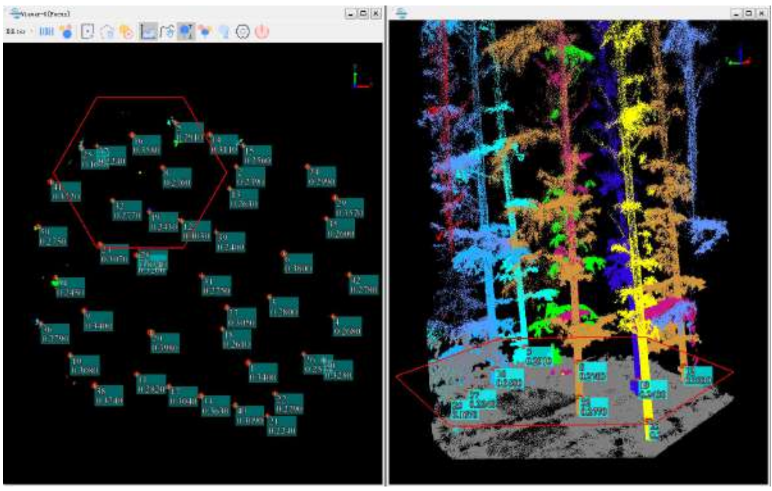
Clear All Seeds

Function Description: Clears all seed points in the window.

Profile

Function Description: When the TLS Editor toolbar is opened, the point cloud window will change to 2D display mode. The profile tool can show if the seed points are accurate in 3D. Click the Profile Tool button and a new window will appear. Click the mouse left button to select a polygon, and all points within the polygon will be displayed in the new window in 3D.

> Note: Since version 5.0, the method of drawing the profile area has been modified as follows: click the left mouse button, drag and release the mouse to complete the drawing of the profile area.



Pan Profile

Function Description: After drawing the profile area in the main window, users can translate the profiled area by clicking this tool, and see the profile data in real-time.

Measure Individual Tree Attributes

Function Description

Click this *Measure Individual Tree Attributes* button on the toolbar to control whether to display the Measure Individual Tree Attribute window.

> For descriptions of individual tree measurements, please refer to Individual Tree Attribute.

Point Cloud Segmentation from Seed Points

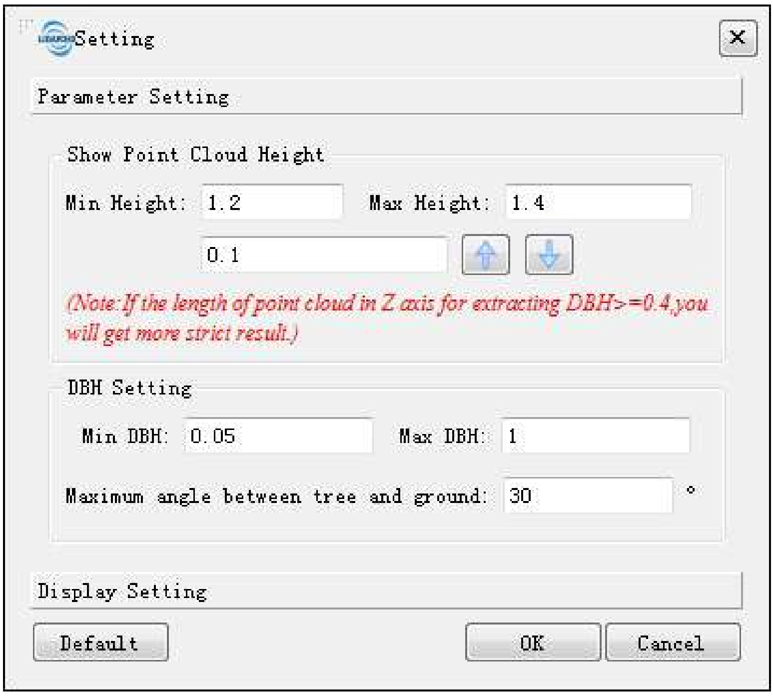
Function Description: Segment the point cloud data based on the edited seed points. Refer to PCS with Seeds. The tree heights will be recalculated after tree segmentation.

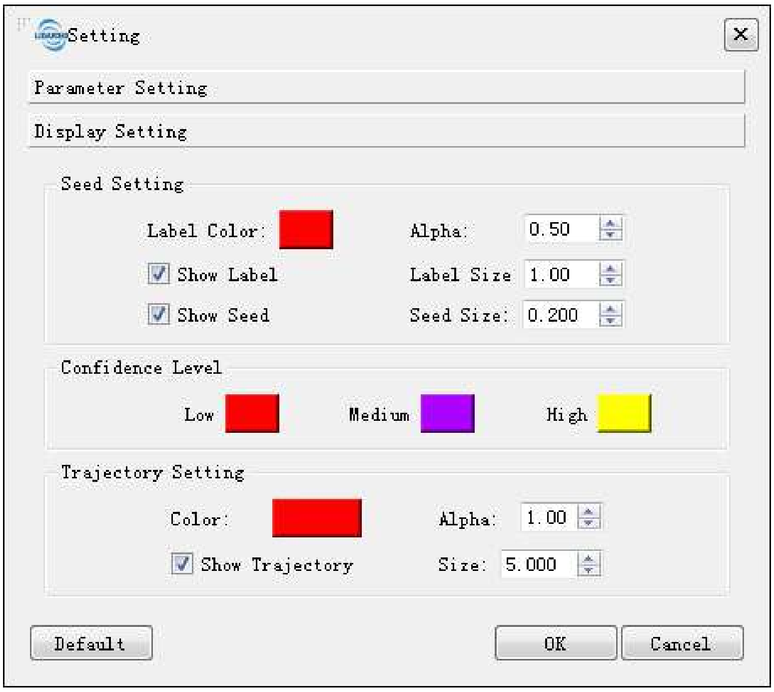
Clear Tree ID

Function Description: If the point cloud data has been segmented already, the TreeID should be removed prior to reperforming individual tree segmentation. Refer to Clear Tree ID.

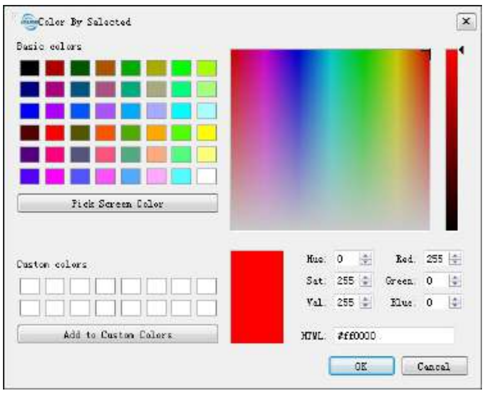
Parameters Settings

Function Description: Includes seed point setting and height setting for point cloud display.





- **DBH Configuration**：
  - **Minimum DBH (Meter) (Default Value is "0.05")**: The minimum DBH value based on user's prior knowledge. If the DBH of a tree is smaller than this value, the fitted DBH will be treated as an error.
  - **Maximum DBH (Meter) (Default Value is "1.00")**: The maximum DBH value based on user's prior knowledge. If the DBH of a tree is larger than this value, the fitted DBH will be treated as an error.
  - **Maximum Tree Inclination Angle (Degree)(Default Value is "30")**: The maximum tree inclination angle based on user's prior knowledge. If the inclination angle of a tree is larger than this value, the corresponding points will not be treated as a tree. Note that this threshold is only effective to the method of **Fit Column**.
- **Seed Setting**: Seed point setting includes the color, alpha and size of the seed points, and whether the seed point ID and its label size are shown.
  - **Color (Default Color is "Red")**: Click , and then the dialog for selecting colors will pop up as shown below. Any color can be selected as the seed point color from this window.



  - **Alpha (Default Value is "0.5")**: Transparency of the seed point, ranging from 0 to 1, with 0 being completely transparent and 1 being opaque. The default value is 0.5. Click , and the value will be incremented or decremented by 0.1; or one can directly enter a specific value.

- **Display Label**: Set if display the label of the Seed ID in the window.
- **Label Size (Default Value is "1")**: The label size, ranging from [0, 100). Click , and the value will be incremented or decremented by 1; or you can directly enter a specific value.
- **Show Seed ID (Optional)**: Use the check box to determine if the seed point ID is displayed in the window.
- **Size (Default Value is "0.2")**: The seed size, ranging from [0,100). Click , and the value will be incremented or decremented by 1; or one can directly enter a specific value.

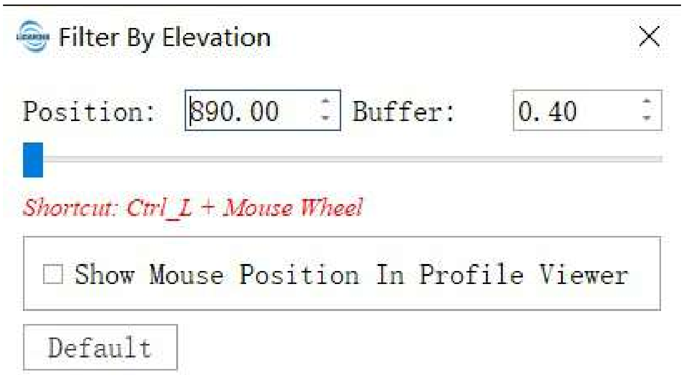
* **Confidence Level**: Estimate the confidence level of batch fitted DBH values, and visualize the fitting results with different confidence levels using different user-defined colors.
  - **Low Confidence Level**: The DBH fitting results is poor, and users need to visually examine and edit the results to improve the fitting accuracy. Users can set the visualization color.
  - **Medium Confidence Level**: The DBH fitting results is medium, and users may need to visually examine and edit the results if necessary. Users can set the visualization color.
  - **High Confidence Level**: The DBH fitting results is very good. Note that if only a single tree was fitted, the default confident level will be set as high.

    > Note that if the visualization height range is larger than 0.4 m when fitting the DBH in batch, LiDAR 360 will use a more strict method to estimate the confidence level. This method usually performed better for trees with a long trunk.

* **Camera Distance of Profile View (Default Value is 20)**: When drawing a profile, the distance from the observation point to the display area of the profile window is convenient for users to view the editing area or editing effect according to the distance according to the job needs.

Filter by Elevation

Function Description: Point cloud display height and whether to show mouse position in profile viewer.

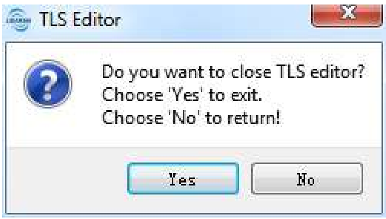


* **Point Cloud Display Settings**:
  - **Position**: The height of the point cloud in the profile viewer.
  - **Buffer**: The thichness of the point cloud in the profile viewer.
  - **Show Mouse Position in Profile Viewer**: When opening a profile, whether to display the real-time position of the mouse in the profile when the mouse moves in the profile window.

> Note: The Ctrl_L + Mouse Wheel scrolling shortcut can only be used when the filter by elevation window is active, and the profile view is the current focus window. After closing the window, the parameters will keep as before it close.

Exit

Function Description: To exit TLS Seed Point Editor, click the exit button and a prompt window will pop up. Click “Yes” to close the TLS Seed Point Editor, or click “No” to return to the editor window.



> Note: If the point cloud has been segmented, you need to Clear Tree ID before being segmented again. Otherwise, the message "All files have been segmented, please clear tree ID first!" will prompt.

Undo and Redo

Function Description: During the seed point editing process, if there is an editing error or misoperation, press [ctrl+z] to undo the operation, and press [ctrl+y] to resume the operation.

> Note: In a seed point editing, the undo and redo functions are limited to 20 times respectively.

```
@inproceedings{
    author={ Macdonald E, Mochan S},
    title={Protocol for stem straightness assessment in Sitka spruce},
    booktitle={Journal of Bacteriology,176(17):5578-82},
    year={2000}
}
```

Individual Tree Editor

Functional Overview

Individual Tree Editor can extract and edit individual trees in large data points cloud, and can quickly locate and filter out user-concerned individual trees. It is mainly used to solve over-segmentation, under-segmentation and missing segmentation and other cases, and can also repair fine of under-story trunk.

Usage

Load the point cloud data needs to be edited in the window and make sure to activate that window.

Click *TLS Forest* > *Individual Tree Editor* or *ALS Forest* >*Individual Tree Editor* to activate the editing bar in current window. Some functions of different modules maybe different.

The buttons in the individual tree editor tool bar, from left to right, are: start/end editing, profile, open the individual tree attributes file, individual tree filter, settings and exit.

Please note that if there is no special need, you should normalize the point clouds which need to be edited. Please refer to Normalization. After editing is complete, the original point cloud can be restored by reverse denormalization. The profile window takes the plane with Z value of 0 as the datum, and for the normalized point cloud, the datum is the ground. For the unnormalized point cloud, the point cloud is far from the datum due to the influence of elevation, which may cause the point cloud and the cut polygon may not exist in the profile view. Therefore, normalized point cloud data can be better edited. In addition, functions such as removing support brackets and re-segmentation under this module need normalized point clouds to get correct results.

Start/End Editing

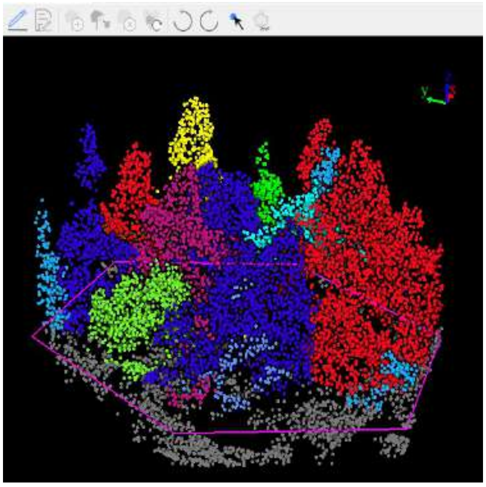
Click *Edit* > *Start Editing*

Select the data to be edited and click "OK" to render the point cloud as displayed by tree id. Other functions of the individual tree point cloud editing tool bar will become available.

The selected data cannot be removed from the window during editing. After editing, click "End Edit" to end the editing.

Profile

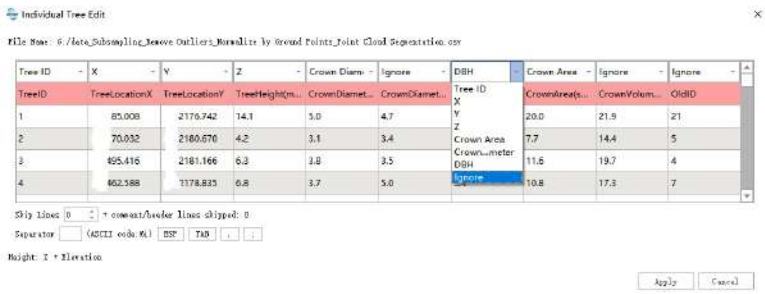
Functional Description: Click the "Profile" button to open the profile window and draw a hexagonal area in the window where the point cloud is located. This area is the individual tree point cloud area to be edited. The individual tree editor toolbar appears above the profile window.From left to right: Load/remove edit area, Save, Create individual tree, Merge individual tree, Delete individual tree, Re-segment/remove support bracket, Back up, Redo, Select point, Hide tree.



> Note: 1. Please pay attention to the individual tree boundary when picking up the hexagonal area, because the incomplete part may affect the result, or try to avoid modifying the boundary when editing. None of the editing functions in this module will have an effect on hidden trees, so you can hide trees you don't want to edit to avoid affecting the results.
>
> 1. Since version 5.0, the method of drawing the profile area is modified to: Click the left mouse button, drag, and release the mouse to complete the drawing of the profile area.

Open Individual Tree Attributes File

Functional Description: Individual tree attributes file is a CSV file generated after individual tree segmentation. Please note that this function recognizes and skips several rows of the table header by default. If necessary, it can be adjusted by itself. Before clicking apply, ensure that the table header field is consistent with the corresponding column in the CSV file.

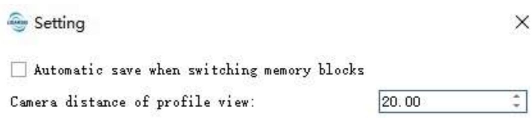


Individual Tree Filter

Functional Description: Before performing individual tree filtering operations, open the individual tree attributes file. This function can be set according to the parameter range, quickly screen interested individual tree. When the filter is enabled, the editor's main window sets the filtered individual tree color in the scene to transparent, and the profile window automatically applies the filter to hide the corresponding individual tree in the edit area. Currently, there are two filtering modes. Trees that match the filtering range are kept display in display mode, while other trees will be hidden. In hidden mode, the effect is reversed.

Settings

Functional Description: This includes profile distance, auto save and other settings.



- **Profile Distance**: When drawing the profile, please observe the distance between the point and the display area in the profile window for the users to view the editing area or editing effect according to the distance conveniently. For example, for ALS forest, the section distance can be set to hundreds of meters. And for TLS Forest, it can be set to tens of meters.
- **Auto Save**: It is used to auto-save the edited area when switching the profile areas.

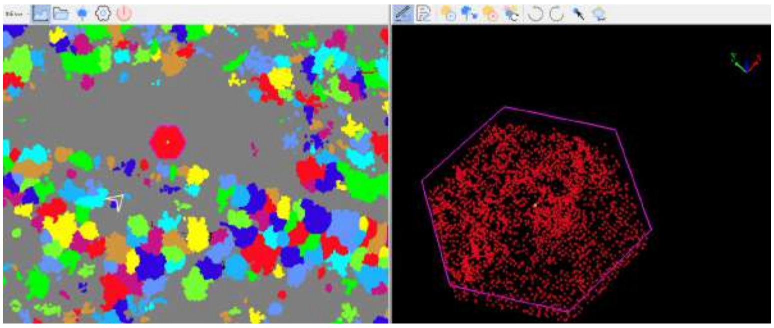
Load/Remove Editing Area

Functional Description: Click *Load Editing Area* button to set the area in the hexagon as the area to be edited. The data for editing the individual tree point cloud is the data in this area. All editing tools are available in the profile window. Enter the editing state at this time, selects the section area in the main window again, it will automatically choose the data in the selected area to be the new editing data.

Click “Remove Editing Area” button, and this action will exit the editing directly if no operation is done to the point cloud file; Otherwise the system will prompt you whether to save the point cloud data. Choose yes to save the point cloud data to a file, and you can see that the large data point cloud in the main window has been modified. Or you can choose no to exit the individual tree point cloud that is being edited without saving the data.

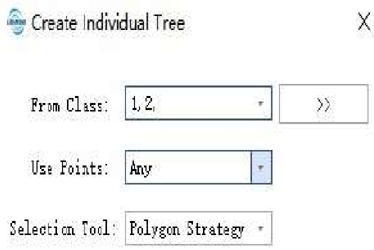
Save

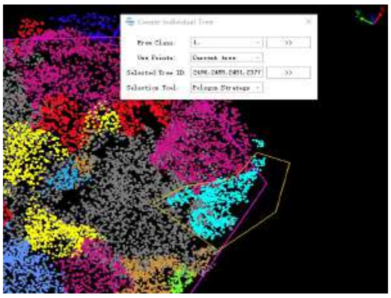
Functional Description: Click the "Save" button to save the modified individual tree point cloud data to the file. At the same time, you can see that the big data point cloud in the main window has been modified accordingly. Back up data if necessary, because it cannot be restored after being saved.



Create Tree

Functional Description: This function can effectively extract the selected point cloud and set it as the new tree ID. Click the "Create Tree" button and will appears the window shown as below. Select the category to create the tree point in the category drop-down list box, choose the category to participate in the point cloud segmentation from the filter point drop-down list box,select the whole category by default, select the scene job tool from the selection tool, select the polygon by default, then select the area in the profile window, double-click the end, you can see the color of the selected area changed, which means the tree is created. This function is often used in conjunction with other functions, such as creating a new tree ID for a point below a certain height and deleting it to remove the ground portion, or making an individual tree in a selection to be a new individual tree and segmenting it again.

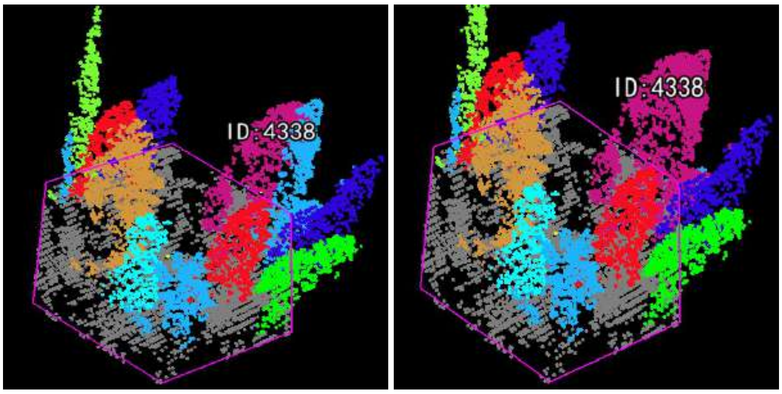




- **From Class**: Only specific classes of points are used to filter . Please note that the tree ID has nothing to do with the category, and it does not have to be classified as a tree to have a tree ID.
- **Use Points**: Only the points with specific tree IDs are used, avoiding making changes to other trees. There are three options:
  - **Any**: Applicable to all visible points.
  - **Current Tree**: Applicable only to the point cloud corresponding to the selected tree ID in the list. When selecting this option, pick the tree ID in the window to the input list by activating the pick tool. After picking, select "Pick seed point" to switch back to this function.
  - **No Tree ID**: Applicable only to the point with tree ID of 0.
- **Selection Tool**: Select the points within the area by different selection tools.

Merge Tree

Functional Description: It is used to manually solve the over-segmentation problem. Click "Merge Tree" button, use the mouse to pick up tree in the profile window, if the picked tree ID is not 0, mark the tree ID at the mouse position.Then pick up other trees, and they will be combined into the first picked individual trees. You can see that in the profile window, the color of the picked trees will change to the color of the first picked individual trees, right-click the mouse to end the pickup.

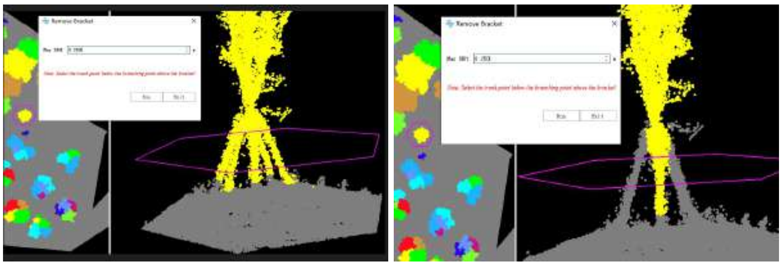


Delete Tree

Functional Description: Click "Delete Tree" button and use mouse to pick up trees in the profile window. If the picked tree’s ID isn’t 0, you can mark it as 0 and delete the tree.

Remove Bracket

Functional Description: This function only exist in the individual tree editor of TLS Forest, and can remove the support brackets of the tree trunk by semi-automatic interaction. Click the left mouse button to select a point on the trunk below the branch point above the support bracket as the reference point, process the point cloud below the height of this point, and set the tree ID of the point cloud corresponding to the support bracket to 0. The principle of the algorithm is the trunk and branches grow at the same time according to the height, and remove the growing branches. Therefore, when the lowest height of the support bracket is the same as the lowest height of the trunk, the result will be better.

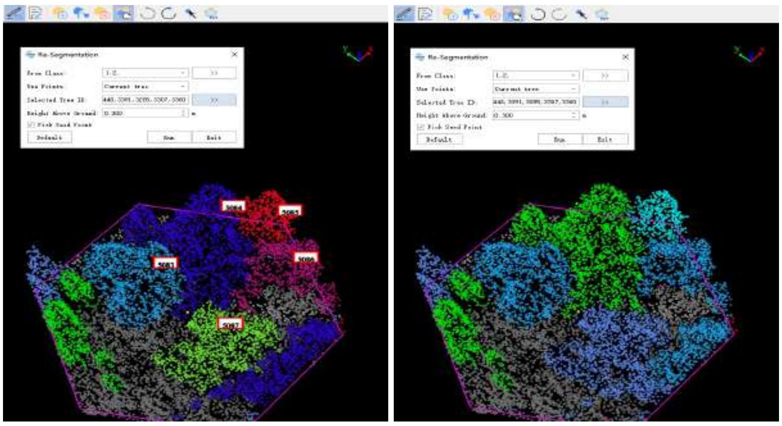


- **Max DBH**: Maximum DBH of the extracted trunk. Appropriate DBH settings can better extract the trunk. If the value is smaller, the extracted trunk will be finer, and if it is larger, more support bracket point clouds will be retained. When you open the individual tree attributes file and click the reference point, the DBH displayed on the input box will automatically update to a slightly larger DBH value for the corresponding tree in the file.

Re-segmentation

Functional Description: This function can only solve the problem of over-segmentation and under-segmentation of trees through semi-automatic interaction in the editing of the individual tree editor area under the ALS Forest. The algorithm process spatial search to the point clouds based on the seed points picked up in the view, and one seed point generates at most one individual tree. For over-segmentation, a point with obvious spatial characteristics (have a certain distance from other trees and easy to distinguish, generally the top of the tree) can be picked as the seed point, and a point closely related to this point in space will be classified as a new individual tree. For under-segmentation, multiple seed points need to be picked up and multiple single trees are generated by the same algorithm at each seed point position.

Click “Re-segmentation” button, and the function dialog box will pop up, click the left mouse button to pick up points as the seed points in the profile after setting the corresponding parameters, click "Run" to re-segment the point clouds in the editing area. The segmented tree IDs contain the biggest tree ID of the big data point cloud and the IDs above it, the tree segmented by this function will not be saved to the individual tree attributes file. Before using this function, you can hide the trees that don’t need to be divided, set the usage category and set of points to obtain higher precision segmentation effect.



- **From Class**: Only specific classes of points are used to filter. Please note that the tree ID has nothing to do with the category, and it does not have to be classified as a tree to have a tree ID.
- **Use Points**: Make targeted editing based on the point set of the selected tree ID. It is usually used for under-segmentation of an individual tree or under-segmentation of multiple individual trees. There are three options to use set of points:
  - **Any**: Applicable to all visible points.
  - **No Tree ID**: Applicable only to the point with tree ID of 0.
  - **Current Tree**: Applicable only to the point cloud corresponding to the selected tree ID in the list. When selecting this option, pick the tree ID in the window to the input list by activating the pick tool. After picking, select "Pick seed point" to switch back to this function.
- **Minimum Height**: Filter points below the minimum height to prevent dividing points on the ground. When the individual tree point cloud and the low grass bush on the ground are jagged and interleaved, the tree generated by the algorithm will grow downward and outward, resulting in

segmentation errors. This phenomenon can be effectively avoided when the parameter is set above the height of the ground low grass bush.

- **Pick Seed Point**: You can pick up seed points under the checked state. If you switch to tools such as "Pick Point" and "Measurement", the check box will be automatically unchecked. At this time, you can use these tools without quitting the function. If you manually check it, you can switch back to the mode of picking seed points.

Undo

Functional Description: Click *Undo* button to undo the last option. The maximum times of undoing is 20. Any operation before the last 20 steps cannot be undone.

Redo

Functional Description: Click *Redo* button to redo the last option. The maximum times of undoing is 20. Any operation before the last 20 steps cannot be undone.

Pick Point

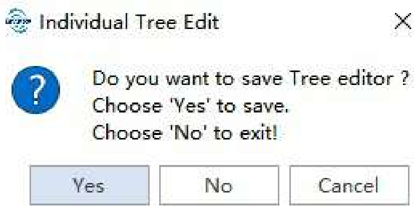
Functional Description: Click the "Pick Point" button and use the mouse to pick up the point cloud in the profile window. The attribute information of the point will be displayed.

Hide Tree

Functional Description: Click "Hide tree" button, use the mouse to pick up the point cloud in the profile window, and hide the tree with the corresponding tree ID of the point in the scene. The hidden tree can avoid being modified.

Exit

Functional Description: Click the "Exit" button to close the profile window and exit from individual tree editor directly if no editing operation has been done; Otherwise the system will prompt you whether to save the data. If you select Yes to save the data in the edit area to the file, you can see that the big data point cloud in the main window has been modified. Or you can select No to exit the editing area directly without saving the data. At the same time, close the profile window and exit from individual tree editor .


Individual Tree Edit
Do you want to save Tree editor ?
Choose 'Yes' to save.
Choose 'No' to exit!
Yes
No
Cancel

DBH Measuring

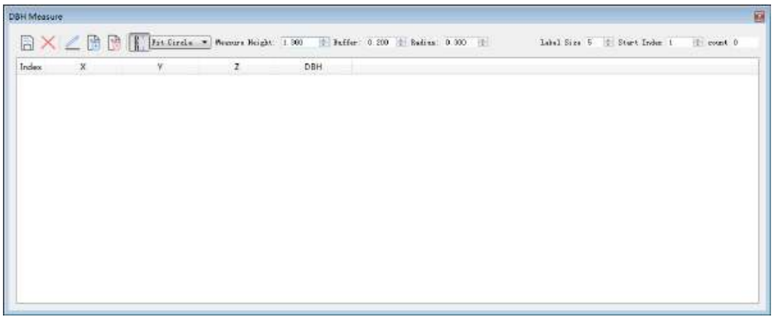
Summary

The DBH measurement tool can measure the tree DBH on the TLS forestry point cloud data. And support to export the measurement results as txt, asc, neu, xyz, pts, and csv formats.

Usage

Load the point cloud data to be edited into the window, and set this window as the active window.

Click *TLS Forestry* > *DBH Measuring* to open the DBH measuring window.



Steps

1. Activate the DBH measuring button (Initially activated). Use the left mouse button to click the roots of the trees in the scene to measure DBH in sequence. This function will select the point cloud of the corresponding range according to the set parameters and use the method of fitting the circle to fit DBH. The fitting results are displayed in the 3D window and the DBH measurement window at the same time.

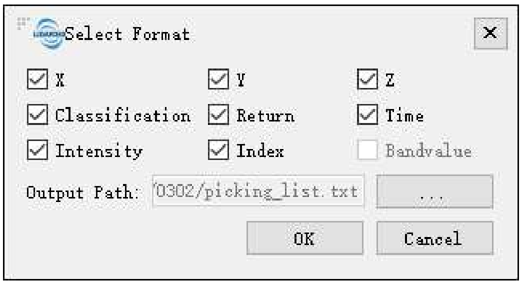
2. Click to select any row in the list, and click the button to delete this measurement result.

3. Click the Start Edit botton , and double-click the cell in the table to change the selected field.

4. Click the Add Field botton, and the following window will pop up. Currently, it supports the following types of custom fields: integer, float, text, date, and enum. After clicking the "OK" button, the new field will be displayed in the list window.

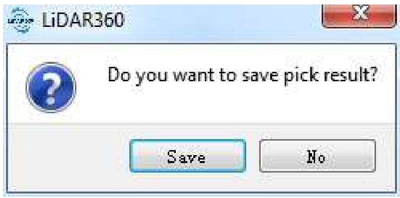
5. Click the Remove Field button to delete the added field selected in the Remove Field dialog (only the fields created by the users can be removed).

6. The selected results can be exported in multiple formats such as txt, asc, neu, xyz, pts, and csv. Click the button, the "Select Method" dialog box pops up (as shown).



7. Click the  button to select the output path and select all the fields need to be export. Then, click "OK" button to finalize the exporting.

8. If the users do not save the selected result, when exiting this function, the software will pop up the following interface, click "Save" to save the point, click "No" to exit the function.



Settings

- **Measuring Height**: This function uses a cylindrical area to select the point cloud, and fits the points obtained with a planar circle to get the DBH. The measuring height corresponds to the height from the fitting circle to the selected tree root.
- **Buffer Height**: The height of the cylindrical area to select the point cloud.
- **Buffer Radius**: The radius of the cylindrical area to select the point cloud.
- **Label Size**: The display size of the label for displaying DBH measurement results in the three-dimensional window.
- **Total Number**: The total tree of measurement result (cannot be manually modified).

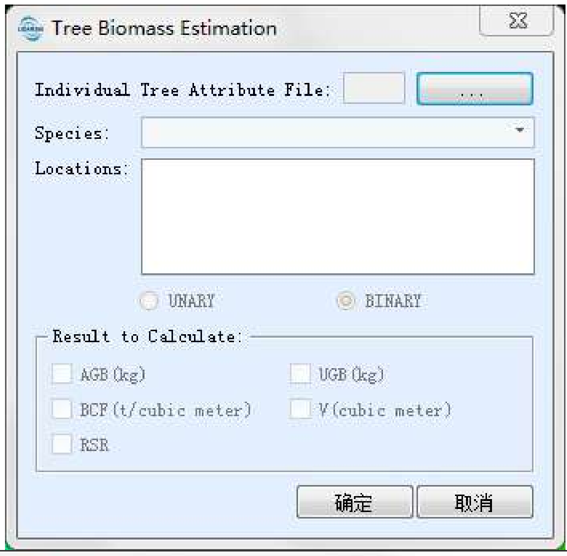
Biomass Estimation

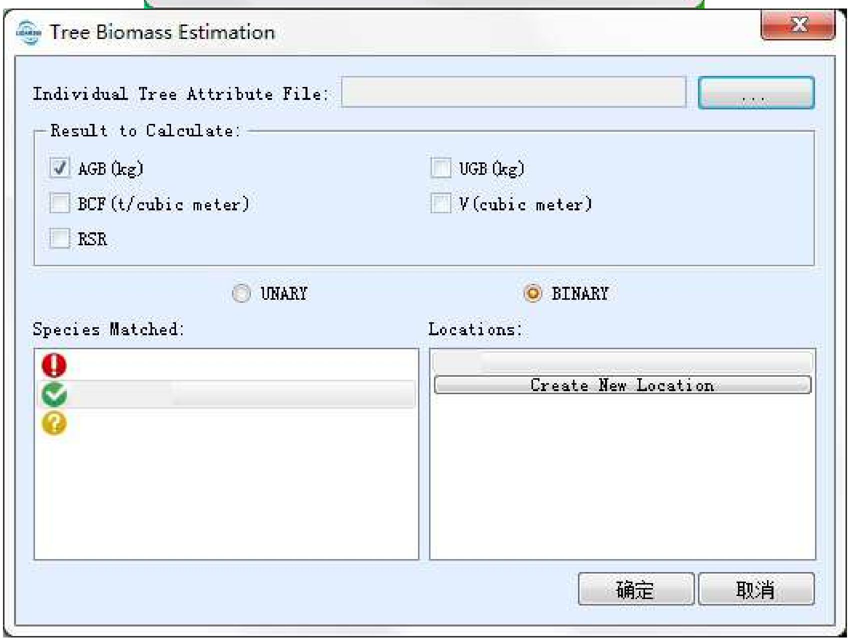
Function Overview

This function estimates the biomass of a region using a biomass model. For an introduction to biomass models, refer to the Biomass Model Manager. The input for this function is an individual tree attribute file, which can be the result from individual tree segmentation or a custom CSV file containing species (optional), diameter at breast height (DBH), and tree height fields. Refer to the appendix for the Individual Tree Segmentation Result File Format.

Usage

Click *Ground Forestry > Biomass Estimation*.





North American Database Interface:

Parameter Settings

- **Input Individual Tree Attribute File**: Refer to the appendix for the Individual Tree Segmentation Result File Format. Depending on whether the input data includes a "species" field, there are two interfaces. After execution, the results will be appended to the end of the file by column.
- **Select Database**: Currently, three databases are supported: Chinese Database, North American Database, and Custom Database. The Chinese Database will not be displayed in non-Chinese environments.
    - **NBEL North American Database**: Contains over 7000 biomass formulas for the North American region, referenced from the USDA. When switching to the North American Database, users can enter one or more filter conditions as needed to find the desired formulas and then execute the calculation.
        - **Filter Equations**: Users can select one or more conditions from "Region", "Species", "Author", "Part", and "Formula ID/Serial Number", then click the "Filter Formulas" button. The filtered formulas will be listed in the table below. Each row in the table represents a formula, and each column represents the details of the formula.
        - **Add, Export, Save, Delete Formulas**: Double-click a row in the **Filtered Formulas Table**, or single-click a formula in the **Filtered Formulas Table** and click the add button on the left to add the formula to the **Selected Equations**; the export button saves the formulas in the elected equations table, the import button imports the formulas saved last time for direct calculation; the delete and clear buttons reduce the formulas in the selected equations table.
        - **Calculate Results**: Each formula in the selected equations table will be added as a new column to the individual tree attribute file.
    - **Custom Database**: Contains all user-defined formulas. When switching to the Custom Database, users can directly select species-region-formula in sequence and then execute the calculation.
- **OK**: Execute the calculation. The results of the biomass calculation will be added as new columns to the individual tree attribute file.

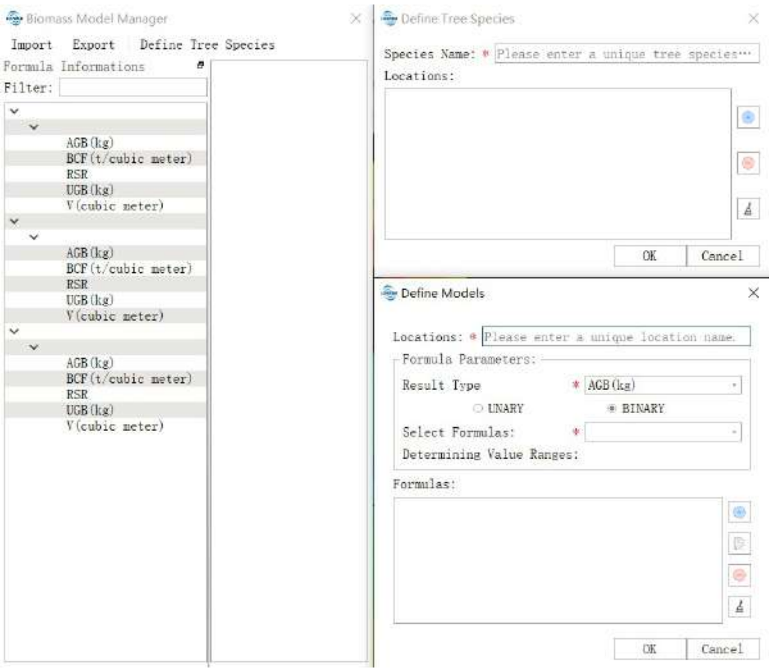
Biomass Model Manager

Functional Overview

Biomass refers to the total dry weight of organic matter per unit area at a certain moment in ecology. It is usually expressed in kilograms per square meter (kg/m^2) or tons per hectare (t/hm^2). The usual method for measuring biomass is to harvest all the materials within a unit area and then dry and weigh each part separately. For carbon storage statistics, the parts that need to be harvested include aboveground parts (trunk, leaves, flowers) and belowground parts (soil, dead wood, litter, etc.). In actual operation, it is difficult to calculate underground biomass and harvesting samples requires a lot of work and can cause serious environmental damage. Therefore, regression formulas are usually used to calculate the total biomass within a unit area sample statistically. Then forest-scale biomass can be obtained by multiplying with forest area. Considering that different tree species have different growth curves due to differences in light exposure and humidity they receive; therefore different biomass inversion models must be constructed for different tree species using basic parameters such as tree height (H) and diameter at breast height(D). Researchers have done extensive research on formulae for biomass models of different species mostly based on H & D as basic inversion parameters. This function supports inputting/deleting/querying/modifying dozens of tree species models based on China's forestry industry standard.

Usage

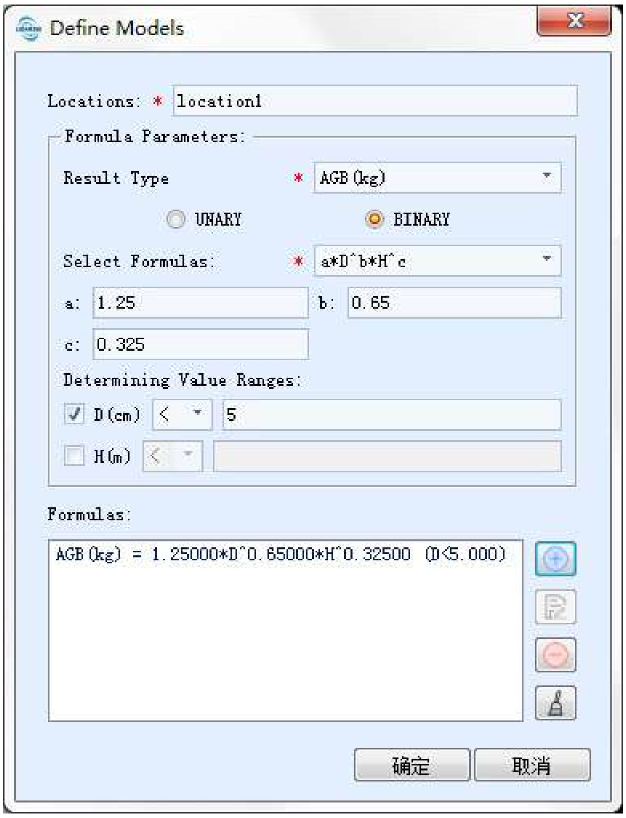
Click *Ground-based Forestry* > *Forest Biomass Model Management*.



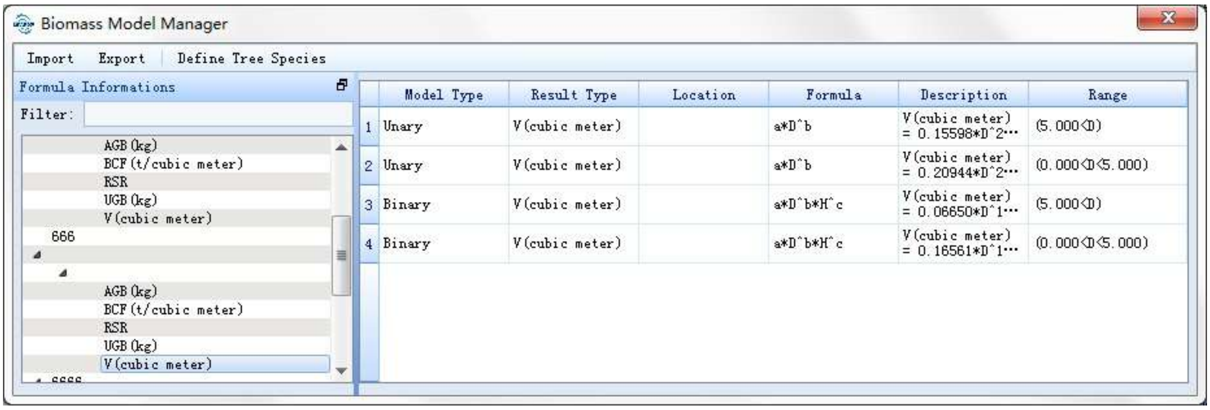
Parameters Settings

- **Import**: You can import custom database files which should only come from exported biomodel database files via this function.

- **Export**: Export data as database file.
- **Define Tree Species**: Open custom tree species function. This function can input custom tree species models into the database.
- **Species Name**: Enter a custom tree species name that cannot be duplicated with other tree species names in the database.
- **Locations**: Each tree species can support multiple regions to enter different biomass models for each region. Click on the add button on the right to add a region and corresponding biomass model.
- **Locations (required)**: Enter the region name.
- **Result Type (required)**: Enter target parameters, including AGB (aboveground biomass), UGB (belowground biomass), BCF (biomass conversion factor), V(volume) etc.
- **Model Type(required)**: Support univariate and bivariate models. Univariate model only contains D while bivariate model contains both D and H. The formula will change according to different selected model types.
- **Select Formulas(required)**: Select calculation formulae. Currently, only two bivariate formulas and one univariate formula are supported.
- **Formula Parameters(required)**: Depending on which formula is selected, the number of formula parameters may vary but no more than three at most.
- **Formula Range(optional)**: Since trees accumulate biomass at different rates during different growth periods, it may be necessary to segmentize height of trunk for biomodels processing such as treating trunks below 5cm diameter separately.



-**Model Management**: After entering all biometric formulas, you can view and manage them in Model Management interface by clicking on each individual tree species or its respective regions where all supported formulas are displayed along with detailed information about each type of publicized data including their types, parameters,result types,and applicable ranges.



@inproceedings{ LY/T 2654-2016 LY/T 2657-2016 LY/T 2659-2016 LY/T 2656-2016 LY/T 2661-2016 }

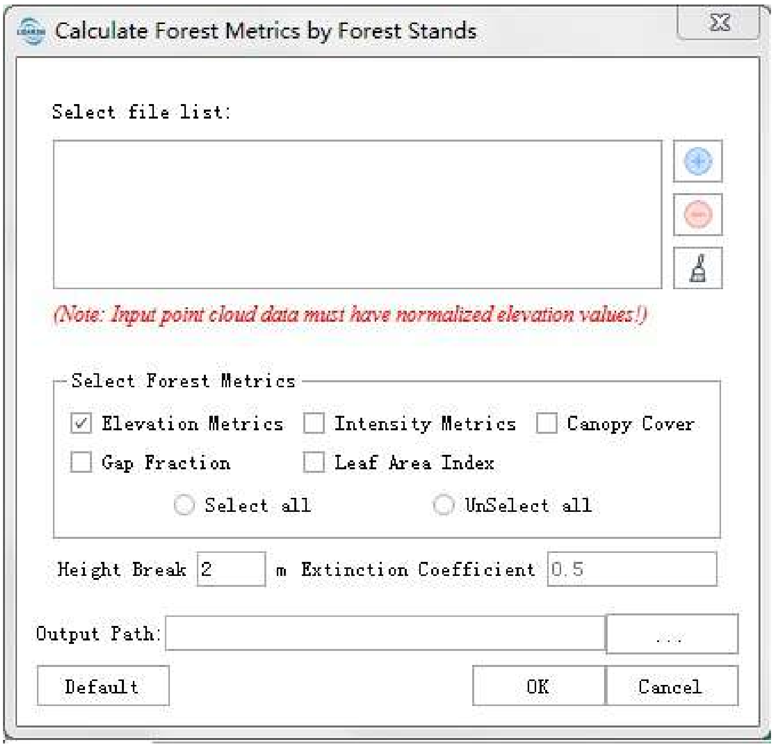
Calculate Forest Metrics by Grid (TLS)

Function Overview

Calculate forest parameters for each point cloud data, including gap fraction, Theory Of Canopy Cover, and LAI(leaf area index). Each point cloud data generates a CSV file.

Usage

Click *TLS Forest* > *Calculate Forest Metrics by Grid*



Parameter Settings

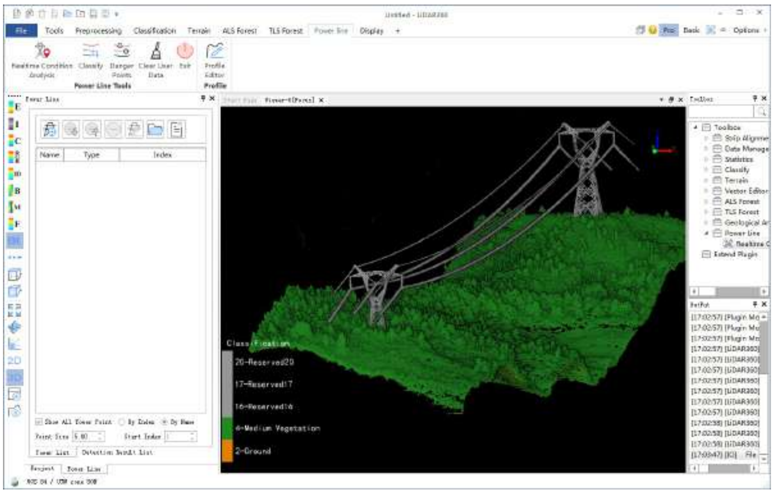
- **Input Data**: Ensure each input point cloud data is normalized. Input files can be individual point cloud data files or point cloud datasets. The data to be processed must be opened in LiDAR360 software.
- **Select Forest Parameters**: Supports Gap Fraction,Canopy CoverandLeaf Area Index。
- **XSize (m) (Default: "2")**: Grid size in the X direction.
- **YSize (m) (Default: "2")**: Grid size in the Y direction.
- **Height Break (m) (Default: "2")**: Threshold for dividing point clouds into different layers; only effective when Canopy Cover is selected.
- **Voxel Size Setting**:
  - **Voxel Size (m) (Default: "2")**: Grid size in the X direction.
  - **YSize (m) (Default: "2")**: Grid size in the Y direction.
- **Voxel Size Setting**:
  - **Voxel Size (m) (Default: "0.25")**: Grid size for dividing point clouds into three-dimensional grids within the statistical unit.

    - **Cofficient (Default: "2")**: Automatically calculates the size of the three-dimensional grid within the statistical unit based on the average point cloud spacing.
- **Output Path**: Output path where each input point cloud data file will generate a corresponding TIFF file, usable as independent variables in regression analysis.

> Note: The height variable function is only available when point cloud data is loaded in the software. Otherwise, the software will prompt "There is no point cloud data meet the conditions of calculation!" If the maximum Z value of the point cloud exceeds 200 or the difference between the maximum and minimum Z values exceeds 200, the software will prompt whether to proceed with non-normalized data. Clicking "YES" allows the data to be processed, while clicking "NO" excludes such data from processing, prompting the user to select data that meets the conditions.

Power Line

The Power Line module is used for processing and analysis of power line point cloud, including Settings,Mark Tower,Classify,Dangerous point detection and Clear Danger Points.



- Settings
- Mark Tower
- Classify
- Detect Danger Points
- Clear Danger Points

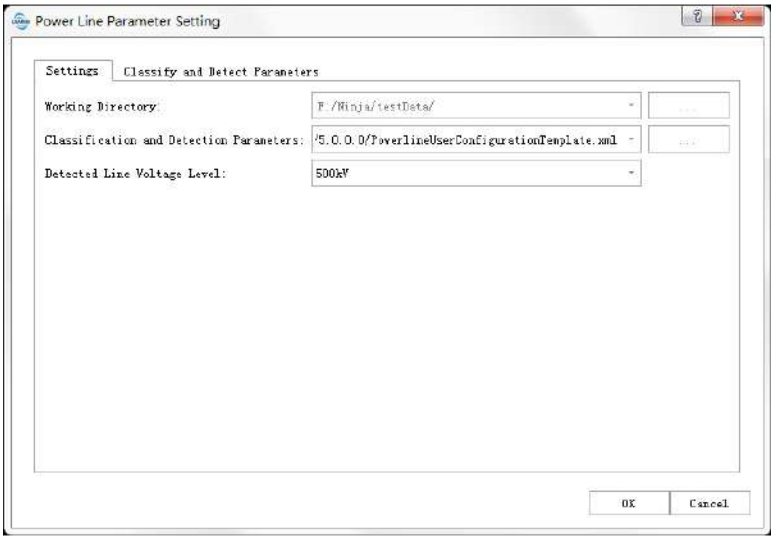
Settings

Summary

This setting will pop up automatically when powerline module is turned on. Powerline module parameter settings include common settings and detection parameter settings. Common settings include working directory settings, point cloud coordinate systems settings, class and detection parameters settings, detected line voltage level settings; detection parameter settings include class settings, detection types, and safety distance settings.

Common Settings

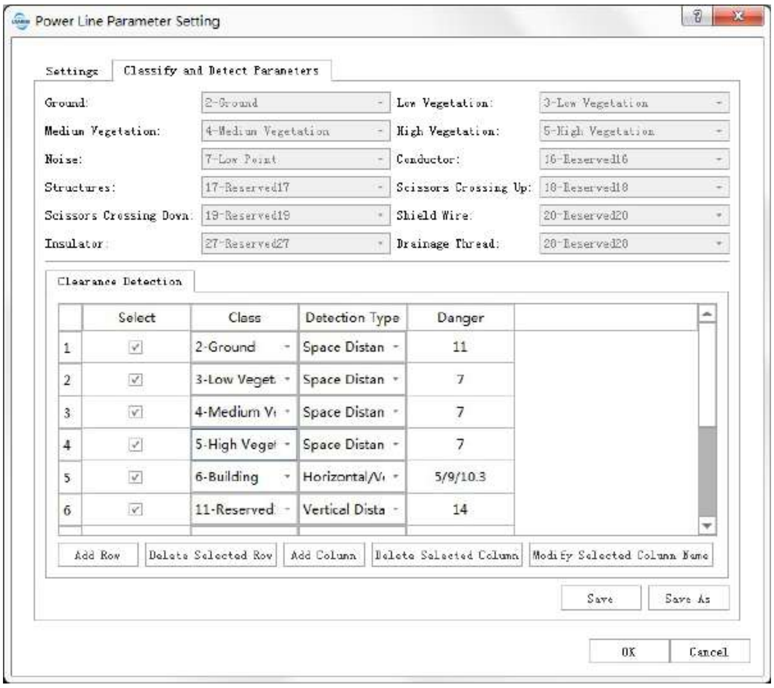
The user can set the working directory, point cloud coordinate system, class and detection parameters, detected line voltage level and add configuration tables. The software can record the user's last 5 input history records in settings of the working directory, point cloud coordinate system and classes and detection parameters. User can select the most recently used item in the drop-down list. The working directory is used to manage the project. The prj.ini file in the working directory records the point cloud coordinate system, classes and detection parameters, and voltage level of the current project.



- **Working Directory**: Used to save the intermediate and final results.
- **Classes and Detection Parameters**: Set the path of the xml format configuration file.
- **Detected Line Voltage Level**: The voltage level of the detected powerline.

Detection Parameter Settings

Displays key detection classes and safety distance detection parameters. User can add voltage levels and then configure their corresponding detection parameters, including clearance distance analysis and crossing analysis. Clearance distance analysis includes detections for different classes of objects, as well as different safety distance and distance detection methods. For crossing analysis, user only needs to set the classes to be detected, and user does not need to set a safety distance.



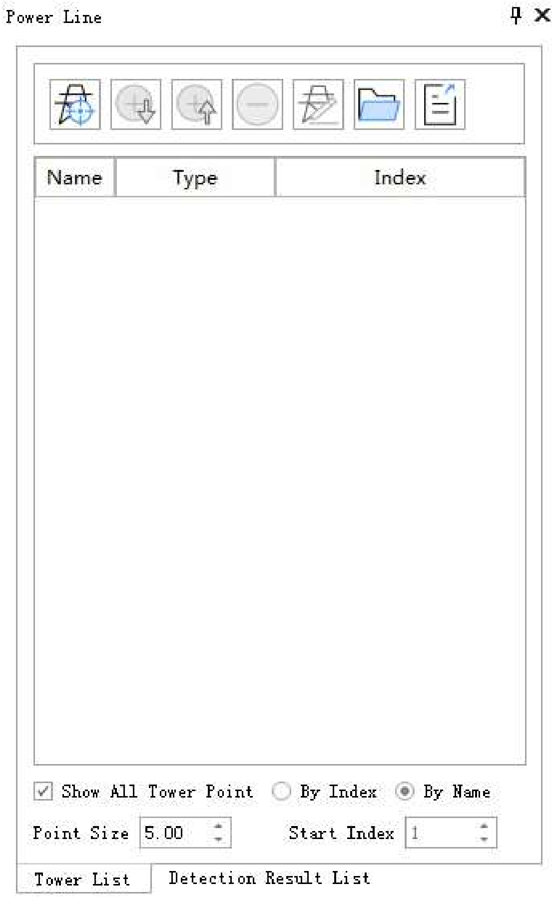
- **Select**: Whether to detect this class.
  - **Checked (default)**: Detect the danger point in this class.
  - **Unchecked (default)**: Do not detect the danger point in this class.
- **Class**: Select the number (from 1 to 32) of the class in the drop-down menu.
- **Detection Type**: Select the detection type in the drop-down menu.
  - **Clearance Distance**: Judge the point whether to be a danger point by the clearance distance. If the distance is less than this value, it is a danger point.
  - **Vertical Distance**: Judge the point whether to be a danger point by the vertical distance. If the distance is less than this value, it is a danger point.
  - **Horizontal Distance**: Judge the point whether to be a danger point by the horizontal distance. If the distance is less than this value, it is a danger point.
  - **Horizontal Distance / Vertical Distance**: Firstly judge the point whether to be a danger point by the horizontal distance. If the distance is less than set horizontal value, it is a danger point. If the distance is larger than set horizontal value, judge the point whether to be a danger point by the vertical distance.
  - **Horizontal Distance / Clearance Distance**: Firstly judge the point whether to be a danger point by the horizontal distance. If the distance is less than set horizontal value, it is a danger point. If the distance is larger than set horizontal value, judge the point whether to be a danger point by the clearance distance.
  - **Horizontal Distance / Vertical Distance / Clearance Distance**: Firstly judge the point whether to be a danger point by the horizontal distance. If the distance is less than set horizontal value, it is a danger point. If the distance is larger than set horizontal value, judge the point whether to be a danger point by the vertical distance. If the distance is less than the set vertical distance, it is a danger point. If the distance is larger than the set vertical distance, judge the point whether to be a danger point by the clearance distance then.
- **Add Row**: Add a new detection class record.
- **Delete Row**: Delete the selected row of detection class.
- **Add Column**: Add a new column for a severity level threshold. I.E., severe, important, and general.
- **Delete Column**: Delete the selected column.

- **Change Column Name**: Change the name of selected column.
- **Save**: Save the configuration to the current xml file.
- **Save as**: Save the configuration as a new xml file.

Mark Tower

Summary

After clicking in the point cloud data and choosing the tower type, user can save the result into LiTower file. After adding the position information of the tower, the software will automatically generate the index and name information of the tower. Index is a number starting from the starting value and incrementing by 1. By default, name is the same as Index. Tower type includes "None", "Tension Tower", and "Straight Line Tower". Double-click the row for each tower to jump to that tower in the displaying window. By checking or unchecking the checkbox in front of "display all tower points" to display or hiding the tower points in the window. Point size can be modified.



- **Display All Tower Points (checked by default)**: Display the tower names or indexes in the window. Uncheck this option to hide the all the tower names or indexes.
- **By Index**: Display the tower index.
- **By Name**: Display the tower name.
- **Point Size (meter) (default value is "5.0")**: Set the red marker sphere and text font size of the tower.

Select Tower

Click this button to start to pick the tower points. Click again to exit this function and save the

Click this button to start to pick the tower points. Click again to exit this function and save the current tower information to the tower.LiTower file in the working directory.

Add Tower After

Add a tower record after currently selected record.

Add Tower Before

Add a tower record before currently selected record.

Delete Tower

Delete the current selected tower record.

Modify Tower Position

Modify the position of the selected tower.

Input Tower File

Input the tower file and renew the tower list in LiTower, txt, csv, and kml file.

Export Tower File

Export the information in the current tower list to LiTower or kml file.

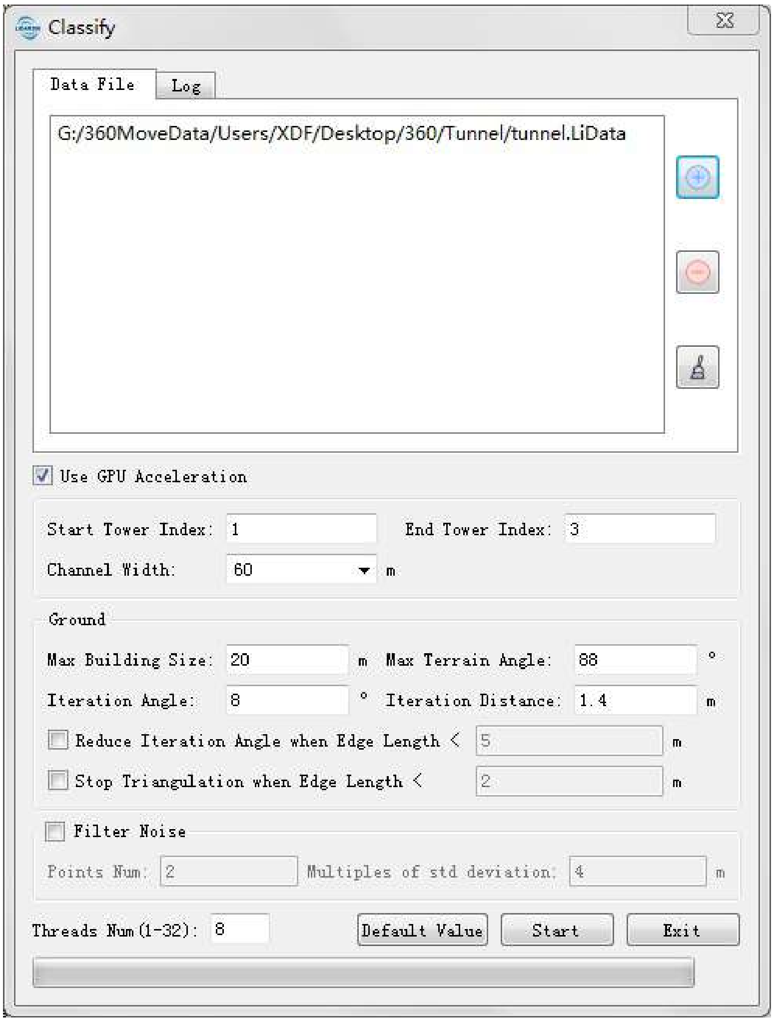
Classify

Summary

According to the tower file, clip and classify the point cloud data. Classify the point cloud into transmission tower, powerline, ground points, noise, and other selected classes.This function can process multiple data simultaneously. Click to add the data to be processed. Click to remove the selected data. Click to clear the data list. This function can perform segmentation, classification, and noise removal in one step, or user can select the function needed by checking the box in front of the name of each function.

Usage

Add the point cloud data, and click *Power Line* > *Classify*.



Settings

- **GPU Acceleration (Unchecked by default)**: Use GPU to accelerate when voltage is more than 500KV.
- **Corridor Width (meters) (default is "60")**: Width extending to both sides of the powerline. The software can record the user's current settings and automatically restore the settings when opened again.
- **Starting Tower Index**: Towers with index numbers smaller than this value are not involved in the calculation. The default value is the minimum Index in the tower file.
- **Ending Tower Index**: Towers with index numbers larger than this value are not involved in the calculation. The default value is the maximum Index in the tower file.
- **Classify Ground Points**: Please refer to the Classify Ground Points in the Classify toolbox for the detailed information of parameter settings. The software will not classify the ground points if this option is unchecked.
- **Remove Noise (Checked by default)**: Classify the noise points. The software will not classify the noise if this option is unchecked.
- **Number of Thread (default value is "4")**: Set the number of thread (1 to 32) to run this function. The software can record the currently setting and when the user open this function again, the setting will be kept.
- **Default**: Restore the default parameter settings.

> Note: The segmented data will be stored in the output path folder. The name of the file will be (Smaller Tower Index_Larger Tower Index). In order not to impact the following steps, it is highly recommended not to change the file name. After the automatic classification, user need to check the accuracy of the result and manually modify it in the Profile Tools.

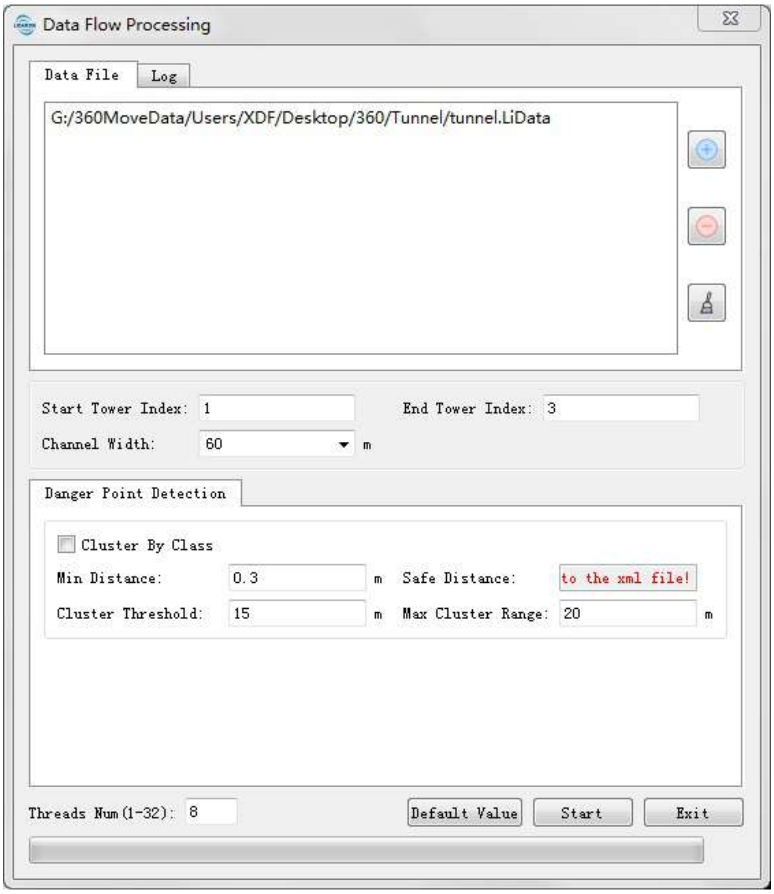
Detect Danger Points

Summary

Batch process the point cloud file to generate the danger points images and reports based on the tower file. The interface of detect danger points function is shown as follow. This This function can process multiple data simultaneously. Click to add the data to be processed. Click to remove the selected data. Click to clear the data list.

Usage

Click *Power Line > Detect Danger Points*



Settings

- **Starting Tower Index**: Towers with index numbers smaller than this value are not involved in the calculation. The default value is the minimum Index in the tower file.
- **Ending Tower Index**: Towers with index numbers larger than this value are not involved in the calculation. The default value is the maximum Index in the tower file.

- **Corridor Width (meters) (default is "60")**: Width extending to both sides of the powerline. The software can record the user's current settings and automatically restore the settings when opened again.

- **Detection Parameters**: Detect the clearance danger points according to the customized

  - **Cluster by Class**: Unchecked by default. Used to control whether to cluster the result by class.If this option is checked, the danger points will be clustered into different groups if they are in different classes; otherwise, the class information will not influence the clustering process. The software can record the user's current settings and automatically restore the settings when opened again.
  - **Min Distance (meters) (default value is "0.3")**: Points with clearance distance less than this distance are not detected as danger points, as they are considered to be noise. The software can record the user's current settings and automatically restore the settings when opened again.
  - **Safety Distance (meter) (default value is "4.5")**: If the distance between the detected point and the powerline point is greater than the minimum distance and less than or equal to the safe distance, the point is considered to be a danger point. If the xml file is not set, user needs to enter the safety distance; otherwise, this parameter is extracted from the xml file. When using the xml to set the parameter, user can set different safety distances for different classes, and for different severity level.
  - **Cluster Threshold (meter) (default value is "15.0")**: The maximum spatial separation distance when the detection results are clustered. This value should be smaller than the maximum clustering range. Three-dimensional Euclidean clustering is used to cluster the danger points. The software can record the user's current settings and automatically restore the settings when opened again.
    - **Maximum Clustering Range (meters) (default value is "20.0")**: After the detection results are clustered, if the length along the power line direction is greater than this value, they are divided into multiple dangerous point clusters. The software can record the user's current settings and automatically restore the settings when opened again.
- **Number of Thread (default value is "4")**: Set the number of thread (1 to 32) to run this function. The software can record the currently setting and when the user open this function again, the setting will be kept.
- **Default**: Restore the default parameter settings.

Clear User Data

Summary

Clear all the danger point detection result in the point cloud. After execute this function successfully, all the danger points (in red) will be changed into non-danger point.

Usage

Power Line > Clear

Settings

- **Number of Thread (default value is "4")**: Set the number of thread (1 to 32) to run this function. The software can record the currently setting and when the user open this function again, the setting will be kept.

Geological Analysis

Geological Analysis provides automatical as well as manual analysis tools for flow direction, flow accumulation, fill, dip and strike.

- Raster Flow Direction
- Raster Flow Accumulation
- Raster Fill
- Query Dip and Strike

Raster Flow Direction

Summary

The tool use D8 algorithm to calculate the raster of flow direction for each pixel.

D8 algorithm is to build flow direction from each pixel to downhill neighbouring pixel of the most steep angle.The output of pixel value is within 1-255 of Integer format.

> Note:
> 1.If the pixel Z value is the lowest among the eight neighbouring pixels, it will be assigned to the minimum value among its neighbourings. And flow direction is to this pixel. If multiple neighbouring pixels have the same lowest value, it will still be assigned to the minimum value among its neighbourings.But its flow direction will be definded via one of the following method. This can filter abnormal low noisy point.
>
> 2.If the z-value of a pixel changes the same in multiple directions, and the pixel is part of a sunken point, the flow direction of the pixel will be considered undefined. At this time, the value of the pixel in the output flow direction raster will be the sum of these directions. For example, if the z-value changes the same to the right (flow direction = 1) and downward (flow direction = 4), the flow direction of the cell is 1 + 4 = 5. You can use the sink tool to mark pixels with undefined flow directions as sunken points.
>
> 3.If the z-value of a pixel changes the same in multiple directions, and the pixel is not part of a depression, a lookup table that defines the most probable direction will be used to specify the flow direction. See Greenlee (1987).
>
> 4.The output D8 descent rate raster data will be calculated based on the z-value change rate of the path length between the pixel centers and expressed as a percentage. For adjacent pixels, this is similar to the percentage slope between pixels. When crossing a flat area, the distance becomes the distance to the nearest pixel with a lower elevation. The result is a graph of the percentage increase in elevation in the steepest descent path from each pixel.
>
> 5.When calculating the D8 descent rate raster data of a flat area, in order to improve performance, the distance to the diagonally adjacent pixel (1.41421 *cell size) is approximately calculated as 1.5* cell size.

Description

The key to obtaining the hydrological characteristics of the surface is to determine the flow direction from each pixel in the grid. This can be done with the flow direction tool.

The tool takes the surface as input and outputs a raster showing the direction of flow from each pixel. If the output rate of decline raster data option is selected, an output raster expressed as a percentage will be created, showing the maximum rate of change in elevation from each pixel along the flow direction to the length of the path between the pixel center. If the Force all edge cells to flow outward is selected, all cells at the edges of the surface raster will flow outward from the surface raster.

The flow direction is determined by the steepest descent direction or the maximum descent direction from each pixel. The flow direction is calculated as follows:

```
maximum_drop = change_in_z-value / distance * 100
```

Calculate the distance between the centers of the pixels.

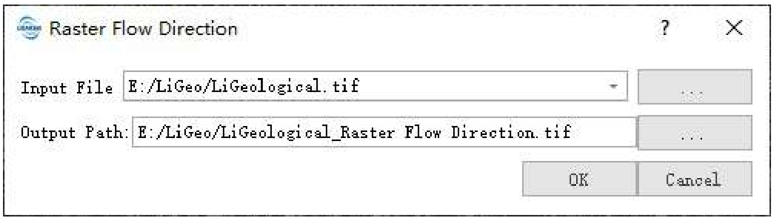
1.If the pixel size is 1, the distance between two orthogonal pixels is 1, and the distance between two diagonal pixels is 1.414 (the square root of 2). If multiple pixels have the same maximum descending direction, the range of adjacent pixels will be expanded until the steepest descending direction is found. After finding the direction of the steepest descent, the output pixel will be encoded with a value representing that direction.

2.If all neighboring pixels are higher than the pixel to be processed, the pixel to be processed will be regarded as noise and filled with the lowest value of its neighboring pixels. The pixel to be processed has a flow direction towards itself. However, if the single pixel sink point is located near the actual edge of the grid or has at least one NoData pixel as an adjacent pixel, it cannot be filled due to insufficient neighborhood information. To treat a pixel as a real single-pixel sink, all neighborhood information must exist.

3.If two pixels flow into each other, they are both meeting points and have an undefined flow direction. This method of obtaining flow direction through a digital elevation model (DEM) was introduced by Jenson and Domingue (1988).

Usage

Click *Geological Analysis >Raster Flow Direction*.



Parameters

- **Input File**：Select input tiff file.
- **Output Path**：Select output path.

```
@inproceedings{
    author={Sch\"{o}Greenlee, D. D.},
    title={Raster and Vector Processing for Scanned Linework},
    booktitle={Photogrammetric Engineering and Remote Sensing },
    year={1987},
}

@inproceedings{
    author={Sch\"{o}Qin, C., Zhu, A. X., Pei, T., Li, B., Zhou, C., & Yang, L},
    title={An adaptive approach to selecting a flow partition exponent for a multiple flow direction a
lgorithm},
```

```
    booktitle={International Journal of Geographical Information Science},
    year={2007},
}
```

Raster Flow Accumulation

This tool is used to calculate raster of flow accumulation by pixel.

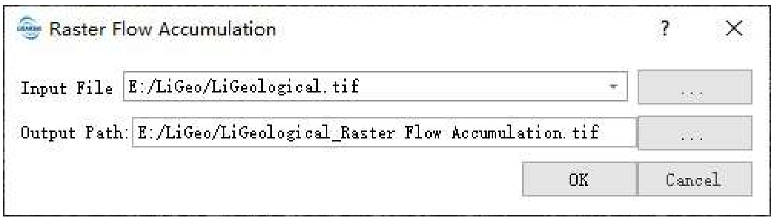
> Notes:
>
> 1.The result will be a flow accumulation raster, each pixel value is calculated by all the uphill pixels by cumulative weight.
>
> 2.D8 flow direction modeling algorithm is used in this tool.
>
> 3.If the Raster Flow Direction is not used for the input raster, the flow accumulation might be annular and go into endless loop.
>
> 4.Pixels of undefined flow direction will only take in uphill pixels, they will not form any downhill flow. For D8 algorithm, if the pixel is not of value 1、2、4、8、16、32、64 or 128, we consider the pixel flow-undefined.
>
> 5.Flow accumulation is based on the amount of uphill pixels, the pixel itself will not be considered in.
>
> 6.The large value pixels can indicate rivers.
>
> 7.The low value pixels can indicate ridges.

Summary

This tool is used to calculate raster of flow accumulation by pixel. The pixel value is the cumulative weight of all the uphill pixels.

Usage

Click *Geological Analysis >Raster Flow Accumulation*



Parameters

- **Input File**：Select input tiff file.
- **Output Path**：Select output path.

```
   @inproceedings{
       author={Sch\"{o}enson, S. K., J. O. Domingue},
       title={Extracting Topographic Structure from Digital Elevation Data for Geographic Information System
Analysis},
       booktitle={Photogrammetric Engineering and Remote Sensing},
       year={1988},
```

```
}

@inproceedings{
    author={Sch\"{o}Tarboton, D. G., R. L. Bras, I. Rodriguez-Iturbe},
    title={On the Extraction of Channel Networks from Digital Elevation Data},
    booktitle={Hydrological Processes},
    year={1991},
}
```

Raster Fill

Raster Fill is used to improve the raster by filling in the depression of the raster.

Depression pixel refers to the pixel lower than other surrounding pixels, which is unable to define the flow direction.The pour point is the boundary pixel with the lowest elevation relative to the confluence area of the depression pixel. If the depression is filled with water, the water will pour out from that point.

1. Z defines the maximum elevation difference between depression pixel and pour point, which decides the minimum depth of the depression area need to be filled.

   For example, suppose the elevation of the pour point in a depression area is 210 feet, and the deepest point of the depression is 204 feet (depth:6 feet). If the z limit is set to 8, this depression will be filled. However, if the z limit is set to 4, the area will not be filled because the depth exceeds the limit value, and it is regarded as a valid depression pixel.

2.All qualified depression pixels will be modified to the elevation of pour point.

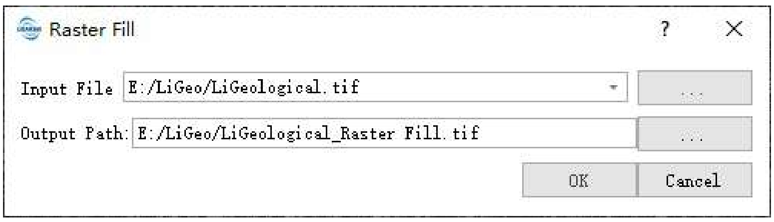
3.The more the depression points, the longer the processing time.

Summary

The Raster Fill tool uses functions equivalent to the focus flow, flow direction, depression, watershed, and area fill tools to locate and fill depressions. The execution of the tool will iterate until all depressions within the specified z limit are filled. While filling the depressions, other depressions may be created at the boundaries of the filled area, and these depressions will be removed in the next iteration.

Usage

Click *Geological Analysis> Raster Fill*。



Parameters

- **Input File**：Select input tiff file.
- **Output Path**：Select output path.

> Note: If the raster data is of integer type, the output filled raster data is also of integer type. If the input raster data is of floating point type, the output raster data is also of floating point type.

```
@inproceedings{
    author={Sch\"{o}Planchon, O.,Darboux, F},
    title={A fast, simple and versatile algorithm to fill the depressions of digital elevation models},
```

```
    booktitle={Catena},
    year={2002},
}

@inproceedings{
    author={Sch\"{o}Tarboton, D. G., R. L. Bras,I. Rodriguez–Iturbe},
    title={On the Extraction of Channel Networks from Digital Elevation Data},
    booktitle={Hydrological Processes},
    year={1991},
}
```

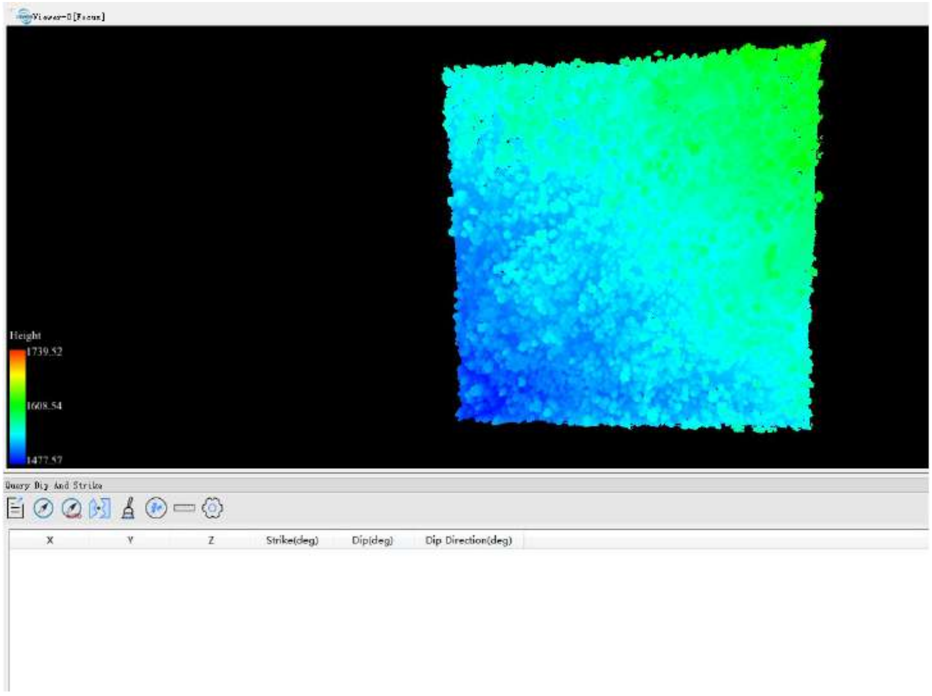
Query Dip and Strike

Summary

This tool is used to calculate dip and strike automatically or based on the point cloud selected manually. Related analysis tool and result report are also available.

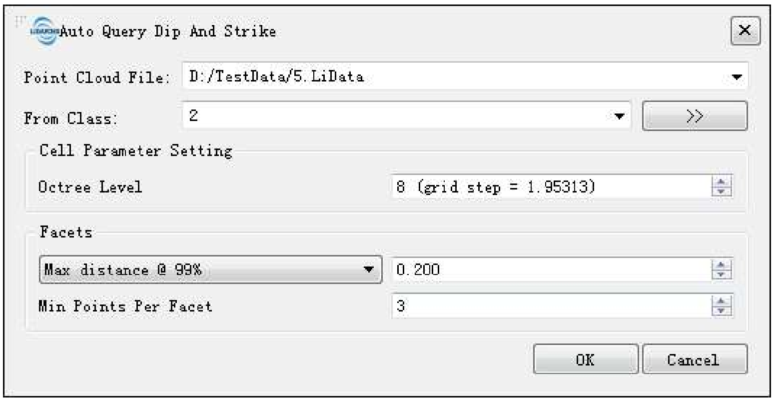
Usage

Load the point cloud in the viewer. Click *Geological Analysis* > *Query Dip and Strike*, and the toolbar will show in the viewer.



In the Query Dip and Strike window, there are several buttons: Export Data, Query Dip and Strike, Automatically Query Dip and Strike, Clear data, Statistic Report,Adjust radius and Settings.

 **Export Data**：Save data in the table as CSV file.
 **Query Dip and Strike**： By clicking this button, a round selection box will show in the point cloud viewer.You can also adjust the radius of the circle. After selecting the point cloud by the round box, the Dip and Strike info will show as 3D model in the viewer. Meanwhile, a query record will add in the table below.
 **Auto Query Dip and Strike**：This tool can automatically calculate Dip and Strike, and add the record in the table below.

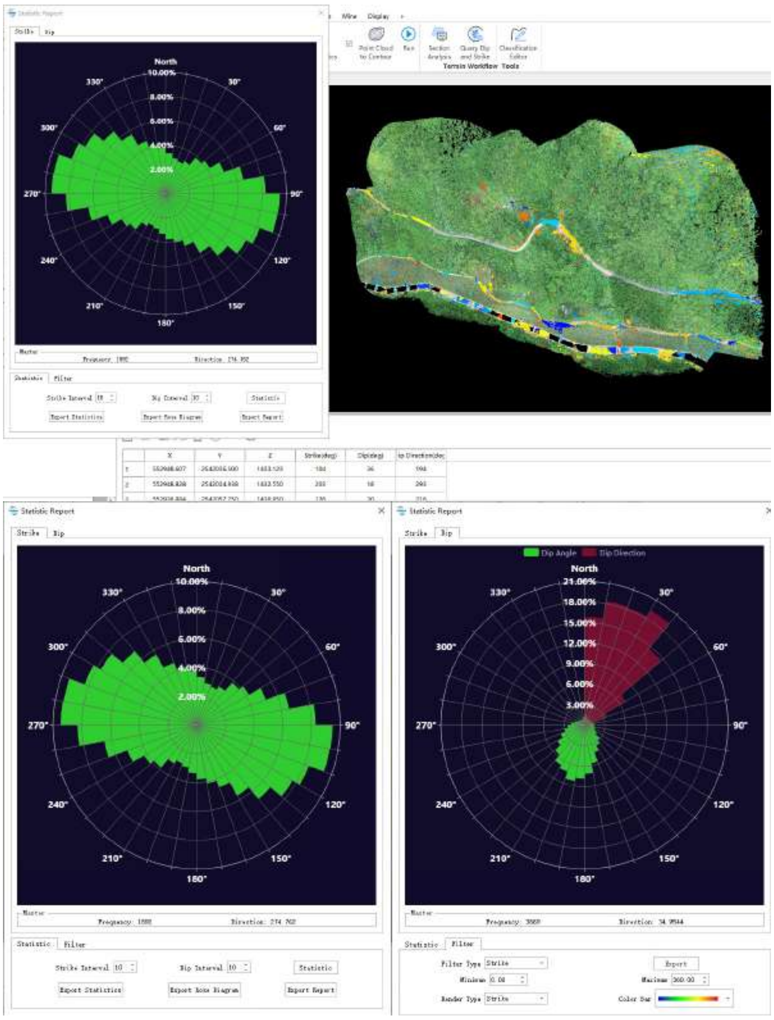


- **Point Cloud File**：Select point cloud file in the drop-down list.
- **From Class**：Select the class of point cloud to be calculated, default setting is ground point.
- **Cell Size**：Create an octree for the point cloud. The higher the level, the fewer the number of point clouds contained in each octree node. The specific results of the segmentation are as follows: the larger the value is set, the finer the queried facet, otherwise the larger the facet.
- **Facets**：Parameters of Facets.
  - **Facets Type**：Different type of Facets.
    - **Max Root mean square error**：The maximum root mean square of the distance between the point cloud of the cluster and the fitted plane in the segmentation.
    - **Max distance@68%**：The maximum distance of the top 68% of the distance between the point cloud of the cluster in the segmentation and the fitted plane.
    - **Max distance@95%**：The maximum distance of the top 95% of the distance between the point cloud of the cluster in the segmentation and the fitted plane.
    - **Max distance@99%（by default）**：The maximum distance of the top 99% of the distance between the point cloud of the cluster in the segmentation and the fitted plane.
    - **Max distance**： The maximum distance between the point cloud of the cluster in the segmentation and the fitted plane.
  - **Distance Threshold**： The larger the threshold, the more points will be segmented into a facet, which means there will be more large facets.
  - **Min Points Per Facet**：The minimum points of a facet. The larger the value, the more the points of a facet.
    - **Convex Hull**: Uses the Convex Hull as the shape of each generated structural surface.
    - **Concave Hull**: Uses the Concave Hull as the shape of each generated structural surface.

Merge: Click the function button to select a polygon in the window as the target polygon. Then pick other polygons in the window, which will be merged into the target polygon. Simultaneously, the corresponding data in the table below will be deleted. Right-click to finish.

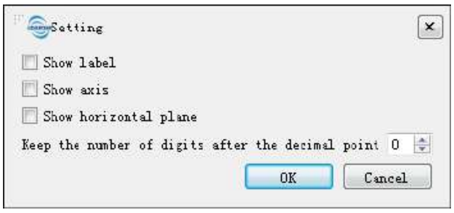
Clear Data：Clear data in the viewer and the table.

Statistic Report：Click the button to do statistics and filter on the dip and strike. Set the interval for dip and strike statistics. The rose diagrams and corresponding statistical reports can also be generated. When filtering, you can choose different filtering types and rendering types, and filter by setting the minimum and maximum angle threshold.



Length Measurement: Select multiple surfaces to calculate the distance between them.

Settings：Checking or unchecking the box to decide whether to show label, axis and horizontal plane.



Photo

The photo module can recover the detailed 3D geometric structure of objects based on overlapping image data, merge point clouds, and generate a series of standard mapping products. The image module can process nadir, oblique, and multispectral images. Core functions include feature point extraction and matching, block adjustment, self-calibration of cameras, generation of DEM(Digital Elevation Models)/DSM(Digital Surface Models) , intelligent orthoimage mosaicking, orthoimage generation, seamline editing, and more.

Photo Module Workflow

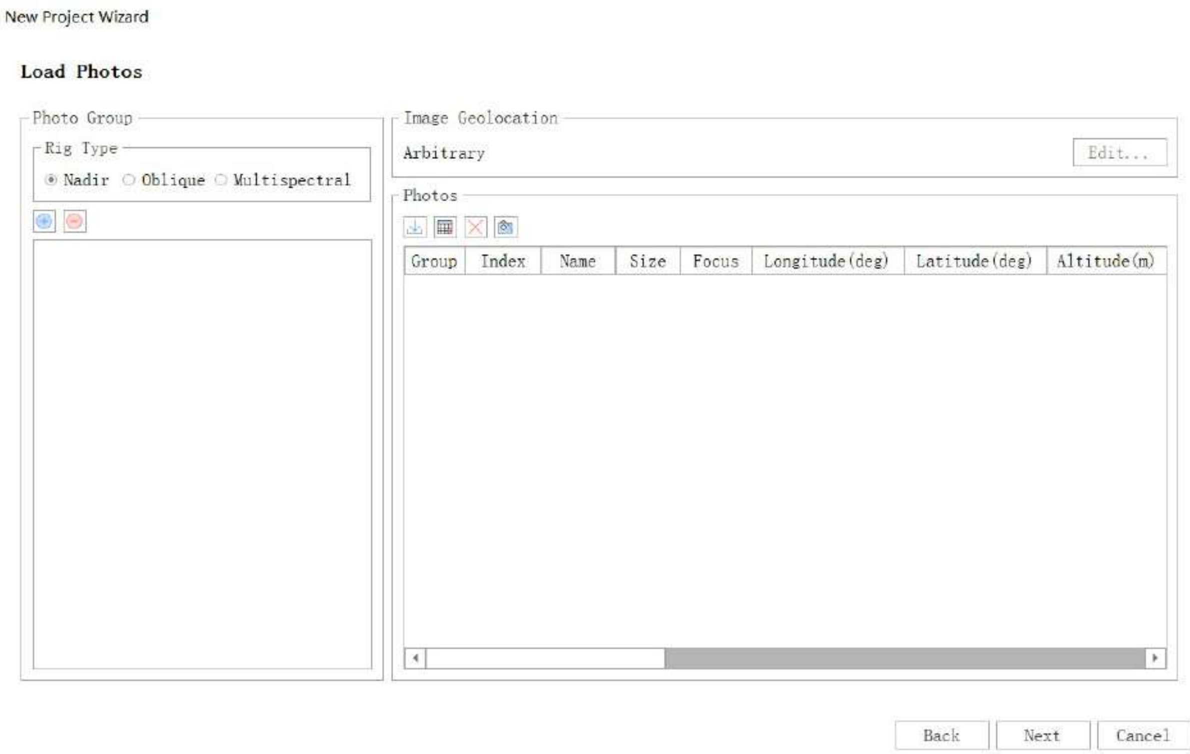
- New Project
- Import POS
- Camera Calibration
- Ground Control Points
- Export Camera
- Photo Workflow Builder
- Image and Point Cloud Registration
- Update Alignment
- Undistort
- Defogging
- Generate Report
- Dom Edit
- Tie Points and Image Filtering

New Project

Function Overview

Create a new image project to execute subsequent computational processes. The main operations include selecting a path, adding camera groups, POS operations, and setting the coordinate system.

Usage



1. Click the " New Project" button to open the new project location.
2. Select the path for the new project and click "Next" to enter the image loading page.
3. The image loading page requires the following steps. After completing the data and POS settings, click "Next" to select the output coordinate system.
   - **Select Rig Type**: Choose the corresponding data type based on the input image data. Currently, nadir images, oblique images, and multispectral images are supported.
   - **Add Photo Group**: Image data within each camera group uses a common set of camera parameters. Click the plus sign to add a group and select at least one image to add. Click the minus sign to delete an added group. After adding a group, right-click on the group and select "Add Image" to add more images to the group. At least five images need to be added to execute the algorithm process properly.
   - **Photo Operations**: Image operations include POS operations, deleting images, and camera calibration.
     - **POS Operations**: If the imported image data includes EXIF latitude and longitude information, POS information will be imported automatically. To import external POS files,

click the "Import POS" button and select the POS file to import. Refer to the Import POS - Import by Filename section for specific operations. Click "Clear POS" to remove the imported POS. Imported POS data can be viewed in the image list.

- **Delete Images**: Select one or more images from the image list and click the "Delete Images" button to remove the added images from the camera group.
- **Camera Calibration**: Refer to the Camera Calibration page for specific operations. When the camera type is multispectral, the main band can be set here.

- **POS Coordinate System**: If there is no POS data, the coordinate system is set to a local coordinate system. If POS information is present, the corresponding coordinate system for the POS needs to be set. When data falls within the latitude and longitude range, the WGS84 coordinate system is recommended by default. To set manually, click the "Edit" button, select "More" in the coordinate system dropdown list, and choose the appropriate coordinate system in the pop-up dialog box.

4. Select the output coordinate system. This coordinate system is for the final output and can only be a projected coordinate system. If POS was not set during image import, it can only be set to a local coordinate system here. If the POS data coordinate system is WGS84, the corresponding UTM projected coordinate system will be recommended. To set manually, select "More" in the coordinate system dropdown list, and choose the appropriate coordinate system in the pop-up dialog box.

5. After confirming the settings, click "Finish" to complete the creation of the new project. The new project will be added to the software.

Import POS

Function Overview

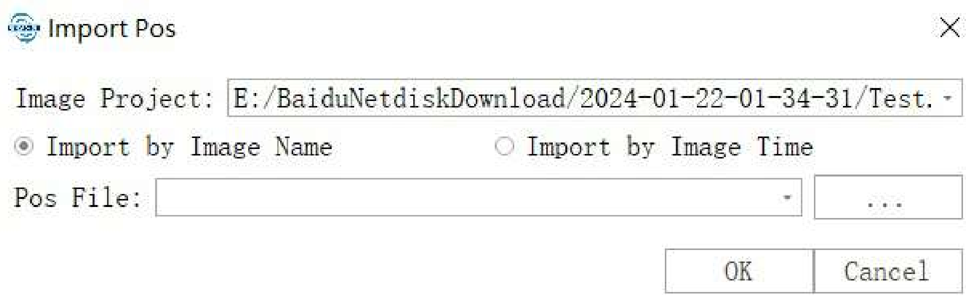
In the processing workflow, POS information can be used to accelerate image processing and determine the position of images in the real world.

Purpose

POS information serves three main purposes:

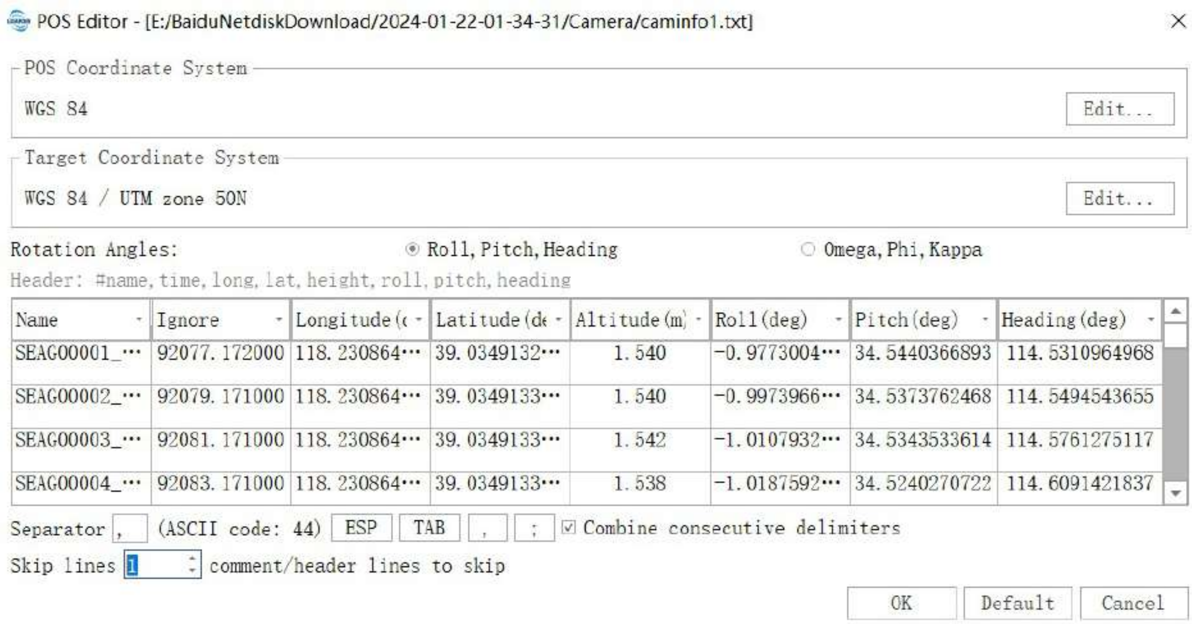
1. **POS information can be used in the image matching process for pair selection (georeferencing), increasing processing speed.**
2. **Used in the alignment updating process.**
3. **As a weighting factor, it participates in bundle adjustment optimization, enhancing the robustness of the processing results.**
    - When processing large-scale, high-resolution images, only considering image information for image alignment may sometimes not yield an optimal result (due to outliers caused by mismatches, unavoidable errors in image distortion parameter estimation, etc.). To improve the robustness of image alignment results, a strategy of joint adjustment of POS and image is used during the alignment process, making the setting of POS accuracy crucial.
    - Note: When the external camera's internal parameter accuracy is insufficient or the original image quality is poor, the camera position in the image alignment results may deviate significantly from the corresponding GPS value. Adjusting the POS weight can significantly improve the image alignment results (e.g., adjusting the camera distance accuracy from the default 10 meters to 1 meter).

Usage



Click the "Import POS" button. First, select the project file to import the POS into from the dropdown list (only project files that have been added to the software can be selected). Then, based on the format of the POS to be imported, choose to import by filename or by time. Importing by filename means the POS file contains a column with the corresponding image names, which can be matched with each image. Importing by time uses the camera shutter time file (.cam) and trajectory file (.traj) to interpolate the trajectory and determine the camera's POS information.

Import by Filename

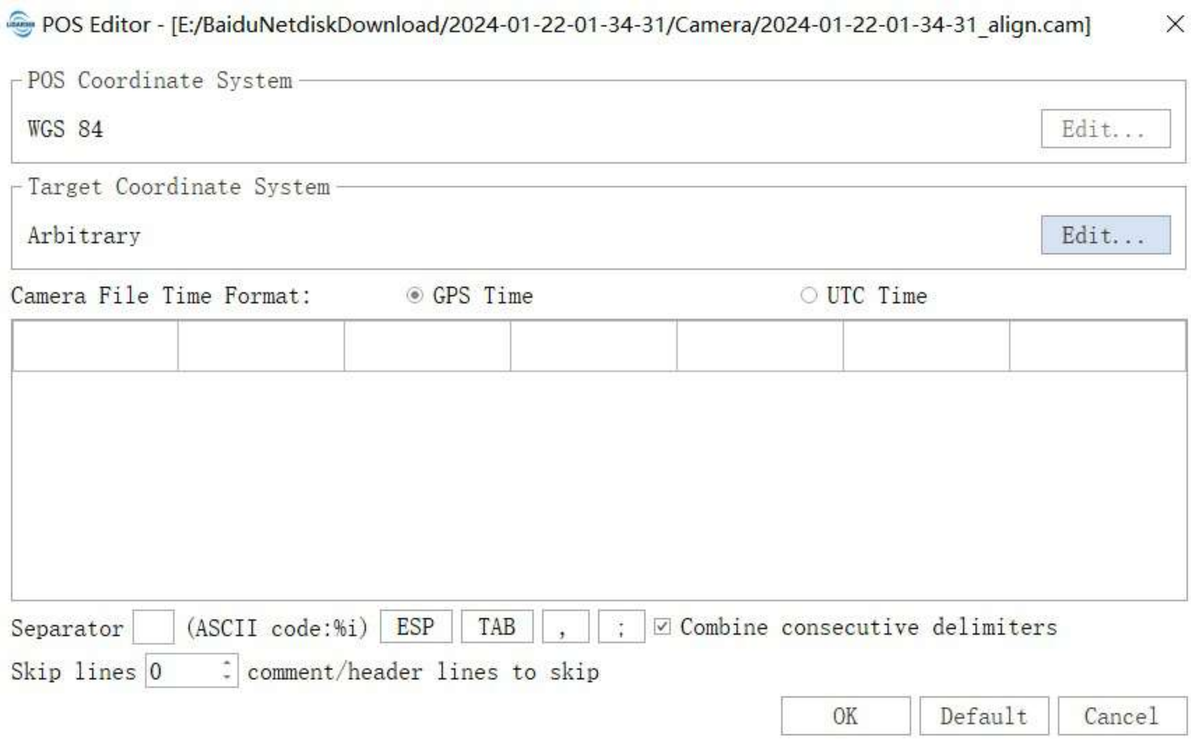


Select "Import by Filename," choose the POS file to import (supports TXT and CSV files), and click "Next." A dialog box will appear to set the POS format.

- **Set POS Coordinate System and Target Coordinate System**: Refer to the Coordinate System section.
- **Separator Adjustment**: If the file data is not displayed correctly in the table, adjust the separator (optional). If the POS file is not displayed effectively in the list after loading, adjust the separator at the bottom of the table to effectively separate and display the data. Currently, TAB, "\t", "\n", "," separators, and combinations of these separators are supported. By default, POS data will be separated using combined separators. If the file separator is not among the four types mentioned above, you can enter an appropriate separator in the separator text box to effectively divide the data.
- **Rotation Angle Type (optional)**: Set the angle type based on the imported data. If you do not want to use angle information, set the column header for the angle column to "Ignore."
- **Skip Invalid Rows (optional)**: Adjust to ignore the first N rows of the POS file. By default, no rows are skipped.
- **Column Headers (required)**: If the column headers do not match the current column data, click the dropdown list of column headers to select the appropriate column header for the current column. If the selected POS coordinate system is a geographic coordinate system, column headers should select longitude, latitude, and altitude; if the selected POS coordinate system is a projected coordinate system, column headers should select Easting, Northing, and altitude.

After confirming that the POS information for each photo is set correctly, click "Confirm" to complete the import and save the POS information into the project file.

Import by Time



Select "Import by Image Time," choose the corresponding camera shutter time file (.cam) and trajectory file (.traj), and click "Next." A dialog box will appear to set the POS format. The first column is the file name column of the imported images, for easy correspondence with the camera shutter time file. The next six columns are the POS position and angle columns. After successfully interpolating the trajectory file by selecting the time column, the specific values will be displayed in these columns.

- **Set POS Coordinate System and Target Coordinate System**: Refer to the Coordinate System section.
- **Separator Adjustment**: The operation is the same as importing by filename.
- **Skip Invalid Rows (optional)**: Adjust to ignore the first N rows of the camera shutter file. By default, no rows are skipped.
- **Camera Time Type (required)**: Set the time type of the camera shutter time file, which is divided into GPS time and UTC time.
- **Column Headers (required)**: The time column must be selected to obtain position and angle data by interpolation.

After confirming that the interpolated POS information for each photo is correct, click "Confirm" to complete the import and save the POS information into the project file.

Coordinate System

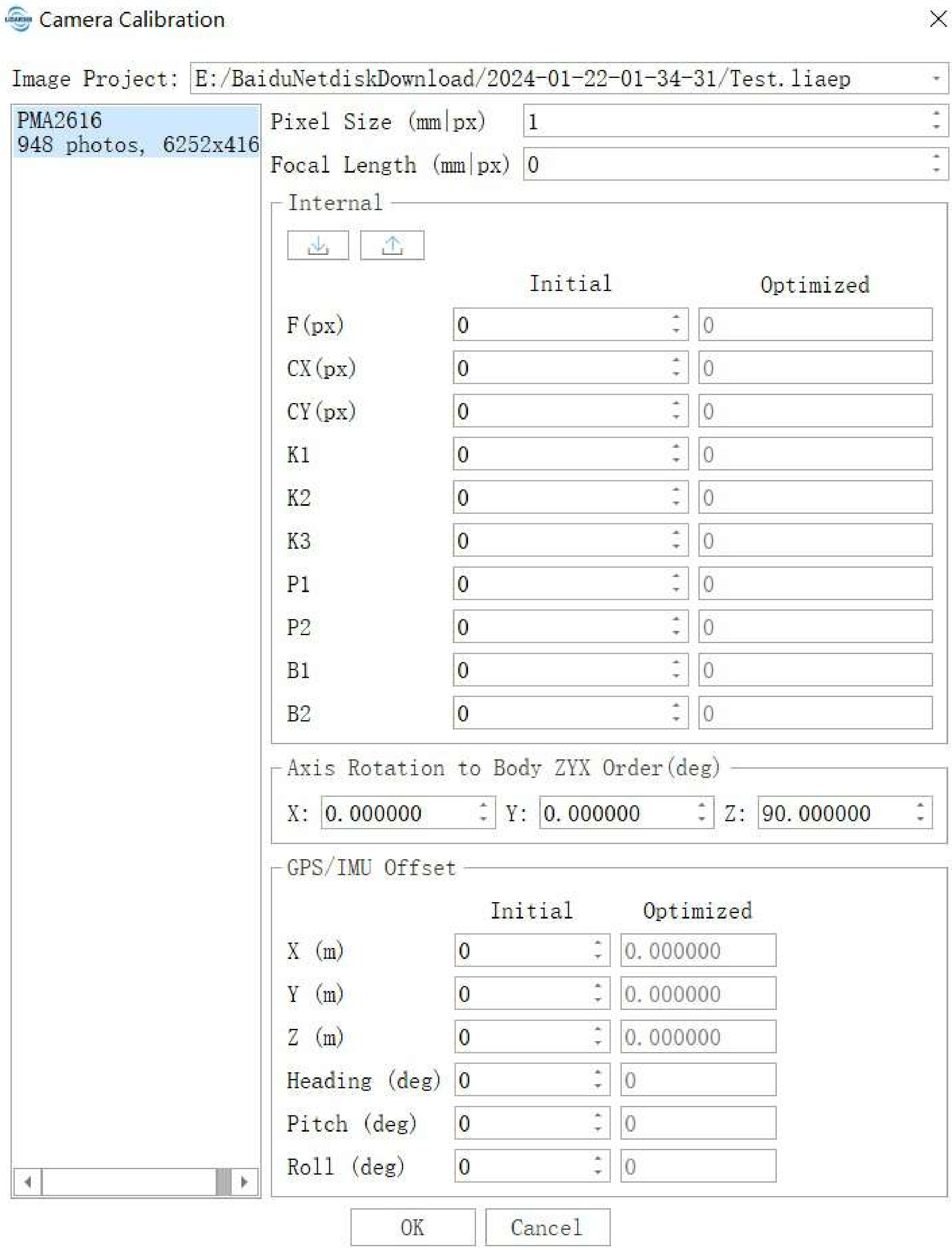
Two coordinate systems need to be set: the POS data coordinate system and the target coordinate system. The POS data coordinate system indicates the coordinate system in which the POS file was collected; the target coordinate system indicates the coordinate system used in subsequent processing steps and must be a projected coordinate system. After selecting the POS data coordinate system, the target coordinate system will automatically be recommended as the corresponding UTM projected coordinate system. To change the coordinate system, click "Edit," and in the popup dialog box, select "More" in the coordinate system dropdown list. Then, select the appropriate coordinate system in the subsequent dialog box.

Camera Calibration

Function Overview

Camera lenses inevitably have some distortion, which affects image processing, especially in high-precision measurement applications. Therefore, it is often necessary to calibrate the camera parameters to improve calculation accuracy using the calibrated camera parameters. This function allows using the calibrated camera parameters as initial parameters in image alignment.

Usage



1. Select the image project that needs camera calibration. Only image project files that have been opened in the software are supported.
2. Switch to the camera group that needs to set camera parameters on the left.
3. You can manually edit the source data, including various parameters such as pixel size, focal length, camera intrinsic parameters, installation angles, and installation errors.
4. If the image alignment operation has been completed for the image project, the optimized parameters will be filled in the optimization column.
5. Camera intrinsic parameters support import and export operations. Importing intrinsic parameters will fill in the source parameters, and exporting intrinsic parameters can choose to export source

parameters or optimized parameters.
6. If the selected image project file is multispectral data, the main band of the project can be set here.

Camera Parameters

The camera model determines the transformation relationship between the image space coordinate system and the image plane coordinate system. The origin of the image space coordinate system is at the projection center. The Z-axis points in the direction of the camera's line of sight, the X-axis points to the right, and the Y-axis points downward. The origin of the image plane coordinate system is at the top-left corner of the image plane, with the center coordinate of the top-left pixel being (0.5, 0.5). The X-axis points to the right, and the Y-axis points downward. The image plane coordinate system is measured in pixels.

The variables involved are as follows:

(X, Y, Z) - Coordinates of a point in the image space coordinate system

(u, v) - Coordinates of the projection point in the image plane coordinate system (in pixels)

(cu, cv) - Principal point offset

(K1, K2, K3) - Radial distortion parameters of the camera

(P1, P2) - Tangential distortion parameters of the camera

(b1, b2) - Affine and non-orthogonal parameters

(w, h) - Width and height of the image (in pixels)

f - Focal length of the camera (in pixels)

The transformation formula is as follows:

$x = X / Z$

$y = Y / Z$

r = √(x^2+y^2)

x' = x(1+K r + K r + K r) + P (r + 2x) + 2P xy

y' = y(1+K r + K r + K r) + P (r + 2y) + 2P xy

u = w*0.5 + c + x'f + x' b + y'b

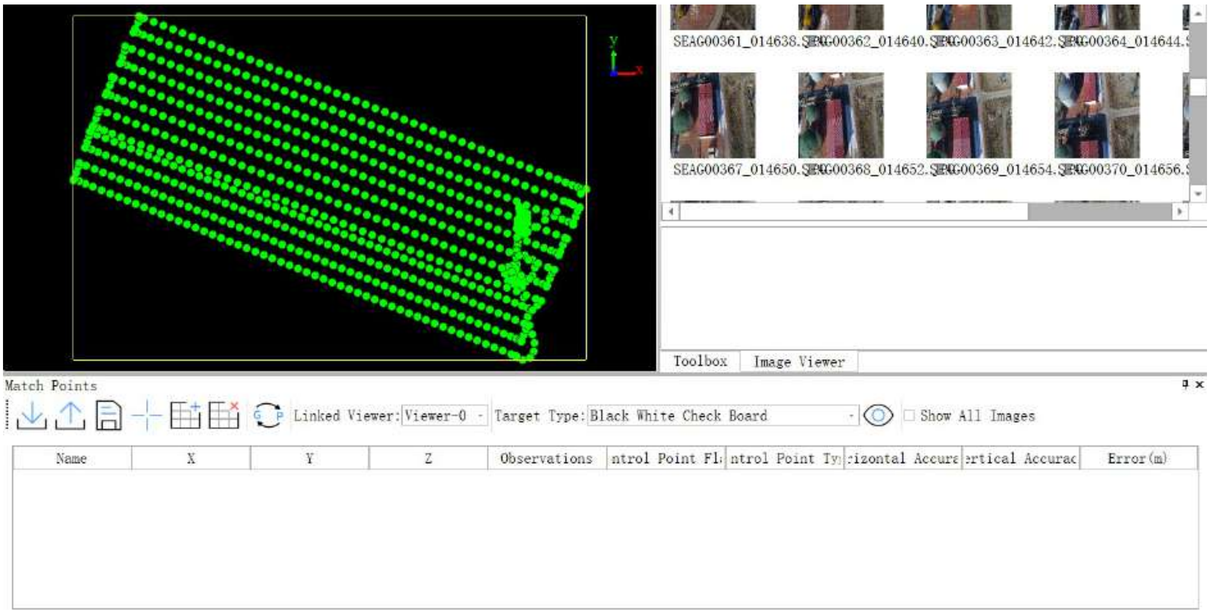
v = h*0.5 + c + y'f

Ground Control Points (GCP)

Function Overview

Using Ground Control Points (GCPs) can improve the quality and absolute accuracy of image alignment. After editing the GCPs, you can use the update alignment function to update the image alignment results to the GCP coordinates. It is recommended to perform control point editing operations on projects that have already completed image alignment.

Usage



The basic process of marking ground control points is as follows:

1. Click the Ground Control Point button and select the image project file to be edited in the pop-up dialog. Only image project files that have been added to the software can be selected.
2. Click OK to start two windows: the control point list window and the image window.
3. Import or mark the data objects in the window to obtain the coordinates of the control points. All imported or marked control points will be listed in the control point list window.
4. Select a control point from the list, choose an image from the candidate images, and click the corresponding observation position of the control point in the single image window to complete a marking.
5. Repeat the process to complete the marking of all control points. Click the save button to save the marking results into the image project file for subsequent operations.

Control Point Prediction

For image projects that have completed image alignment, when a control point is selected, the images and their predicted points where the control point might appear will be added to the image window based on the control point coordinates, alignment results, and existing observations for marking.

> Note: The coordinate system of the control points must be set correctly, otherwise, prediction failure or incorrect subsequent alignment updates may occur.

Control Point List Window

Import Control Points

Two types of control point files are supported: control point coordinate files (.txt or .csv) and control point observation export results (.xml). The control point observation export result is generated through the export function and can directly import control point coordinates, attributes, and observations together. For control point coordinate files, select the control point file in the import interface, and set the format in the pop-up dialog.

- Coordinate System: Set the coordinate system of the imported control points.
- Skip Rows: Skip the number of invalid rows at the beginning of the file (such as comments), default is no skip.
- Column Header: Click the column header, and specify the attribute of each column in the drop-down list, such as serial number, latitude and longitude, elevation.
- Separator: Effectively split the data using the separator. Supports four types of separators: space, tab, comma, and semicolon, and their combined separators. The default is the combined separator. If the file separator is not one of the above four types, you can enter a custom symbol in the separator text box.

  > Note: Importing control points will clear all control points in the current control point list.
  >
  > ## Export Control Points
  >
  > Export the existing control points and their attributes, observations, and other information to an XML file, which can be imported again using the import function.
  >
  > ## Save
  >
  > Save the existing control points and their attributes into the image project file.
  >
  > ## Select Point
  >
  > In the connected window, select the position of various data objects, and use the coordinate position of the selected point to update the coordinates of the currently selected control point.
  >
  > ## Add Point
  >
  > Add an empty control point.
  >
  > ## Delete Point
  >
  > Delete the currently selected control point.
  >
  > ## Coordinate System

> Set the coordinate system and mode of the control points. To modify the coordinate system, select more in the drop-down box, and choose the corresponding coordinate system in the pop-up dialog. Control point modes are divided into rigid and non-rigid. High-precision control points are recommended to use the non-rigid mode, while low-precision control points are recommended to use the rigid mode.
>
> ### Linked Viewer
>
> The select point function requires setting a connected window to operate under that window. Therefore, when there are multiple windows, you need to select the window here.
>
> ### Target Type
>
> Perform image target detection on the selected control points. In the drop-down box, you can select the type of target to be detected. Click the detect button to start detection. Successfully detected images will directly add the results to the control point observations. Note: The target detection function searches near the predicted value of the control point and may fail to detect.
>
> ### Show All Images
>
> Check this option to add all images as candidates to the image window.
>
> ### Control Point List
>
> The control point list lists all current control points. Selecting a row will jump to the coordinate position of that control point. Double-click the point name cell, coordinate cell, or plane elevation accuracy cell to modify the corresponding value. The drop-down boxes under the identification and type columns can switch the identification and type.

Image Viewer

In the image window, candidate images are listed in the upper half of the window, including images with prediction information (red dot in the upper right corner) and images with observation information (green dot in the upper right corner). The lower half is the image display window, where the red crosshair indicates the predicted point coordinates. Clicking on the image will add the observation value, displayed as a green crosshair. Right-clicking on the image display window can clear the current observation.

Control Point Explanation

A control point contains the following information:

- Name: The unique ID information of the control point.
- Coordinates: Distinguish based on the selected coordinate system. For local coordinate systems or projected coordinate systems, it is X, Y, Z; for geographic coordinate systems, it is latitude, longitude, and height.
- Observations: The number of image observation points manually added or identified through target detection.

- Control Piont Flag: Divided into control points and check points. Control points are included in calculations during subsequent alignment updates, while check points are only used for accuracy checks in the report generation.
- Type: Divided into XYZ, XY, and Z, which are plane elevation control points, plane control points, and elevation control points, respectively.
- Horizontal and Vertical Accuracy: Set the horizontal and vertical accuracy, which affects the weight of the control point during calculations.
- Error: The distance between the control point coordinates and the intersection point of the observation values.

  > Note: An effective control point requires at least two observations. The update alignment function requires at least four control points.

Export Cameras

Function Overview

Exporting camera exterior orientation involves exporting the exterior orientation information after image alignment into third-party formats such as BlocksExchange XML format (Smart3D Block-AT) (*.xml) and Inpho Project File (*.prj). This allows third-party software to utilize the results of the image alignment.

Usage

1. Click on the Export Cameras function and select the image project file you want to export.
2. Choose the export path and select the desired format. Two formats are supported.
3. After confirming everything is correct, click OK to export.

One-Click Workflow

Function Overview

The one-click workflow allows you to use newly created or imported image project (.liaep) files, along with point clouds or externally generated DSM/DEM images, to perform image alignment and orthoimage generation.

> Note: The point cloud and image project used must be in the same coordinate system and location to execute the combined point cloud processing workflow; otherwise, the resulting product coordinates will not match, making processing impossible.

Usage

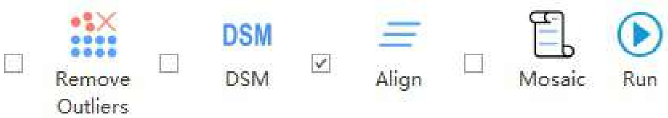


Photo Workflow Builder

Combined Point Cloud Workflow

The combined point cloud workflow uses point clouds to generate DSM/DEM, which is then used in the image processing workflow to generate orthoimages.

1. Load the point cloud and image project to be used into the window.
2. Select the processes to execute based on actual needs. For orthoimage generation using point cloud DSM, it is recommended to select smoothing, resampling, denoising, DSM, image alignment, and orthoimage; for orthoimage generation using point cloud DEM, it is recommended to select smoothing, resampling, denoising, ground point classification, DEM, image alignment, and orthoimage.
3. Click the run button and set the parameters for each function in the pop-up dialog box. Note that in the orthoimage parameters interface, if using point cloud-generated DSM, select DSM-based stitching; if using point cloud-generated DEM, select DEM-based stitching. Refer to other modules for point cloud parameter descriptions: SmoothPoints, Subsampling, Outlier Removal, ClassifyGroundPoints, DSM, DEM. Refer to the Parameter Settings section for image alignment and orthoimage parameters.
4. Confirm that the parameters are correct, then click OK to execute the process.

Non-Point Cloud Workflow

When only completing the image alignment workflow without generating orthoimages, or when providing external DSM/DEM, you can directly perform image alignment and orthoimage workflow without using point clouds.

1. Load the image project to be used into the window.
2. Based on actual work requirements, only select image alignment and orthoimage workflow, and

deselect all point cloud-related function buttons.

> Note: The orthoimage workflow requires prior completion of the image alignment workflow or the use of an already aligned image project.

3. Click the run button and set the image alignment parameters and orthoimage parameters in the pop-up dialog box. Note that in the orthoimage parameters interface, you need to select custom model stitching and set the path of the external DSM/DEM. Refer to the Parameter Settings section for other parameter descriptions.
4. Confirm that the parameters are correct, then click OK to execute the process.

Parameter Settings

Image Alignment

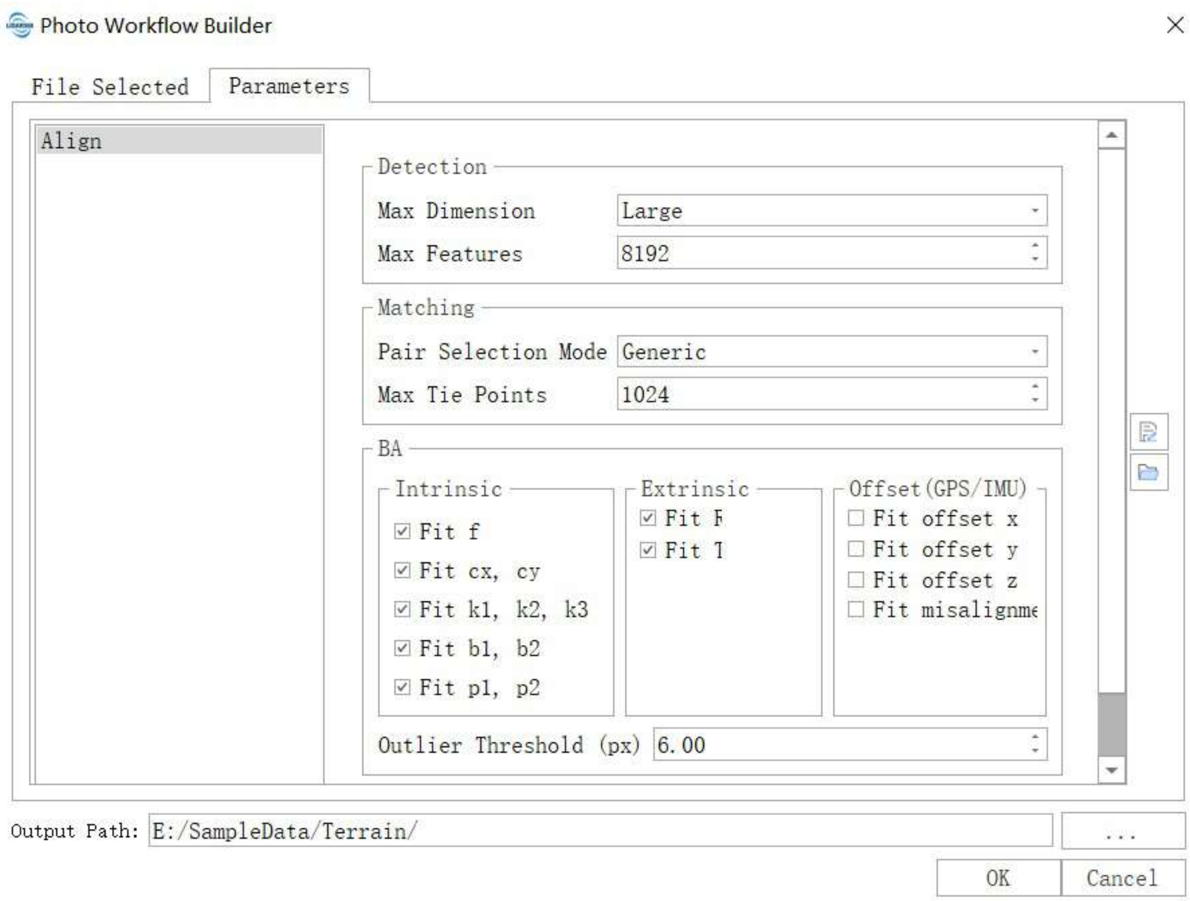


Image alignment is the process of recovering the camera's position and attitude at the time of shooting and the sparse 3D scene structure based on the original images, geographical location information (GNSS and GCP, optional), and initial camera intrinsics (optional). The image alignment results can be used for subsequent DOM/DSM generation processes.

Feature Point Detection

- Maximum Image Size: There are three supported maximum image sizes: "Large," "Medium," and "Small." Depending on the selected maximum image size, different levels of sampling (up-sampling or down-sampling) will be performed on the original images during processing.

- Large (default): The large size detects the most feature points, resulting in more robust processing outcomes compared to "Medium" and "Small" sizes, and takes the longest time.
- Medium: The medium size detects fewer feature points than the large size and takes less time.
- Small: The small size detects the fewest feature points and takes the shortest time.

> Note: Within a certain range, an increase in the number of feature points enhances the robustness of the processing results (too many feature points can introduce more noise). The algorithm balances processing efficiency and robustness, and in most cases, the default "Large" option yields better results.

- Maximum Number of Features (default "8192"): The maximum number of features refers to the upper limit of feature points per image involved in subsequent processing steps. The higher the upper limit, the longer the subsequent processing takes, and vice versa.

> Note: Efficiency should not be the only consideration, as too few features can affect the robustness of subsequent processes. The algorithm balances efficiency and robustness, and in most cases, the default value "8192" yields better results.

Feature Point Matching

- Pair Selection Mode: To enhance the efficiency of feature point matching, a pre-matching image pair set is selected before the actual matching, and subsequent matching is performed only between pairs in this set. Three pair selection modes are supported: "General," "Brute Force," and "Geo-Reference."
  - General (default): In "General" mode, the original images are down-sampled, and pre-matching is performed on low-resolution images to narrow down the matching range, followed by fine matching.
  - Brute Force: In "Brute Force" mode, exhaustive matching is performed on all images (this process is time-consuming).
  - Geo-Reference: In "Geo-Reference" mode, based on the geographical location information of the images (read from EXIF or imported externally), only the nearest subset of images is selected for pre-matching to narrow down the matching range.

> Note: The "Geo-Reference Mode" is effective only when geographical location information is available. In most cases, the default "General" option yields good results. If the image alignment results are significantly incorrect, try the "Brute Force" mode.

- Maximum Number of Tie Points (default "1024"): The maximum number of tie points refers to the maximum number of tie points between two images during processing. Too few tie points can affect the accuracy of the results or even cause processing failures. In most cases, the default "1024" option yields better results.

Bundle Adjustment

- Optimize Intrinsics (default: all optimized): Supported intrinsics include camera focal length (f), principal point (cx, cy), radial distortion parameters (k1, k2, k3), tangential (decentric) distortion parameters (p1, p2), and image plane distortion parameters (b1, b2). In the absence of initial intrinsics, the software's internal self-calibration algorithm can provide good intrinsic estimates. It also supports using pre-calibrated camera intrinsics. If the imported intrinsics are highly accurate, intrinsics optimization can be deselected to keep intrinsics fixed during bundle adjustment.

- Optimize Extrinsics (default: all optimized): Supported extrinsics include rotation parameters/exterior orientation angles (R) and translation parameters/exterior orientation elements (T). Without accurate externally imported camera extrinsics, be sure to select optimization; otherwise, image alignment will fail.
- Estimate Mounting Errors (default: none estimated): Supported mounting parameters include GPS mounting errors (offset values along x, y, z axes, gps -> camera) and IMU mounting errors (heading, pitch, roll angular offsets, imu -> camera). The default mounting parameters are 0 (i.e., GPS and IMU are assumed to have no mounting errors). GPS mounting errors can be accurately estimated only when GPS precision is high and accurate control points (GCP) are available. If GPS precision cannot be guaranteed or there are no control points, do not select this option, as it will severely impact the alignment results.
- Outlier Threshold (default "6.00"): The outlier threshold is the upper limit of reprojection error during bundle adjustment (in pixels). Feature points and their corresponding scene points exceeding this threshold will be excluded to effectively suppress the impact of outliers on the adjustment results.

Orthoimage

The orthoimage process utilizes the interior and exterior orientation elements of the images and the generated Digital Elevation Model (DEM) or Digital Surface Model (DSM) to orthorectify the original images and stitch and merge the corrected images. The module involves processes such as distortion correction, orthorectification, image exposure compensation, color enhancement, image stitching, and merging.

Generation Modes

The generation modes include three types based on the data source: generated DSM, DEM, and externally imported data. Generally, DSMs have higher precision, making it easier to restore correct pixel positions during orthorectification, resulting in higher-quality orthoimages. For example, in urban areas with significant building height differences, using DEM-generated orthoimages often results in tilted buildings, whereas in relatively flat areas, the effect of using DEM or DSM for orthorectification may not differ significantly.

- DSM-based stitching: Stitch orthoimages based on DSM generated from point clouds. In most cases, it is recommended to generate DSM and use this mode for the best results.
- DEM-based stitching: Stitch orthoimages based on DEM generated from point clouds. This mode is effective in forests with tall tree canopies. Since DEM is used, the positional accuracy of ground objects in the orthoimages may decrease.
- Custom model stitching: Use custom DEM or DSM for pixel-by-pixel stitching; DSM is recommended.

  > Note: 1) In combined point cloud workflows, a corresponding DEM or DSM must be generated, or users can import the corresponding DEM or DSM; otherwise, the

  process will be interrupted due to lack of data. 2) The coordinate system of the imported DEM or DSM will be automatically converted to the current project coordinate system; otherwise, it needs to be set during import.

Parameter Options

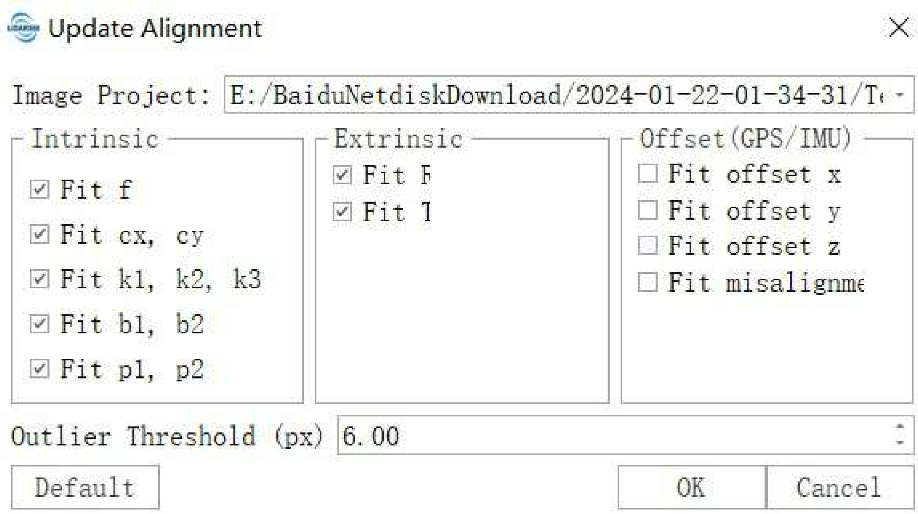
- Fusion Level (default: high): Includes five options: "High," "Medium," "Low," "Very Low," and "None," where "None" means no fusion. When the width and height of a single orthoimage are large (exceeding 3000 x 3000 pixels), the higher the fusion level, the better the fusion effect. When the generated orthoimages are small (less than 1000 x 1000 pixels), a high fusion level can easily lead to overall darkening, in which case, lower the fusion level appropriately. Color correction (default enabled): If enabled, the module will readjust the brightness, contrast, saturation, and white balance of the original images during orthorectification. In most cases, enabling this feature helps generate clear, well-lit, and vibrant orthoimages. If the input images are multispectral, this feature is disabled.
- Maximum Tilt Angle (default 60 degrees): Images with tilt angles greater than this value will be excluded from orthorectification, stitching, and merging. Images with smaller tilt angles are preferred as they produce better quality orthoimages and consume less computational resources, resulting in higher efficiency.
- Resolution: This parameter defines the spatial resolution of the output DOM (unit: meters). The smaller the value, the higher the resolution, and the longer the processing time.

Update Alignment

Function Overview

If image alignment is already completed, clicking the "Update Alignment" button can align the photogrammetric results to POS and control points. This feature can also be used to update the image alignment status after filtering tie points or deleting images.

Usage



1. Click "Update Alignment." In the pop-up dialog box, select the image project for which you want to perform the update alignment.

   > Note: Only image projects that have completed the image alignment step can perform the update alignment operation.

2. Select the parameters to be optimized. These parameters are the same as the bundle adjustment parameters in the image alignment settings. Refer to the Image Alignment parameters page for detailed explanations.

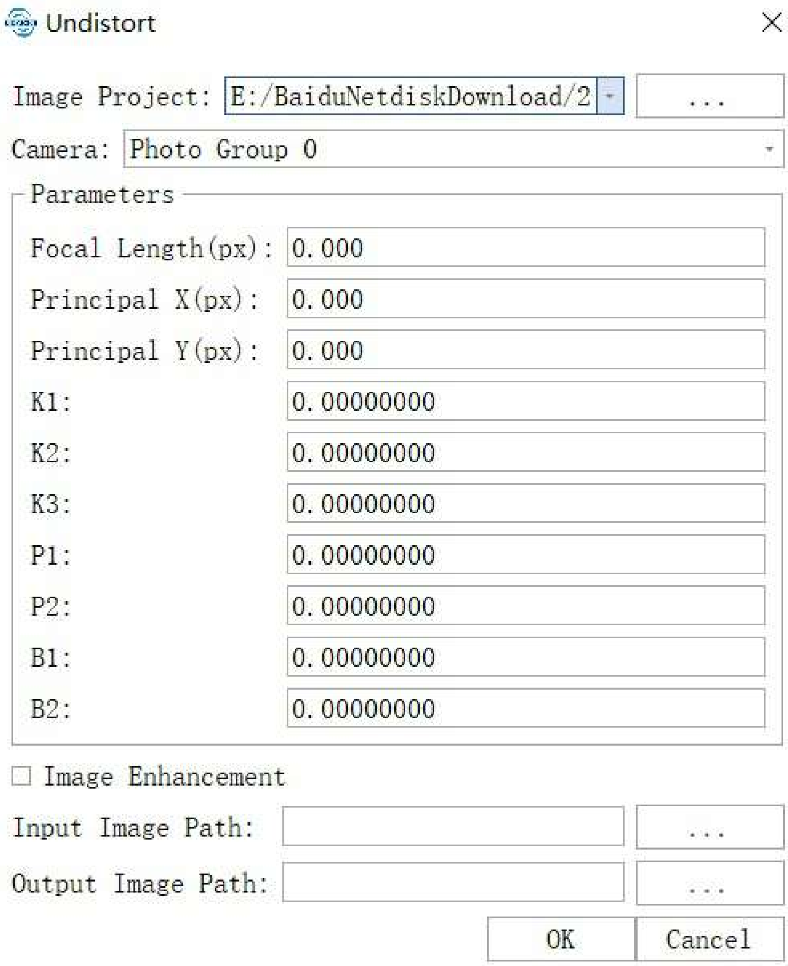
3. After confirming the parameters are correct, click "OK" to start the update alignment process.

Undistort

Function Overview

The Image Undistort dialog box can correct a selected set of images for distortions caused by camera or lens errors. This requires setting the camera's intrinsic parameters and distortion parameters in the dialog box.

Usage



1. To use the camera intrinsic parameters obtained from the completed image alignment, select the desired image project in the Image Project row, and switch to the camera group whose intrinsic parameters you wish to use in the Camera row. The selected camera parameters will automatically populate the parameter list below.
2. If you do not use the camera parameters from an existing image project, you will need to manually input the parameters, including camera focal length, pixel size, principal point offset, CCD correction, radial distortion, and tangential distortion. Refer to the Camera Calibration - Camera Parameters section for definitions of the camera parameters.
3. Check "Image Enhancement" to improve the clarity and contrast of the output image.
4. Select the image directory that requires distortion correction; images in this path will be searched for

correction.

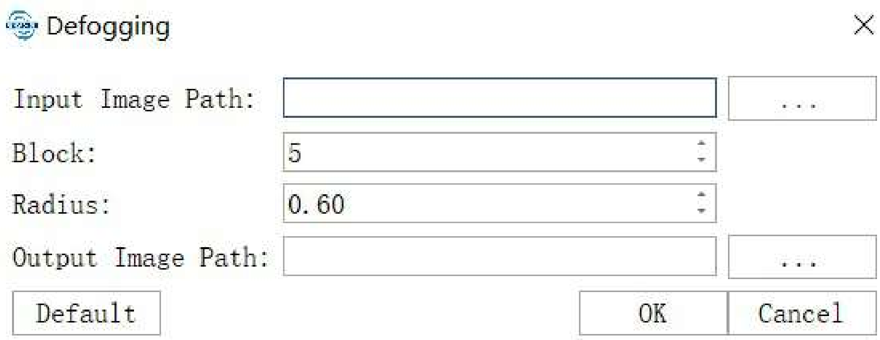
5. Choose the output directory; the undistorted images will be saved to this path.
6. After ensuring all parameters are correctly entered, click "OK" to start the distortion correction process.

Defogging

Function Overview

Defogging makes images clearer and enhances image quality.

Parameter Settings



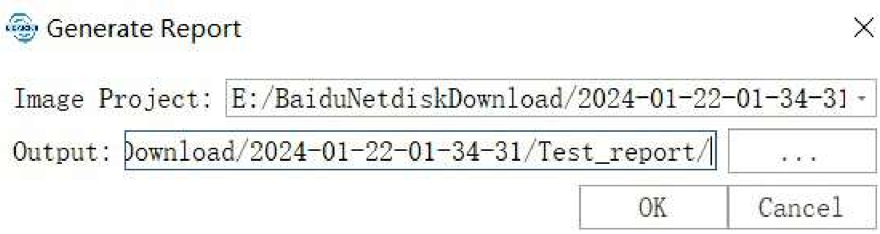
- **Input Image Path**: Users can select the folder corresponding to the image sequence to be processed.
- **Block** : The image is processed in blocks. The larger the block size, the fewer the number of image blocks; the smaller the block size, the more image blocks are generated.
- **Radius**: The filter radius for guided filtering. The filter radius affects the quality of the filtered image. The smaller the filter radius, the clearer the image details and the stronger the sense of layers; the larger the filter radius, the more blurred the image details and the weaker the sense of layers.
- **Output Image Path**: Save the images after fog removal in the corresponding folder.

Generate Report

Function Overview

After completing the image alignment or orthorectification steps, you can generate a quality report. Typically, the quality report includes basic project information, parameter settings for completed steps, results information, and runtime details.

Usage



1. Click on the "Generate Report" function and select the image project file you wish to operate on. Ensure that the project has completed image alignment or orthorectification steps within this module to generate a quality report.

2. Choose the export path folder and click "Export" to confirm the export.

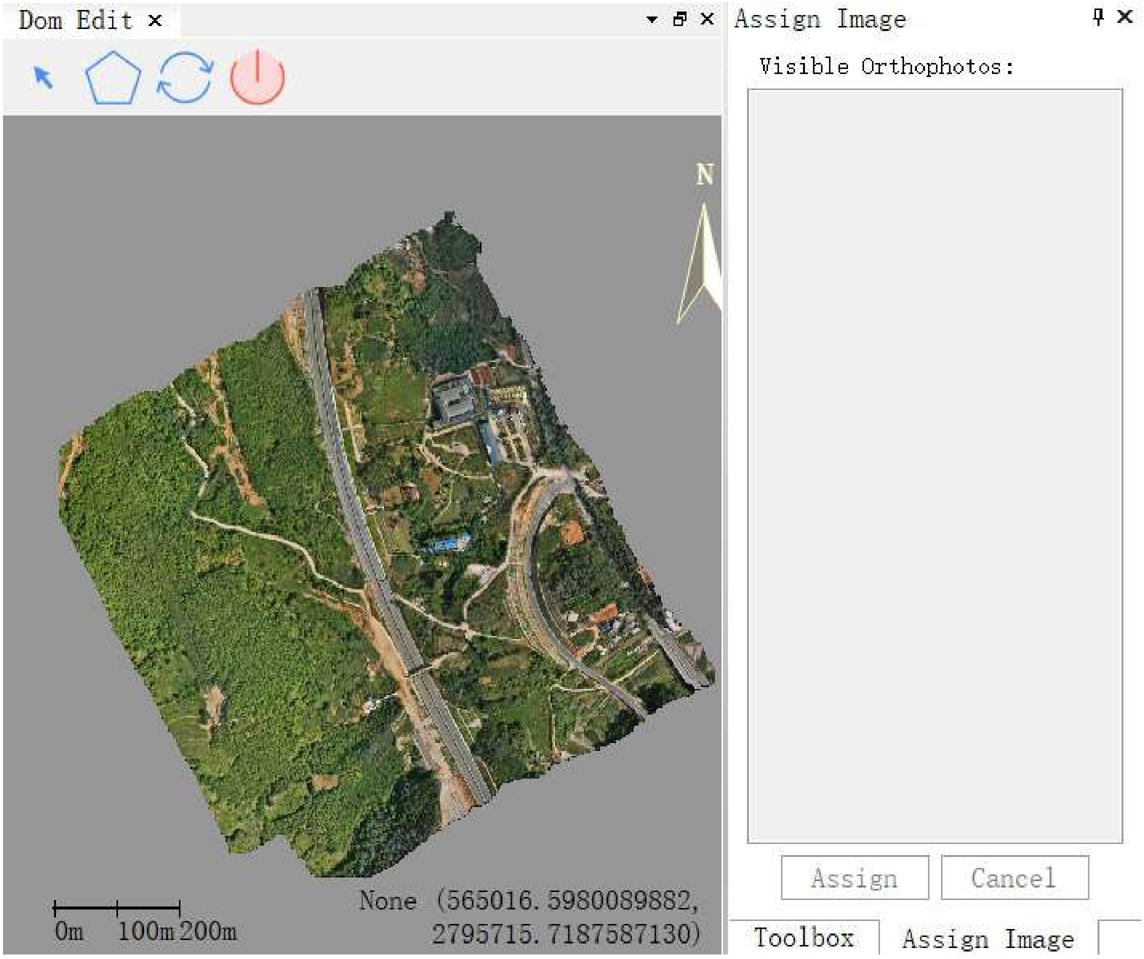
This process will compile and export a comprehensive quality report containing essential details about your project's processing and results.

Dom Edit

Function Overview

After completing the orthophoto image processing, inspect the quality of the DOM (Digital Orthophoto Map). If obvious seamlines or poor-quality areas are found, you can edit specific areas to adjust the seamlines for better results.

Usage



1. Click the DOM Edit function and select the image project file that needs the DOM Edit function. Note that the image project must have completed the orthophoto process to execute the DOM Edit function.
2. Toggle Display: In the right-click menu within the DOM Edit window, toggle the display of polygons to show or hide the drawn polygons, and toggle the display of seamlines to show or hide the current DOM seamlines.
3. Draw Editing Area:
   - Draw Polygon: Select the Draw Polygon tool in the DOM Edit window toolbar. Click consecutively with the left mouse button to draw the area to be edited. Right-click to finish and draw the polygon.
   - Select Polygon: Select the Selection tool in the DOM Edit window toolbar and select the polygon

to be operated on.

  - Edit Polygon: After selecting the polygon, you can insert a vertex at the mouse position from the right-click menu. Move the mouse over a vertex and right-click to select the Delete Vertex function to delete that vertex.
  - Delete Polygon: After selecting the polygon, click Delete Polygon in the right-click menu.

4. Assign Images: Right-click in the selected polygon area and choose Assign Images from the right-click menu. All overlapping images in this area will be displayed in the mosaic panel. Choose an image with better quality from the list and click the Assign button below. If you cancel, you will need to reassign the images.
5. Update Image: Click the Update Image tool in the DOM Edit window toolbar, select the output path for the DOM, and update the assigned polygon area to export it as a new DOM.

Filter Tie Points

In the results of image alignment, some points may be incorrect or low-precision due to feature point mismatches or other factors. You can use the filtering tool to filter out these points and update the alignment with the filtered results to improve the accuracy of the image alignment.

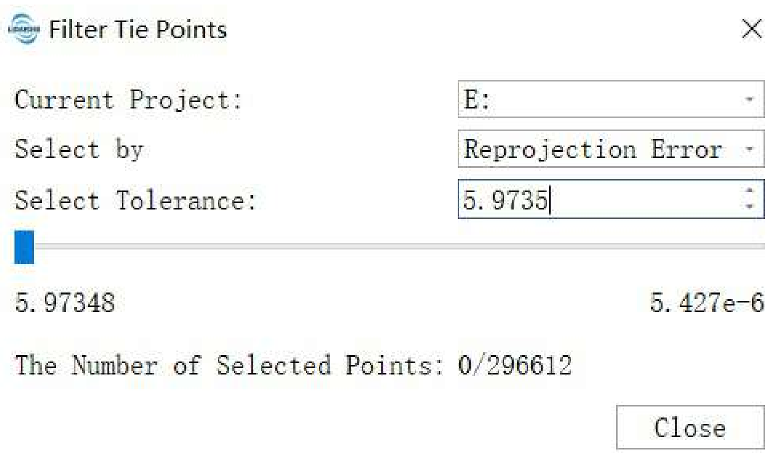
The tie point filtering tool provides two filtering methods:

- **Selection Tool Filtering**
- **Attribute-Based Filtering**

Selection Tool Filtering

Use selection tools such as polygon and rectangle to select tie points or images in the 3D view. Click the **Image Selection/Tie Points Selection** tool, and choose **Image Selection** or **Tie Points Selection** to activate the selection tool. Click **Graphics Tool** to choose the selection tool you need, and draw a shape in the 3D view to make a selection. The **Subtract** tool can remove newly selected images or tie points from the already selected objects. The **Invert** tool can invert the selection across the entire image project. The **Clear** tool can clear all current selections.

Attribute-Based Filtering



- **Current Project**: The image project where the tie points need to be filtered, which must be added to the 3D view.
- **Filter Attribute**: The attribute used for filtering tie points, with the maximum and minimum values displayed at the ends of the slider.
- **Filter Threshold**: Specifies the threshold for the attribute used for filtering.
- **Filtered Point Count**: The number of tie points selected based on the current attribute and threshold.

Filter tie points that meet specified conditions based on their attributes. The attributes of tie points include:

- **Reprojection Error**: The average reprojection error of the tie point in all its observable images. If the average reprojection error is greater than the specified threshold, the tie point is selected.
- **Triangle Angle**: The intersection angle formed by the projection centers of the two images to which the tie point belongs and the tie point itself, measured in degrees. If the intersection angle for any two

images of the tie point is smaller than the specified threshold, the tie point is selected.

> The intersection angle is the angle formed by the projection centers of two images and the tie point. It is defined as α, and the minimum of α and 180-α is taken as the intersection angle. This can be used to filter out tie points with too large or too small intersection angles.

- **Image Count**: The number of images that can observe the tie point. If the number of images is less than the specified threshold, the tie point is selected.
- **Image Observations**: The number of observations corresponding to the images that can observe the tie point. If the number of observations in all observable images is less than the specified threshold, the tie point is selected.
- **Distance**: The distance from the tie point to the image projection center. If the distance from the tie point to any of its observable image projection centers is greater or less than the specified threshold, the tie point is selected.
  - **Filter Points with Distance Less Than**: When using the **Distance** attribute, selects tie points with distances less than the threshold.
  - **Filter Points with Distance Greater Than**: When using the **Distance** attribute, selects tie points with distances greater than the threshold.

Deleting Tie Points

After selecting the tie points to be deleted, click the delete tool to remove the selected tie points.

Undo and Redo

Click the **Undo Tool** to restore the last deleted tie points; click the **Redo Tool** to delete the last restored tie points.

Mine

Provides tools related to mines, including extract tunnel points, compute normal vectors, trangulation modeling, poisson modeling and other functions.

- Tunnel
  - Extract Tunnel Points
  - Compute Normal Vectors
  - Surface Reconstruction
  - Triangulation Modeling (Poisson)
- Open-pit Mine
  - Volume Change Analysis
  - Extract Toes And Crests
  - Inter Ramp Compliance
- MeshEditor
  - Mesh Editor

The functions of Open-pit Mine include: Volume Change Analysis Report， Extract Toes and Crests， Inter Ramp Compliance.

- Volume Change Analysis Report
- Extract Toes And Crestes
- Inter Ramp Compliance

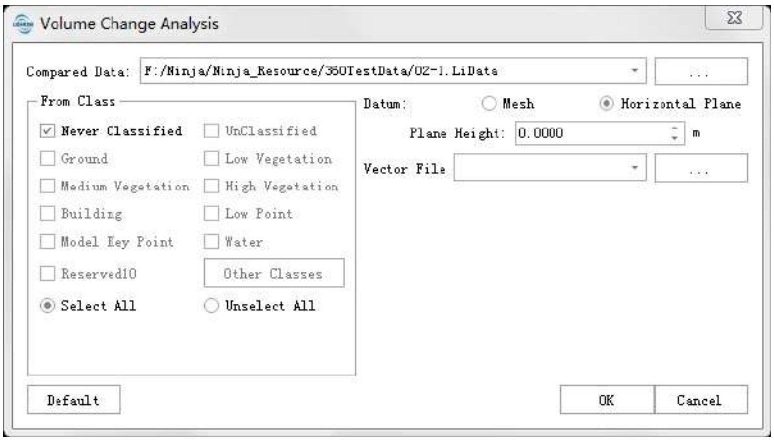
Volume Change Analysis Report

Functions Overview

This function is mainly used to calculate the point cloud fill and cut volume based on the model or plane and generate a report.

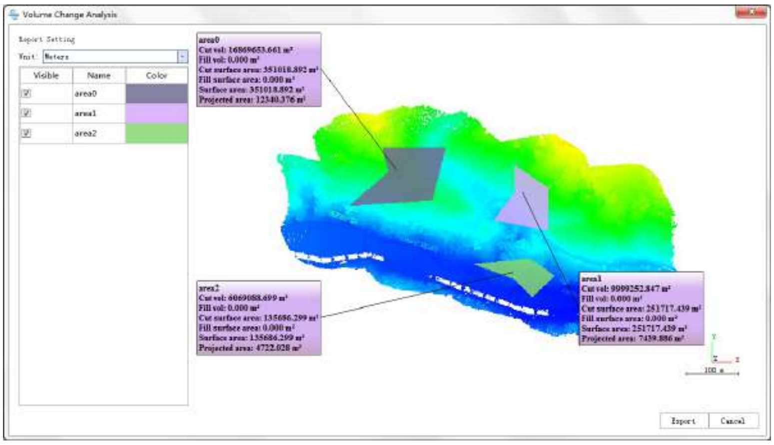
Usage

Click *Mine* > *Volume of closed model*.



Parameter settings

- **Enter the data to be compared**： The data to be compared can be point cloud or model data . If the data is already open in 360, the user can select from the drop down menu. You can also use the button on the right to import directly.
- **Datum**：
  - **Model**： Select model as datum. A model file must be selected at this point.If the model data is already open in 360, the user can select from a drop down menu.You can also use the button on the right to import directly. Currently only models in ".LiTin" format are supported.
  - **Flat**： Select the plane as datum.At this moment, you need to specify the plane height. The amount of cut and fill will be calculated based on this level.
- **Choose a vector file**： The user can select the vector file to load into the LiDAR360 software from the drop-down menuvector file， they can also select ... button to load an external vector data file。 The file needs to contain several closed polygons. These polygons will be used as the boundaries of each area concerned by the user for fill and cut calculations, and the calculated results will also correspond to the polygons one-to-one.
- **Minimum area (square meters) (default "1.0")**： Polygon areas smaller than this area will be discarded.
- **Cut and fill report**： Click OK and after the calculation is completed, the volume change analysis report interface will pop up.The calculated cut and fill of each area is displayed in the form of a table. Each polygonal area will be drawn on the basemap with floating labels showing its cut and fill and surface area.



- **Report settings**：Units can be changed in the report settings interface. The first column of the table can change the visibility of the area. Double-click the 2 and 3 columns to change the visibility, fill and cut information and color of the corresponding area, respectively.
- **Export**：Export report to html format.

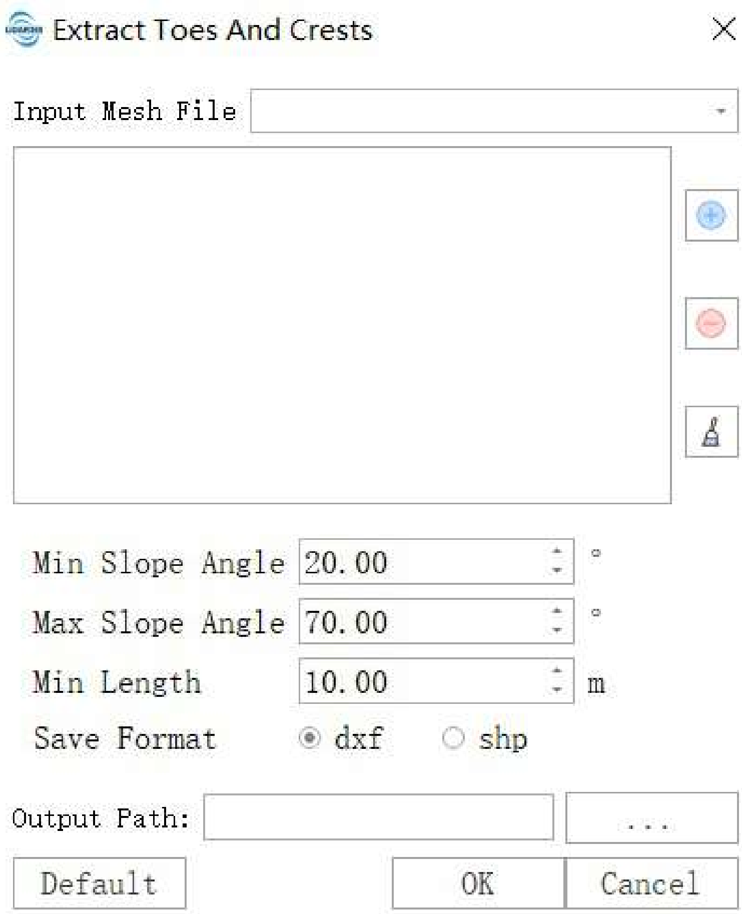
Extract Toes and Crests

Functional Overview

Automatic extraction of mine toes and crests based on triangulation model.Toes and Crests will be saved in different layers.

Usage

Click *Mine> Extract Toes and Crests*, the parameter interface will pop up, set the parameters to execute the extraction.



Parameters Settings

- **Input Mesh File**: Select from the project layer catalog or select model files from a folder.
- **Min Slope Angle (°)**: Minimum step slope angle.
- **Max Slope Angle (°)**: Maximum step slope angle.
- **Min Length (m)**: Shortest slope line length.
- **Smooth Iteration**: Number of smoothing iterations for the triangulation model.
- **Save Format**: The format in which the extraction result slope line is saved (supports .shp, .dxf).
- **Default**: Click this button to restore all parameter default values.

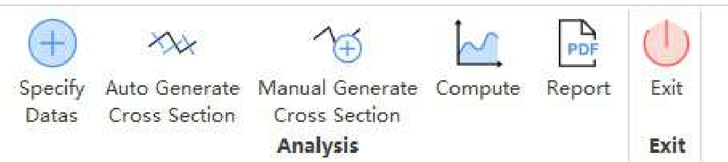
Inter-Ramp Compliance Analysis

Functional Overview

This function quickly calculates the step slope angle and bench width to generate a profile based on the interest area, mine trend line, and completed surface model of the top and bottom lines of slopes.

Usage

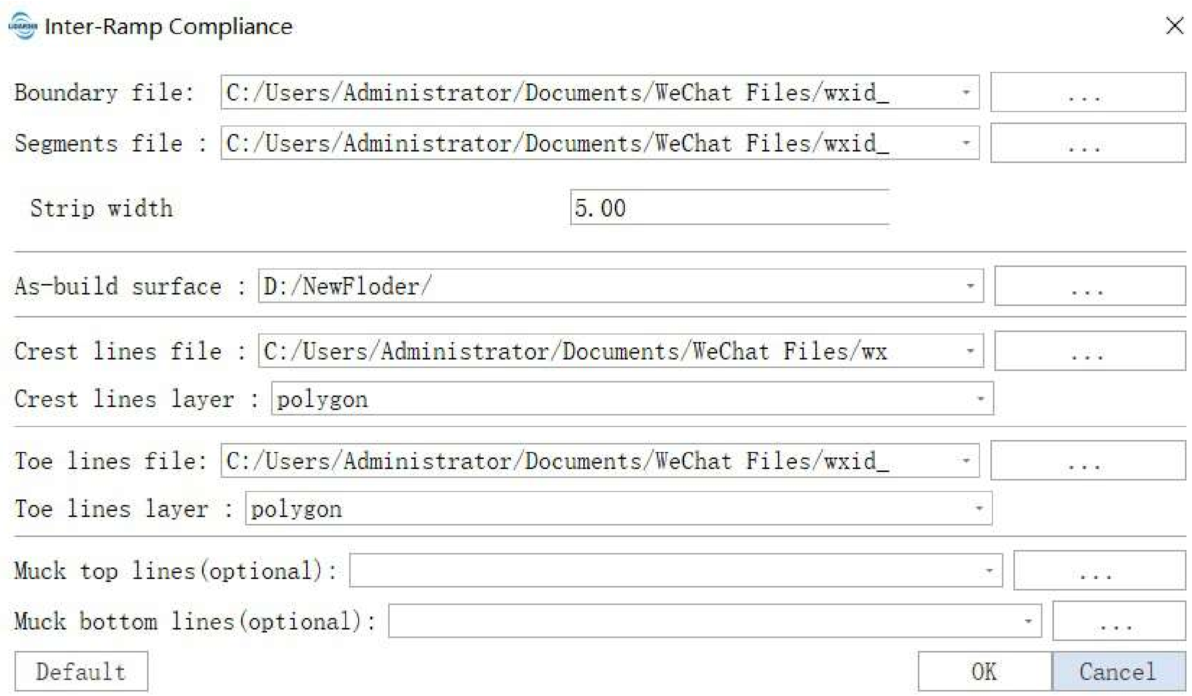
Click *Mine* > *Inter-Ramp Compliance Analysis* in the function to open the page:



The options are specify data , auto generate cross section , manual generate cross section , compute , reports and exiting.

1. Specify Datas

Click on specify data button to pop up a dialog box for data selection and parameter setting:



- **Boundary File**: The polygon range file of an interest area, can be draw in vector editor model.
- **Segments File**: The trend line file of an open-pit mine in the area, can be draw in vector editor model.
- **Strip Width (m)**: The width parameter for generating a single mesh strip in the interest area based on the polygon and trend lines.
- **As-build surface**: A current 3D model of an open-pit mine in obj format.
- **Crest line file**: The corresponding file for the top line of an open-pit mine slope.

- **Crest line layer**: The corresponding layer for the top line of an open-pit mine slope.
- **Toe line File**: The corresponding file for the bottom line of an open-pit mine slope.
- **Toe line layer**: The corresponding layer for the bottom line of an open-pit mine slope.
- **Muck top lines(optional):**The upper boundary file corresponds to a mud area in an opencast mining operation.
- **Muck bottom lines(optional):**The lower boundary file corresponds to a mud area in an opencast mining operation.

Click OK, and yellow grid strips will be generated.

2. Auto Generate Cross section

Clicking on auto generate cross section

- Cross section every strips: Generate cross-section every few strips.

Click OK to generate blue cross section position lines.

3. Manual Generate Cross Section

Clicking on manually generate cross section allows you to interactively add or delete cross section position lines within windows.

The currently selected strip location is highlighted in red. Confirm with the left mouse button. If there is no cross-sectional position line within the currently selected strip, a new one will be added; if it already exists, it will be deleted.

4. Compute

Clicking on compute calculates step slope angles and bench widths to generate profiles. The generated results can be previewed in the profile window on the right side.

5. Reports

Clicking on reports pops up a dialog box:

- **Author**: Report author.
- **Output Path**: Report save path. Set parameters, click OK, then PDF reports and CSV tables will be generated at the target location.

6. Exit

Click exit to

Mesh Editor

The mesh editor function allows for the editing and modification of triangular mesh models, including selection tools, editing tools, display settings, etc. This tool can be used to edit and optimize automatically generated models.

Usage

Click on *Mine* > *Mesh Editor* to open the function page:

It is mainly divided into the following parts:

- Editor
- Select
- Edit
- Setting

Use the editor tool to select a model object and enter edit mode. Use the select and edit to modify and optimize the model. Save edited objects by clicking and exit from model editing.

Editor

Control mesh editing, supporting the editing of one mesh object at a time.

Start Edit

Click Edit > Start Edit to select a model file for editor. During the editing process, the edited model data cannot be removed.

Save

Click Edit > Save

to save your edits and overwrite the original file.

End Edit

Click Edit > End Edit

If there are any modifications made to the model file, you will be prompted with a message asking if you want to save your changes:

Click "Yes" to save your edits and overwrite the original file. Click "No" to cancel your edits and exit. Click "Cancel" if you wish to continue editing this model object.

Selection Tools

The selection tools for the currently edited model include triangle face selection from left to right, including selection tools (rectangle, lasso, polygon), selection mode (select through, select visible), whether to select back faces, substract, select components, expand, shrink, flood fill select, hide selection and unhide selection. There are also options for select all, unselect all, invert selection tools.

Selection Tools

Switch between current selection tools via a drop-down menu:

 Rectangle Select

 Lasso Select

 Polygon Select

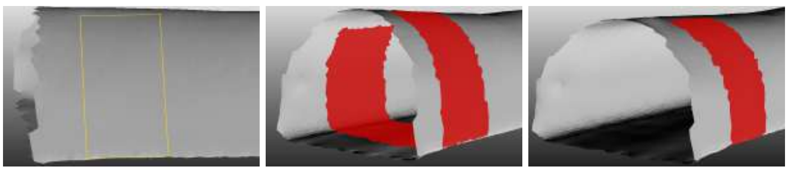
Selection Mode

Two modes can be switched between:

 Select Visible: In this mode only triangles visible in the window can be selected using the chosen tool.

 Select Through: In this mode all triangles within a selected area can be selected using the chosen tool.

The difference is shown below:



From left to right: selected area, select through, visible selection

Select Back Faces

 Select back faces. Whether or not to enable back face selection is enabled by default.

Substract

Substract: When this mode is enabled, selecting a region with the chosen tool will deselect any previously selected triangles within that region.

Select Component

Select component: This function selects all triangles belonging to the same component as the currently selected triangle face. For example, if one triangle of a floating object is selected and this function is executed, then all triangles of that floating object will be selected.

Expand

Expand: This expands the range of currently-selected triangle faces outward from their current position.

Shrink

Shrink: This shrinks the range of currently-selected triangle faces inward from their current position.

Flood Fill Select

Flood fill selections: This selects all connected triangles in an approximately flat plane relative to the currently-selected face(s).

Hide Selection

Hide selected items: This function makes the currently-selected triangle faces invisible, making it easier to select obscured or internal triangles.

Unhide Selection

Cancel hiding: This function restores visibility to all previously-hidden triangle faces.

Select All

Select all: This selects all faces of the currently-edited object.

Unselect All

Deselect all: This deselects all faces of the currently-edited object.

Invert Selection

 Invert selection: This function inverts the current selection. Currently-selected items will be deselected and vice versa.

Editing Tools

Editing operations for the currently edited model, from left to right include undo, redo, delete, fill all holes, fill single hole, bridge, smooth and decimate.

Undo

 Undo - undo the last edit operation.

Redo

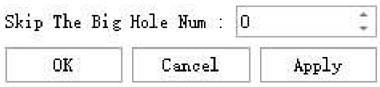
 Redo - re-execute the last edit operation.

Delete

 Delete - delete the currently selected triangle face.

Fill All Holes

 Fill all holes. Click on the function button to pop up a parameter interface:



- **Skip The Big Hole Num**: Skip the number of holes with more edges. The skipped holes will be displayed in green and those being filled will be displayed in red.

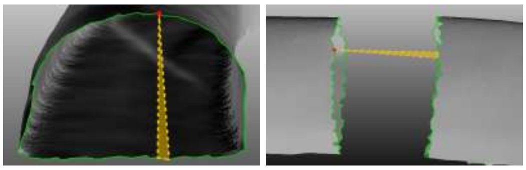
  Click apply to execute filling all holes. After filling is complete, click OK to exit and save the current patching result or click cancel to exit without saving it.

Fill Single Hole

 Fill a single hole. Enter single-hole filling mode and select a hole by moving your mouse over it. The selected hole will turn red. Confirm by clicking on the left mouse button to fill the selected hole.

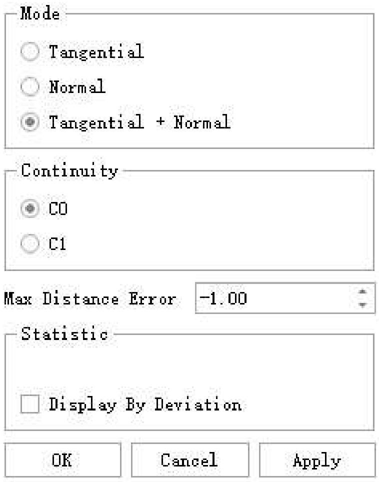
Bridge

 Bridge. Click on bridge to enter bridging mode. Use the left mouse button to select a point on the hole and then select an edge of the hole to generate a single triangle face. The selected points and edges can come from either the same or different holes, as shown below:



Smooth

 Smooth - click on this function button to pop up a parameter interface:



- **Mode**

  Tangential: smooth towards local triangle tangent direction.

  Normal: smooth towards local triangle normal direction.

  Tangential+Normal: smooth in both directions.

- **Continuity**

  C0: maintain continuity of local point position smoothing.

  C1: maintain continuity of local point normal smoothing.

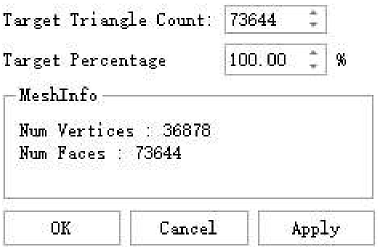
- **Max Distance Error** : maximum allowable displacement between smoothed and original vertex positions. If exceeded, corresponding vertices will not be changed. Default is -1, with no limit on vertex movement distance.

- **Statistic** : source category for processing broken lines corresponding point cloud, generally mesh category.

  Display by deviation: whether or not model displays according to size of post-smoothing vertex displacement amount; valid only when not displaying double-sided models.

Decimate

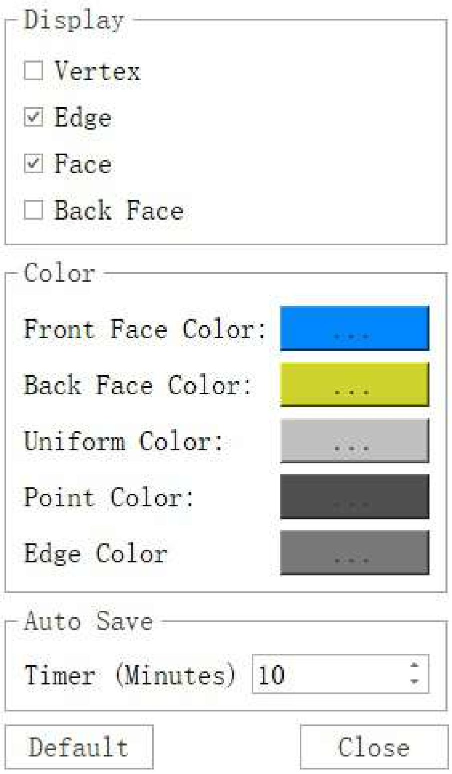
 Decimate - click on this function button to pop up a parameter interface:



- **Target Triangle Count**: simplify the number of triangles to the set value.
- **Target Percentage**: control the target number of triangles as a percentage.

Settings

Click on the settings button to open the settings interface:



- **Display**

Vertex: Whether to display vertices of the mesh model.

Edge: Whether to display edges of the mesh model.

Face: Whether to display faces of the mesh model.

Backface: Whether or not to distinguish between front and back faces when displaying models.

- **Colors**

Front Color: The color used for displaying front-facing polygons when backface display is enabled.

Back Color: The color used for displaying back-facing polygons when backface display is enabled.

Single Color: The color used for displaying models when backface display is disabled.

Vertex Color: The color used for displaying vertices of a model.

Edge Color:: The color used for displaying edges of a model.

- **Auto Save**

Time (minutes): How often should cache be saved while editing a model, in case program crashes and you need to recover your work.

The function of tunnel includes: Extract tunnel points, Triangulation modeling, Compute normal vectors, Poisson modeling.

- Extract tunnel points
- Compute normal modeling
- Poisson modeling
- Surface Reconstruction

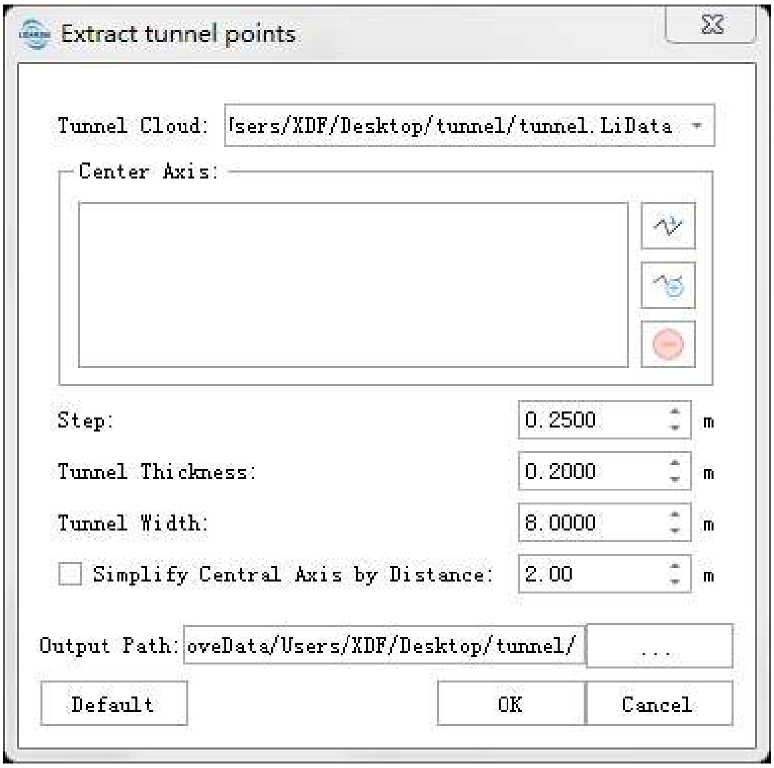
Extract Tunnel Points

Summary

Generate the center axis by importing from files or manually drawing, and then remove the noise inside the tunnel along the center axis, and generate a new file. The algorithm first extracts the roadway point cloud in a certain range around the center axis, and slices the roadway point cloud vertically along the center axis at the certain step. Then calculate the outer ring (roadway boundary) of each slice, remove the internal noise, and finally merge the results. This function is suitable for caves, shield tunnels, underground tunnels and other data.

Usage

Click *Mine* > *Extract Tunnel Points* 。



Settings

- **Input Data**:The input file can be a single point cloud data or a point cloud dataset, which must be opened in the LiDAR360 software.
- **Center Axis**: Currently, two methods of "import from files" and "manual drawing" are supported to obtain the center axis. The algorithm will extract the tunnel point cloud around these center axes and denoise, and finally generate a new file.
    - **Import Center Axis from Files**: Supports center axis files in SHAPE format and ASCII format. The center axis file must contain at least two columns, namely: X and Y. After the center axis file is imported, it will be displayed in the scene. The center axis needs to be fitted with the point

cloud, otherwise the function will not work properly.

  - **Manual Drawing**: The user can use the interactive polyline as a medium to automatically calculate the center axis. Click this button to start drawing a polyline in the view with the left mouse button, double-click to end the drawing of the current polyline, and add it to the list of polylines. Select a polyline in the list and click the button to delete the corresponding polyline.

- **Step (m) (Default Value is "0.25")**: The tunnel is longitudinally sectioned along the center axis with this step length, and each segment of the point cloud after sectioning is individually denoised. The step size affects the efficiency and precision of the algorithm. The smaller the step size, the lower the efficiency and the finer the denoising effect. When the tunnel surface consistency is good and the direction change is gentle, the value can be increased appropriately. Good results can be obtained by keeping this value between 0.2 and 3.0.

- **Tunnel Thickness (Default Value is "0.2")**: The thickness of the tunnel refers to the average distance from the inner wall of the tunnel to the surface of the tunnel. Points in the tunnel greater than this distance will be counted as noise points.
- **Tunnel Width (m) (Default Value is "8")**: The tunnel width refers to the maximum width of the contour section in the tunnel, which determines the extration range of the point cloud on both sides of the center axis. This value needs to be greater than the maximum values of the net width of the entire tunnel to avoid the inability to obtain the tunnel point cloud during the calculation process.

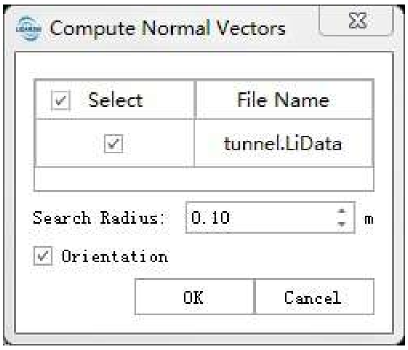
Compute Normal Vectors

Summary

Calculate the normal vector of the tunnel. The algorithm will search for adjacent points within a certain radius for each point, and then calculate the local plane and normal direction based on these points. Finally, use the minimum spanning tree to optimize the direction of the normal (optional). The generated normal will be stored as an additonal attribute, and the addtional attribute field is "Normals".

Usage

Click *Mine* > *Compute Normal Vectors*。



Settings

- **Input Data**: The input file can be a single point cloud data or a point cloud dataset, which must be opened in the LiDAR360 software. File format: *.LiData.
- **Search Radius (m) (Default Values is "0.5")**: Neighborhood search radius. The algorithm obtains neighboring points through neighborhood search for each point, and then calculates the normal direction based on these points. If this value is too large or too small, it will cause the normal calculation error, and when it is too large, it will also cause the algorithm efficiency to decrease. This value can be set to 5 times the average dot spacing to get the best results.
- **Orientation (Default is checked)**: When this option is checked, the minimum spanning tree will be used to optimize the consistency of the normal direction.

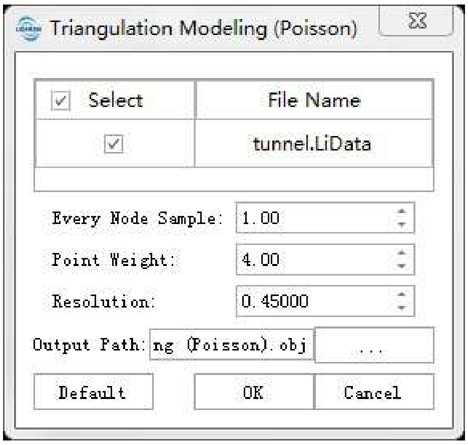
Triangulation Modeling(Poisson)

Functional Overview

Triangulation Modeling(Poisson) is used to generate a 3D model file in .obj format using point clouds. Prior to running this function, Tunnel Normal Computation must be performed on the point cloud. Set parameters and preview results in the window on the right-hand side of the screen. Adjust density parameters for optimal results before saving.

Usage

Click *Mine* > *Tunnel* > *Triangulation Modeling(Poisson)*.



Parameters Settings

- **Point Cloud File**: Input one or more point cloud data files. File format: *.LiData.
- **Every Node Sample (default "1.5")**: The minimum number of points retained in each node after octree reconstruction is used to weaken noise interference with reconstruction results. A larger value provides greater noise resistance but ignores more details; set between 1.0 and 5.0 for small amounts of noisy point clouds, and between 15 and 20 for large amounts.
- **Point Weight (default "2.0")**: The weight of interpolated points in the Poisson implicit equation determines how closely the reconstructed implicit triangular mesh surface matches the original point cloud.
- **Resolution (default "0.4")**: The resolution of the reconstructed implicit triangular mesh surface determines its density; higher resolutions result in denser meshes that preserve more detail, while lower resolutions produce sparser meshes.
- **Density**: After reconstruction, adjust this parameter during preview generation to confirm final reconstruction results.

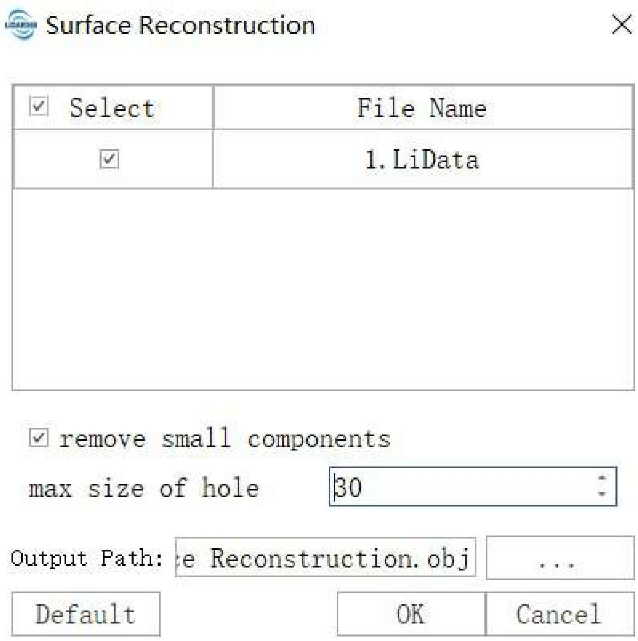
Surface Reconstruction

Functional Overview

This function is used for fast 3D modeling based on point clouds. Prior to executing this function, it is usually necessary to perform preprocessing such as denoising, sampling and smoothing on the raw data. This function does not require calculation of normals or generation of 3D model files.

Usage

Click *Mine* > *Tunnel* > *Surface Reconstruction*.



Parameters Settings

- **Input Point Cloud Data**: Input one or more point cloud data files. File format: *.LiData.
- **Remove Small components**: Whether to remove small objects such as floating debris.
- **Max size of hole**: Used for filling smaller holes.

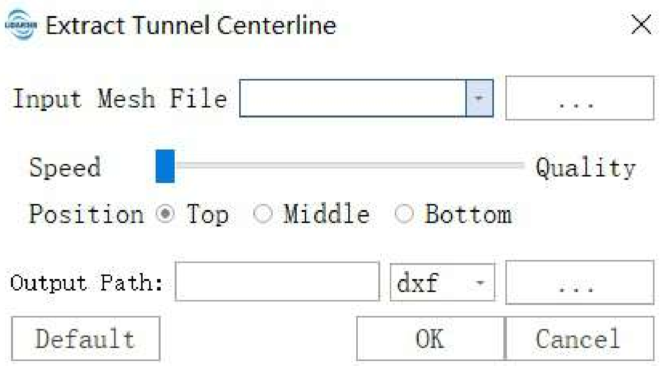
Extract Tunnel Centerline

Overview

This function extracts the tunnel centerline based on a triangulated mesh model. The tunnel model can be generated using surface reconstruction features or created using other software.

Usage

Click *Mine* > *Tunnel* > *Extract Tunnel Centerline* to open the parameter settings interface and execute the extraction.



Parameter Settings

- **Input Mesh File**: Select the model file from the project layer directory or choose from a folder.
- **Speed-Quality**: Adjust the slider to balance between extraction speed and quality.
- **Position**: Set to upper or lower, determining whether the result line will adhere to the upper or lower surface of the tunnel.
- **Output Path**: Specify the location and format for saving the centerline results (supports .shp, .dxf, .gpkg, .csv).
- **Default**: Click this button to restore all parameters to their default values.

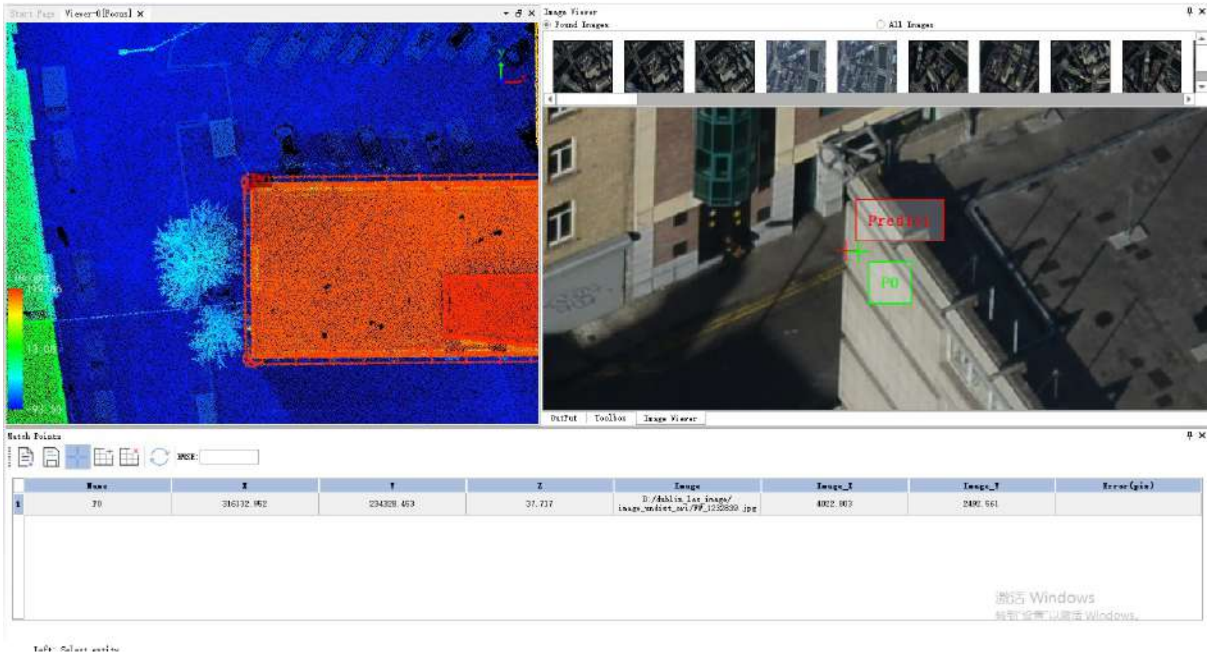
Building modeling

The function of Building Modeling realizes automatic extraction of building model based on classification point cloud, manual editing and storage of automatically extracted model results. In the process of building model editing, the point cloud and image data are used as the background, and the real-time preview effect of the building model is used as the auxiliary, the building model is constructed by manually editing the two-dimensional vectorization object of the building model quickly and accurately. At the same time, the results of building model support the generation of OBJ, FBX, CityJSON format for third-party software interconnection.

Modeling Process

- Extract Building Footprint
- Subsampling
- Remove Outliers
- Classify Ground Points
- Classify by Deep Learning
- DSM
- DOM
- Building Model Registration
- Auto Create Buildings
- Project Textures form Photos
- Attributes Caculate
- Building Editor
- Matched Photos Editor
- Materials Editor

Building Model Registration

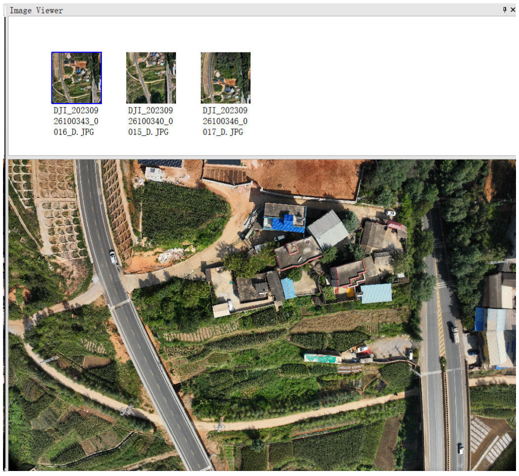


By using point clouds, models, or imported control points, select corresponding image control points in the image window. After collecting four pairs of control points, the image project will be registered based on the four pairs of control points.

Usage

- Image Viewer
- Match Points

Image Viewer



Display a list of images in the image project file and the currently selected images in the list.

Find Image

Show the images found in the image list based on the current selected 3D point or the center point of the building surface.

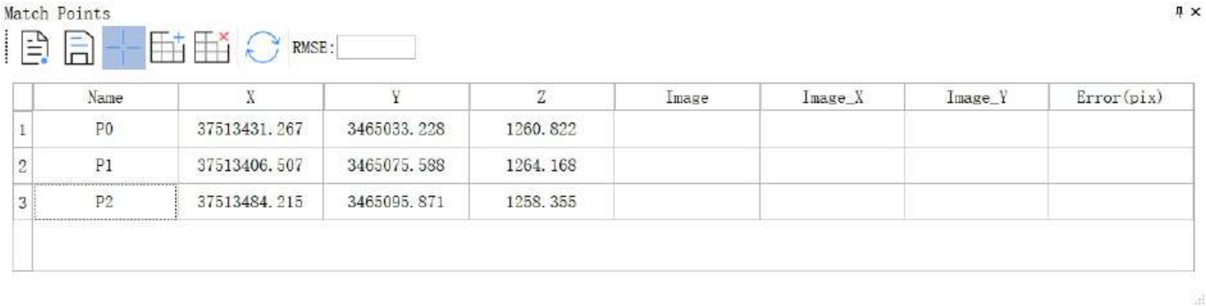
All Image

Show all images in the image project.

Save

Save the current texture editing result.

Match Points List



	Name	X	Y	Z	Image	Image_X	Image_Y	Error(pix)
1	P0	37513431.267	3465033.228	1260.822				
2	P1	37513406.507	3465075.588	1264.168				
3	P2	37513484.215	3465095.871	1258.355				

Import

Import matched points file to the match point list.

Export

Export match points from the match point list to a file.

Click Selection

Enable click selection function, select 3D points in the point cloud or model window, and select 2D image points in the image window.

Add Point

Add a new line of match point record.

Delete Point

Delete currently selected match point record.

Save

Save current match result to image project file. If precise alignment is checked, you need to select a point cloud file and perform precise alignment based on the current alignment result.

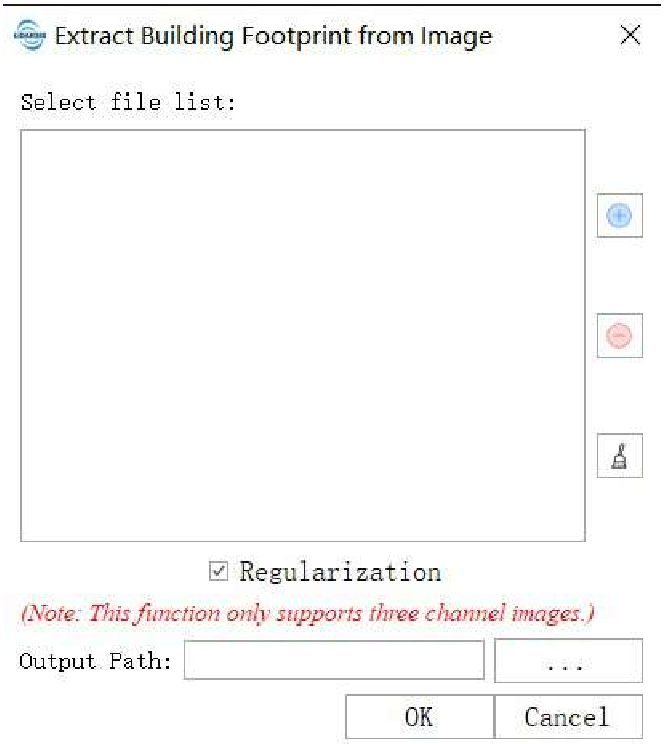
Extract Building Footprint from Image

Description

This function uses deep learning to extract building footprint from images. The input file must be a DOM (digital orthophoto image) or a three-channel color image produced by aerial photography, such as an orthophoto image. It cannot be a single-channel image such as DEM or DSM. The output is a gpkg vector file containing multiple building footprints.

Usage

Click on *3D Building > Extract Building Footprint from Image*.



Parameters

- **Select file list**：Select image data either from already opened software or from a folder. Currently supported formats include tif, tiff, jpg, jpeg, and png.
- **Simplify（default“Yes”）**：When simplification is checked, the building outline will be streamlined to some extent, eliminating the jagged edges in the original outline.
- **Regularization（default“Yes”）**：Whether to perform regularization on the generated building outline. After regularization, the angle between any two adjacent sides of the building outline will not exceed the angle threshold.
    - **Angle Threshold（default“10”）**：Line segments with angles smaller than this threshold will be merged into a straight line.
- **Delete（default“10”）**：If checked, contours with an area smaller than this value will be deleted.
- **Output Path**：The path where the output contour vector file will be saved.

Create building models automatically

Functional Overview

Based on the general architectural model description, the Lidar360 automatic modeling function generates level-of-detail(LOD)2 architectural models. For the LOD 2 architectural model, the main expression is the shape of the roof, and the wall is a vertically stretched plane. Automatic building modeling is based on ALS point cloud data and building vector footprint. The vector footprint is mainly used to determine the scope of each building and constrain model generation. Automatic modeling creates a high-precision Lod2 model of a building, and the resulting model can be modified in detail with manual editing tools.

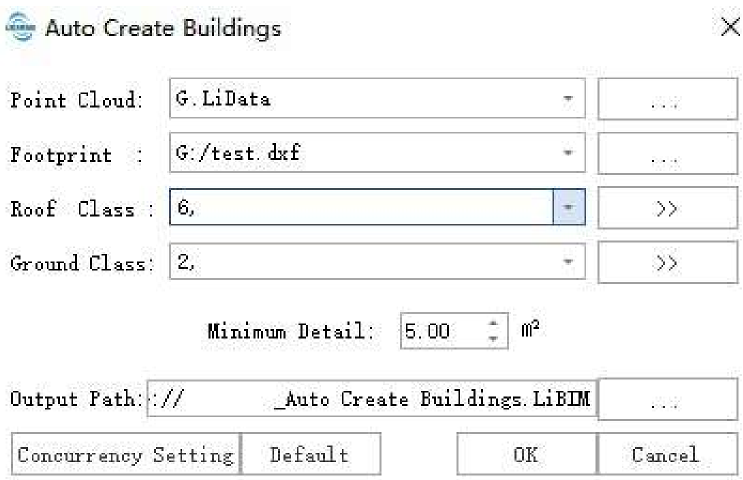
Usage

Process of creating building models automatically is as follows:

1. Use the classification tool to classify ground points.

2. (Optional) Use the classification tool to classify buildings.

3.Click*3D Modeling function page >Automatic building model*button, pop up a dialog box:



1. Set parameters and click OK to start the execution.

Parameters Settings

- **Point cloud file**: Select point cloud data from the software opened or from the folder. Point cloud data should be classified by the ground points at least.
- **Building footprint file**: Building vector footprint file supports shp, dxf, gpkg format file, can be generated through the Lidar360 vector editing module or use the existing building footprint file.
- **Roof type**: It is used to determine the use of point cloud when extract the shape of the roof. Building category is selected by default. If building classification is not performed on the data, you can co-opt the creation points, unclassified points, or user h household to customize the category of roof.
- **Ground Class**: It is used to determine the floor elevation of a building. The ground point category is used by default.
- **Minimum detail area**: It is used to extract the roof shape for surface filter parameters, a smaller value

can extract more details, and the error will increase correspondingly; a larger value can extract less details, the error rate is small. It can be set to 0 if no filtering is performed.

- **Output Path**: The path to save the model file.

Editor

Control the start and end of building model editing, and support to edit one building model object at a time. The building model objects that can be edited include: select building model files for editing, new sketch building model file.

Select Building Model File to Edit

Click Editor>Select Building Model File to Edit

The interface for selecting Building Model object data pops up. The Building Model object data that has been loaded into the editing window will appear in the selection list. Select the data to be edited.

If there is no building model object data in the edit window, a dialog box is displayed to select the building model file and load the data from disk for editing.

After starting editing, other functions of the Building Model editing tool bar will be activated.

During the editing process, the edited Building Model data cannot be removed from the editing window.

New Sketch Building Model

Click Editor>New Sketch Building Model File to Edit

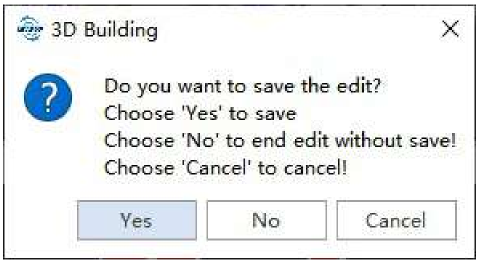
A temporary edit model object is constructed in the edit window. Manual extraction of buildings and manual editing after extraction can be carried out. At the same time, the edited building model data is loaded into the directory tree as a temporary object in the window.

After starting editing, other functions of the Building Model editing tool bar will be activated.

During the editing process, the edited Building Model data cannot be removed from the editing window.

Save

Functional Description: Click*editor > save*. If the sketch is not saved, the Save File interface will pop up, otherwise it will be saved to the original file.



End Edit

Click Editor > End Edit

After ending edit, a prompt box will pop up to save the editing results. After the user makes a selection as required, the user can exit the editing function, and the editing module of the building model becomes unavailable.

Utility Tools

Basic Function: including undo, redo and setting.

Undo

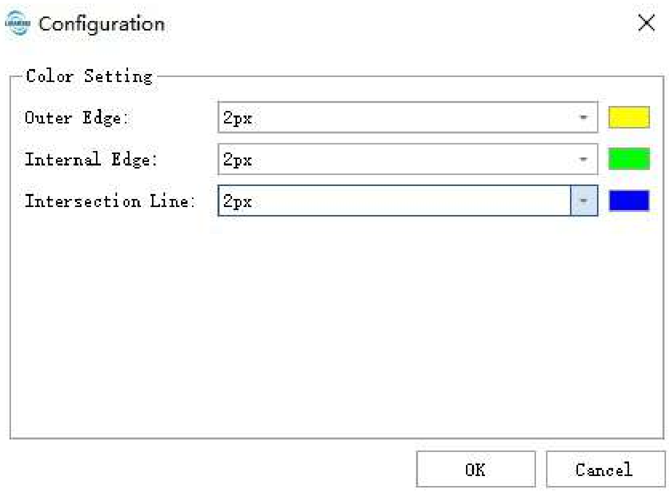
Undo the last edit.

Redo

Redo the last edit.

Settings

Configure building model to edit modular parameters.



- **Color settings**: Define the color of three linetypes, which are OuterEdge, InternalEdge, IntersectionLine.
- **Width settings**: Define the color of three linetypes, which are OuterEdge, InternalEdge, IntersectionLine.

Create the building model manually

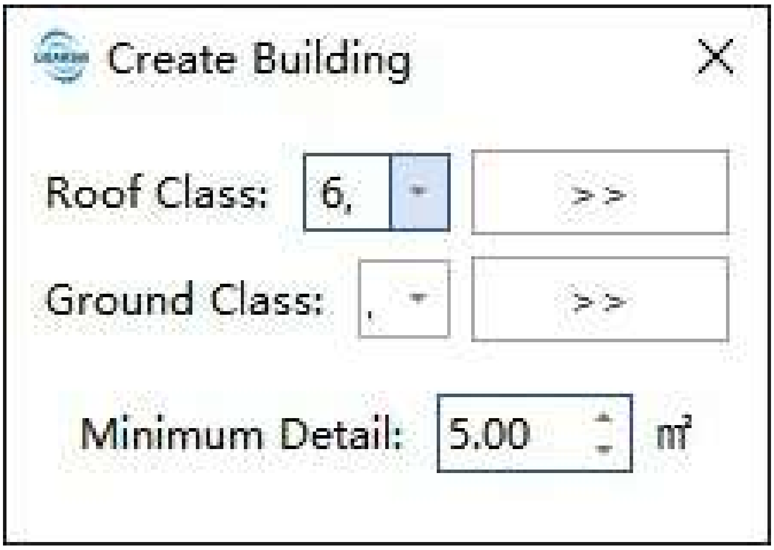
Functional Overview

Building model generation is carried out by drawing the footprint range of a single building, which is mainly used to automatically create buildings that are missing from the extracted building model function or the building vector footprint. The point cloud data is consistent with the functional requirements of creating building models automatically, and at least ground point classification should be carried out.

Usage

The process of manually creating a building model is as follows:

1. Click *3D modeling feature page* > *Editing*Button, Create or Select a building model file.
2. Click *3D modeling feature page >Create the building model* button to pop up the dialog:



1. Set parameters.
2. In the point cloud window, click the left mouse button to draw the boundary contour of the target building. Right mouse button can go back to the previous point, double click to confirm, and execute the generation of building model.

Parameters Settings

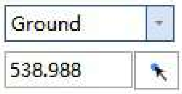
- **Roof type**: It is used to determine the use of point cloud when extract the shape of the roof. The selection of building category is defaulted. If the data is not classified by building, you can co-opt the created point, the unclassified point, or use it to customize the category as the roof.
- **Ground Class**: It is used to determine the floor elevation of a building. The ground point category is used by default.
- **Minimum detail area**: It is used to extract the roof shape for surface filter parameters, a smaller value can extract more details, and the error will increase correspondingly; a larger value can extract less details, the error rate is small. It can be set to 0 if no filtering is performed.

Building height model

The building height model is an auxiliary feature for building editing and is used to change the height of the overall building.

Functional Overview

Building ground elevation: Determine a new building ground elevation by manually inputing or selecting point cloud ground points.



Building elevation: Input manually and determine the overall height of the building.

Building Editor

Description

Choose a generated building model file or create a new building model file to start editing. This function supports manual creation of building models and adjustment of building model details.

Usage

- Editor
- UtilityTools
- CreateBuildings
- BuildingElevation
- BuildingEditorTools
- AssistantTools
- ShowingOrHidingTools
- BuildingModelsList

Building editing tool

Edit the endpoints, line segments and patches on the 2D vectorized objects of the building model.

Move Edge Vertex

Move the endpoint on the building. The operation steps are as follows.

1. Click the left mouse button to select the endpoint on the building.
2. Move the mouse to a new position.
3. Click the left mouse button to finish moving the endpoints of the building.
4. Click the right mouse button to end the move point operation.

Insert Edge Vertex

Insert a new node on the line segment of the building. The operation steps are as follows.

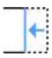
1. Click the left mouse button to select the line segment.
2. Move the mouse to a new position.
3. Click the left mouse button to select the insertion point location, or click the right mouse button to go back to step 1.
4. Click the right mouse button to end the insertion point operation.

Remove Edge Vertex

Delete the endpoint on the building. The operation steps are as follows.

1. Move the mouse to determine the currently-selected endpoint through the snap flag.
2. Click the left mouse button to delete the currently-selected endpoint.
3. Click the right mouse button to delete the point operation.

Move Edge

Move the line segment on the building. The operation steps are as follows.

1. Click the left mouse button to select the line segment on the building.
2. Move the mouse to determine the new position of the line segment, click the left button to complete the move, or click the right button to go back to step 1.
3. Click the right mouse button to end the move line operation.

Align Edge

Move the line segment you want to edit vertically or parallel through the selected reference line. The operation steps are as follows.

1. Move the mouse to determine the currently-selected endpoint through the snap flag, and click the left mouse button to select the reference line.
2. Move the mouse to determine the position of the edit line by capturing the flag, and click the left mouse button to select the edit line, or click the right mouse button to go back to Step 1.
3. Move the mouse to locate the new edit line and click the left mouse button to move the edit line, or right-click the mouse button to go to Step 2.
4. Click the right mouse button to end the alignment line operation.

Apply Intersection Line

Turn the adjacent step edges of two faces into intersecting line. The operation steps are as follows.

1. Move the mouse to determine the line segment through the snap flag that meets the operation conditions of applied cross line. Click the left mouse button to complete the operation of applied cross line.
    i. Right click to end the operation of applied cross lines.

Remove Patch

Delete patches. The operation steps are as follows.

1. Move the mouse, determine the patch to delete by snap identification, and click the left mouse button to delete the patch
2. Click the right mouse button to end the delete-face operation.

Split Patch

Split patch. The operation steps are as follows.

1. Click the left mouse button to select Patch to be edited.
2. Click the left mouse button to draw the first point of the split line, or click the right mouse button to end the current editing and go back to step 1.
3. Click the left mouse button to draw the second point of the split line to end the operation, or click the right mouse button to end the current editing and go back to step 1.

Merge Patches

Merge two adjacent surface patches The operation steps are as follows.

1. Click the left mouse button to select Patch 1
2. Click the left mouse button to select Patch 2 and merge Patch 2 onto Patch 1, or click the right mouse button to go back to step 1.
3. Click the right mouse button to exit the operation of merging patches.

Add Hole

Add an inner ring to building patch to form a void. The operation steps are as follows.

1. Click the left mouse button to select Patch to be edited.
2. Click the left mouse button to select points and draw a polygon. Double click the left mouse button to finish the drawing or click the right mouse button to return to a point of the polygon.
3. Right click to add face operation.

Split Building

Split the building from one to two. The operation steps are as follows.

1. Click the left mouse button to draw the first point of the split line.
2. Click the left mouse button to draw the second point of the split line, or click the right mouse button to end the current editing and go back to step 1.
3. Left click to select one side of the split line as a new building, and jump to the corresponding location at the same time, or right-click to end the current editing, and go back to step 2.
4. Right click to end the operation of dividing buildings.

Fit Patch

Fitting building roofs and adjusting the slope of individual roofs. The operation steps are as follows:

1. Select the point cloud category used for fitting the roof, the default is the building type point cloud.
2. Click the left mouse button to select the building roof to be fitted.

Extrude Building

Generating a new model, the specific operational steps are as follows:

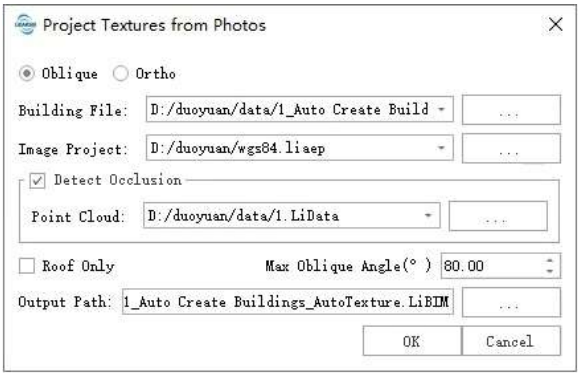
1. Open profile viewer and select the range of building profile to be constructed in the main viewer.
2. Open the function of Extract buildings, draw the outline of the building in the profile viewer, and double-click the left mouse button to finish drawing, and right-click to go back.
3. Move the mouse in the main Viewer, click the mouse to select and determine the two boundaries of the building, and complete the construction of the building.

Project Textures from Photos

Description

The project textures from photos function automatically applies textures to the roofs or walls of buildings using imported Aerotriangulation projects or orthophotos.

Usage



Parameters

- **Model**：ncludes two modes: Oblique and Ortho. Choose Oblique mode if using image project results for texture mapping, and choose Ortho mode if using an orthophoto.

Oblique Mode

- **Building File**：The building file (*.LiBIM) to be texture mapped.
- **Aerotriangulation projects**：The image project file (*.LiAep) used for texture mapping.
- **Detect Occlusion**： Whether to use a point cloud file for obstruction detection during texture mapping to reduce the likelihood of wall features being obstructed by other objects.
- **Point Cloud**：The point cloud file (*.LiData) used for occlusion detection.
- **Roof Only**：Whether to apply texture mapping only to roofs. This option is suitable for image projects that only contain nadir photos.
- **Max Oblique Angle**：Only use photos with angles between the shooting angle and wall slope less than this value for texture mapping to prevent excessive warping of wall textures.
- **Output Path**：The path to where the generated texture mapped building file would be saved.

Ortho Mode

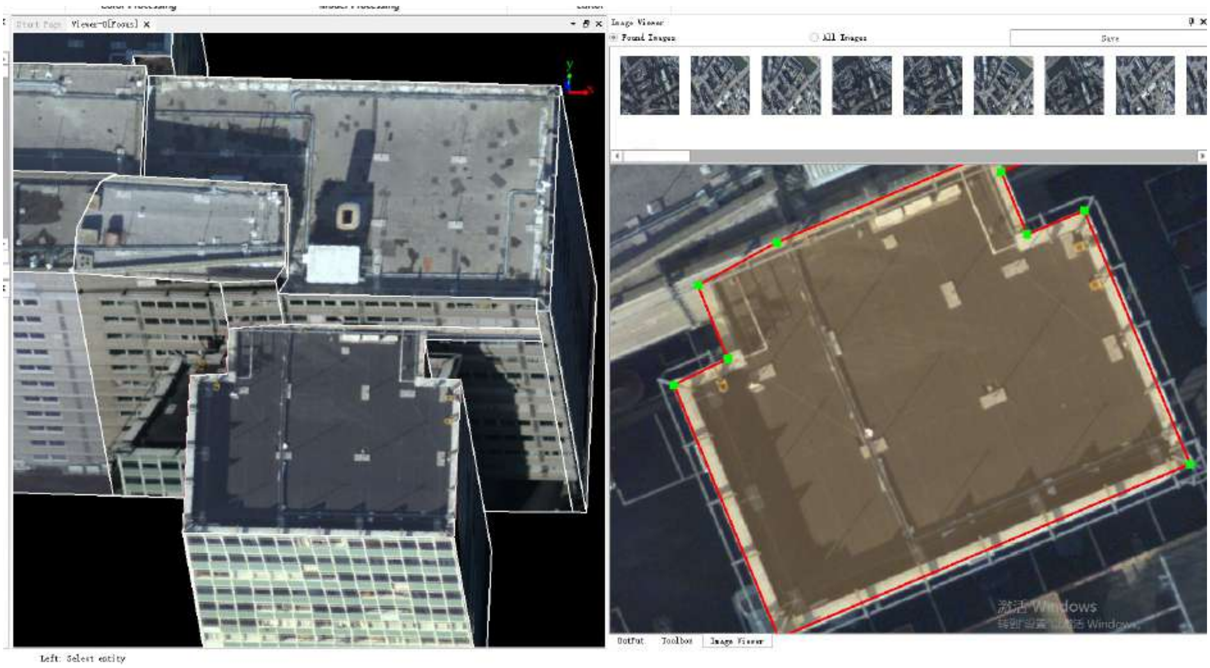
- **Building File**：The building file (*.LiBIM) to be texture mapped.。
- **DOM**：The orthophoto file (*.tiff) used for texture mapping。
- **Output Path**：The path to where the generated texture mapped building file should be saved.

  > Ortho mode can only apply texture mapping to roofs.

Matched Photos Editor

Description

The Matched Photos Editor in LiDAR360 enables editing of already textured LiBIM data. By selecting each building patch, choosing the desired image from the candidate list, and adjusting the endpoint position on the image, the texture of the building patch can be updated.



Usage

Select the building object to adjust texture in the building model list.

- Building Model List
- Image Viewer

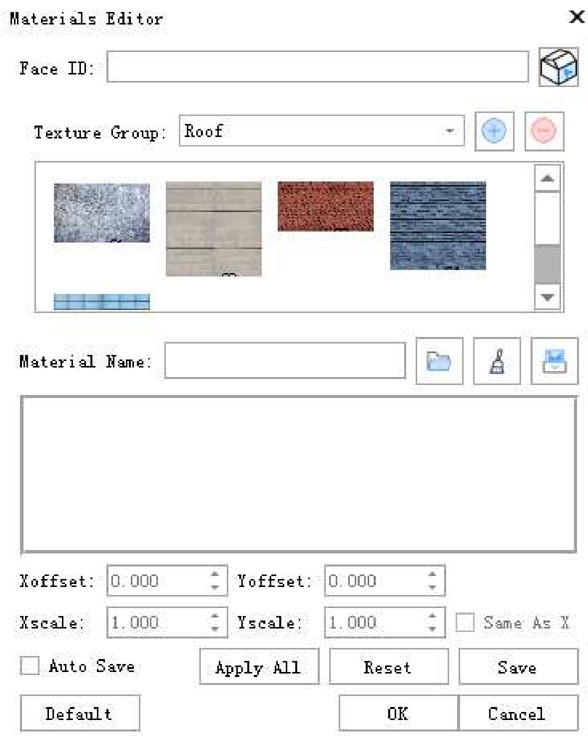
Materials Editor

Description: Edit the materials of building patches.

Process

1. Click the button to open the Materials Editor viewer, select or load the building data file to be edited, and the currently viewer containing the selected building data file is the workspace of this function.

2. After selecting building model data, Building Models List and materials editor interfaces will pop up.



3. (Optional) Click button to activate selection tool, left-click on a patch in this function's workspace, to select the patch and edit its material.

4. (Optional) Click button to clear current texture or image.

5. Select desired materials.

   5.1 Method one：Choose required materials from library by selecting group where they are located and clicking on them in material editor list interface.

   5.2 Method two：Choose required materials from local files by clicking button, pop up file selection dialog box, and set image file as material for the selected patch.

   5.3 (Optional) Click button to add currently selected material to library.

6. Adjust material control parameters.

   6.1 Method one：Set X,Y offset and X,Y scaling ratio of material in model materials editor interface.

6.2 Method two：Interactively control parameters by the interactive sphere that appears on the selected patch in workspace.

Parameters

- **X Offset**: Set the offset value of X-axis position of material origin point in texture image, valid range is 0-1.0, default value is 0.
- **Y Offset**: Set the offset value of Y-axis position of material origin point in texture image, valid range is 0-1.0, default value is 0.
- **X Scale**: Set scaling ratio along X-axis direction for texture image，default value is 1.
- **Y Scale**: Set scaling ratio along Y-axis direction for texture image，default value is 1.
- **Same as X**: Keep same scaling ratio between Y axis and X axis for current face's textures.
- **Auto Save**：Automatically save modifications made during previous building's materials editing when a new building has been chosen to edit.
- **Apply All**：Apply current patch's materials settings to other patches with same semantics within current building model.
- **Reset**：Restore current building's materials settings from LiBIM files.
- **Save**：Save current modifications into LiBIM files.
- **Default**: Reset all controls back to their default values.

Basic Function

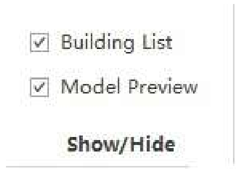
The tool is used for auxiliary window linkage during building model editing.

Auxiliary - Window Linkage

A new window and the window is linked with the main window to support the editing of architectural models.

Show/Hide

The functions of this tool include model preview, and opening and closing the building list.



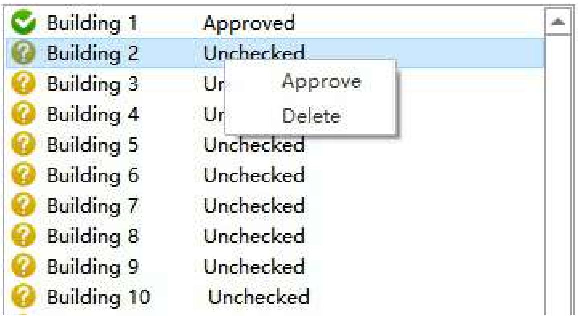
Model Preview

Open or close the building model preview window.

Open and Close Building List

Open or close the current list of buildings being edited.

Building Models List



This function displays the list of building models would be edited and records whether the current building model has been checked. Also it supports selection, deletion, and checking of building models.

Select

Left-click to select the building model in the list that you want to edit, and the editing viewer will turn to the position of the selected building model object.

Delete

Right-click on the building model list item, and left-click to select delete. It will remove the building model list item from the building list and remove the corresponding building model from the editing viewer.

Check

Right-click on the building model list and select "Checked". The corresponding building model item will display "Approve" as the suffix, and the icon in front of the list item will turn green with a check mark. The viewer will automatically jump to the next item, and the editing window would also jump to the position of the next building model.

Recompute

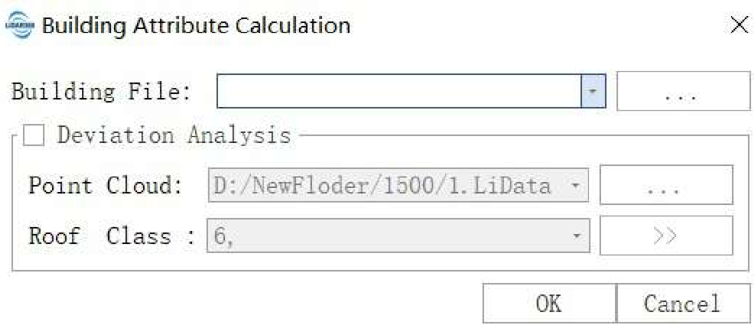
Right-click on building list,the list of building construction parameters will pop up. After adjusting the parameters, re-fit the new building.

Attributes Calculate

Description

The Attribute Calculation function automatically calculates the attributes of building models, such as building height, eave height, base elevation, roof orientation and roof shape based on the input data. The results will be output to the attribute table of the building file.

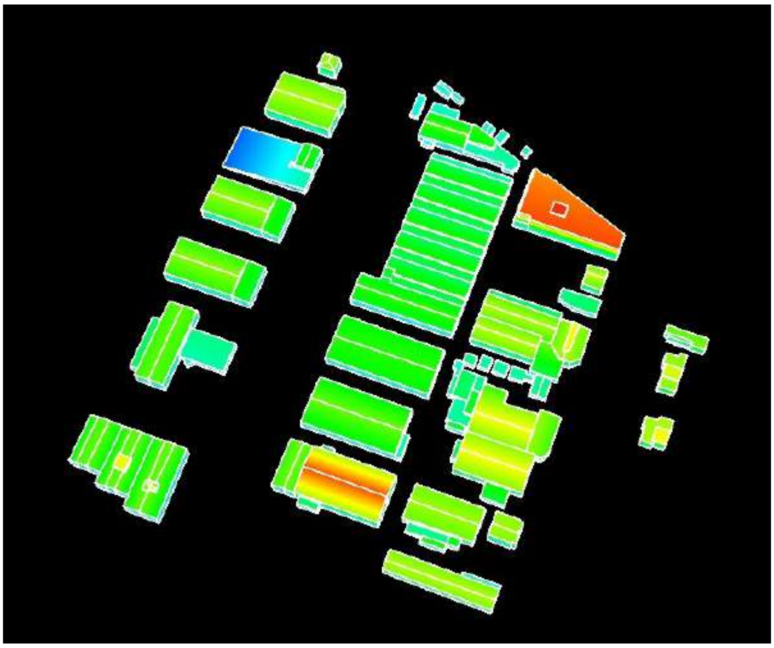
Usage



Parameters

- **Building File**: Input format is LiBIM building model file. You can select LiBIM data that has been opened in LiDAR360 software through the drop-down box or load LiBIM data by clicking on the button on the right.
- **Deviation Analysis (Optional)**: This function mainly calculates RMSE error values from points in point clouds to building models. By default, it calculates RMSE errors from building points (category 6) to building models.

The result of Building Attribute Calculation is shown in the figure below.



D:/code/data/Building/bds_ed1.LiBIM

	BLDG HEIGHT	EAVE HEIGHT	BASE ELEV	ROOF DIR	ROOF FORM	ROOF AREA	RMSE	ROOF PERIMETER
1	10.1292	4.2390	530.3198	nan	Hip	1633.2988	0.4163	177.7412
2	8.8038	4.1050	530.4596	287.4143	Gable	1278.4382	0.4892	154.0623
3	7.7086	7.0576	531.5613	287.1822	Gable	1344.7988	0.2269	158.2917
4	7.8654	4.4142	531.4487	287.4439	Gable	4441.6433	0.3248	298.1643
5	3.2265	2.5723	537.0738	129.1170	Gable	52.5579	0.5950	29.9624
6	3.1566	nan	537.0532	nan	Flat	19.0106	0.4572	18.7840
7	3.3110	2.8918	536.8526	211.9078	Gable	42.1643	0.4744	28.0272
8	3.0992	nan	537.3420	nan	Flat	32.6564	0.3230	23.5182
9	3.1824	nan	537.4614	nan	Flat	16.4622	0.5268	16.3328
10	8.6141	6.6843	532.4203	287.1156	Gable	122.5950	0.7851	43.3045
11	4.0882	nan	536.6829	nan	Flat	164.6554	0.4868	58.4310
12	3.5539	2.8457	537.4094	17.0795	Gable	91.3578	0.3895	38.3443
13	3.3764	nan	537.3031	nan	Flat	55.4418	0.2551	34.7371
14	3.3088	nan	537.2603	nan	Flat	42.6029	0.2010	28.1301
15	3.9458	nan	536.6133	nan	Flat	30.3163	0.1567	22.0970
16	5.9765	2.7567	534.1654	197.7521	Gable	425.5805	0.1628	104.9272

Tutorials

- Getting Started Tutorial
- Point Cloud Data Basic Operations Tutorial
    - Data Management
    - Statistics
    - Classify
- Strip Alignment Tutorial
- Terrain Tutorial
- Forestry Tutorial
- ALS Regression Analysis
- Power Line Dangerous point detection

Videos

1. LiDAR360 Installation and License Activation
2. LiDAR360 Overview and Loading Data
3. LiDAR360 Window Management
4. LiDAR360 Measurement Tools
5. LiDAR360 Selection Tools
6. LiDAR360 Profile Tools
7. LiDAR360 Camera Roam in 90 Seconds
8. Point Cloud Generation
9. LiDAR360 Strip Adjustment Module
10. LiDAR360 Classify Ground
11. LiDAR360 Classify Ground by Selected
12. LiDAR360 Terrain Module
13. LiDAR360 Tree Segmentation
14. LiDAR360 ALS Forestry Module
15. LiDAR360 TLS Forestry Module

Appendix

This chapter introduces some key terms, file formats, shortcut keys and ways of high-performance graphics mode adjustment.

File Formats

This section introduces the data formats supported by LiDAR360 and the specific file formats involved in various functional modules, such as POS file, control point file, seed point file and so on.

LiData

LiData is a point cloud data file format defined by LiDAR360. It consists of a public header block, variable length records, and point data records.

This format can be exchanged with other common point cloud data format files, including LAS, LAZ, E57, PLY, ASCII, etc.

When loading common point cloud data formats (including: LAS, LAZ, E57, PLY, ASCII, etc.) into LiDAR360, a LiData file named after the same name will be generated, and the subsequent operations are based on that LiData file.

LiData file has a version number to indicate its version. Currently, the most updated version of LiData is V2.0.

Starting from LiDAR360 V4.0, the all the LiData versions before V2.0 are supported. For LiDAR360 V3.x, the highest supported version is LiData V1.9. The differences between LiData V1.9 and V2.0 are listed as follows:

Features	LiData V1.9	LiData V2.0
Classfication	0-31 （32 classes in total)	0-255 (256 classes in total)
Classfication Flag	Not supported	Supported
Scanner Channel	Not supported	Supported
Near Infrared	Not supported	Supported
Scan Angle Rank	In degree, in the range of [-90°~+90°]	In in range of [-30000, +30000] (unitless) which corresponds to the degree of [-180°, +180°]
Additional Attribute	Not supported	Supported

Additional Attributes are stored in the LiAtt files

LiAtt File Format

LiAtt is the additional attribute file format of LiData. It consists of a file header and additional attribute data records.

The additional attribute is supported since LiData V2.0.

The additional attributes are matched with the corresponding LiData file by the file name.

When the non-matching LiData and LiAtt files are manually modified to have the same file name, the software may still be able to recognize all the additional attributes in the LiAtt file, but the corresponding additional attribute values may be incorrect. Modifying the LiAtt may destroy the LiAtt file structure and cause the additional attribute file to no longer be recognized when opening the LiData file next time.

Deleting LiAtt files does not affect the normal use of LiData files. But after LiDAR360 loads LiData, deleting the corresponding LiAtt files may cause some operations to fail and cause unknown errors.

LAS

LAS file format Support the LAS file in the following formats:

LAS 1.1

LAS 1.2

LAS 1.3*

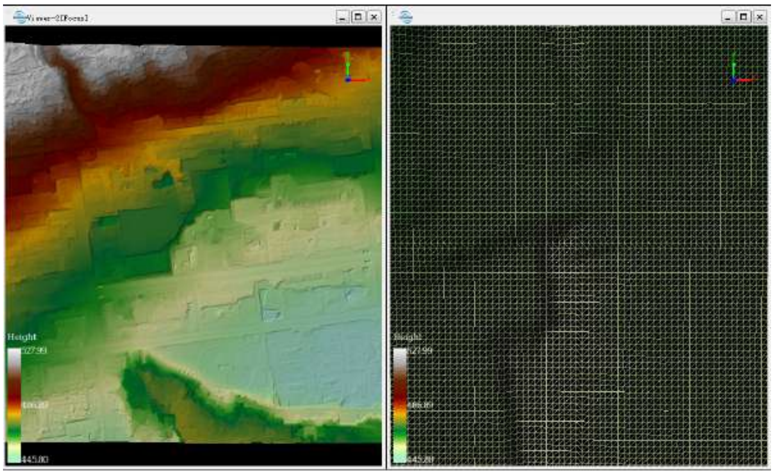
LAS 1.4*

-
    - Limitation for LAS 1.3 and LAS 1.4:

      Not support waveform data currently.

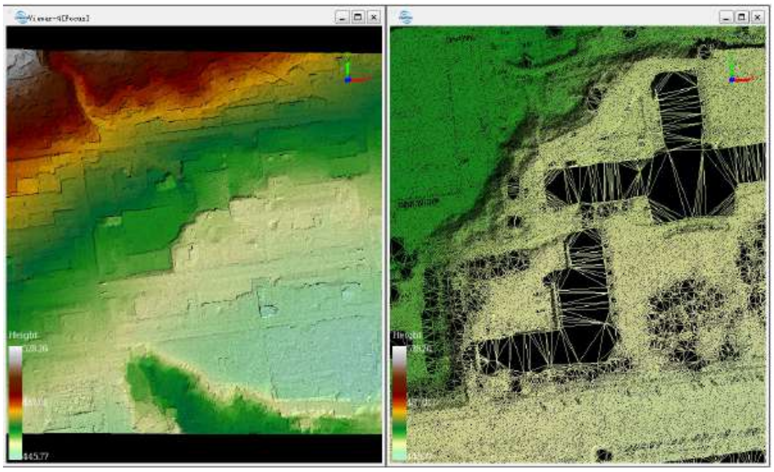
LiModel

LiModel file stores triangulated regular network models generated by DEM or DSM. It saves regular grid nodes and block-organized triangulated regular network models according to quadtree. DOM texture information can be superimposed on the model. LiModel could be transformated from massive data of DEM or DSM. It supports editing operations including flatten height, smooth height, noise points removal and so on. After editing, it can be exported to Tiff file for generating contours.



LiTin

The LiTin file is generated by the irregular 2.5D triangulation model based on the point cloud. It can improve quality of contour lines by editing operations including flatten height, smooth height, vertex addition and removal, etc. It organizes data in full memory mode, which takes up large memory and causes low rendering efficiency. So it is suggested to generate LiTin file into tiles.



Clipping Point Cloud File Format

Summary

The file is a text file that contains a list of scopes used to load multiple circles or rectangles.

Clip by Circle File Format

Each circle consists of three values separated by commas: the X coordinate, the Y coordinate and the radius of the center.

The following table is an example of a circular extent file:

```
322610.51, 4102305.22, 50

322685.86, 4102400.5, 50

322820.45, 4102510.21, 100

322850.35, 4102655.33, 100

323000.00, 4103000.00, 60
```

The above values can be stored in a text file (such as "extents.txt") and loaded into the LiDAR360 software.

Clip by Rectangle File Format

Each rectangle consists of four values separated by commas: X minimum, X maximum, Y minimum and Y maximum.

The following table is an example of a rectangular extent file:

```
322601.255, 322801.255, 4102309.655, 4102409.655

322548.966, 322600.110, 4102310.180, 4102360.180

322539.155, 322600.255, 4102309.655, 4102359.655

322745.950, 322780.110, 4102204.660, 4102250.180

322875.224, 322975.224, 4102028.660, 4102128.660
```

The above values can be stored in a text file (such as "extents.txt ") and loaded into the LiDAR360 software.

Extract by Time File Format

Each extraction range consists of two values separated by commas: starting GPS time and ending GPS time.

The following table is an example of extracting range files by GPS time:

```
526494.500,527494.500

527494.500, 528494.500

527494.500, 528494.500

528494.500, 529494.500

529494.500, 530494.500
```

The above values can be stored in a text file (such as "extents.txt ") and loaded into the LiDAR360 software.

Homologous Points File Format

Homologous points file is a comma separated text file. The first row is a file header. Each row consists of 9 columns: ID, reference coordinate (X,Y,Z), alignment coordinate(X,Y,Z).

```
 ID     ref_X        ref_Y        ref_Z      align_X       align_Y    align_Z
P01, 322500.1100, 4102499.9600, 2613.1400, 322500.0089, 4102499.9856, 0.0000
P02, 322999.8400, 4102499.7900, 2614.3400, 322999.9283, 4102499.9585, 0.0000
P03, 322999.8300, 4102000.3200, 2554.4100, 322999.8738, 4102000.1457, 0.0000
P04, 322500.3700, 4102000.1600, 2490.7400, 322500.1140, 4102000.0595, 0.0000
```

POS File

The POS File contains information such as GPS time, longitude, latitude, height, roll, pitch, heading, GridX and GridY. GPS time, longitude, latitude, height, roll, pitch and heading must be necessary, while longitude and latitude or GridX and GridY include at least one. If POS file doesn't include GridX and GridY, we need to set projection coordinate system when we input trajectory files. GridX and GridY can be calculated according to projection coordinate system, longitude and latitude in strip alignment module.

The trajectory information stored in its file is structured as follows:

Data	Units	Type
GPS time	seconds	double
longitude	degree	double
latitude	degree	double
height	meters	double
roll	degree	double
pitch	degree	double
heading	degree	double
GridX(Optional)	meters	double
GridY(Optional)	meters	double

The examples the POS file is shown below. Example 1(not including GridX，GridY):

```
380954.000,112.5311950876, 26.8969520123,378.543,  7.1701230000,  3.0890110000,-39.4065340000
380954.008,112.5311938923, 26.8969533249,378.537,  7.2001860000,  3.0914780000,-39.4034150000
380954.016,112.5311926975, 26.8969546376,378.531,  7.2368710000,  3.0936380000,-39.4011190000
380954.024,112.5311915034, 26.8969559507,378.525,  7.2683090000,  3.1015050000,-39.3975470000
380954.032,112.5311903098, 26.8969572641,378.518,  7.3007560000,  3.1115160000,-39.3929590000
380954.040,112.5311891169, 26.8969585779,378.512,  7.3269790000,  3.1179720000,-39.3878260000
380954.048,112.5311879247, 26.8969598920,378.506,  7.3525870000,  3.1180460000,-39.3804020000
380954.056,112.5311867331, 26.8969612065,378.500,  7.3745730000,  3.1151630000,-39.3713830000
```

Example 2(including GridX，GridY):

```
383207.336,112.5421590662,26.9034172036,313.865,3.538615,2.660518,-67.848653,653147.099716932,2976670.6235468
9
383207.344,112.5421572108,26.9034177865,313.861,3.533299,2.659177,-67.840828,653146.914649722,2976670.6858765
4
383207.352,112.5421553554,26.9034183697,313.857,3.522385,2.658042,-67.828619,653146.729582108,2976670.7482394
3
383207.36,112.5421535001,26.9034189529,313.854,3.512757,2.659231,-67.816251,653146.544524429,2976670.81060244
383207.368,112.5421516447,26.9034195363,313.85,3.502656,2.662677,-67.807435,653146.35945655,2976670.87298749
383207.376,112.5421497892,26.9034201198,313.846,3.502243,2.664987,-67.803265,653146.174378605,2976670.9353835
383207.384,112.5421479336,26.9034207035,313.843,3.500293,2.668456,-67.80232,653145.989290462,2976670.99780155
383207.392,112.5421460783,26.9034212874,313.839,3.501546,2.671267,-67.797563,653145.804231844,2976671.0602421
2
383207.4,112.5421442231,26.9034218713,313.835,3.496569,2.674773,-67.789195,653145.619183163,2976671.12268281
383207.408,112.542142368,26.9034224554,313.832,3.483849,2.676885,-67.774991,653145.434144147,2976671.18514579
383207.416,112.5421405129,26.9034230395,313.828,3.471533,2.676137,-67.765536,653145.24910513,2976671.24760876
```

```
383207.424,112.5421386577,26.9034236237,313.824,3.47028,2.675779,-67.760612,653145.064056049,2976671.3100827
383207.432,112.5421368024,26.9034242079,313.82,3.475101,2.677064,-67.761833,653144.878997039,2976671.37255652
383207.44,112.5421349471,26.9034247923,313.817,3.476053,2.681571,-67.761664,653144.69393776,2976671.43505249
```

OUT File

Out file is a binary file that stores trajectory information. The following table shows the format of the POSPac SBET file provided by Applanix. For details, refer to the PosPac quick start guide.

The trajectory information stored in its file is structured as follows:

Data	Units	Type
time	seconds	double
latitude	radians	double
longitude	radians	double
altitude	meters	double
x velocity	meters/second	double
y velocity	meters/second	double
z velocity	meters/second	double
roll	radians	double
pitch	radians	double
platform heading	radians	double
wander angle	radians	double
x body acceleration	meters/second^2	double
y body acceleration	meters/second^2	double
z body acceleration	meters/second^2	double
x body angular rate	radians/second	double
y body angular rate	radians/second	double
z body angular rate	radians/second	double

Control Point File Format

The control point file contains a list of control points in TXT format. The first row is the file header, while other rows store comma separated X, Y, Z coordinates of control points. The following table is an example of a control point file:

```
X, Y, Z
473575.563, 291005.332, 127.244
473576.899, 291004.245, 126.328
473576.899, 291004.243, 126.317
473576.899, 291004.245, 126.328
473576.899, 291004.243, 126.317
```

Notes Elevation Points File

The notes elevation points file is a comma-delimited CSV file. The first row is the file header, while other rows store X, Y, Z and Label (separated by comma).

The following figure shows an example of notes elevation points file:

```
X, Y, Z, Label
322539.46, 4102000.01, 2489.21, 2489.21
322551.33, 4102009.72, 2489.55, 2489.55
322562.85, 4102000.03, 2489.74, 2489.74
322563.16, 4102019.38, 2489.98, 2489.98
322511.58, 4102056.04, 2492.86, 2492.86
```

Sample Data File

The sample data file is a text file (*.txt) delimited by commas that contains a file header in the first row. The first two columns of each row are the X and Y coordinates, followed by a number of dependent variables. Multiple dependent variables can be stored, but only one is used for each regression analysis.

The dependent variable is tree height in the following example of a sample data file:

```
X,Y,Height
322859.25,4102463.86,33
322862.25,4102459.35,31.5
322864.56,4102462.49,32
322874.58,4102463.50,35
322655.52,4102192.25,21.1
```

ALS seed point format

Files in csv or custom treedb format, as shown in the example below:

```
TreeID,TreeLocationX,TreeLocationY,TreeLocationZ
1,322971.5,4102497.5,47.387
2,322549.5,4102496.5,49.42
3,322678.5,4102495.5,48.456
4,322716.5,4102494.5,34.366
5,322516.5,4102489.5,23.726
```

TLS seed point format

Files in csv or custom treedb format, as shown in the example below:

```
TreeID,TreeLocationX,TreeLocationY,TreeLocationZ,DBH
1,-5.135,-21.219,1.303,0.244
2,-1.693,-30.778,1.3,0.365
3,3.249,-26.096,1.3,0.389
4,-3.852,-28.103,1.304,0.309
5,4.89,-30.851,1.301,0.299
```

CHM Segmentation Result

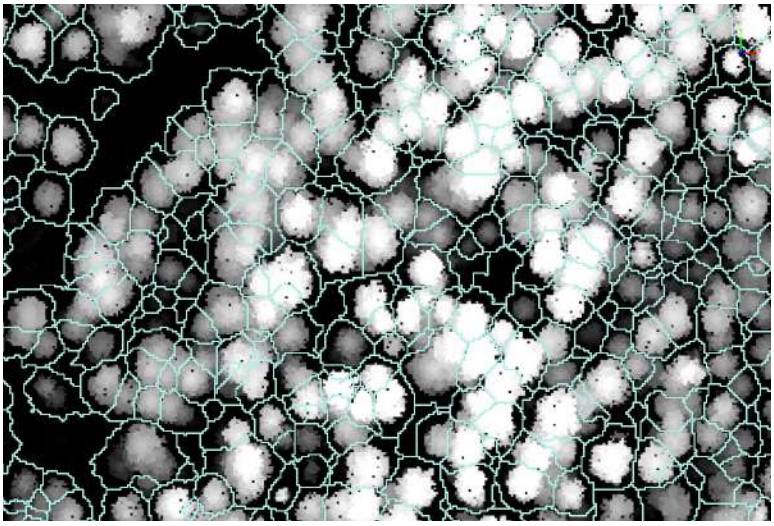
The CHM segmentation result contains a comma-separated .csv table file and a polygon-type .shp vector file.

The .csv table contains the attributes of tree ID, x, y coordinate position, tree height, crown diameter and crown area.

The following table shows an example of the segmentation result of CHM:

```
TreeID, TreeLocationX, TreeLocationY, TreeHeight, CrownDiameter, CrownDiameter(S-N),CrownDiameter(E-W),CrownA
rea
1, 322716.24, 4102494.69, 36.165, 8.982, 9.865, 8.099, 63.36
2, 322751.21, 4102499.9, 41.282, 4.491, 5.132, 3.850, 15.84
3, 322519.35, 4102499.3, 32.008, 4.708, 4.712, 4.704, 16.64
4, 322742.15, 4102497.7, 26.956, 8.347, 7.536, 9.158, 54.72
5, 322892.26, 4102499.5, 45.493, 7.792, 7,967, 7.617, 47.68
```

The SHP file contains the geometric extents of each tree segmented. The attribute table contains the ID, x, y coordinate position, tree height, crown diameter and crown area attributes of each tree.



C:/ALS Forest Sample Data/Forest_Outliers Removal_CHM Segmentation.shp

	TreeID	X	Y	TreeHeight	Diameter	Area
1	33	322716.250	4102494.750	17.719	8.272	53.750
2	7	322751.250	4102499.250	8.767	3.141	7.750
3	9	322813.750	4102499.250	29.870	2.931	6.750
4	1	322510.750	4102499.250	13.972	6.358	31.750
5	2	322519.750	4102499.250	13.099	7.756	47.250
6	4	322573.250	4102499.250	7.661	3.568	10.000
7	6	322733.750	4102499.250	19.508	5.140	20.750
8	16	322742.750	4102498.250	12.150	4.686	17.250
9	19	322746.750	4102497.250	19.537	4.686	17.250
10	8	322766.250	4102499.250	17.510	4.853	18.500
11	12	322838.750	4102499.250	32.976	7.878	48.750
12	10	322815.250	4102499.250	31.832	3.568	10.000
13	5	322578.250	4102499.250	11.403	6.154	29.750
14	13	322892.250	4102499.250	32.772	7.797	47.750

ALS Forest Results of Point Cloud Segmentation & PCS with Seeds

The ALS Forest results of point cloud segmentation and PCS with seeds are comma-separated .csv table files containing the attributes of TreeID, x, y coordinate position, tree height, crown diameter, crown area and crown volume.

The following table shows an example of the results of the ALS individual tree segmentation:

```
TreeID, TreeLocationX, TreeLocationY, TreeHeight, CrownDiameter, CrownArea, CrownVolume
1, 322511.52, 4102089.78, 60.606, 10.85, 92.04, 3240.171
2, 322511.81, 4102015.14, 53.785, 13.15, 135.09,4358.651
3, 322537.43, 4102062.51, 46.667, 12.45, 126.76, 3272.472
4, 322529.42, 4102073.12, 45.197, 4.071, 13.019, 342.992
5, 322525.07, 4102101.07, 43.861, 3.105, 7.57, 107.672
```

TLS Forest Results of Point Cloud Segmentation & PCS with Seeds

The TLS Forest results of point cloud segmentation and PCS with seeds are comma-separated .csv table files containing the attributes of TreeID, x, y coordinate position, tree height, DBH, crown diameter, crown area and crown volume.

The following table shows an example of the results of the TLS individual tree segmentation:

```
TreeID, TreeLocationX, TreeLocationY, TreeHeight, DBH
1, 136547.147, 289995.532, 6.498, 0.081
2, 136562.037, 289985.496, 5.652, 0.112
3, 136543.853, 290001.586, 6.115, 0.078
4, 136547.766, 289997.909, 5.868, 0.081
5, 136547.127, 289980.102, 5.217, 0.097
```

Format of Tree Attribute Information from TLS Point Cloud Segmentation

Individual tree attribute information from TLS point cloud segmentation is recorded in comma-separated .csv table files containing the attributes of tree ID, x, y coordinate position, tree height, DBH, crown diameter, crown area, crown volume, crown base height and straightness.

The following table shows an example of the individual tree attribute information from TLS point cloud segmentation:

```
TreeID, TreeLocationX, TreeLocationY, TreeHeight, DBH, CrownDiameter, CrownArea, CrownVolume, CBH, Straightne
ss
1, 136547.147, 289995.532, 6.498, 4.394, 12.964, 132.006, 120.961, 1.221, 7
2, 136562.037, 289985.496, 5.652, 3.008, 6.234, 30.524, 32.742, 1.351, 3
3, 136543.853, 290001.586, 6.115, 4.123, 8.627, 58.447, 78.947, 1.234, 4
4, 136547.766, 289997.909, 5.868, 4.464, 9.574, 71.984, 43.512, 1.471, 6
```

```
5, 136547.127, 289980.102, 5.217, 6.070, 11.155, 97.727, 307.870, 1.186, 1
```

Seven Parameters Calculation File Format

Control points should have at least three pairs and should be saved correspondingly in source and targeted coordinate files. Supported coordinate formats are spatial Cartesian coordinate system (X, Y, Z) and geographic coordinate system (B, L, H) (in the format of degree or degree:minute:second). Each point should be written in one row, and the geographic coordinates should be written in format of latitude, longitude, and height, separated by comma.

Example (degree:minute:second):

```
24:20:52.04982,116:40:30.53733,75.01
24:20:16.93056,116:40:25.28056,78.57
24:19:41.28740,116:40:18.60240,83.92
```

Section Result File Format

Two-Dimensional Section Result File Format

The 2D section file format can be ASCII file (.txt), vector file (.shp), dxf file (.dxf), and hdm format.

- csv file is separated by comma. Names can help to distinguish they are cross section or longitudinal section (CrossSection is the name for points in cross section and LongiSection is the name for points in longitudinal section by default). The section result file contains three columns, including name, distance to origin (in longitudinal section, the origin is the starting point. In longitudinal section, the origin is the intersection of cross section and longitudinal section. To left is negative, to right is positive), and elevation. Name suggests in which cross section this point is. E.g., CrossSection1(0) indicates this point is in the cross section named CrossSection1, and this cross section and the longitudinal section 0 are orthogonal. Elevation records the true hight of this point in the three-dimensional coordinate system. Following is the part of exported two-dimensional section result file:

```
Name,Distance,Height
CrossSection1(0),39.510,0.025
CrossSection1(0),39.413,0.085
...
CrossSection2(0),33.884,0.000
CrossSection2(0),32.447,0.033
...
CrossSection7(1),43.774,0.000
CrossSection7(1),42.662,0.025
```

- hdm (south). This format file is a comma-separated text file that records the mileage of each section and the distance perpendicular to the central axis. The format can be described as follows:
  - One hdm file can contain multiple cross sections.
  - Each section line starts with BEGIN field, followed by section mileage, section serial number (can be omitted), and ends before the next BEGIN field. For example, BEGIN, 0.000:1, it means that the mileage of this section is 100.000, and the serial number is 1.
  - The section data is then recorded and separated by commas. There are 2 fields in each row. The first field is the offset and the second field is the elevation. The offset is divided into positive and negative points. Mileage direction), the left deviation is negative, the right deviation is positive.

```
- For the cross section without a middle stake point, the offset takes the first point on the left as the bas
e point, and both are positive.


- The example is as follows, including 3 hdm data with mid-stake point section:

        BEGIN,0.000:1
        -13.265,90.242
        -12.617,90.284
        0.000,90.173
        13.819,89.890
        18.002,89.631
        19.329,88.435
```

```
        20.581,88.404
        20.721,89.645
        23.318,89.641
        BEGIN,7.229:2
        -13.918,90.280
        -7.584,90.219
        0.000,90.106
        1.415,89.919
        16.547,89.944
        BEGIN,27.542:3
        -24.713,90.563
        -22.875,90.290
        -7.484,90.045
        0.000,89.819
        7.535,89.624
        21.322,89.731

    Note: The middle pile is not necessarily the center point of the cross section, and the left and right wi
dths of some cross sections are different
```

- hdm (latitude ground). This file is a comma-separated text file that records the pile number, the height of the middle pile and the distance to the middle axis of each section. One hdm file can contain multiple cross sections. Each section line occupies three lines, the first line is the mileage and the height of the middle pile, the second line is the offset and elevation in the left direction of the line, and the third line is the offset and the elevation in the right direction of the line. Then record a section line.

```
  Mileage pile number Middle pile height
  Left Offset Elevation Left Offset Elevation Left Offset Elevation Left Offset Elevation Left Offset El
evation
  Right offset Elevation Right offset Elevation Right offset Elevation Right offset Elevation Right offs
et Elevation
  0    288.08
      5.93    0.18    21.79    3.45    32.07    4.49    36.32    8.21
      4.22    -0.09    10.03    0.08    12.31    -5.28    26.9    -5.28
  0    456
      4.22    -0.09    10.03    0.08    12.31    -5.28    26.9    -5.28
      5.93    0.18    21.79    3.45    32.07    4.49    36.32    8.21
  ...
```

- When the export format is a vector file, distance and height will be connected as X and Y values respectively to form a series of polygons. Each polygon corresponds to a section. The section name is saved in the file as an attribute value.
- When the export format is dxf (2004), each section will be automatically arranged into a table with N rows and three columns according to their respective coordinate values, and the vertical sections will form a single row. Below each section is a text label (optional), the label content includes left and right width, middle pile height difference, fill and excavation area, etc. When exporting, you can choose horizontal and vertical scales, whether to add elevation symbols on the survey section, and the prefix of the fill and cut mark text. The scale information will be marked in the lower right corner of the graph. The label text and the cross-section coordinates are in different layers.

Three-Dimensional Section Result File Format

The 3D Section result file format can be ASCII file (.csv), vector file (.shp) and dxf file (.dxf).

- ASCII file. A comma-separated text file. There are 4 columns in the file, namely: name, X coordinate, Y coordinate, and Z coordinate. Points with the same name belong to the same section.

  Name,X,Y，Z CrossSection1(0),323128.788,4104260.073,0.025 CrossSection1(0),323130.481,4104261.752,39.413 ... CrossSection2(0),323245.294,4104234.559,0.073 CrossSection2(0),323245.659,4104234.934,0.103 ... CrossSection7(1),323429.209,4104472.832,0.000 CrossSection7(1),323430.965,4104472.877,0.011

- Vector file. X, Y, and Z will be sequentially connected as coordinate values to form a series of polygons. Each polygon corresponds to a section. The section name will be saved in the file as the attribute value.

- dxf format file. X, Y, and Z will be sequentially connected as coordinate values to form a series of polygons. Each polygon corresponds to a section.

Note: When exporting multiple files, all vertical sections and cross sections will be output as separate files; when exporting a single file, all sections belonging to the same vertical section will be integrated into one file, and all vertical sections will remain Will be output as a separate file.

Matrix Format

The matrix file has four rows and four columns, each row of data is separated by a comma.

```
a00,a01,a02,a03

a10,a11,a12,a13

a20,a21,a22,a23

a30,a31,a32,a33
```

See example below:

```
0.99, 0.085, 0.86, -100

0.99, 0.085, 0.86, 100

0.99, 0.085, 0.86, 100

0, 0, 0, 1
```

Centerline File

A centerline file is a comma-separated CSV file containing two columns: X and Y.

Below is an example of a centerline file:

```
X, Y,
322539.46, 4102000.01
322551.33, 4102009.72
322562.85, 4102000.03
322563.16, 4102019.38
322511.58, 4102056.04
```

LiBIM File Format

LiBIM is a file format for saving 3D Building Modeling data. It consists of a file header block and a model data records.

It can import and display the data content through LiDAR360 software, and edit the 3D building model based on the data. It also can support the functions like converting LiBIM to OBJ, converting LiBIM to FBX, converting LiBIM to CityJson.

ID List of Json Callable Functions for Batch Processing

The following table shows the Plugin ID and Action ID of each Json callable function available in LiDAR360.

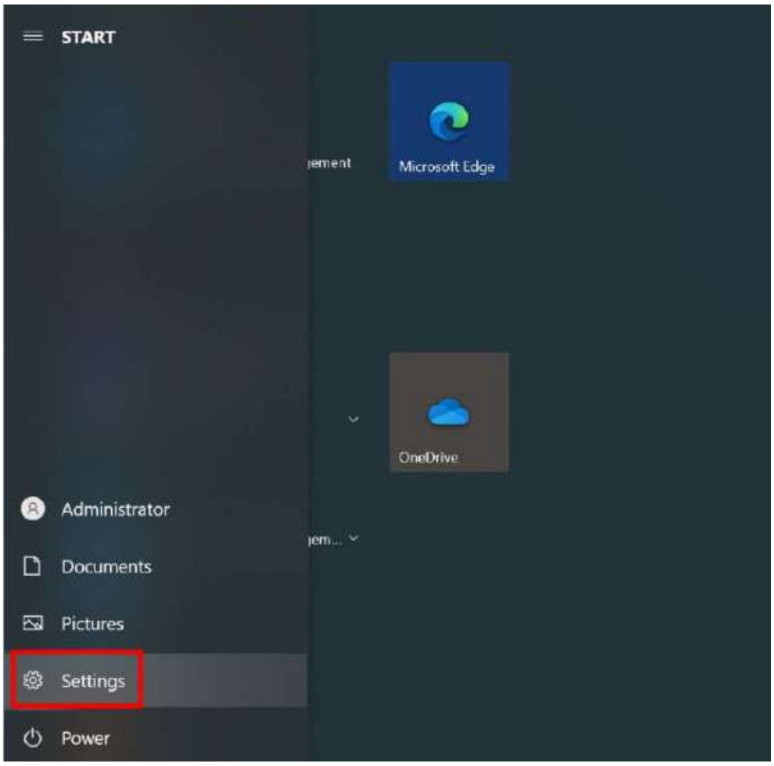
Function Name	Plugin ID	Action ID
Remove_Outliers	0	0
Normalize_by_DEM	0	1
Normalize_by_Ground_Points	0	15
Tile_by_Range	0	2
Tile_by_PointNumber	0	16
Merge	0	3
Extract_Point_Cloud_Boundary	0	4
Subsampling	0	5
PCV_Rendering	0	6
Define_Projection	0	7
Reproject	0	8
Extract_Color_from_Image	0	9
Subdivision	0	10
Transformation	0	11
Clip_by_Circle	0	30
Clip_by_Rectangle	0	31
Clip_by_Polygon	0	32
Convert_to_LiData	0	40
Convert_to_Las	0	41
Convert_to_ASCII	0	42
Convert_to_TIFF	0	43
Convert_to_Shape	0	44
Convert_to_DXF	0	48
Extract_by_Class	0	60
Extract_by_Elevation	0	61
Extract_by_Intensity	0	62
Extract_by_Return	0	63
Classify_Ground_Points	1	0
Extract_Median_Ground_Points	1	5
Classify_by_Attribute	1	10

Classify_Low_Points	1	11
Classify_Below_Surface_Points	1	12
Classify_Isolated_Points	1	13
Classify_Air_Points	1	14
Classify_byHeightAboveGround	1	15
Classify_byMinElevationDifference	1	16
Classify_Buildings	1	20
Classify_Powerlines	1	21
Classify_Model_Key_Points	1	17
Classify_by_Machine_Learning	1	25
Classify_by_Trained_ML_Model	1	26
DEM	2	0
DSM	2	1
Point_Cloud_to_Contour	2	20
Generate_TIN	2	23
Elevation_Metrics	4	0
Intensity_Metrics	4	1
Canopy_Cover	4	2
Leaf_Area_Index	4	3
Gap_Fraction	4	4
Linear_Regression	4	5
Support_Vector_Machine	4	6
Fast_Artificial_Neural_Network	4	7
Point_Cloud_Segmentation	4	10
Generate_Seeds_from_Layer_Stacking	4	14
Clear_Tree_ID	4	11

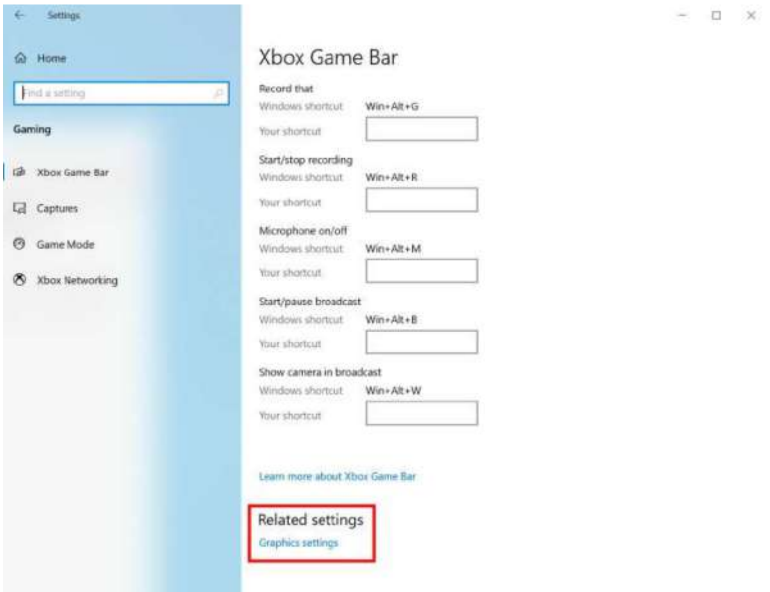
High-Performance Graphics Mode Adjustment

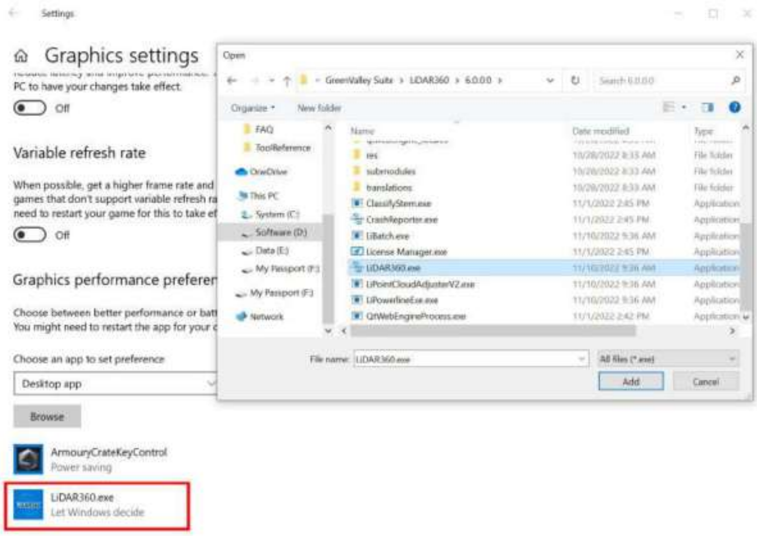
1. Open system settings

1. Choose game settings - > Graphics settings



2. Click to browse - >choose“LiDAR360.exe" - > Options drop-down menu, select "High Performance"




Settings
Graphics settings
PC to have your changes take effect.
Off
Variable refresh rate
When possible, get a higher frame rate and
games that don't support variable refresh ra
need to restart your game for this to take ef
Off
Graphics performance preferen
Choose between better performance or batt
You might need to restart the app for your c
Choose an app to set preference
Desktop app
Browse
ArmouryCrateKeyControl
Power saving
LiDAR360.exe
Let Windows decide
Open
GreenValley Suite > LiDAR360 > 6.0.0.0
Organize
New folder
FAQ
ToolReference
OneDrive
This PC
System (C:)
Software (D:)
Data (E:)
My Passport (F:)
Network
Name
Date modified
Type
res
submodules
translations
ClassifyStem.exe
CrashReporter.exe
LiBatch.exe
License Manager.exe
LiDAR360.exe
LiPointCloudAdjusterV2.exe
LiPowerlineExe.exe
QtWebEngineProcess.exe
File name:
LiDAR360.exe
Add
Cancel

FAQ

This chapter introduces the frequently asked questions and solutions in the use of LiDAR360. It contains Installation and License Activation FAQ, Platform FAQ, Strip Alignment FAQ, Terrain FAQ, Forestry FAQ and Power Line FAQ.

Hardware requirements and supported operating systems

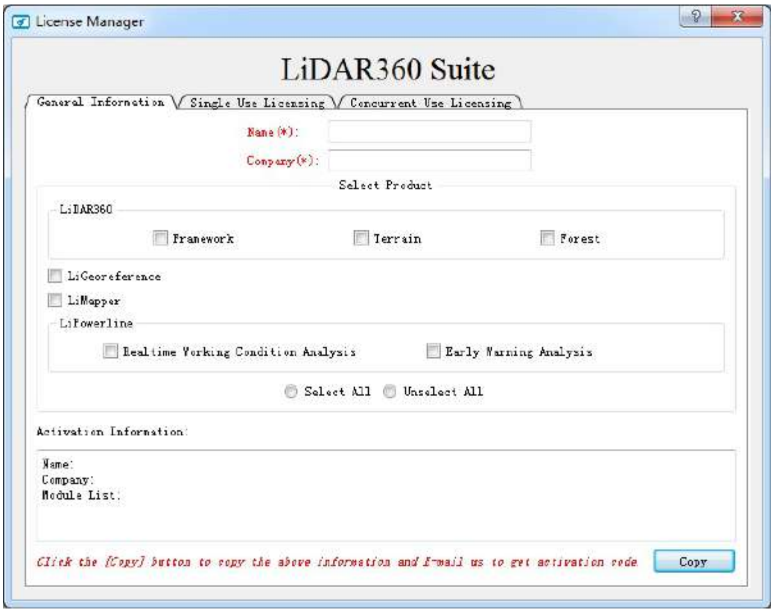
- **RAM**: at least 8G or more.
- **CPU**: Intel® Core™ i5/i7; Dual-core processor.
- **Display Adapter**: NVIDIA graphics card recommended, video memory no less than 2GB.
- **Operating Systems**: Windows 7 (64 bit), Windows 8 (64 bit), Windows 10 (64 bit), Microsoft Windows 11 (64-bit), or Windows Server 2012 or later.

Why does LiDAR360 pop up the message indicating the trial license expired after installation?

- The LiDAR360 trial license fail might be caused by the following reasons:
  - **System Time Changes**: During the trial period, some reasons that may cause system time change, resulting in expired trials.
  - **Trial Expiration**: LiDAR360 has 30-day trial period for each version. If the same version has been installed for 30 days, the trial has expired.
  - **Other Reasons**: Please contact us by email info@greenvalleyintl.com.

How to activate LiDAR360?

- **Send Activation Information**: Run the software as administrator, click ***Help > Activate License*** in the menubar, or double-click the License Manager.exe in the installation directory.



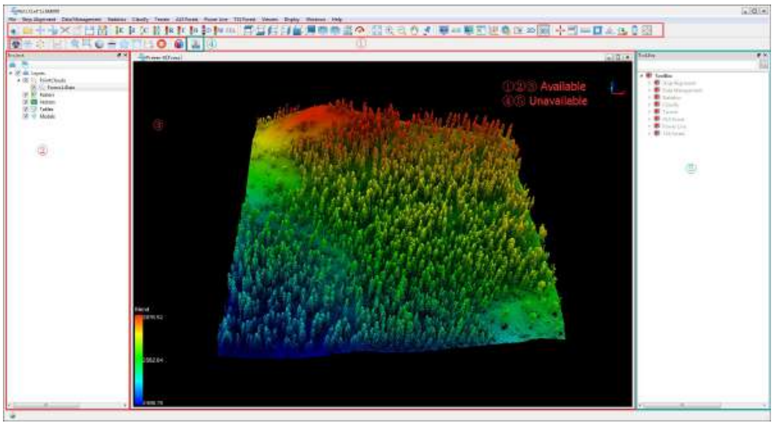
  - Fill in Name (Mandatory);
  - Fill in Company (Mandatory);
  - Select the appropriate module(s);
  - Click *Copy*;
  - Send an email with the copied content to info@greenvalleyintl.com.
- **Activate License**:
  - Copy the license code to your local drive. Pleaser refer to the License Manager for license activation.

Does LiDAR360 support Windows Server System?

- Windows Server is supported.
  - Please install and activate LiDAR360 using the administrator account. The software must be installed in the folder to which all users have access.

Can I continue to use LiDAR360 after the trial period ends?

- When the trial period ends, you can still use the LiDAR360 software as a point cloud visualization tool. The major software functions, such as the viewer, measurement tool and selection tool in the toolbar will still be available. If you still need a full license or a license for a specific module, please contact us info@greenvalleyintl.com for more information on purchasing the software.



How to view the help manual?

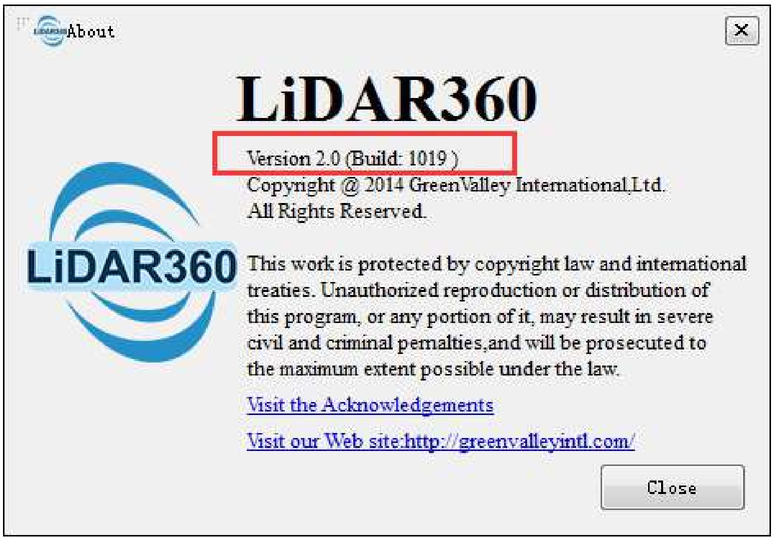
- Click the Help button in the upper right corner of the software interface.

How can I use the authorization code on another computer if it cannot be revoked due to the broken of current computer?

- Please sent an email to us through info@greenvalleyintl.com with the corresponding authroization code. We will revoke the code in the activation server for you so that it can be used in another computer.

How to check the version information of LiDAR360 software?

- You can check the version information of LiDAR360 in ***Help > About***, 2.0 is the software version and 1019 is the build date.



How to change the menu language?

- Click *Display->Language* in the menu bar to switch between English, Chinese, Japanese, French, Korean, and Spanish.

Why does the software exit abnormally?

- Based on our experience, the major two reasons of abnormal exits are the insufficient disk space or memory space in your computer and the existence of opened instance of screen word capturing software. If you meet any other scenarios that lead to abnormal exits, please contact us info@greenvalleyintl.com.

What are the supported data formats in LiDAR360?

- LiDAR360 can import the following data formats:
    - **Point Cloud**: LiData files (*.LiData custom point cloud format), LAS files (*.las,*.laz), ASCII files (*.txt, *.asc, *.neu, *.xyz, *.pts, *.csv), PLY files (*.ply), E57 files (*.e57), PCD file (*.pcd).
    - **Raster**: Image File(*.tif,*.jpg).
    - **Vector**: Vector File(*.shp).
    - **Table**: Table File(*.csv)
    - **Model**: Proprietary Model File(***.LiModel**), Proprietary TIN File***.LiTin**), OSG Model File(*.osgb, *.ive, *.desc, *.obj).

- LiDAR360 can export the following data formats:
    - **Point Cloud**: LiData files (*.LiData custom point cloud format), LAS files (*.las,*.laz), ASCII files (*.txt, *.asc, *.neu, *.xyz, *.pts, *.csv), PLY files (*.ply), E57 files (*.e57), PCD file (*.pcd).
    - **Raster**: Image File(*.tif,*.jpg).
    - **Vector**: Vector File(*.shp, *.dxf)
    - **Table**: Table File(*.csv)
    - **Model**: Proprietary Model File(***.LiModel**), Proprietary TIN File(***.LiTin**).

How to fix drag and drop in windows 8 and windows 10?

- Go to RUN, or search "regedit.exe" then run as administrator. On Registry Editor go to: HKEY_LOCAL_MACHINE\SOFTWARE\Microsoft\Windows\CurrentVersion\Policies\System, double click "EnableLUA", and change the value from 1 to 0. Restart windows, and the problem will be solved. **Reference**.

Why is the EDL visual effect not obvious if multiple data are added in the same window?

- It's recommended to show distant multiple data in separate windows with EDL effect. Or remove unwanted data in the current window.

What is the unit of point cloud data in LiDAR360?

LiDAR360 V5.2 and later supports user-defined unit settings. You can set custom units through "Display Settings" -> "Measurement Settings" in the upper right corner of the window. Currently supported units are meters, feet, decimeters, centimeters, millimeters, and inches.

How to subsample the point cloud uniformly?

- LiDAR360 provides **subsampling methods** by *Minimum Points Spacing*, *Sampling Rate*, or *Octree*. The subsampling by *Octree* is an uniform method.

- Click ***Data Management > Point Cloud Tools > Subsampling***, then select the sampling type *Octree*.

How to *Pick Rotation Center* while using the *Pick Multi-Point* tool?

- Hold the Ctrl key and pick rotation center by left-click.

Why are point cloud transformation results incorrect in LiDAR360?

- This is because different units are used in each program. In LiDAR360 degree units are used. In order to carry out a correct transformation, the units must be degrees.

Can lidar data or software be built into a GIS Platform?

- Yes. Our software can generate shapefiles (.shp) and .tiff files which can be integrated into a GIS system. In our software you can also create new polygons, lines as well as generate contour lines, DEM and other digital models.

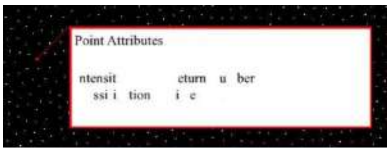
What memory(RAM) requirement is there to process large datasets in LiDAR360?

- We recommend at least 16GB of RAM for processing large datasets.

Why there are very large error after rectifying? How to solve it?

- You can view the residuals for all the homologous points pairs and remove the homologous points pairs with large residuals. After that, redo the rectifying process.

Why does the dialog box text appear incomplete or garbled when viewing single point cloud information?



- Update the discrete graphics card driver to the latest, and then run the software in high-performance graphics mode. To adjust the high-performance display mode under the Win10 operating system, please refer to **Set high-performance display mode**.

Why is DJI L1 data a straight line when opened through LiDAR360 (data display is incorrect)?

- This situation is because LiDAR360 supports the display of projected coordinates, and it is necessary to convert geographic coordinates to projected coordinates
- Data management -> point cloud format conversion -> Las conversion to LiData to select the

projection coordinates, and the converted data can be displayed correctly by re-importing it into the software.

The startup time of LiDAR360 software is too long, can I troubleshoot computer performance problems?

- Analysis of the reason for this situation is that the configured group authorization cannot be connected, and the software startup caused by the time delay is too slow. The solution: (1) Find the bit_config.xml file in the folder and delete it; (2) Where the software configures the authorized IP address of the group, find the corresponding file and delete it.

What are the supported POS formats by LiDAR360?

- ASCII format (*.pos) and binary format (*.out) are currently supported.
  - *.pos file contains GPS time, longitude and latitude, flight height, roll, pitch and heading. An example is shown as follows:

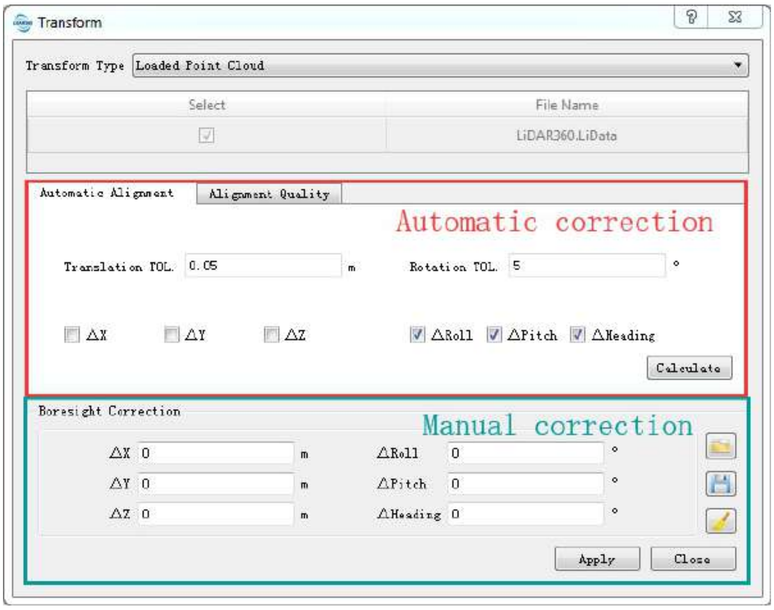
Info | Select Coordinate

GPSTime	Longitude	Latitude	Height	Roll	Pitch	Heading
380954.000	112.5311950876	26.8969520123	378.543	7.1701230000	3.0890110000	-39.4065340000
380954.008	112.5311938923	26.8969533249	378.537	7.2001860000	3.0914780000	-39.4034150000
380954.016	112.5311926975	26.8969546376	378.531	7.2368710000	3.0936380000	-39.4011190000
380954.024	112.5311915034	26.8969559507	378.525	7.2683090000	3.1015050000	-39.3975470000
380954.032	112.5311903098	26.8969572641	378.518	7.3007560000	3.1115160000	-39.3929590000
380954.040	112.5311891169	26.8969585779	378.512	7.3269790000	3.1179720000	-39.3878260000
380954.048	112.5311879247	26.8969598920	378.506	7.3525870000	3.1180460000	-39.3804020000

  - *.out file is a proprietary format of the company Applanix. Please refer to OUT File.

Is the strip alignment manual or automatic in LiDAR360?

- There are both manual and automatic methods in LiDAR360. And the results will be displayed in real-time.



What is the time reference in POS file supported by LiDAR360? Is the GPS time necessary?

- The only requirement is that the time stamps in POS file and point cloud data files have the **same reference** (e.g., GPS Time or UTC Time). If they are not the same, you have to convert one to the other. Generally, time in point cloud counts in SOW(seconds of a week) or SOD(seconds of a day).

Sometimes, the point clouds after strip alignment are partially aligned, but partially not, and in some area they are even distorted. Why?

- This may occur if the POS file is of low accuracy. The principle of the *Strip Alignment* module is to correct **boresight errors** between laser scanner system and GPS/INS system. The precondition is that the POS accuracy for the acquired data meets the standard specifications. Otherwise the resulting point clouds are partially distorted. Boresight correction alone doesn't guarantee the aligned results with high accuracy.

Why there is no output when I try to clip the point cloud data with the POS file?

- Please check the GPS starting and ending time of both the POS file and the point cloud data. And see if there are any overlap between the GPS time of two files.

The data collected by the DJI L1 device, the track time reference is inconsistent with the point cloud GPS time reference, such as the track time in weeks and seconds, and the point cloud time information when the GPS is adjusted? (The range of GPS weeks and seconds is 0~604800, which can be used to distinguish GPS weeks and seconds and adjust GPS time)?

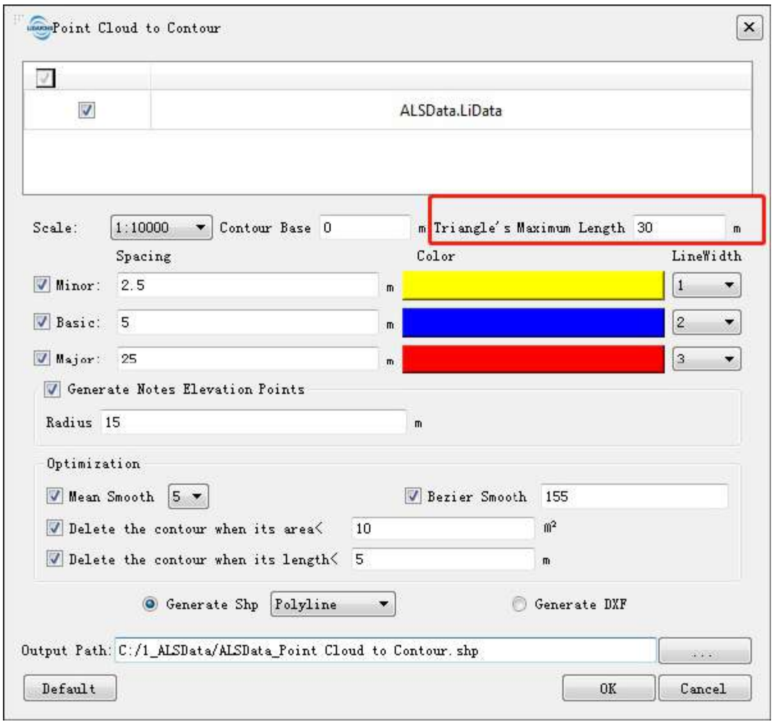
- The GPS time conversion process is as follows: Preprocessing -> Air Belt Adjustment, input the trajectory file, the trajectory file of the DJI L1 device is in .out format, after inputting the trajectory file, the GPS time conversion and coordinate system selection dialog box will pop up, check the box. Select "GPS time conversion", select the source GPS time, and "Convert to", select the data collection date for the acquisition date, and convert the trajectory time to the point cloud time.

The actually scanned LiDAR point density is much larger than the DEM point density requirement of the production. Can data be subsampled?

- If the actually scanned LiDAR point density is much greater than the production demand, you can use the **Subsampling** tool to subsample the point clouds. This function provides three types of subsampling methods: "Minimum Points Spacing", "Sampling Rate" and "Octree".

Some of the contour lines generated by the function "Point Cloud to Contour" are broken. Why?

- The broken contour lines may be caused by the lack of ground points. The contours can be contiguous by increasing the parameter "Triangle's Maximum Length" on the interface. For example, set 30 meters up to 50 meters or an even larger value.



During DEM processing, the result of this process has .tiff format. Can I convert this into 3D data?

- You can convert your .tiff DEM into our custom 3D model format (LiModel). Go to ***Data Management > Conversion > Convert TIFF to LiModel(../ToolReference/DataManagement/Conversion/ConvertImagetoLiModel.md)***.

How to check and improve the quality of DEM generated by LiDAR360?

```
- You can view the DEM effect through Terrain->Section Editing->DEM Preview. For unqualified DEMs, you can op
timize the DEM through terrain patching and inserting breaklines, regenerate the DEM, and check the accuracy
of the generated DEM through Terrain->DEM Accuracy Check. Check and output report.
```

How do you generate a smoother contour for a DEM?

- Perform a smoothing operation using smooth height tool. Save the data. Then Convert the LiModel back to .tiff format, ***Data Management > Conversion > Convert LiModel to TIFF(../ToolReference/DataManagement/Conversion/ConvertLiModeltoImage.md)***. From here perform a contour operation ***Terrain > Raster to Contour(../ToolReference/Terrain/RastertoContour.md)***.

How to choose the interpolation method for generating DSM?

- It is recommended to choose IDW interpolation for forestry area, and TIN interpolation (Spike Free TIN) for urban area.

```
   @inproceedings{
        author={Zhao X Q, Guo Q H, Su Y J and Xue B L},
        title={Improved progressive TIN densification filtering algorithm for airborne LiDAR data in forested
 areas},
        booktitle={ISPRS Journal of Photogrammetry and Remote Sensing,117:79-91},
        year={2016}
   }
```

Can LiDAR360 achieve specific area classification, such as lake, river area?

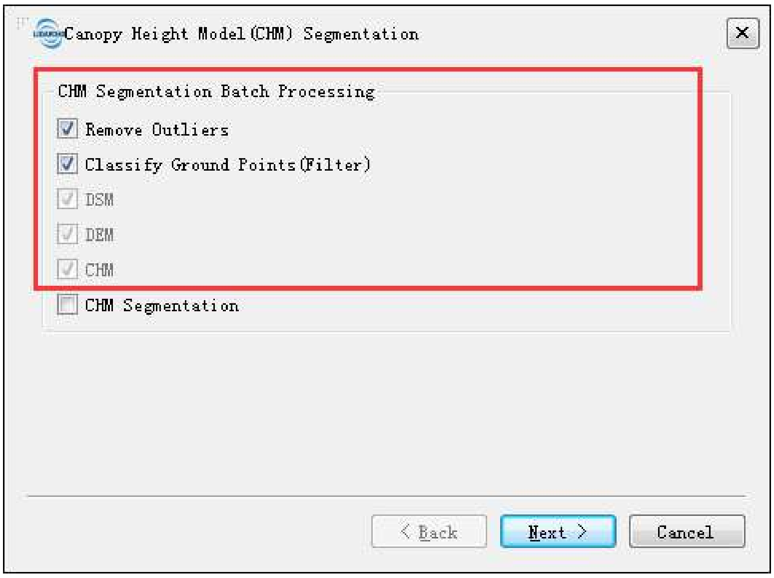
- You can go through Classification -> Section Editing -> Start Editing -> Generate TIN -> Add Breaklines -> Leveling Areas (Lakes)/Double Breaklines (Rivers) -> Set Initial Category, Target Category, such as Water Body -> Add, complete lake area classification

Can LiDAR360 achieve uniform assignment of elevations for point clouds within a given vector range?

- You can select the area to be unified elevation through Classification->Section Editing->Start Editing->Generate TIN->Add Breakline->Closed Area Breakline->Polygon, you can specify the elevation, or you can pick the triangle mesh patch with the mouse point uniform elevation

Can LiDAR360 generate CHM in batches?

- LiDAR360 can generate CHM in batches (***ALS Forest* > *Batch Process* > *Canopy Height Model(CHM) Segmentation***).



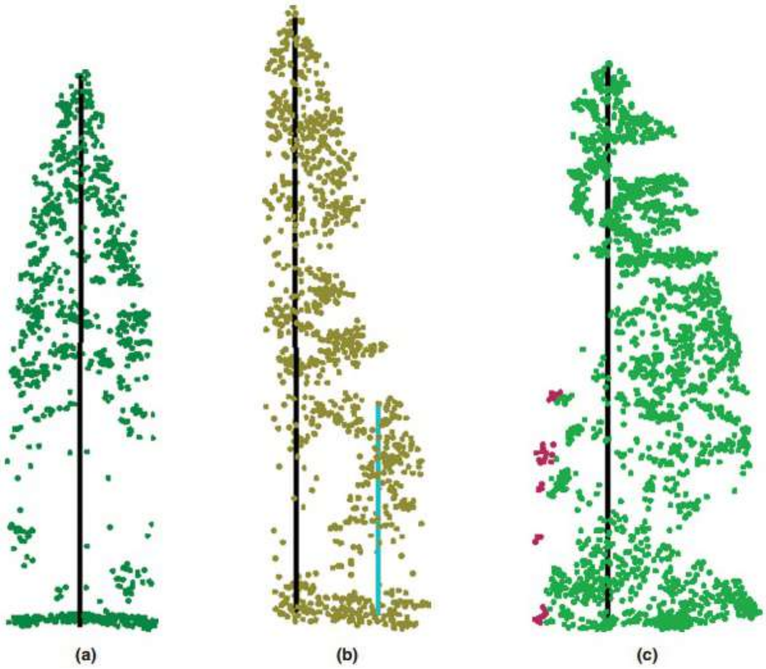
How to evaluate the accuracy of individual tree segmentation results?

- For evaluation of individual tree segmentation accuracy, please refer to Li et al's article on (Li et al., 2012). By comparing with field measurement, the number of correctly segmented trees, the number of falsely segmented trees, and the number of missed trees are calculated according to the following formulae: recall (r), precision (p), and F-score (F). Recall indicates the tree detection rate, precision indicates the correctness of the detected trees, and F-score is the overall accuracy taking both commission and omission errors into consideration. The values of r, p and F vary from 0 to 1.

$$\mathrm{r} = \frac{TP}{TP + FN}$$

$$\mathrm{p} = \frac{TP}{TP + FP}$$

$$\mathrm{F} = 2 \times \frac{r \times p}{r + p}$$



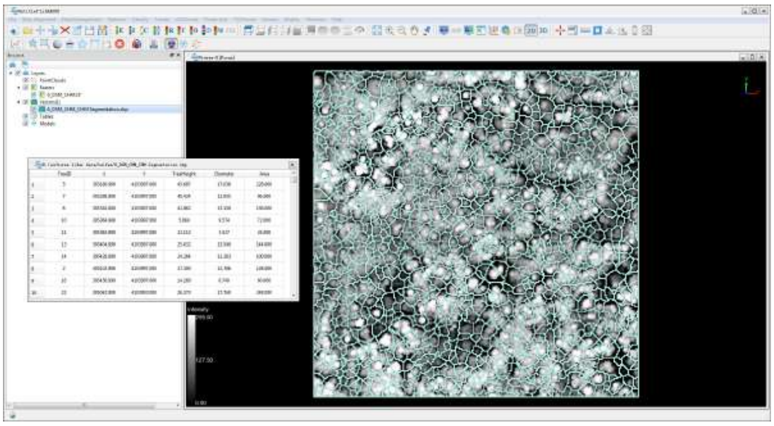
（a）Correctly detected tree (True Positive, TP)（b）undetected tree (False Negative, FN) （c）falsely detected tree(False Positive, FP)

What parameters affect the CHM segmentation accuracy and how should these parameters be set?

- The accuracy of **CHM segmentation** are affected by CHM resolution and Gaussian smoothing factor.
  - CHM is the difference between DSM and DEM. The CHM resolution is determined by the resolution of DSM and DEM. Generally speaking, this value should not exceed one-third of the crown width, and the range can be set to 0.3-1m. Usually, the resolution of 0.5-0.6m can get a higher segmentation accuracy.
  - Sigma is the Gaussian smoothing factor (default value is "1"). The greater the value is, the smoother the results are. The degree of smoothness can affect the number of trees being segmented. In the case of under-segmentation, it is recommended to reduce this value (e.g. 0.5); and in the case of over-segmentation, it is recommended to increase the value (e.g. 1.5).
- Moreover, beside the algorithm parameters, the CHM segmentation result can also be largely influenced by tree density and tree species. If the algorithm does not work well in s certain study area, users can try to use other segmentation algorithms to get the best segmentation result.

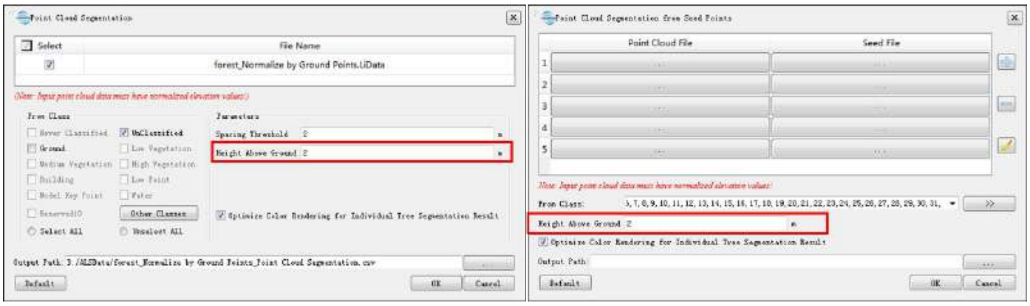
How to remove some results with small crown area after running CHM segmentation?

- After running CHM segmentation, a shp file containing the tree boundary is obtained. The attribute table includes the ID, location, height, crown diameter and crown area of each tree. The data can be imported into a third-party software (e.g. ArcGIS), and you can remove the segmentation results with a small crown area according to the crown area attribute.



How to make point cloud data below 2m be participated in segmentation?

- The parameter "Height Above Ground" on the **Point Cloud Segmentation** and **Point Cloud Segmentation from Seed Points** interfaces indicate that points below the value are not considered as part of the tree and will be ignored during the segmentation process. The default value is 2m. If you need to make the point cloud below 2m participate in the segmentation, the value can be reduced appropriately.



Which kind of segmentation method for coniferous and broad-leaved mixed forest data can be used to obtain higher accuracy?

- For coniferous and broad-leaved mixed forest data, **CHM Segmentation** is recommended.

The growth of trees in the area is not the same, how to achieve higher segmentation accuracy?

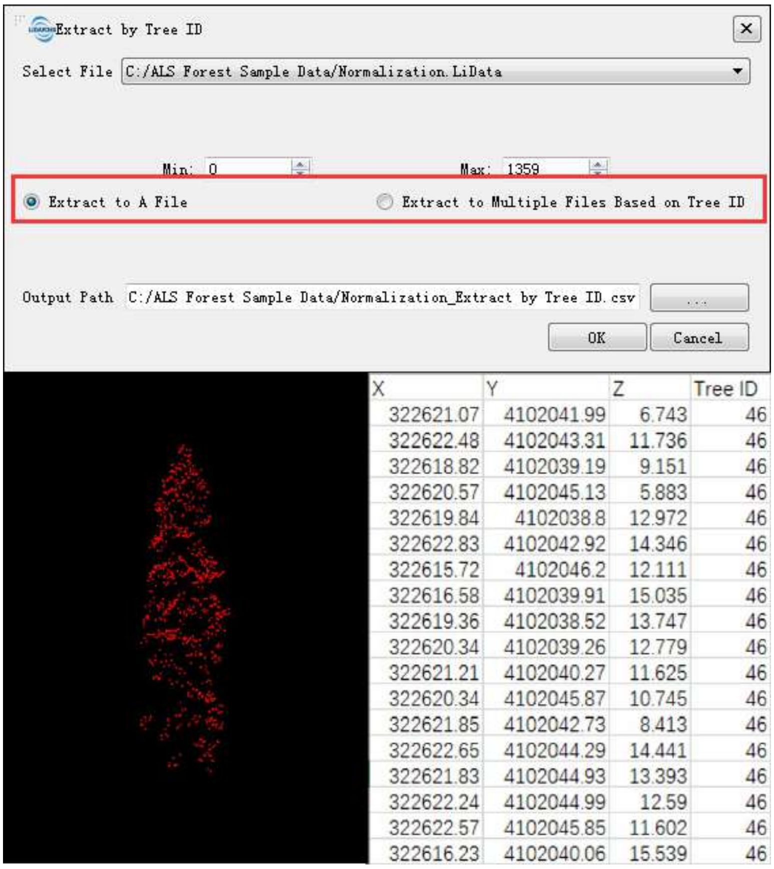
- It is recommended to clip the point cloud according to the growth as different data by **Select Tools** or **Clip Tools** and handle them separately.

If the tree canopy is covered in weeds, can tree identification still be performed in LiDAR360?

- Yes, it can.

How to export individual tree segmentation results to third-party software for analysis?

- Click ***ALS Forest* > *Extract by Tree ID***, the segmented point cloud can be exported to CSV file for subsequent analysis using other software. LiDAR360 supports exporting each tree as a separate CSV file or exporting all points as one file. The exported CSV file is shown in the figure below, which contains X, Y, Z coordinates and tree ID information.



X	Y	Z	Tree ID
322621.07	4102041.99	6.743	46
322622.48	4102043.31	11.736	46
322618.82	4102039.19	9.151	46
322620.57	4102045.13	5.883	46
322619.84	4102038.8	12.972	46
322622.83	4102042.92	14.346	46
322615.72	4102046.2	12.111	46
322616.58	4102039.91	15.035	46
322619.36	4102038.52	13.747	46
322620.34	4102039.26	12.779	46
322621.21	4102040.27	11.625	46
322620.34	4102045.87	10.745	46
322621.85	4102042.73	8.413	46
322622.65	4102044.29	14.441	46
322621.83	4102044.93	13.393	46
322622.24	4102044.99	12.59	46
322622.57	4102045.85	11.602	46
322616.23	4102040.06	15.539	46

How many sample data files are required for regression analysis?

- The number of sample data used for regression analysis is not clearly defined. In general, under the premise of ensuring the accuracy of plot location and measurement, the greater the number of sample data is, the higher the accuracy of regression analysis can achieve. Sample sites should be randomly selected and representative to cover different forest types within the study area. A sample size less than 30 is referred as a small sample, and a sample size of 30 or greater is referred as a large sample. To ensure regression analysis accuracy, the recommended sample size should be greater than or equal to 30 (you should also consider the size and complexity of the study area).

How to choose the independent variables involved in the regression analysis?

- The elevation percentiles obtained from LiDAR data are generally the independent variables for regression analysis. However, specific elevation percentiles are not all the same for different study areas.

Can trained regression models be used for other data?

- Yes. Click ***ALS Forest > Regression Analysis > Run Existing Regression Model***. Available regression models (**Linear Regression**, **Support Vector Machine**, **Fast Artificial Neural Network** and **Random Forest Regression**) can be used to estimate forest metrics.

Can I export the independent variables (e.g. elevation percentiles) generated by LiDAR360 into third-party software (e.g. SPSS, R) for regression analysis?

- Yes. Independent variables provided by LiDAR360 including **Elevation Percentile**, **Elevation Density**, **Intensity Percentile**, **Leaf Area Index**, **Canopy Cover** and **Gap Fraction**. Among them, Elevation Percentile, Elevation Density and Intensity Percentile are in CSV formats, and they can be imported into third-party software such as SPSS directly. Leaf Area Index, Canopy Cover, and Gap Fraction are in TIF formats that can be converted to text format by ArcGIS and imported into third-party software.

```
@inproceedings{
    author={ Li W K, Guo Q H, Jakubowski M K and Kelly M},
```

```
    title={A new method for segmentation individual trees from the LiDAR point cloud},
    booktitle={ Photogrammetric Engineering and Remote Sensing,78(1):75-84},
    year={2012}
}
```

What if the TLS Stem Extraction runs in a flash or there is no change in the classification result?

-The function of TLS Stem Extraction is based on deep learning and needs to use GPU. Make sure the latest driver has been installed on your computer.

-For Windows7 System, the driver need to be installed includes：

- 451.67-desktop-win8-win7-64-international-whql.exe
- GeForce_Experience_v3.20.3.63.exe
- Please select the appropriate driver according to your own operating system for your Nvidia GPU and installation.
- If you still cannot run this function after installing the latest driver, please check whether the required system patches are installed. For example, Windows 7 needs to install the following patches:
- Windows6.1-KB3068708-x64.msu
- Windows6.1-KB3080148-x64.msu
- Please make sure all the required patches are correctly installed on your computer.

After single tree classification, which column in the result table is the height of the tree? What if not?

- After single trees are divided, there is no tree elevation value in the result table, only X and Y coordinate values. To get the elevation attribute of each tree in the table, you can: first use the DEM tool to generate the DEM of the plot, and then use the "Foundation Forestry -> Tree Attribute -> Extended Single Tree Attribute" tool to make the DEM file according to the single tree attribute. The coordinate position of the tree divided by the wood, and the elevation value can be assigned to the CSV table.

Why is the tree flat when the scanned data is viewed in the cross-section? Is there any way to improve it (the scan is scanned according to the normal trajectory), in this case, it is easy to have problems fitting the DBH through the software?

- Considering the fact that the tree trunk is flat in the tree scan results, it may be because the sapling plants are relatively small. For large-scale woodland data, it is recommended that the scanning trajectory be more curved; during data calculation, in LiBackPack-BP In the software, first uncheck the smoothing option, and the data will not be smoothed.

There is a big difference between the results of single-tree segmentation and the actual number of single-trees. It is ruled out that the DBH fitting is inaccurate and non-parameter setting problems?

- Exclude the DBH fitting accuracy and parameter setting problems, then the cause of this problem may be the inconsistency between the original data unit and the LiDAR360 parameter setting unit. For example, the original data unit is feet, while the LiDAR360 data processing unit is unified as meters. Unit conversion is required before data processing.
- Conversion method: If the original data is in the las format, directly convert "las to LiData", and select "feet" as the source unit; if the original data format is LiData, first "convert to las" and then perform the above operations.

LiDAR360 Version Release Notes

v8.0 - 8/15/2024

- **Preprocessing**：
    - Add 3D Control Point Report:
        - Add automatically targets detection and control points matching.
        - Add defining transformation models in control point reports.
    - Add DJI L1 / L2 Reconstruction, supporting UAV Processing workflow.
    - Optimize Trajectory Adjustment, supporting massive point cloud and image combined adjustment, target detection from point cloud / image and matching control point to improve data precision.
    - Optimize Data Registration:
        - Add Define UCS to change point cloud coordinate by local cartesian coordinates.
        - Add support for importing control points as reference data option for point pairs registration.
- **Data Management**：
    - Add Adjust Point Cloud Color.
    - Add Unit Conversion, supporting all platform formats.
    - Add Convert ASCII to Vector / Convert Vector to ASCII.
    - Add support for defining projection / reprojecting for raster, vector and ascii file.
    - Add support for selecting target coordinate from layers.
    - Add Convert LiTIN / LiModel to OBJ / OSGB.
    - Add 3D affine parameters in Transformation Calculation.
    - Optimize Normalization to mitigate object cracking caused by steep slope.
    - Optimize Segment by Attribute, supporting using additional attributes for statistic.
    - Optimize Convert to TIFF, supporting fixing holes.
    - Optimize Raster Subdivision,supporting the generation of tile boundary lines.
    - Optimize Extract by Return, supporting 1~15 return number.
- **Classification**：
    - Add support for road classification and outline extraction.
    - Add support for classify roof and wall in Classify Buildings.
    - Optimize Classify by Deep Learning, supporting new Scenes.
    - Optimize Classify Ground Points, supporting Concurrency processing.
    - Optimize Classify by Custom Deep Learning:
        - Add support for setting a trained model as a new function.
        - Add support for image labeling, training and inferencing.
        - Add Classify Tunnel by deep learning trained model.
        - Optimize point cloud deep learning model.
    - Optimize Classification Editor:
        - Add automatically saving setting.
        - Add semi-auto classify by SAM.
        - Add seed point selection for classify by select area operation.

        - Optimize Simulate Ground Points tool, supporting random distribution.
        - Optimize memory usage.
- **Forestry**：
    - Add Forests Stands Delineating.
    - Add built-in tree model library for Generate Tree Model, supporting 70 Tree Species.
    - Optimize Individual Tree Attributes Calculation:
        - Add support for trunk curvature calculation.
        - Add support for trunk section and trunk volume calculation.
        - Add support for forked DBH extraction and management.
        - Optimize tree height and DBH calculation.
        - Optimize forestry setting option, supporting individual tree attributes setting.
        - Optimize statistic individual tree attributes, supporting one-click extraction of multiple attributes.
    - Add Stand Analysis and report, supporting Thinning Analysis and Standing Tree Volume Analysis.
    - Add View Azimuth and Distance.
    - Add support for exporting individual tree attributes as table / vector file.
    - Add support for point cloud display by tree attributes by treedb linked.
    - Add CHM Pits Filling to fix holes in coniferous forests CHM.
    - Optimize Individual Tree Report, adding tree number, area information, pano image and supporting pdf format.
    - Optimize Regression Analysis, adding scatter plot result.
    - Optimize Trunk Based Segmentation to improve result accuracy.
    - Optimize Extract by Tree ID , supporting extract individual tree to LiData / Las format.
- **Terrain**：
    - Add REM.
    - Add Flood Analysis.
    - Add View Analysis.
    - Add Sky View Factor.
    - Add Annual Insolation.
    - Add Solar Radiation.
    - Optimize LiTIN / LiModel Editor, supporting undo and redo operation for each step.
    - Optimize LiModel Editor, supporting adding various breaklines.
    - Optimize Contour Editor, supporting topology checking and select area filtering.
    - Optimize DOM, supporting color correction.
    - Optimize generate contour functions, supporting more simplify and smooth methods.
    - Optimize DEM, supporting add point cloud name to results in standard model, expanding scale to 1:5000.
    - Optimize Section Analysis, supporting quality inspection and more analysis result.
    - Optimize Generate Elevation Annotation, supporting diamond sampling.
    - Optimize Change Detection, supporting missing data area annotating.
- **Mine**：
    - Add Extract Tunnel Centerline.
    - Add various section analysis results for tunnel.
- **Building**：
    - Add support for attributes editor in Building Editor.
    - Add support for building attributes calculation, extension, filtering and so on.

- Optimize building attributes calculation, supporting calculating rmse between point cloud and model.
- Optimize operation of Matched Photos Editor.
- Optimize LiBIM file rendering to reduce lag when handling large data.

- **New Photo Module**
  - Add New Project, support for infrared and multispectral data.
  - Add Align for aerial triangulation processing, support GPU acceleration.
  - Add Filter Tie Points, supporting automatic / semi-atuo selection.
  - Add DOM Editor.
  - Add Ground Control Points, supporting control points picking and automatic detection.
  - Add Camera Calibration.
  - Add Undistort.
  - Add Defogging.
  - Add support for image project tie points rendering, supporting display by projection error and so on.
  - Optimize Align to Point Cloud, supporting improve accuracy by camera calibration, control points picking and automatic detection.
- **New Distributed Module**
  - Add various functions for distributed computing.
  - Add support for large data and multiple data sets.
  - Add support for dynamic configuration of compute node priorities.
  - Add support for multi-instance computation, monitoring, and management of compute nodes.
  - Add support for merging results.
- **Vector**
  - Add Extract Feature by SAM.
  - Add Trace Toe Tool.
  - Add Attach to Ground.
  - Add Respace Geometry.
  - Add Orthographic Draw Line.
  - Add Round Corners.
  - Add Trim Polyline.
  - Add Convert to Polyline / Polygon.
  - Add support for exporting label me format for image deep learning.
  - Optimize and merge some tools.
  - Optimize drawing curve tools, supporting set parameters to generate curve.
  - Optimize vector layer management, supporting layers modify setting.
  - Optimize Copy Parallel, supporting copy distance setting.
  - Optimize vector setting, supporting symbology and snapping setting.
  - Optimize vector editor operation, adding operation prompts.
- **Other**
  - Add unit setting, supporting processing and generating result in different unit.
  - Add measurement setting, supporting snapping different data type to improve measurement accuracy.
  - Add Line Above / Below, Plane Above / Below / Inside Selection.
  - Optimize Volume Measurement and report, supporting point cloud image, adding basic

information and image setting.
    - Optimize Profile View, supporting setting linked viewer setting.
    - Optimize vector file display, supporting show or hide control by layer.
    - Optimize project directory tree:
        - Add image project display control, supporting adjusting image size, exporting and so on.
        - Optimize table / vector(shp / gpkg) / LiBIM attributes table management, supporting attributes calculation.
        - Optimize table file display, supporting dynamic adjusting label size and so on.
    - Optimize option setting, supporting cpu selection, unit settings, more forestry settings.
    - Optimize Batch Processing, supporting more functions.

v7.0 - 6/10/2022

- **Preprocessing**:
    - Add mirror angle correction in trajectory adjustment for livox laser
    - Add hybrid adjustment, improve point cloud accuracy
    - Optimization data registration process, supports registration of point clouds and models
    - Add auto split trajectory for boresight
    - Add Z-value recording in projection coordinates when trajectory solving
- **Data Management**:
    - Add point cloud color brightness and contrast control tool
    - Add smoothing line tool, provide multiple smoothing methods
    - Add batch point clouds to PLY tool
    - Add building model export to DAE
    - Add smoothing meshes tool
- **Classification**:
    - Add deep learning urban/Rural scene classification
    - Support multiple segment classification for classify by height above ground
- **Terrain**:
    - Add construction progress analysis tool
    - Add contour consistency check tool
    - Add generate elevation annotation points on contour lines
    - Support external aerial projects for generate DOM
    - Optimize TIN to DEM tool to solve the problem of external interpolation outside boundary
    - Optimize dimming of mountain shadows
    - Section analysis
        - Support tunnel section parameterized design
        - Support generating section based on slope
        - Support distance display for section comparison
- **Forestry**:
    - Add biomass estimation and biomass model management
    - Add deep learning-based individual tree segmentation to improve tree detection
    - Add forestry attribute custom format to simplify attribute usage

    - CHM segmentation adds east-west and north-south crown width parameter extraction
- **Mine**:
    - Add inter-ramp compliance tool
    - Optimize toes and crests extraction, support auto-separating of toes and crests
    - Add mesh editor tools, allow automatic and semi-automatic fill holes, hide selection, model decimate, and smooth tools
    - Optimize mesh surface reconstruction to improve speed and effectiveness
    - Mesh editor
        - Support both front and back mesh display, and overlay display of points, lines, and surfaces
        - Support mesh selection tools such as lasso, polygon, and rectangle
        - Support mesh back faces selection and invert selection
        - Support mesh visible and through selection
        - Support selection expand and shrink, component selection, and flood selection
        - Support hide selection, select all, and unselect all
        - Support filling all holes and filling single holes
        - Support bridge tool
        - Support mesh smooth
        - Support mesh decimate
- **3D Building**:
    - Add extracting building footprint from image
    - Add building model registration with image
    - Add projecting textures from photos
    - Add manual image texture mapping, material texture mapping, and material library management
    - Add building attribute management, including adding, deleting, and editing attributes
    - Add attributes calculate, including volume, height, and house shape
    - Improve building model editing
        - Add split building
        - Add fit patch
        - Add extrude building
        - Building split supports polyline segmentation
        - Add file management after deletion
- **UAV Processing**:
    - Supports one-click processing of trajectory adjustment, pos process, cut point cloud by trajectory, cut overlap, smooth points, subsampling, remove outliers, classify ground points, classify by deeep learning
- **Vector Editor**:
    - Add merge polygon
    - Support gpkg data format
    - Add vector text labeling
    - Add intersect polylines
    - Add copy parallel
    - Add extract building footprint from Mesh
    - Supports large data LOD drawing, improving editing efficiency
    - Support show vertices settings

- **Platform**:
  - Add vector display line width setting
  - The directory tree adds trajectory and imageproject management
  - Support selection of attribute fields to display, data saving as new, attribute editing, and attribute filtering
  - The selection and clipping tool supports saving remaining data
  - Volume measurement reporting support and saving the current scene graph
  - The rolling screen tool supports multiple data types such as point clouds, models, vectors, and tables
  - Mesh editing, LiBIM, LiModel, LiTIN, and other editing supports shortcut key settings
  - Support gradient background color setting for rendering scene

v6.0 - 8/10/2022

- **New Modules and Features**:
  - Added voxel down-sampling when loading data is converted to LiData
  - Added strip adjustment function for multiple sorties
  - Added conversion based elevation projector plane coordinate
  - Added support for KML vector format
  - Vector Drawing
    - Added arc drawing
    - Added the drawing of one point circle, two point circle and three point circle
  - Added building modeling module
    - Added automatic building of LOD2 level model based on base map
    - Added semi-automatic building of LOD2 level model
    - Added building model editing tool
      - Added vertex moving tool
      - Added vertex deleting tool
      - Added vertex adding tool
      - Added line removal tool
      - Added line alignment tool
    - Added building model plane editing tool
      - Added sheet merge tool
      - Added sheet removal tool
      - Added the face intersection line tool
      - Added the tool for add faces
    - Added building ground elevation adjustment
    - Support the output of building models in obj, fbx, cityjson, and other formats
    - Support LiBuilding data consolidation
  - Forestry module
    - Added individual tree segmentation and matching tools
    - Added tree 3D modeling
    - Added export of individual tree report
    - Added tree trunk extraction tool based on TLS data
    - Added tool for extracting crown base height of individual tree

        - Added TLS porosity calculation
        - Added TLS cover degree calculation
        - Added TLS individual tree editor
            - Added semi-automatic support removal tool
            - Added individual tree attribute filter check
        - Added airborne forestry individual tree point cloud editing tool
            - Added semi-automatic individual tree point cloud segmentation
            - Added attribute-based filter check
            - Added the operations for creating, deleting, and merging individual tree point cloud
    - Terrain module
        - Added DOM production tool based on DEM\DSM
    - Added Custom Deep Learning Classification Tools
        - Support customized process for training and reasoning tasks
        - Support model management
        - Support algorithm update
- **Enhancements**:
    - Shortcut keys are supported for clipping tool selection
    - Point cloud fitting plane can be used as the measurement datum for volume measurement
    - Split window rendering mode is supported for point cloud rendering
    - The efficiency of point cloud conversion to LiData data is improved by 30%
    - The efficiency of common tools such as smoothing, denoising and resampling are optimized, and the parallel mode is improved by 50%-120%
    - Model data architectural semantic rendering is supported
    - Project Management Window
        - Import of *.ligeo and* .mmprj project data is supported
    - Clipping tools
        - “Tile by polygon” supports buffer settings
        - Single point drawing of rectangular areas is supported for rectangular clipping
    - Control of object move tool is supported for manual translation and rotation
    - Classification
        - The classification of tower and power line is expanded through deep learning of relevant classification
    - Terrain
        - Whether the section is controlled by interpolation is supported for road section analysis tool

v5.4 - 07/25/2022

- **New Modules and Features**:
    - Added the function of extracting by additional attributes
    - Added the function of segment by attribute
    - Volume Measurement
        - Added the triangulation method for volume measurement for more accurate calculation
        - Added volume measurement of solid model
    - Projection and coordinate conversion
        - Added geoid option

        - Support independent selection of horizontal projection and vertical projection
        - Added four parameters and geoid joint conversion
        - Added geoid surface model fitting
        - Added model data conversion options to the manual rotation and translation and the manual registration tools
    - Added the function of classification by additional attributes
    - Added model smoothing function
    - Vector Editing
        - Added point break, line break, merge, delete functions
        - Added Bspline and BezierSpline drawing
        - Added Bspline and BezierSpline shaping
        - Added 2D and 3D vector drawing elevation mode settings
        - Added polyline smoothing tool
        - Added highest and lowest points snapping
    - Added the function of manual tree species marker
    - Added individual tree attribute settings and the support of selection of calculation methods such as tree position and crown width
    - Vector editing
        - Support extracting toes and crests
        - Support volume change analysis
- **Enhancements**:
    - Added gallery mode to the common editing tools in the interface, which supports floating display of tools
    - The classification class supports 0-255 class selection
    - Support 2D and 3D data linkage
    - Custom tab settings window supports tool searching
    - Data rendering
        - Added raster, vector, model, and table data to display by window
        - Display by window supports point cloud, raster, vector, model, table data filtering by value range
        - Added raster data hillshade feature rendering
        - Support data to save the last right-click attribute rendering settings
    - Multi-point selection supports coordinate positioning
    - Strip alignment supports the use of selected categories for boresight error calculation
    - Trajectory adjustment supports the import of multiple trajectories
    - Control point report supports import label display and export report format setting
    - Point cloud subdivision functions support preview
    - Boundary extraction supports extraction to the same vector file
    - Classification editing
        - Added the semi-automatic classification of buildings and noise points
        - Support 3D editing mode
        - Support joint image editing
        - Support block editing by range, scale and external import range
        - Improved the function of ground point simulation
    - TLS seed point editing
        - Support profile and main window position displaying

        - Added shortcut keys for the main window height filter function
        - Profile view window supports range line display
        - Optimized DBH batch extraction function to improve tree recognition accuracy
    - Clear TreeID tool supports height filtering
    - Optimized the display of geostatistical rose diagram v5.3 - 03/20/2022

- **New Modules and Features**:
    - Added building and vegetation categories based on deep learning
    - Added DEM Edge Alignment
    - Added DEM accuracy assessment
    - Added CSF filter
- **Enhancements**:
    - Section Viewing and Editing
        - Added vertical stretch display
    - Section classification editing
        - Added ground point simulation
        - Added DEM preview
        - Support frame selection of multiple blocks to construct and edit block data
    - Color rendering
        - Added real-time contour rendering
        - Improved the convenience of rendering point cloud attributes
    - Ground point classification supports terrain scene selection
    - Support coordinate and projection transformation during point cloud data import
    - Manual translation and rotation tools support setting the rotation center
    - Improved the efficiency of generating contour lines by 2-10 times
    - Optimized the efficiency of functions such as denoising, smoothing, ground point classification, noise filtering, etc. Support multi-threading and memory resource settings in the options
    - Optimized the image rendering operation experience and improve the fluency
    - Solved the problem of shortcut keys customization conflict, support unified control

v5.2 - 11/19/2021

- **New Modules and Features**:
    - Breakline
        - Point cloud profile editor supports breakline drawing
        - Improved LiTIN editor breakline drawing
        - Support adding breaklines to DEM
        - Support adding breaklines to contour lines
    - Added extract water area tool
    - Support point clouds in PCD format
- **Enhancements**:
    - Section
        - Support create section by adding model data
        - Support adding section at any position

        - Optimized create section by point cloud and fix the missing section caused by the missing point cloud
        - Optimized export of section to meet production requirements
    - Optimized Projection and Coordinate Conversion, add support for storage of user-defined conversion models, parameters can be created and applied directly
    - The measurement tool supports unit setting
    - Optimized the efficiency of the control point report function with an average increase of 40%
    - Generated CHM function to support big data processing
    - Strip alignment supports SBET format trajectory data
    - Contour function supports class selection
    - Roaming tool supports pause during roaming
    - Added trial application on start page
    - Point cloud data GPS time conversion supports selecting date for conversion
    - Mutual conversion between shp and kml supports geographic coordinate system
    - Subdivision tool supports buffer setting
    - Support report custom settings
    - Optimized the bit rate of the screen recording tool to provide a default value based on the system resolution

v5.1 - 07/29/2021

- **New Modules and Features**:
    - Added mine module
        - Extract tunnel points
        - Compute normal vectors
        - Triangulation modeling
        - Poisson modeling
    - Vector editing
        - Support 3D vector drawing
        - Added tools for shaping elements
        - Added split polygon tool
        - Added semi-automatic extraction of building outlines
        - Added semi-automatic extraction of road shoulders
        - Added intersection drawing line
        - Added section auxiliary drawing
        - Added associated window to assist drawing
        - Added selection file editing
        - Added temporary draft editing
        - Support 3D capture
        - Support 3D selection
        - Support shortcut keys setting
        - Support rollback and redo
    - Added the hover mode, support the use of hover mode under measurement point selection
    - Added cutting by straight line
    - Added dividing by polygon

- **Enhancements**:
  - Profile editing and classifying
    - "Select tool" supports category settings to be individually set and recorded
  - "Select cutting" supports cutting of the triangulation model
  - Individual tree point cloud editing supports directory tree control display and hidden
  - "Cut by polygon" supports inner and outer cutting options
  - Manual translation and rotation support XIYZ component progress control
  - Directory tree
    - Support right-click to open the folder
    - Vector data supports dxf format

v5.0 - 01/29/2021

- **New Modules and Features**:

  - Added geology module
    - Added flow direction function
    - Added flow direction function cumulant
    - Added fill function
    - Added slope/aspect query function
  - Added data IO API for software development
    - Support creating LiData point cloud file
    - Support reading LiData point cloud file
    - Support modifying LiData point cloud file
  - Added powerline module
    - Added deep learning of powerline classification
    - Added danger point detection function
    - Added customization of the danger point detection
    - Support danger point visualization
  - Added point cloud smoothing function
  - Added trunk extraction function
  - Added elevation annotation points generation based on contour lines function
    - Added new format transform tools
      - shp to kml
      - kml to shp
  - Support background grid display in viewers
  - Support extracting point cloud by TreeID
  - Support clear individual tree record and segmentation by TreeID
  - Added calculation of forest metrics
- **Enhancements**:
  - UI
    - Brand new Ribbon user interface style
    - Support free dragging of child windows
    - Support customization of UI pages
    - Support automatic hiding of docked window
    - Support one-click industry application processing

- Strip alignment
  - Added strip adjustment function
  - Support clipping trajectory by the quality and selecting with a brush on a trajectory
- Directory tree
  - More comprehensive point cloud information display
  - Support selecting the version of the LAS file when right-clicking to export
  - Support right-clicking to add additional attributes of point cloud
  - Support right-clicking to rename files
  - Support setting the point size of table data
- Expanded the projection library to support more projections and geoid surface
- Optimized PCV efficiency to support more point cloud calculations for PCV
- Optimized ALS automatic individual tree segmentation
- Improved point cloud attribute expansion of an individual tree to support slope and altitude information
- Optimized individual tree seed point editing
  - Improved method of profile selecting
  - Improved method of adding and deleting seed points
  - Added viewpoint position and direction for easy positioning of trees
  - Real-time update of attribute table corresponding to seed point modification
  - Support undo and redo
  - Support setting profile viewpoint distance
- Optimized individual tree point cloud editing
  - Improved method of profile selecting
  - Support creating an individual tree by specifying a chosen tree
  - Support setting profile viewpoint distance
  - Automatic saving of configuration
- Optimized profile point cloud editing
  - Improved the efficiency of block editing by more than 50%
  - Support mouse position tracking in the profile window
  - Support data exporting
- Added section comparing analysis function in section analysis
- Upgraded LiModel format to solve the problem of coordinate accuracy loss
- Reduced the GPU usage rate of point cloud display by 20%
- Support multiple calls to the same function in batch processing
- Fixed the bug when using cross selection and clipping tool simultaneously
- Improved the efficiency of point cloud conversion to DXF file

v4.1 - 06/08/2020

- **New Features**:
  - Added three-point angle measurement
  - Added registration of UAV and backpack forestry point cloud data
  - Added individual tree attribute information expansion
  - Added detection of forest structure changes
  - Added DBH measurement

- Help menu
  - Added software usage problems and demand feedback channel
  - Added FAQ, quick start, official website link, etc.
- **Enhancements**:
  - Optimized the efficiency of loading multiple files
  - Improved color rendering of point cloud attributes
  - Additional attribute management
    - Support deleting
    - Support component combination rendering
  - Improved profile editing and classifying
    - Support grid scale display in profile window
    - Support directory tree to manage point cloud tiles in memory
  - Support import boundary of measurement area with SHP file in volume measurement
  - Improved the efficiency of filtering function and the density of ground points
  - Support directory tree to manage point cloud tiles in memory in individual tree point cloud editing function
  - Support combined calculation in forestry metrics
  - Support extract point cloud by tree ID, support export the extracted point cloud to LAS and LiData format
  - Support attribute filtering in ALS seed point editing
  - Improved LiTIN editing, support 3D view editing, and support the setting of starting height of broken line
  - Added tunnel cross-section in cross-section analysis
  - Window layout supports multiple layout modes
  - Newly designed UI icons

v4.0 - 12/30/2019

- **New Features**:
  - Added denormalization tool
  - Added classifying close points tool
  - Added manual rotation and translation tool
  - Added convert ASCII to BLH tool
  - Added individual tree editor tool. Support to create, merge, and delete tree
  - Added statistics individual tree attributes tool
  - Added TLS leaf area index tool
  - Added ALS forestry metrics calculation in give area tools
  - Added deviation analysis tool for two data of the same area collected in different times
  - Added change detection tool
  - Added GPS time transformation tool which supports transforming GPS time between GPS coordinated time and GPS week and second
- **Enhancements**:
  - Support online update checking and installing
  - Upgraded the version of LiData to V2.0 which supports all the features of Las1 .4 (except the

waveform information) and the additional attribute of the point cloud file
    - Support the color rendering for all the attributes of point cloud data
    - Support the rotation of 2D view
    - Support fast mode and precise mode in volume calculation and providing the information of the measuring area
    - Support zooming to the layer by double-clicking the file in the directory tree
    - Support recovering the data association in the same path for the project file path
    - Support EDL effect in render to file and save to video tools
    - Support generating automatic alignment report and clearing the data alignment information
    - Improved the profile editor which supports customizing shortcut keys and solves the problem of profile buffer settings
    - Improved the EDL effects in orthogonal projection
    - Improved the cross selection tool which supports data clipping when being used with selection tool, or 2D drawing when being used with Vector Editing.
    - Improved data interaction experience:
        - Improved browsing fluency
        - Support automatically picking the rotation center
        - Improved point picking experience
        - Support measuring components in the measurement tools
        - Support measuring points number in the density measurement tool
    - Support recording the original Z value in normalization tool
    - Improved the data formats supported:
        - Support E57 point cloud format
        - Support additional attributes for LAS, PLY, and ASCII format point cloud files
        - Support choosing the version of las file when exporting point cloud in las format (las 1.2, las 1.3, and las 1.4)
        - Support converting version of LiData (V 1.9)
        - Support converting point cloud in Geographic Coordinate System to projected LiData
    - Improved the calculation method of individual tree crown volume
    - Improved the efficiency of the calculation of forestry metrics

v3.2 - 06/06/2019

- **New Features**:
    - Added brightness settings tool
    - Added ICP registration function
    - Support exporting TIN in DXF format
    - Added raster statistics tool
- **Enhancements**:
    - Optimized EDL display effect in profile editing mode
    - Optimized the ground point filter
    - Optimized the efficiency and effect of "Classify Buildings" tool
    - Support rectangular/circular interactive area selection tool in clipping function
    - Changed band calculator to raster calculator which supports custom formula

    - Shortcut keys and right-clicking menu in ALS individual tree segmentation editing tool
    - Support position uncertainty optimization mode in regression analysis
    - Support corresponding multiple point cloud files to one trajectory in "cut point cloud based on trajectory" tool
    - Profile editing and classifying:
        - Added the lasso selection tool
        - Added the plane detection tool
        - Support clearing the triangular network already been built
        - Support setting the size of brush selection tool
        - Support setting the step length of the profile translation
        - Optimized the efficiency of triangular network generating
    - TLS individual tree segmentation editing:
        - Support ignoring the Z-values of the input seed file
        - Support right-clicking menu and shortcut keys
        - Support hot keys in single tree attributes measuring
        - Support display of LiBackpack trajectory file
    - Manual registration:
        - Support inputting/outputting coordinate transformation matrix
        - Support the preview of transformation
    - Raster and table data:
        - Support color settings
        - Support highlighting objects when they are selected
        - Support centralizing the objects when they are double-clicked
- **Bug Fixed**:
    - Fixed the bug that the order of the vector and table data's attribute table is wrong
    - Fixed the bug that only the displayed data will be used in "batch extraction DBH" tool
    - Fixed the bug that profile selection will disappear in the main window if users edit point cloud in profile window

v3.1 - 12/31/2018

- **New Features**:
    - Profile editing and classifying:
        - Support block editing and classifying
        - Support multiple windows selecting and classifying
        - Add the brush selecting tool
        - Block editing mode supports to undo/redo
        - Added the automatic timing/automatic saving function
        - Added multiple selecting area filter classification/classification setting tools
        - Support shortcut keys switch function
        - Block editing mode supports RAM usage warning
    - Added convex hull and concave hull mode for boundary extraction
    - Added check for updates function

- **Enhancements**:
    - Improved the efficiency of LiData generating
    - Support to recalculate the histogram of point cloud intensity and elevation
    - Moved the rectify to reprojection and conversion module
    - Multiple point selecting tool supports to customize the extended attributes
    - Support projected coordinate systems in JGD2011 coordinate system
    - Support POS files in more formats. Now support POS file with projected coordinates
    - Support converting length units for las and LiData files

v3.0.1 - 12/10/2018

- **New Features**:
    - TLS individual tree segmentation editing:
        - Added circle selection tool
        - Added profile translation tool
        - Added DBH inspector tool
- **Enhancements**:
    - Optimized the color rendering for individual tree segmentation.
    - Optimized the segmentation of the vector boundary of crown in CHM individual tree segmentation.
    - Support the .shp file in section tool.
- **Bug Fixed**:
    - Fixed the bugs already known

v3.0 - 10/16/2018

- **New Features**:
    - Added saving users’ parameters settings as default, and the prompt of the information about each parameter when putting mouse on it
    - Support the spike-free TIN generation algorithm
    - Added the random forest regression model
    - Added the regression predict analyze
    - Added the Projection and Coordinate Conversion Toolset:
        - Seven parameters solution
        - Four parameters solution
        - Vertical datum transformation
    - Added noise classification function
    - Support conversion of raster data to LiData
- **Enhancements**:
    - Improved the ground point filtering algorithm
    - Improved the accuracy and efficiency of the TLS individual tree segmentation tool
    - Upgraded the LiTIN format to support saving the corresponding LiDAR point classes in LiTIN

data and display LiTIN data by classes
    - Improved the editing and flattening TIN data in LiTIN format
    - "Tile by range" supports buffer setting and vector exporting
    - Support subdivisions of Graticules
    - Support generating multiple files when performing polygon clipping
    - Support selecting RGB range when convert data to las format

v2.2 - 06/20/2018

- **New Features**:
    - Multi-threaded batch processing;
    - Support invoking of command line;
    - Measurement of individual tree attributes (total height, CBH, stem straightness. etc.) from TLS data.
- **Enhancements**:
    - Improved regression analysis which allows for the importation of independent variables from external sources;
    - Output elevation and intensity variables as .tif formatted files;
    - Individual tree segmentation from a selected point cloud class;
    - Improved TLS data editing tools that allow users to (1) batch process data to extract DBH measurements and (2) fit DBHs to point clouds selected in the profile view window;
    - Added crown diameter, area, and volume to the output of the TLS individual tree segmentation tool;
    - Improved Section Analysis Tool in the Terrain Module now allows users to: (1) analyze .dxf formatted files, (2) import more than one file at a time and draw multiple section lines. (3) define a step value to simplify sections. (4) export multiple section files
    - Output contour feature as 2D polylines;
    - Editing of vector dataset attributes;
    - Improved profile feature user experience.
- **Bug Fixed**:
    - Tree heights now updated for tree filtering after tree segmentation tool has been run;
    - Projection information of las data can now be read.

v2.1.2 - 04/25/2018

- **Enhancements**:
    - Improved batch extraction function for DBH (Diameter at Breast Height);
    - LiData upgrade.

v2.1.1 - 04/08/2018

- **Enhancements**:

- Support LAZ/LAS 1 .4 file formats;
- Intensity information with ply data now can be read.

v2.1 - 02/05/2018

- **New Features**:
  - Automatic calculation of flight strips matching parameters;
  - Median ground point clouds classification;
  - Supervised classification based on machine learning;
  - Support revising ground point filtering results of a defined region;
  - Support cross section analysis;
  - Vector Editing;
  - Support selected region extends from both end in profile viewer;
  - Tile by Point Number;
  - Support generating TIF images with projection information.
- **Enhancements**:
  - Improved DEM, DSM interpolation algorithm;
  - Improved efficiency of point cloud and TIN generated contour lines;
  - Improved efficiency of individual tree segmentation algorithm (individual tree segmentation from point cloud and individual tree segmentation based on seed point);
  - Support richer terrain product formats, including a new ASC format for raster data and a new DXF format for the contour line.
- **Bug Fixed**:
  - De-redundant function;
  - Multipoint selection;
  - Fixed a bug of Classifying by attribute.

v2.0 - 10/30/2017

- **New Features**:
  - New License Key: The software could be activated on a monthly basis or by module;
  - Forest Module (divided to ALS module and TLS module);
    - ALS Module:
      - Seeds generation algorithm: It's used to generate segmentation seed layer based on CHM, Point Cloud Segmentation and Layer Stacking;
      - Seeds edition: adding/deleting seeds, selecting seeds and segmenting individual tree based on seeds;
      - Added individual tree batch processing based on seed layer;
    - TLS Module:
      - Ground Classification using TLS data;
      - Added the seed point based ground point cloud individual tree batch processing;
      - Added the calculation of DBH;
      - Seeds edition: adding/deleting seeds, selecting seeds and segmenting individual tree

    - based on seeds;
  - Classification Module:
    - Improved performance of ground points filter algorithm;
    - Classification for key feature points. By identifying the key surface points, this allows the down-sampling of relatively flat areas thus improving processing efficiency without sacrificing quality;
    - Added a check box to the interface of ground point filter function to extract model key points after separating ground points;
  - Power Line Module:
    - Added creation of vector output for power line and tower classifications;
    - Real-time measurement of the distances from any given point to the closest power line and tower;
  - Added the setting of point cloud display point size for individual layers;
  - Added automatic identification of point cloud center point and setting it as rotation center;
  - Improved visual effects of TIN;
  - Shapefile layer attribute table now can be displayed;
  - Identifying and locating the corresponding individual tree in the viewer by double-clicking a row in the segmentation result list table.
  - Cross selection tool supports more accurate adjustment of the boundingbox;
  - Added Display Parameters Settings in LOG viewer while the LOG file still stays at system's TEMP folder;
  - Support information collection when the software crashes. You can send crash information in the pop up interface to the mailbox;
  - The cut results for multiple data sources can be saved separately or be merged into a file.

v1.5 - 07/14/2017

- **New Features**:
  - Added power line module. The main functions include marking tower position, cutting the point cloud data based on the tower position, automatic classification of power line and tower, dangerous points detection and report generation;
  - Added selection toolbar which includes polygon selection, rectangle selection, sphere selection, in cut, out cut, save cut and cancel selection;
  - Added editing point cloud classification with real-time TIN;
  - Support merging multiple files into one;
  - Support undo to profile editing and selection function by Ctrl+Z.
- **Enhancements**:
  - CHM segmentation algorithm improvement. Users can adjust more parameters and generate the shape file of tree boundary;
  - Added the residual report to the registration function. Users can choose the points for coordinate Conversion;
  - Cross selection result can be export;
  - The parameters setting for batch processing for multiple files and multiple functions can be imported and exported;

  - Improved profile editing efficiency;
  - Users can choose whether to generate minor, basic and major contour when using "point cloud to contour" and "TIN to contour" functions;
  - Added seven parameters transformation for reprojection function;
  - The results of Clip by Circle and Clip by Rectangle can be exported as one file, or saved as individual files according to the extent of circle or rectangle;
  - The real-time coordinates in the status bar are more accurate;
  - Improved user experience of navigating.

v1.4 - 05/31/2017

- **New Features**:
  - Added "generate TIN", "TIN edit", "TIN to Contour", "TIN to DEM" functions;
  - Added Grid Volume Statistics function;
  - Added "Camera Roam" and "Save to Video functions";
  - Fixed the bug of memory leak when loading .ply data.

v1.3 - 04/17/2017

- **New Features**:
  - Added PLY data import and export to I/O module;
  - Added elevation difference, overlap and density quality reports to strip adjustment module;
  - Added "Resampling based on octree" and "point cloud convert to DXF" to data management module;
  - Power line classification;
  - Support mix and glass coloring modes in rendering;
  - Batch Processing for multi-files and multi-functions;
  - Geometric correction (support point cloud registration based on points or sphere targets).
- **Enhancements**:
  - Speed improvements for contour generation based on point cloud;
  - Users can set the properties of minor, basic and major contours for "point cloud to contour". The elevation attributes can be read;
  - The boundary of concave polygon is shown for area measurements;
  - The resolution of surface model is not limited to 0.5m.
- **Bug Fixed**:
  - Fixed the bug of file opening error when there is an invalid file in multiple .csv files;
  - Fixed the bug of window flash caused by EDL effect in windows 10;
  - Bug fixed for stepwise regression;
  - Bug fixed for software crash caused by opening old version of LiData;
  - Bug fixed where LiDAR360 could not be opened with unstandardized projection settings.

Lite - 02/04/2017

Free version, with main functions including data visualization and grid statistics.

v1.1 - 12/22/2016

- **New Features**:
    - Software platform update: 2D&3D integration, multiple window linkage, rolling screen, display order changed by drag, cross selection,etc;
    - Strip quality check function;
    - Added generating contour lines based on point cloud;
    - Chinese software version;
    - Bug fixed for raster image subdivision and LiModel with texture.

v1.0 - 10/11/2016

First version release of LiDAR360.