For more information contact Synergy Positioning Systems or
visit the Synergy Positioning Systems website at www.synergypositioning.co.nz
All branches: Phone 0800 867 266 Email: info@synergypositioning.co.nz

Instruction Manual
Post-Processing Software

MAGNET Collage

1810 (8H) **U**

INTRODUCTION

Thank you for purchasing our product.

- Please read this Instruction manual carefully, when using this product.
- The specifications and general appearance of the instrument are subject to change without prior notice and without obligation by TOPCON CORPORATION and may differ from those appearing in this manual.
- The content of this manual is subject to change without notice.
- Some of the diagrams shown in this manual may be simplified for easier understanding.
- Always keep this manual in a convenient location and read it when necessary.
- Except as permitted by Copyright law, this manual may not be copied, and no part of this manual may be reproduced in any form or by any means.
- This manual may not be modified, adapted or otherwise used for the production of derivative works.

Symbols

This manual uses the following symbols in explanations:

	: Indicates a precaution for use or an important note to read before work.
	: Indicates a relevant chapter (section) or a chapter (section) to reference.
	: Indicates supplemental information.
[OK] etc.	: Indicates operation icons on the display and window dialog buttons.
{Shift} etc.	: Indicates keys on the keyboard.
<File Dialog>, etc.	: Indicates the title of a view, window or dialog box.

Copyright of the software

- This software is copyrighted by Topcon. Please note that the specifications of this software are subject to change without notice.

Registered trademarks

- Mercury-Evoluto MMS Edition is a product name of Fukui Computer Corporation.
- Windows is a registered trademark of Microsoft Corporation.
- AutoCAD, DWG and DXF are trademarks or registered trademarks of Autoddesk, Inc.
- Other company and product names in this manual are trademarks or registered trademarks of respective companies.

Exporting this product (Relating EAR)

- This product contains software/technology, which are subject to the EAR (Export Administration Regulations). Depending on countries you wish to export or bring the product to, a US export license may be required. In such a case, it is your responsibility to obtain the license. The countries requiring the license as of Aug. 2017 are shown below. Please consult the Export Administration Regulations as they are subject to change.

  North Korea
  Iran
  Syria
  Sudan
  Cuba

  URL for the EAR of the US: http://www.bis.doc.gov/policiesandregulations/ear/index.htm

Exceptions from responsibility

- The manufacturer, or its representatives, assumes no responsibility for any damage, or loss of profits (change of data, loss of data, loss of profits, an interruption of business etc.) caused by use of the product or an unusable product.
- The manufacturer, or its representatives, assumes no responsibility for any damage, or loss of profits caused by usage different to that explained in this manual.
- The manufacturer, or its representatives, assumes no responsibility for consequential damage, or loss of profits due to heavy rain, strong wind, high-temperature and humidity, or storing or use of the product under unusual conditions.
- Product failures caused by rebuilding are out of warranty.
- Cautions and warnings included in this manual do not cover all the possible events.

CONTENTS

1. ABOUT THIS PROGRAM

1.1 Product Overview

The MAGNET Collage program is a post-processing program that runs on the desktop of your personal computer. You can handle the measurement data of various sensor systems shown below in an integrative manner.

Mobile mapping system IP-S3:
Calculation of geographical coordinates at measurement point of laser scanner, generation of panoramic image, and creation of clouds data can be carried out by obtaining the travelling figure and trajectory from the data measured by inertial measurement unit (IMU), GNSS receiver, wheel encoder, laser scanner, panoramic camera, which are mounted on a vehicle.

Ground laser scanner system GLS:
The clouds data can be created by registering scan data measured from multiple occupation points in various methods.

You can also carry out removal of unnecessary points, adjustment of clouds density, and data output in various data formats by integrating the clouds data obtained by these sensor systems.

1.2 Operating Environment

To execute the install of MAGNET Collage, a PC that meets the conditions shown below is required.

Configuration	Requirement
OS	Windows 7 64-bit, Windows 8 64-bit or Windows10 64-bit
CPU	Intel Core i7 (4 cores, 8 threads) or greater
RAM	DDR3 8GB or greater
Storage	MMS module : SSD 160GB or greater recommended Scan module : SSD 20GB or greater recommended
Screen Configuration	32-bit color, 1280x1024 pixels or more
Graphics Card	NVIDIA discrete GPU, VRAM 512 MB or greater
Network	Network environment that can be connected to internet

- To use the link function with "Mercury-Evoluto MMS Edition" of Fukuicomputer, Inc., the use of dual display is recommended.
- To create the clouds, approximately tenfold free capacity of file size of measured scan data is sometimes required for the temp folder (standard: c:/user/AppData/Local/Temp).
- MMS Project
  The capacity of data folder after processing 1-hour run data (urban area run, JPEG quality 75%, shooting distance interval 4m) with a standard setting is approximately 160GB.
- GLS Project
  The size of station data which is measured by GLS-2000 with highest resolution is approximately 3GB to 4GB. The capacity of data folder after processing this station is approximately 20GB.

■ Compatibility between applicable device and data

Mobile mapping system (MMS)

Product Name	Function	Status
IP-S3	Reading of measurement data	Available
	Analysis of trajectory and generation of clouds	Available
	Reading of analysis result of Mobile Master Office	Available
	Data output	Available
IP-S2	Reading of measurement data	Not available
	Reading of analysis result of Geoclean Workstation	Corresponding to IP-S2 and IP-S2 Compact+
	Analysis of trajectory and generation of clouds	Corresponding to IP-S2 Compact+
	Data output	Corresponding to IP-S2 and IP-S2 Compact+

Ground laser scanner system

Product Name	Reading of Measurement Data (Stand-alone data)
GLS-1000	Available
GLS-1500	Available
GLS-2000	Available

Note

Importing ScanMaster datasets

ScanMaster version 3.07 can publish MAGNET Collage format project dataset from ScanMaster format project dataset, but there are several restrictions as follows:

- Collage cannot import Edge, Section or Contour sets of ScanMaster directly. Those data must be converted into polylines beforehand on ScanMaster.
- Mesh data cannot be imported.
- Clouds existed under a Scan Position item in ScanMaster project will be imported into a model in a model project which is published by ScanMaster.
- Clouds existed under a Data item in ScanMaster project will be imported into a model in a model project which is published by ScanMaster.

1.3 Computer Settings

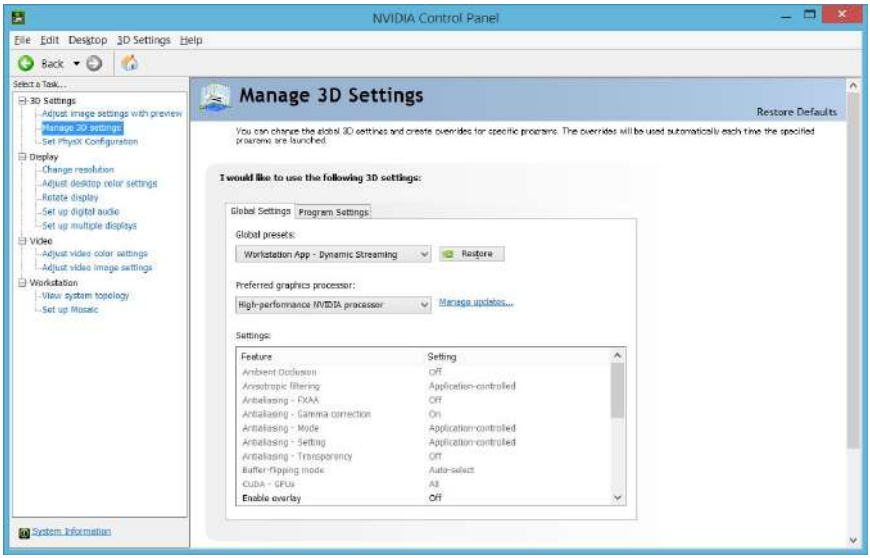
This section describes how to set up your computer to use MAGNET Collage.

■ Select a Graphics Card

If your computer allows you to select a graphics card, ensure that a high performance graphics card is installed and selected.

1. Right-click on the computer desktop and select "NVIDIA Control Panel".

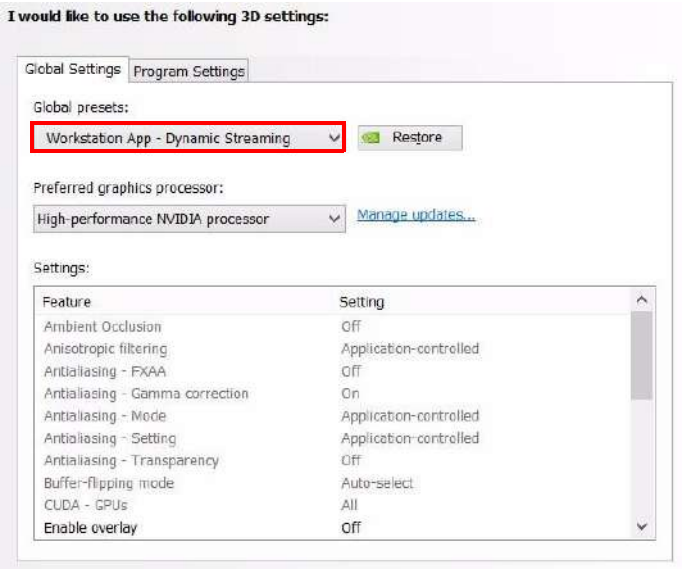
   <NVIDIA Control Panel> appears.



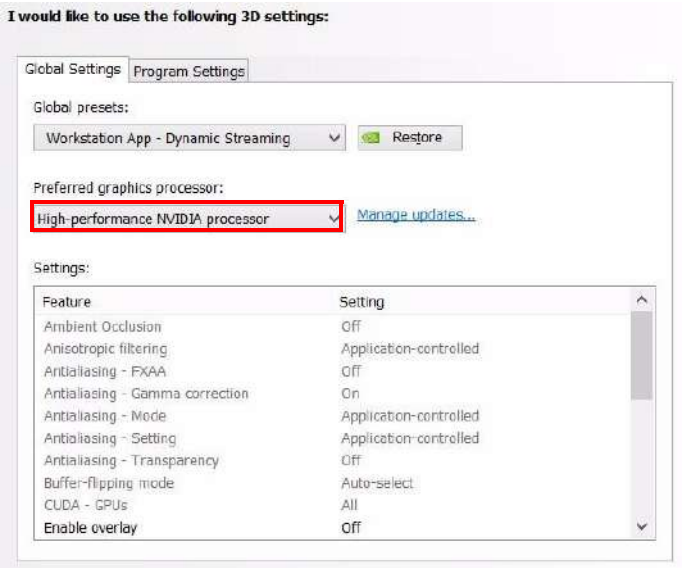
NVIDIA Control Panel

2. Click "Manage 3D settings" from the task menu in the left hand of the screen.

3. If your computer allows you to select a "Global Presets" in the right hand of the screen, ensure that "Workstation App-Dynamic Streaming" is selected.



4. If your computer allows you to select a "Preferred graphics processor" in the right hand of the screen, ensure that "High-performance NVIDIA processor" is selected.



1.4 About License Clauses

For license clauses for the library used in this software, refer to "License.txt" in the folder where this software is installed.

1.5 Account Name of Personal Computer and Data Path

This software does not allow multibyte characters, special characters in some languages and some symbols (\, /, :, *, ?, ", <, >, |) in a relevant data path.
Store the run data folder, base station observation data, geoid file, and output data folder at the place where above characters and symbols are not included with the path. The same goes for the account name of the computer.

1.6 Management of Run Data

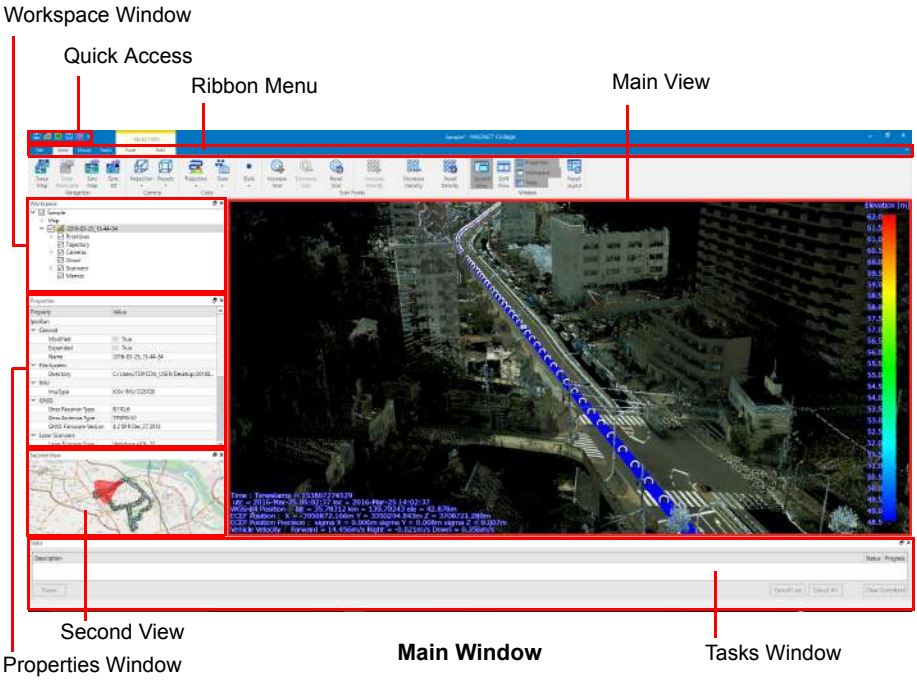
Management, storage, and backup of collected data is the responsibility of the user. Create a backup of the collected run data before processing it.

2. BASIC OPERATION

This chapter describes the features and screens of MAGNET Collage and the items to understand before actually starting operation.

2.1 Description of Display Screen

MAGNET Collage provides an efficient user interface designed in such a way that enables quick and easy data load and display and attribute creation. Understanding the user interface better improves operability, enabling efficient work.



Ribbon Menu : Each functions of workspace operation, switching of display, edit of clouds, segment / operation of primitives, data analysis, and edit of data are classified in the ribbon page.

Quick Access : Frequently used icons are lined up in the upper left of the main window.

[Save] : A workspace is saved.

[Add IPS] : IP-S3 project is added to workspace.

[Add RDM] : RD-M1 project is added to workspace.


[Add GLS] : GLS project is added to workspace.


[Add model] : A model project is added to workspace.

Main/Second View	:	Display screens for the trajectory, point cloud, etc. Two screens are provided, and the two types of the screens can be displayed concurrently. Moreover, the second view can be segmented into two.
Workspace Window	:	The information of project data and map data is displayed.
Properties Window	:	Displays the attribute information of the item selected in <Workspace Window>.
Tasks Window	:	Displays information on various tasks. Also, you can cancel, pause, or resume processing.

Note

- The display locations and sizes of windows other than <Main View> can be customized. While left-clicking the title bar, move the mouse to move the window to another location.

2.2 Mouse Operations

The interface of MAGNET Collage is designed with an emphasis on screen operations with mouse. "Mouse Operations" describes screen operations.
☞ “Keyboard shortcuts”

Table 1: Mouse Operations

View	Operation	Result
Panorama view	Displaying and operating a panorama image	
	Left-drag	Displaying the surrounding area in the panorama image
	Right-drag	Zooming in/out on the center of the image by dragging upward/downward
	Left double-click	Moving to the image adjacent to the clicked point
	Wheel mouse rotation	Zooming in/out on the center of the image
	Ctrl+Left double-click on a point	Moving to the nearest panorama of target point and displayed facing the point
	Alt+Left double-click	Correcting parallax of a panorama image according to the distance to a selected point
	Alt+R key	Restoring a default panorama image
3D view	Displaying and operating in 3D view	
	Left-drag	Rotation display
	Left double-click	Moving to the center of the image
	Right-drag	Parallel translation along the plane where the mouse cursor is located
	Wheel mouse drag	Parallel translation
	Wheel mouse rotation	Zooming in/out the image
	Right and left click + drag	Parallel translation along a surface that intersects at right angle against eye direction of camera
Map view	2D display and operations on a map	
	Left-drag	Parallel translation
	Wheel mouse rotation	Zooming in/out the image
	Left double-click	Moving to the center of the image
	Right-drag	Zooming in/out by dragging upward/ downward

■ Keyboard Shortcuts

Keyboard shortcuts enable you to quickly perform operations and switch screens. The table below describes the keyboard shortcuts for MAGNET Collage.

Table 2: Keyboard shortcuts

Execution location	Shortcut	Result
Menu	Program shortcut	
	Ctrl+N	Opens a new workspace.
	Ctrl+O	Opens a workspace.
	Ctrl+S	Saves the workspace.
	Ctrl+Shift+S	Saves a workspace with a new name.
	Ctrl+Q	Exits MAGNET Collage.
	Ctrl+X	Switches map display windows.
	Ctrl+C	Switches displays between panorama and 3D.
	Ctrl++	Increases the point size.
	Ctrl+ -	Decreases the point size.
	Alt++	Increases the point density.
	Alt+ -	Decreases the point density.
	Ctrl+M	Minimizes the window.
	Ctrl+Z	Shows/Hides <Second View Window>.
Panorama view	Displaying and operating a panorama image	
	F key	Moves to a forward image relative to the traveling direction.
	B key	Moves to a backward image relative to the traveling direction.
	I key	Shows/Hides panorama images.
	Space key	Restore default
	←, →	Sets the image size of the parallax-corrected panoramic image. The specified resolution will be displayed at the bottom of the panorama view. 4000: 4000x2000, 5400: 5400x2700, 8000: 8000x4000
3D view	Displaying and operating in 3D view	
	F key	Moves from the selected panorama image to a forward image relative to the traveling direction.
	B key	Moves from the selected panorama image to a backward image relative to the traveling direction.
	Space key	Restore default

Table 2: Keyboard shortcuts

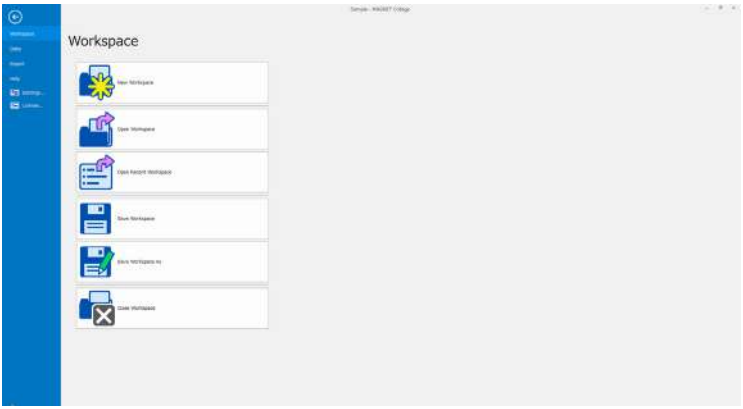
Map view	2D display and operations on a map	
	F key	Moves from the selected panorama image to a forward image relative to the traveling direction.
	B key	Moves from the selected panorama image to a backward image relative to the traveling direction.
	Space key	Restore default

2.3 Reading of Data

With the workspace, GLS data, MMS data, RDM data, and model data can be separately read to be registered in tree format and the data can be displayed simultaneously.

■ Workspace menu

Click "File" tab in the ribbon menu and select [Workspace].



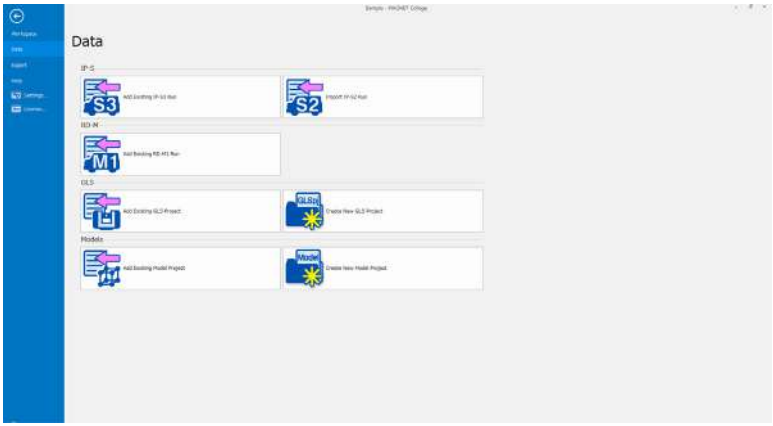
Workspace Menu

New Workspace	:	A workspace is newly created.
Open Workspace	:	An existing workspace is opened.
Open Recent Workspaces	:	The list of recently opened workspaces is displayed.
Save Workspace	:	The current workspace is saved.
Save Workspace As	:	The current workspace is named and saved.
Close Workspace	:	The workspace is closed and a new workspace can be created.

- When you create a new workspace, please save the workspace immediately. If you don't save the workspace as any name, auto-save function for workspace will not be enabled.(☞"2.9 Application Setting"-" Auto Save")

■ Data menu

Click "File" tab in the ribbon menu and select [Data].



Data Menu

Table 3: Data Menu

Data		Description
IP-S	Add Existing IP-S3 Run	IP-S3 run data is added to a workspace.
	Import IP-S2 Run	IP-S2 run data is imported in a workspace.
RD-M	Add Existing RD-M1 Run	RD-M1 run data is added to a workspace.
GLS	Add Existing GLS Project	GLS project that is created by Collage is added to workspace. If you specify a GLS raw data directly, Collage will edit the raw data during GLS data processing. In this case, you can avoid a risk of the raw data modification by making a copy of the raw data to a local disk and adding the copy to workspace.
	Create New GLS Project	GLS project is newly created and added to a workspace. In this case, a copy of GLS data is imported to GLS project.
Models	Add Existing Model Project	Model project that is created by Collage is added to workspace.
	Create New Model Project	Model project is newly created and added to a workspace.

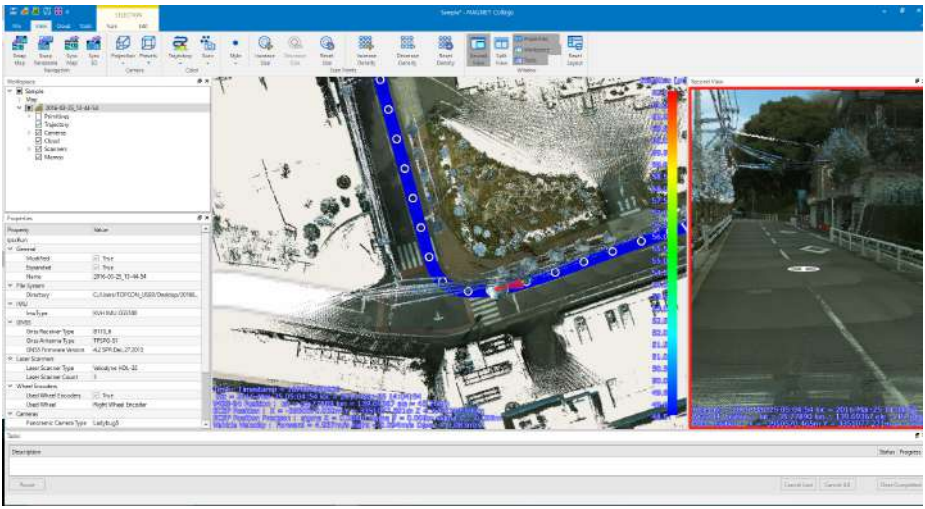
2.4 Data Display

The screen of MAGNET Collage consists of a main view window and various windows. You can change three main screen displays (map view, 3D view, and panorama view) in the <Main/ Second View Window>. In addition, you can change the data display mode by clicking relating icon.

■ 3D data display

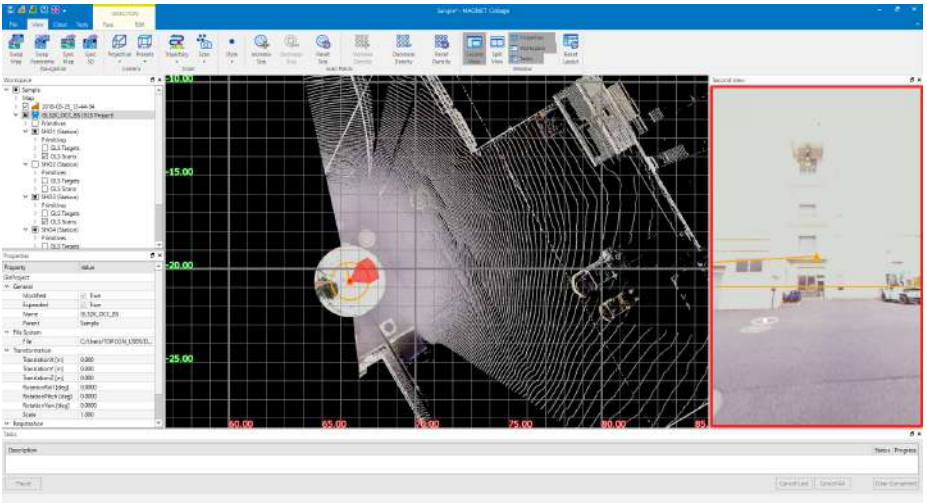
Map view

On the map view, scan, clouds, and panorama etc. are displayed in 2D. IPS, GLS and model data for which geodetic has been referenced are overlapped on the background map.



Map View (Display the IPS Data)

GLS and model data for which geodetic has not been referenced are displayed on the grid of local coordinates.



Map View Displaying GLS Local Coordinate Data

Note

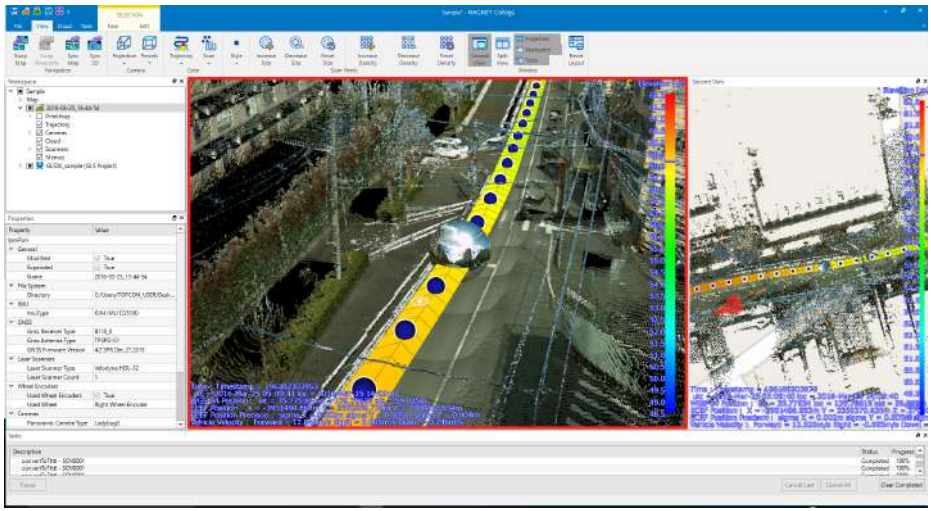
- If the personal computer is connected to the Internet, you can display a background map on the map view. The background map of a location that has been displayed is recorded in the personal computer. Therefore, from the next time, the map can be displayed even when the computer is not connected to the Internet.

The main functions of map view are shown below.

- Scaling up / down and parallel translation of displayed content
- In case of IPS, if you move the mouse cursor on trajectory, the detailed information such as position of the point and time, and the color bar of the currently selected display color are displayed.
- In case of IPS and GLS, clicking a blue ball can select a panoramic image and the selected point changes to the ball display of the panoramic image.
- The camera direction and angular field on panoramic / 3D view are displayed as orange quadrangular pyramids on the map view.
- In case of IPS, pressing {F} key selects the next panoramic image in the travelling direction. Pressing {B} key selects the previous panoramic image in travelling direction.

3D view

On 3D view, scan, clouds, and panorama etc. are displayed in 3D.



3D View Displaying IPS Data

The main functions of 3D view are shown below.

- Scaling up / down, parallel translation, and rotation of displayed content
- In case of IPS, if you move the mouse cursor on trajectory, the detailed information such as position of the point and time, and the color bar of the currently selected display color are displayed.
- In case of IPS and GLS, clicking a blue ball can select a panoramic image and the selected point changes to the ball display of the panoramic image.
- In case of IPS, pressing {F} key selects the next panoramic image in the travelling direction. Pressing {B} key selects the previous panoramic image in travelling direction.
- The compass shows north, south, east and west of the current display location.

Synchronization of map view and 3D view

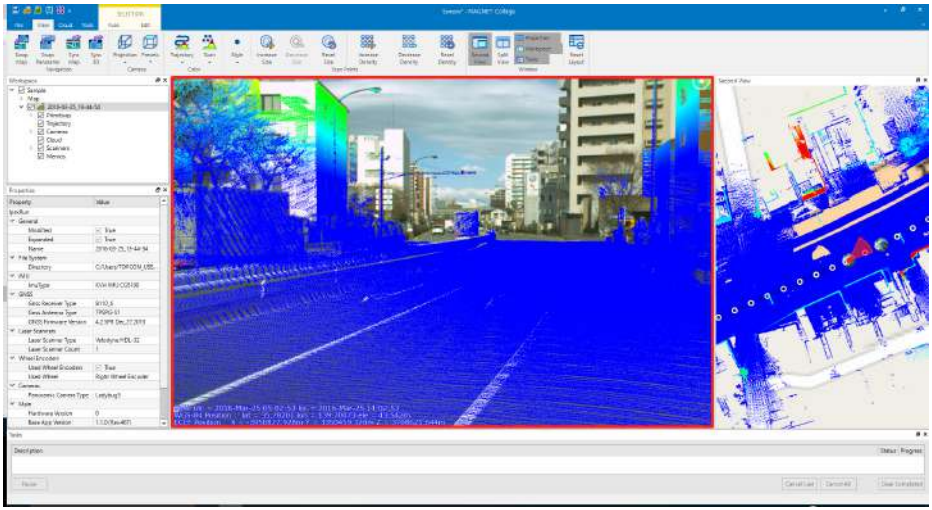
If you click "Sync Map"  in the "View" tab of ribbon menu, the display range of map view is adjusted within the current display range of 3D view. If you click "Sync 3D"  , the display range of 3D view is adjusted within the current display range of map view.

Jump To display position of project data

If you double-click an item on <Workspace Window> or click "Jump To"  in the "Edit" tab, the display range is adjusted so that the selected item is displayed.

Panorama view

In case of IPS and GLS, a panoramic image from a viewpoint of camera can be displayed as panoramic view. To display the panoramic view, creation and array of panoramic image need to be completed.
If scan and generation of clouds are finished, scan and clouds can be displayed at the same time.



Panorama View Displaying Panoramic Image of IPS/GLS-2000 Data

- A panoramic image is a referential image and misalignment may be generated depending on the location. For the point measurement, etc., it is recommended to refer to the intensity of scan/ cloud data.

The main functions of the panorama view are as follows:

- 360-degree rotation about the camera center (origin) and zoom-in/out.
- Rotation display of an image by dragging the left mouse button.
- Zooming in/out an image with wheel mouse.
- Zooming in/out an image by dragging the right mouse button upward/downward.
- Displaying a scan/cloud overlapped with an image.
- Displaying panorama position information.
- Press the {F} key to select the panorama image immediately after the current one relative to the traveling direction. Press the {B} key to select the panorama image immediately before the current one relative to the traveling direction.
- In case of IPS or GLS-2000, correcting parallax of a panorama image by "Alt + Left double-click".
- In case of IPS or GLS-2000, restoring a default panorama image by "Alt + R key".
- In case of IPS or GLS-2000, Showing or Hiding a panorama image by {I} key.

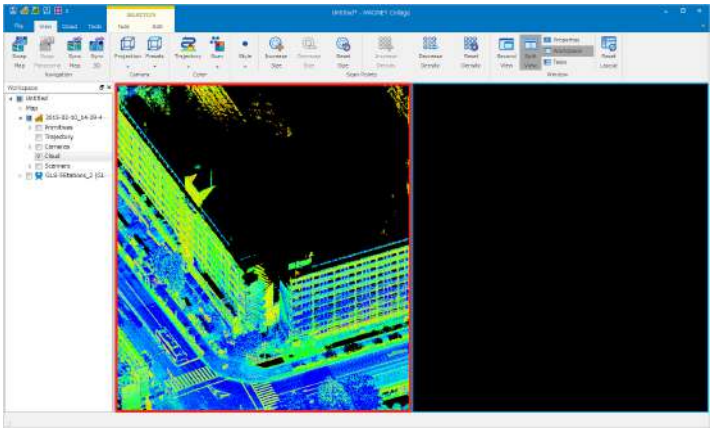
■ Split view

If you click "Split View"  in "View" tab, 3D view can be segmented. If you click "Set Scope" in "Edit" tab, you can limit scope of 3D view.

On the split view, the following operation is enabled.

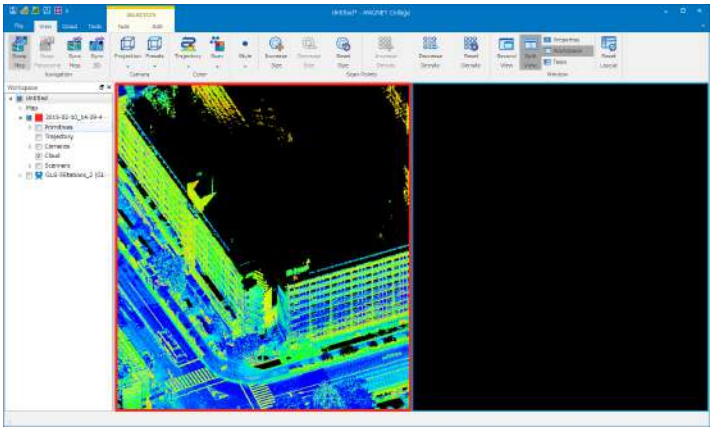
1. Click "Split View" in "View" tab.

   3D view can be segmented. (Red frame view and Blue one)



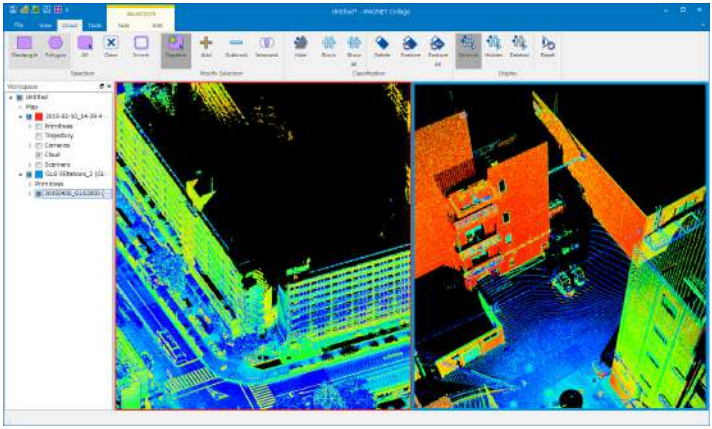
   If you click red frame view or blue one, each view becomes a thick frame.

2. Select IPS project on <Workspace Window> and select "Set scope" on the right-click while red frame view is clicked.



   IPS data will be displayed in red frame view.

3. Select GLS project on <Workspace Window> and select "Set scope" on the right-click while blue frame view is clicked.

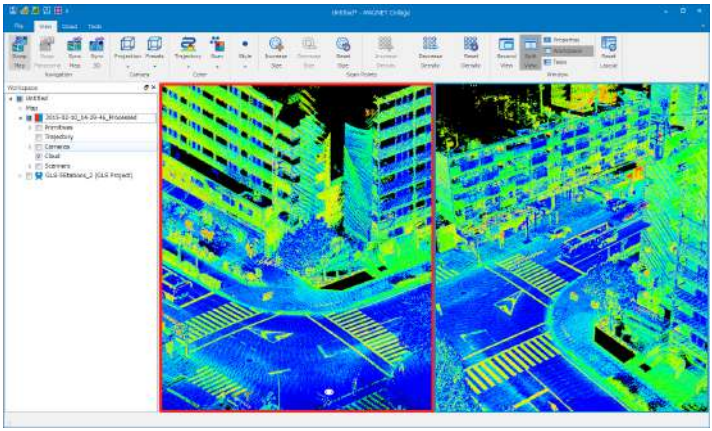


GLS data will be displayed in blue frame view. So, you can see different project data at the same time.

4. To cancel the selected scope, select workspace name on <Workspace Window> and set each scope view.

Note

• You can set red and blue scope to the same project.



■ Property window

On <Property Window>, the detailed information of the item that is currently selected on <Workspace Window> is displayed.

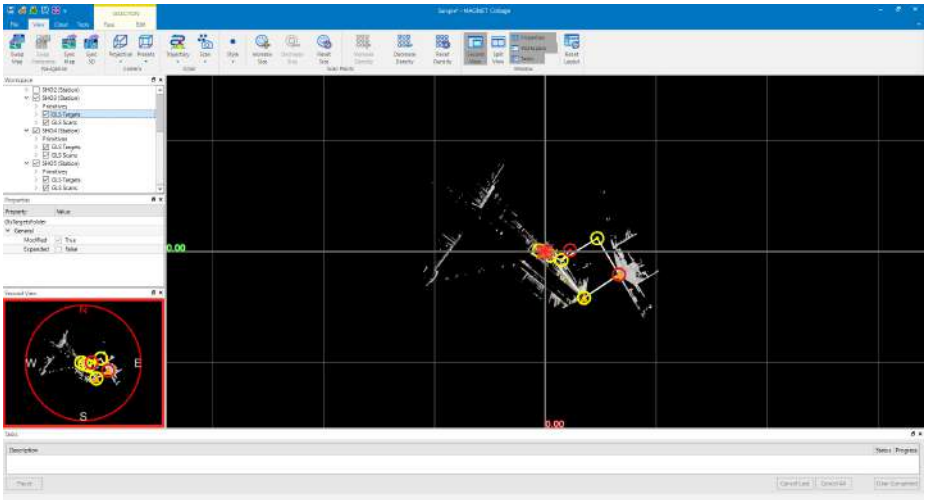
Properties

Property	Value
IpsxRun	
General	
Modified	False
Expanded	False
Name	2016-03-25_13-44-54
File System	
Directory	C:/Users/TOPCON_USER/Desktop/20160325_PN0056_FT/2016-03-25_13-44-54
IMU	
ImuType	KVH IMU CG5100
GNSS	
Gnss Receiver Type	B110_6
Gnss Antenna Type	TPSPG-S1
GNSS Firmware Version	4.2 SPR Dec.27.2013
Laser Scanners	
Laser Scanner Type	Velodyne HDL-32
Laser Scanner Count	1
Wheel Encoders	
Used Wheel Encoders	True
Used Wheel	Right Wheel Encoder
Cameras	
Panoramic Camera Type	Ladybug5
Main	
Hardware Version	0
Base App Version	1.1.0 (Rev.467)
FPGA Version	1.1.0
OS Version	1.1.0
Calibration File	PN0056_1_2016-3-17-10-15-20
Serial Number	PR0056

Property Window

■ Display of GLS data

If you select GLS project on <Workspace Window> and double-click it or click "Jump To"  in "Edit" tab, GLS data will be displayed.



Main Screen Displaying GLS Data

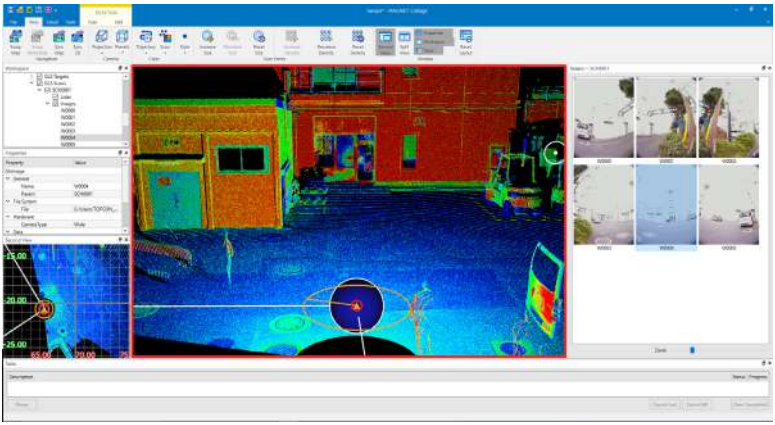
Display of occupation point and target

The occupation point is displayed as an orange triangle and the coordinate position of occupation point on the ground surface is displayed as an orange circle. The position of target is displayed as outline of small ball and the coordinate position of target on the ground surface is displayed as a circle. At this time, the usual target is displayed in yellow and the target that becomes viewpoint backsight point is displayed in red. The occupation point and the target are linked by line segment.

Display of camera image

On workspace, select [GLS project]->[Station]-> image item under GLS clouds and click "Open Gallery"  in "Edit" tab. <Gallery View> that lists the camera images as thumbnail will be displayed.

If you double-click an image on the gallery view, an individual image view window will be opened.

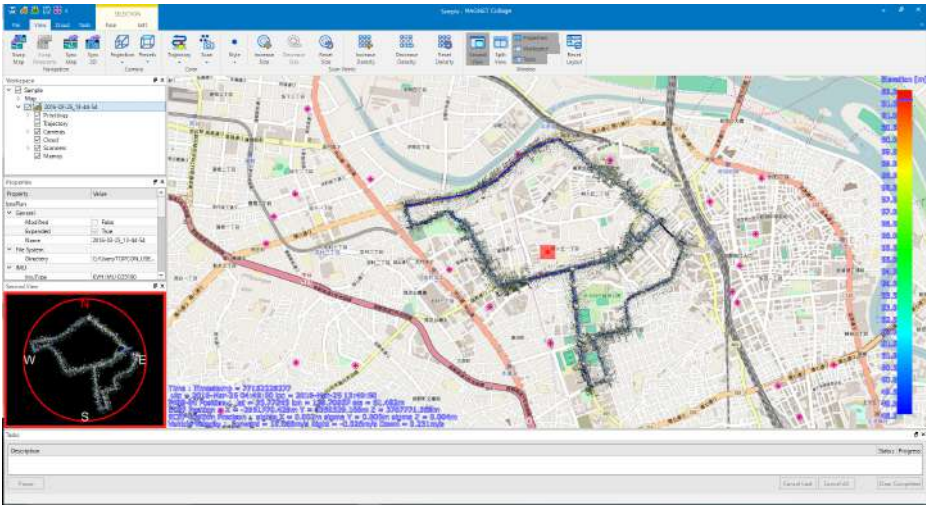


Gallery View

If you double-click an image on <Gallery View>, an individual image display window will be opened.

■ Display of IPS data

If you select IPS project on <Workspace Window> and double-click it or click "Jump To"  in "Edit" tab, the IPS data will be displayed.



Main Screen Displaying IPS Data

Scan and clouds

The measurement data of laser scanner that is displayed in 3D is called scan data. The scan data that is optimized for high-speed processing is called clouds data. These data have following characteristics.

Data	Display	Edit of clouds	Selection of segment
Scan data	Low speed	Not available	Available
Clouds data	High speed	Available	Not available

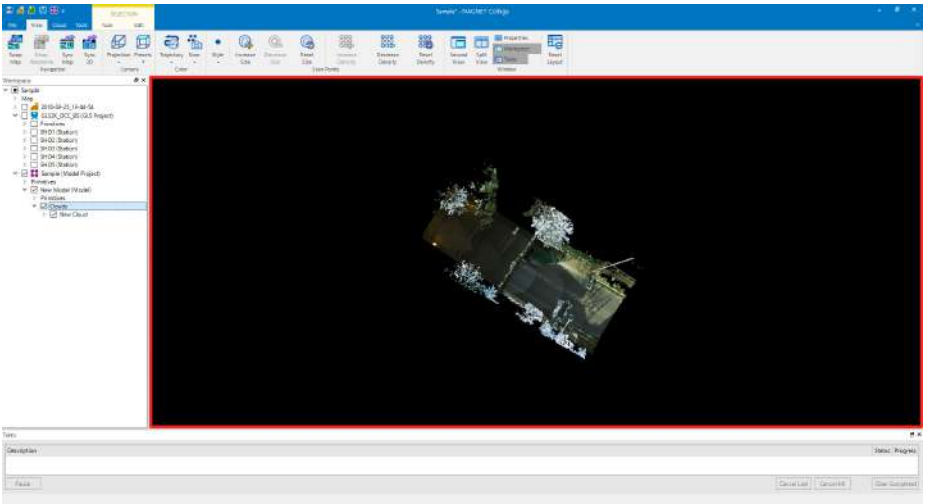
Selection of segment of data and clipping

To edit CAD primitives using scan data, if you select the segment of only target data range and clip it, the work can be carried out very efficiently. To select a segment of data, click "Select"  in "Tools" tab and click start position and end position on the trajectory. Multiple segments can be selected at the same time. Click "Clip"  in "Tools" tab. To cancel the clip, click "Unclip"  . To cancel the selected segment, click "Unselect" 

☞ "4.3 Data Editing"-"Specifying data editing segment"

■ Display of model data

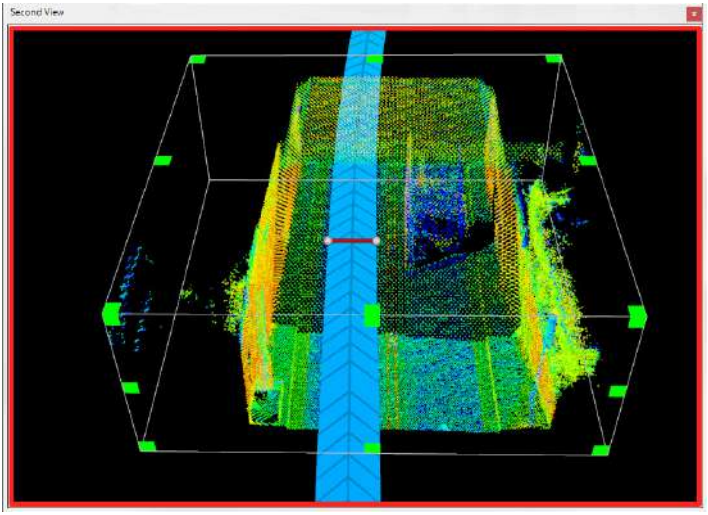
If you select model project on <Workspace Window> and double-click it or click "Jump To"  in "Edit" tab, Model data will be displayed.



Main Screen Displaying Model Data

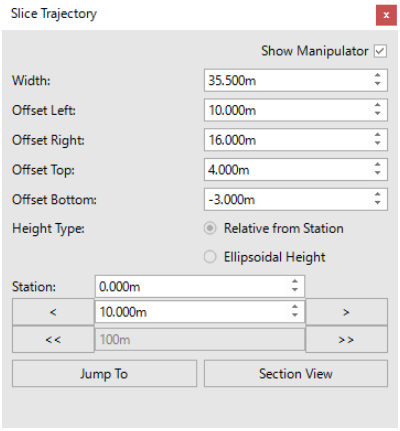
■ Slice view

Scan or clouds can be displayed by cutting it as an arbitrary cross-section surface.



In case of IPS data

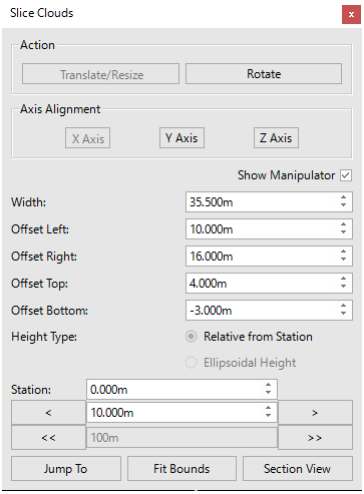
If you select trajectory of IPS project on <Workspace Window> or select polyline item in primitives and click "Slice"  in "Edit" tab, <Slice Window> will be displayed.



IPS Slice Window

In case of GLS data

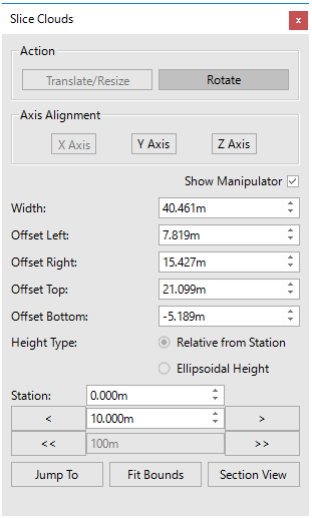
If you select SCN item in GLS clouds of station of GLS project on <Workspace Window> or select polyline item in primitives and click "Slice"  in "Edit" tab, <Slice Window> will be displayed.



GLS Slice Window

In case of model data

If you select clouds item, TIN item or DEM item of model project on <Workspace Window> or select polyline item in primitives or alignment item and click "Slice"  in "Edit" tab, <Slice Window> will be displayed.

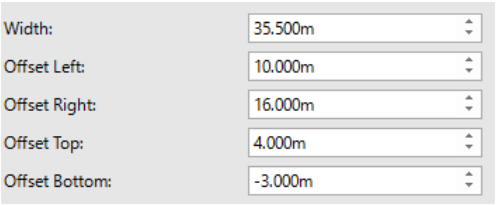


Model Slice Window

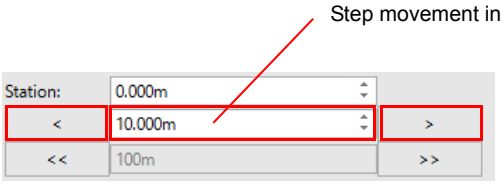
On the slice window, the following operation is enabled.

Setting of slice position

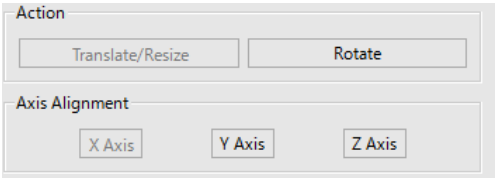
If you hold Shift key and double-click a trajectory, the position can be set to the slice position. If you drag a baton  on the slice window, the slice position can be changed. If you input the distance from starting point in "Station" input field and click [Jump To], the slice position can be moved to the input position.



If you click [<] or [>], the slice position can be moved in a stepped manner by the distance set in the step movement input field.



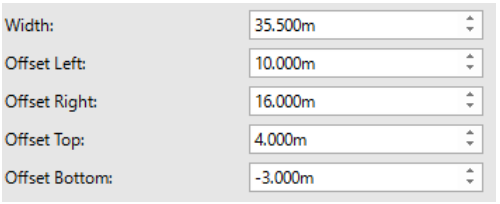
In case of GLS data and model data, the slice position can be arbitrarily specified by parallel translation, resizing, and rotation.



[Translate/Resize] : Execute parallel translation and resizing of 3D view.
[Rotate] : Execute the rotation of 3D view.
[X-Axis], [Y-Axis], and [Z-Axis]: Set the basic axis direction of slice.

Setting of slice range

If you hold {shift} key and drag the size change handle at four corners of slice range, the slice range can be changed. The slice range can be also changed by directly inputting a value in "Width" or "Offsets" input field.



Height specification

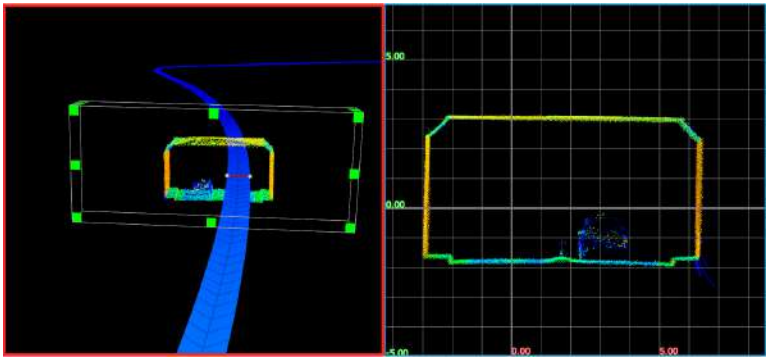
The standard of height of slice range can be specified as "Relative from Station" or "Ellipsoidal Height".

Input range

Setting items	Specified range
Station	0.000 to 10000000.000
Step	0.000 to 999.990
Step x 10	10 times of the input value in "Step"

■ Cross-section view

If you click [Section View] during slice view, <3D View> will be segmented into two and <Cross-Section View> will be displayed.



<Cross-Section View> is the view that displays a slice in 2D by setting the longitudinal direction to vertical direction. The operation same as that of <Map View> is available.

■ Boundary fitting

If you click [Fit Bounds] within slice frame of GLS data and model data, the slice area is adjusted so that entire boundary range of clouds data is included.

■ Calculate point cloud density

Point cloud density can be calculated from selected cloud. To select point cloud, please refer to the following.

In case of IPS data

Right-click cloud in IPS project on <Workspace Window>, then click "Calculate Point Cloud Density"


.

In case of GLS data

Right-click SCN item or Lidar in GLS clouds of station of GLS project on <Workspace Window>, then click "Calculate Point Cloud Density" .

In case of UAS data

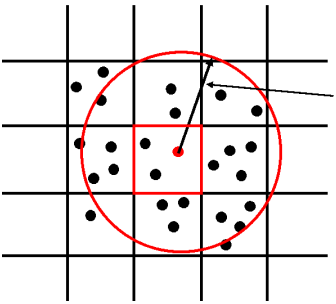
Right-click cloud in UAS project on <Workspace Window>, then click "Calculate Point Cloud Density" .

In case of Model data

Right-click clouds item or Lidar under clouds of Model project <Workspace Window>, then click "Calculate Point Cloud Density" .

If you click "Calculate Point Cloud Density" , <Calculate Point Cloud Density Window> will be displayed.
Set some parameters on < Calculate Point Cloud Density Window >.

Setting items	Description
Neighbor Radius	This function searches for neighborhood points and calculates density. Radius to search for the neighbor points.
Precision	This parameter affects the calculation accuracy of the boundary of the search region. The accuracy of the boundary will be increased from "Low" / "Medium" / "High" in order, but processing time will be longer.
Grid Size	Set the grid size. Points within the specified region are given certain density. If this value is set finely, the grid will be detailed, but the processing time will be longer.



A representative point (red point) is selected automatically in point clouds included within a cell. A circle on the basis of the representative point with the radius of "Neighbor Radius" is drawn. The point cloud density is calculated from point clouds included within the circle, and the density is used as the density of the cell (red cell). It means the calculation is executed once only each cell. In the example on the left, the red cell includes 3 points. Point cloud density calculation is executed once for the representative point (red point).

To display the results

The method of confirming the calculation result of point cloud density is as follows.

1. Select "View" tab on the ribbon menu, click "Scan" and "Density".

   Note

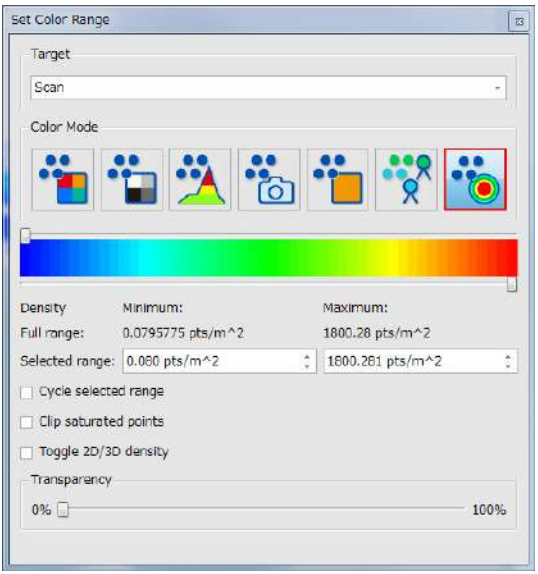
   • The display color range can be changed.

There are two ways to display <Set Color Range Window>.

- Select a run data on <Workspace Window> and select "Edit Color Range..." on the right click.
- Select a run data on <Workspace Window>, select "Edit" tab on the ribbon menu and then click "Color Range".

2. Execute one of the above ways.

   The following <Set Color Range Window> is displayed.



Set Color Range Window

3. Select "Target" as "Scan".

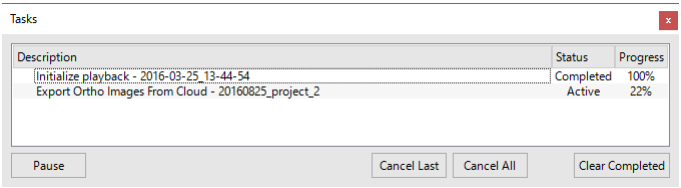
4. By clicking "Density"  in color mode, clouds are displayed in color display by density.

5. Adjust the numeric range in "Selected range".

   Also, if you check "Toggle 2D/3D density", the color range will be displayed with the density per unit volume (such as cubic meter). If it is not checked, the color range will be displayed with the density per unit area (square meters etc.).

2.5 Task Management

With <Task Window>, you can check the progress of various stacked tasks.



Task Window

Description	:	Task name and data folder name are displayed.
Status	:	Task processing status is displayed.
Progress	:	Progress of task is displayed.
[Pause]/[Resume]	:	Currently executed task is paused or the execution of paused task is resumed.

• Some types of tasks cannot be paused.

[Cancel Last]	:	The task at the end of task list is cancelled.
[Cancel All]	:	All tasks in the task list are cancelled.
[Clear Completed]	:	Completed task and cancelled task are cleared.

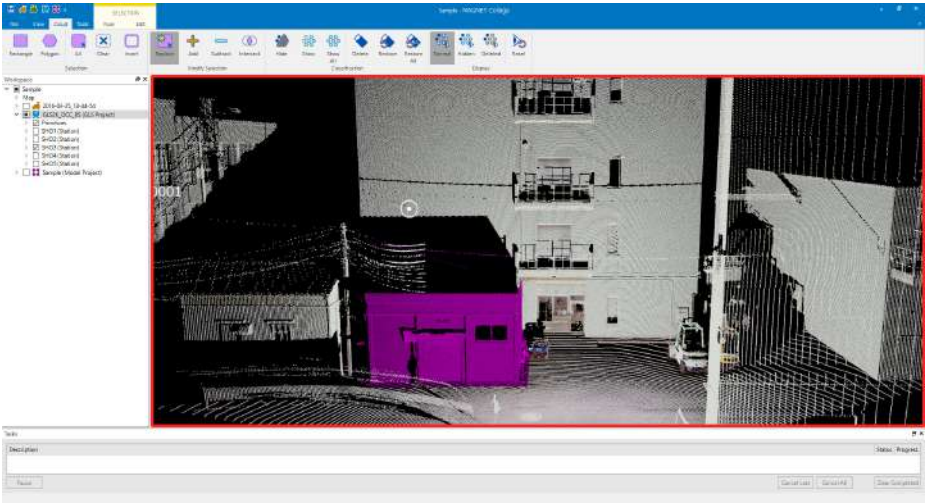
2.6 Edit of Clouds

It is possible to clear unnecessary part from clouds data or add the clouds contained in the selected area to a work space as new clouds.

- To edit clouds, the clouds data is necessary. The scan data cannot be edited.

■ Selection and edit of clouds

Display clouds on <3D View> or <Map View> and click "Cloud" tab in the ribbon menu. To select an area, select selection method and selection mode, and then click area start point and end point in order.



■ Selection methods

Four selection methods (Replace / Add / Subtract / Intersect) shown below are available.

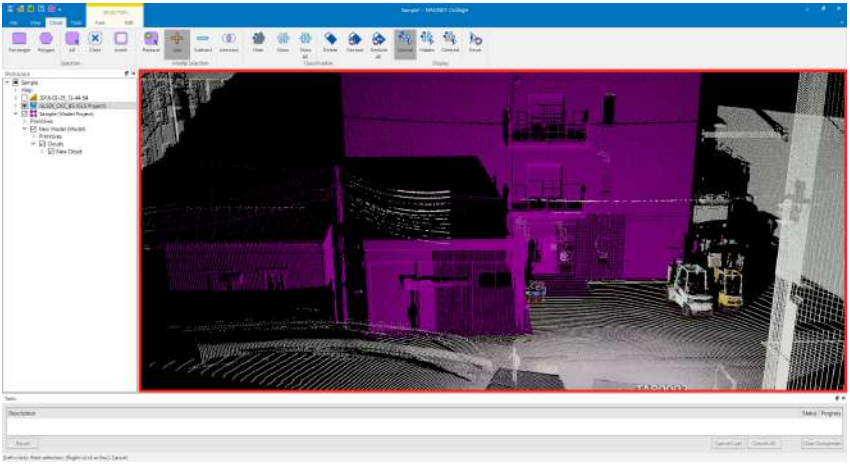
Replacement

If you click "Replace"  , the system enters replacement selection status. This is the most general

selection method. The existing selected area is released and a new area is selected. The area that is selected at the same time is only one.

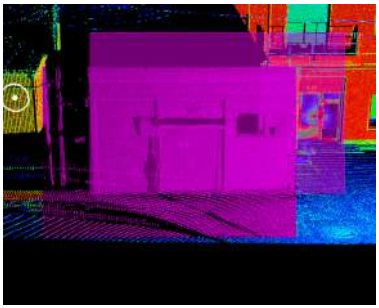
Add

If you click "Add"  , the system enters add selection status. A newly selected area is added without changing the existing selected area.

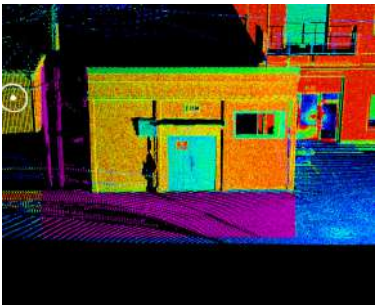


Subtract selection

If you click "Subtract"  , the system enters subtract selection status. For the existing selected area, the area overlapped with the newly selected area is subtracted from the selection.



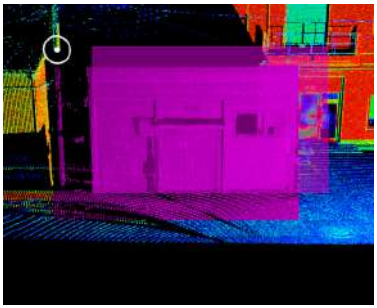
Existing selected area is overlapped with an area to be selected newly



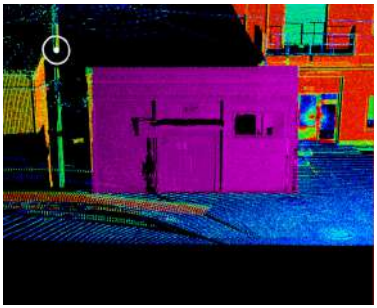
Selection of overlapped area is released by selecting a new area

Intersect

If you click "Intersect"  , the intersect is selected. The existing selected area is selected leaving only the area that is intersected with the newly selected area.



Existing selected area is overlapped with an area to be selected newly



Only overlapped area is selected by selecting a new area

All

If you click  , all displayed clouds are selected.

Clear

If you click  , all selected areas are released.

Invert

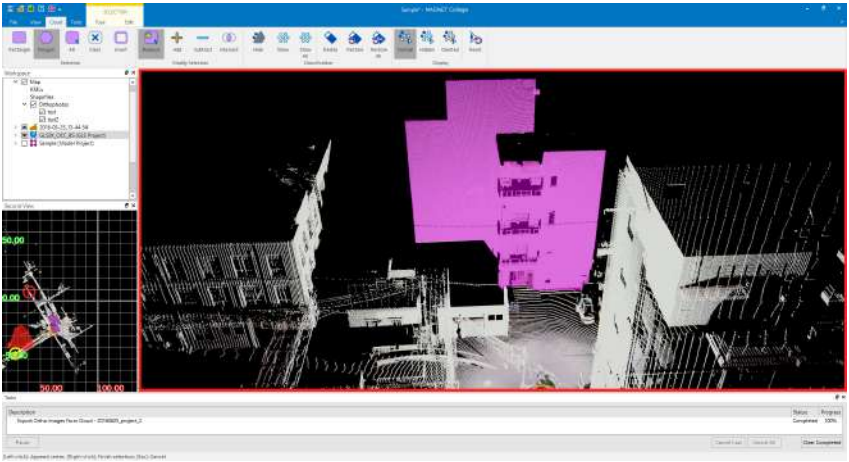
If you click  , the selected area is inverted and unselected area becomes selected status and the selected area becomes unselected status.

Selection mode

If you click "Rectangle"  , the system enters rectangle selection mode. If you click start point and end point, the area is selected.

If you click "Polygon"  , the system enters polygon selection mode. You can specify multiple vertexes.

The area is fixed by clicking the right mouse button.



■ Hide and Delete

If you select an area and click "Hide"  , the area can be hidden. If you click "Delete"  icon, the area can be deleted.

The hidden status of the area is released when the application is quitted.
The deleted status of the area is kept even after the application is quitted.

Selection of display

Normal  : The normal clouds become visible.

Hidden  : Hidden clouds become visible.

Deleted  : Deleted clouds become visible.

Reset  : The visible status is reset to standard visible status.

Edit of Hide and Delete

If you click "Hidden"  , the area in hidden status will be displayed in light blue.

If you select this area partially and click "Show"  , only the selected area can be reset to the standard status.

If you click "Show All"  , all hidden areas are reset to the standard status.

If you click "Deleted"  , the deleted area will be displayed in yellow. If you select this area

partially and click "Restore"  , only the selected area can be restored to the standard status. If you click

"Restore All"  , all deleted areas are restored to the standard status.

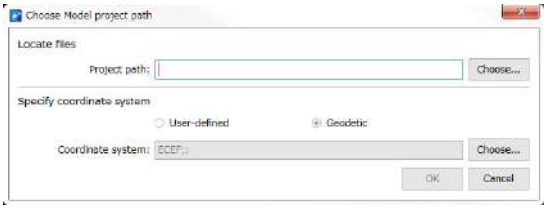
■ Reading from a file

It is possible to read clouds data from a file to add it to the work space. The read clouds will be added to the model project.

Creation of a model

Select an existing model project on <Workspace Window> or click "Create New Model Project" on <Data Menu> in "File" tab.

- The "Project Path" to be set here should be an empty folder.



Click "Create Model"  in "Fuse" tab and add a model to the model project.

Import of clouds file

Select a model to which clouds are added and click "Import clouds"  .
☞"5.1 Data Import"-"Loading the clouds data from file"

■ Creation from currently displayed clouds

It is possible to select all or partial clouds, which are currently displayed, to make new clouds. The created clouds will be added to the model project.

Selection of clouds

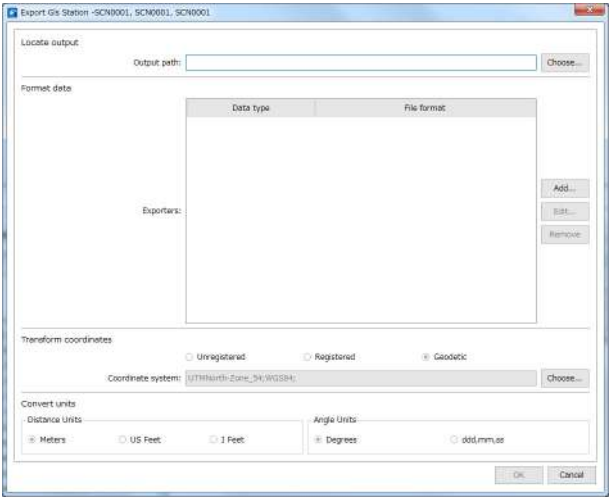
Set the target clouds to selection status using the selection function in cloud tab.

Creation of clouds in selected area

Select an existing model or create and select a model newly. Click "Create Cloud"  in "Fuse" tab.

■ Output to a file

Output the clouds data to a file. Select the target clouds on <Workspace Window> and click "Export"  in "Import/Export" group in "Edit" tab.

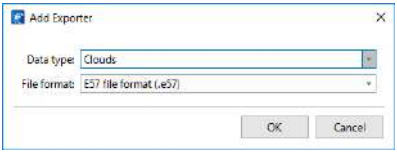


1. Set the destination to output the data.

   Set the folder to output the data in "Output path" input field.

2. Set the data format.

Add items to be output to "Exporters" list. Add items by clicking [Add...] or edit the existing items by clicking [Edit...].

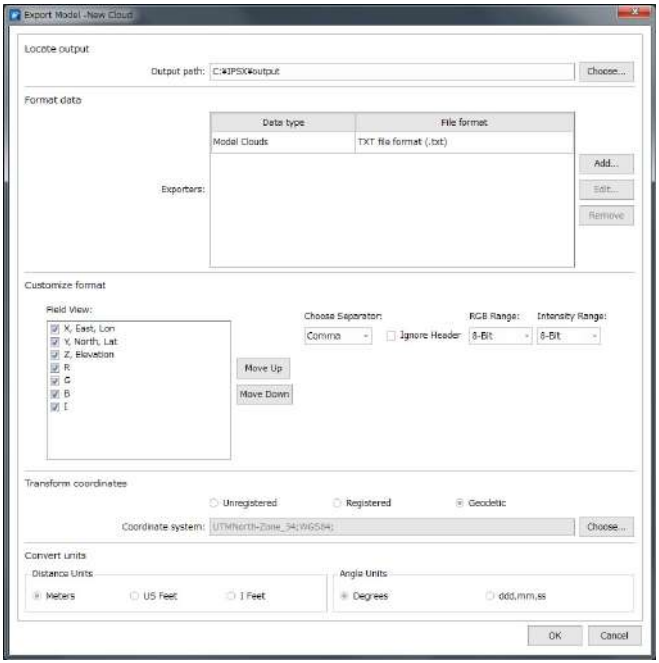


Output Items Add Dialog

Supported data formats: E57, CL3, PCD, RCS, LAS, PTS, TXT, CL3+IJ+ALG

- The file extension of the import files must be lower case.

When you output data in text format, you can customize the fields to be outputted or field order. If you select "TXT file format (.txt)" in the "File format" input field and click [OK], <Export Model - New Cloud> will be displayed.

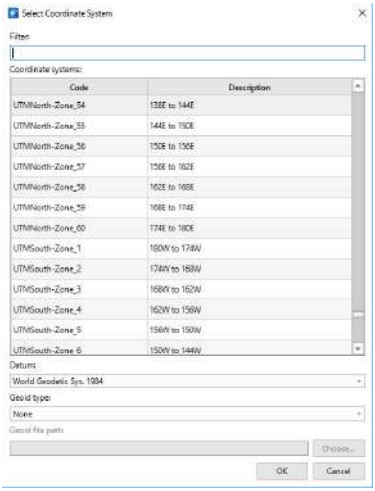


Fields and order : The checked field items will be output in order from the top. The order can be swapped by clicking [Move Up] or [Move Down].

Choose separator	:	You can choose the filed separator from the character of Comma, Tab, Space, Semicolon or Custom. When you choose the custom character, you can specify an arbitrary character.
Ignore Header	:	When this option is checked, the header line doesn't output to the file.
RGB Range	:	You can choose the range of RGB from the options of 0.0~1.0, 8 bit (0~255) or 16 bit (0~65535).
Intensity Range	:	You can choose the range of intensity from the options of 0.0~1.0, 8 bit (0~255) or 16 bit (0~65535).

3. Set the method for transforming coordinates.

In case of clouds for which geodetic is referenced, the coordinate system to be output can be selected by selecting "Global coordinate". If you click [Choose...], <Coordinate System Selection Dialog> will be displayed.



Coordinate System Selection Dialog

Filter	:	If you input a character string, the items of coordinate system list is narrowed down.
Coordinate systems	:	Click and select the target coordinate system.
Datum	:	If there are multiple datums, select the target datum.
Geoid type	:	To apply geoid, specify an already defined geoid or specify "Geoid file path" and select it from a file. If it is not necessary to apply it, specify "None".

4. Set the unit.

   Select distance unit and angular unit for the data to be output.

5. Execute the data output.

   If you click [OK], the data output will be executed.

■ Creating clouds and adjustments

Select the target model and click "Clouds" in the "Fuse" tab. <Process Cloud Dialog> will be displayed.

Process Cloud Dialog

Adjustment of density

The density of the created cloud can be adjusted. Check precision and density and click [Start].

Setting items	Description
Precision	Set the minimum digit of precision coordinate value. If it is set to 0.01m, for example, the coordinate value will be rounded out by the digit of 0.01m.
Density	Set the density for density clouds. If it is set to 0.01m, for example, the density will be adjusted so that the intervals between points become 0.01m.

Filter cloud

The noise of the created cloud can be removed.

☞ Refer to "Point cloud noise removal" in "4.2 Data Processing".

Topographic resampling

The cloud can be performed the topographic resampling. It makes a topographic surface grid with the specified ground sample distance, chooses a point by the specified surface policy and removes the other points in the grid.

Setting items	Description	
Ground Sample Distance	Specify an interval distance for a topographic surface grid.	
Surface Policy	High	Chooses the highest elevation point in the grid.
	Average	Create a point which has an average elevation in a set of points in the grid, and it will be located at centroid of the grid in a horizontal direction.
	Most central	Chooses the nearest point to the centroid of the grid in a horizontal direction.
	Low	Chooses the lowest elevation point in the grid.

Smooth cloud

Smooth cloud filter detects flat surfaces and brings noise points near to each flat surface.

Setting items	Description
Sigma	Sigma is the size of features we want to preserve. All details smaller than sigma will be considered noise and adjusted. Bigger sigma means big smoothing, zero sigma means no smoothing.
Feature preservation	Feature preservation controls the extent of edge detection. The edge sharpness will be increased from "Low"/"Medium"/"High" in order. Precise option is a little better for feature detection, Fast option is a few times faster especially for dense datasets and big Sigma.

Remove redundant points

Remove redundant points can be used to reduce number of points in the dataset by removing extra points on flat surfaces.

Setting items	Description
Density	Density is average point density on flat surfaces that we try to achieve. If density is 0.05m, then every square meter will have ~400 points.
Sigma	Sigma is the size of features we want to preserve. All details smaller than sigma will be considered noise and re-sampled.
Feature preservation	Feature preservation controls the extent of edge detection. The edge sharpness will be increased from "Low"/"Medium"/"High" in order. • This algorithm for edge detection is different from Smooth cloud, different value may need to be used.

■ TIN and DEM

- Before using below functions, it is required that a Model project has been created and point clouds have been imported in the model project previously.

  ☞For the details on Preparation of a new model project, please refer to "2.6 Edit of Clouds" and "5. EDITING POINT CLOUDS".

TIN

TIN (Triangulated Irregular Network) can be generated from selected cloud.

- It is impossible to create TIN in the following conditions:
  - importing point clouds into a model project whose coordinate system is ECEF.
  - Selecting more than one point clouds, they have different coordinate system each other.
  - The number of points in the selected point cloud is 3 or less.

Create TIN

1. Select cloud in a model project.

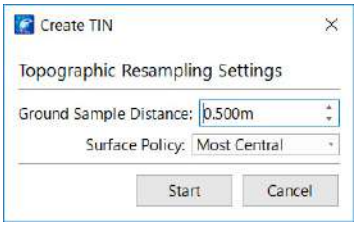
   ☞For editing clouds, please refer to "2.6 Edit of Clouds".

2. Right-click TINs folder in a model project on <Workspace Window>, then click "Create

   TIN" or click "Tool"->[TIN]  on the menu.

   <Create TIN Dialog> will be displayed.

3. Select "Ground Sample Distance" and "Surface Policy" from the drop-down list on the <Create TIN Dialog>.



Create TIN Dialog

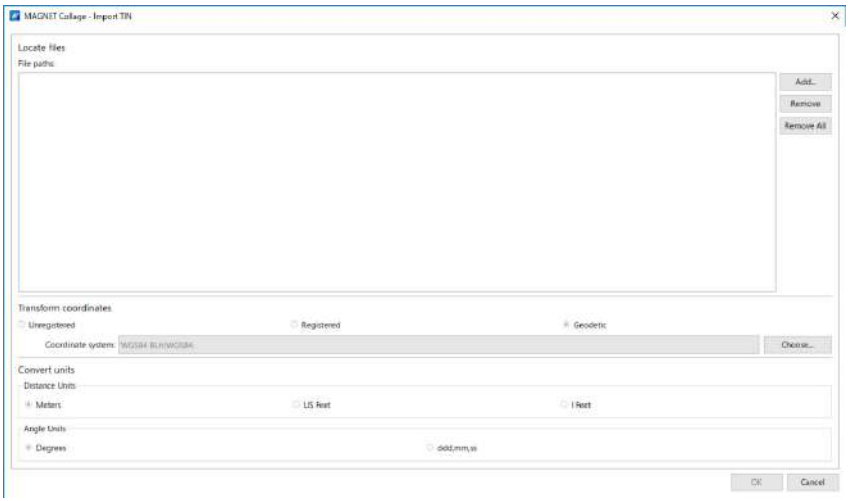
☞ For the details of each parameter, please refer to "Topographic resampling" in "2.6 Edit of Clouds".

4. Click [Start] to generate TIN.

Import TIN

1. Right-click TINs folder in a model project on <Workspace Window>, then click "Import TIN" or click "Edit"->"import TIN"  on the menu.

<Import TIN Dialog> will be displayed.



Import TIN Dialog

2. Click "Add..." and specify the TIN to be loaded.

   ☞ The TIN formats that can be loaded are shown below. If necessary, set up the coordinate system.

Table 4: File Format

Data Type	File Format
TIN	PLY (*.ply)
	Land XML (*.xml)
	Magnet XML (*.mxl)
	Topcon Tin TN3 (*.tn3)

3. When setup is completed, click [OK].

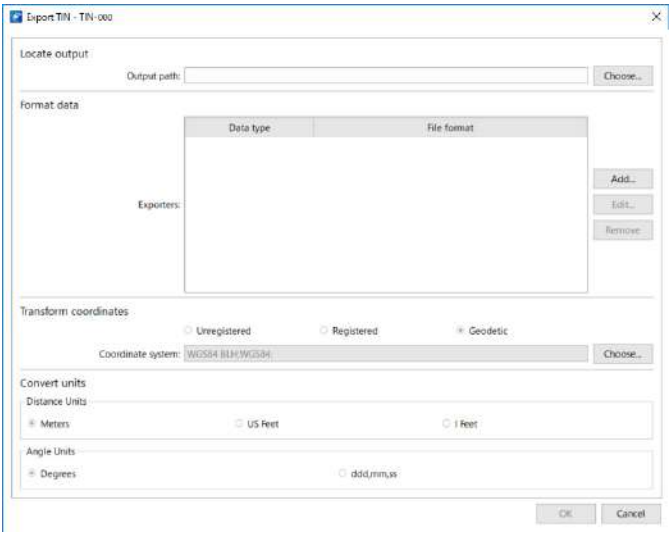
   Importing TIN will start.

Export TIN

Output the TIN to a file.

1. Select the target TIN on <Workspace Window> and click "Export"  in "Import/Export" group in "Edit" tab on the menu.

   <Export TIN Dialog> will be displayed.



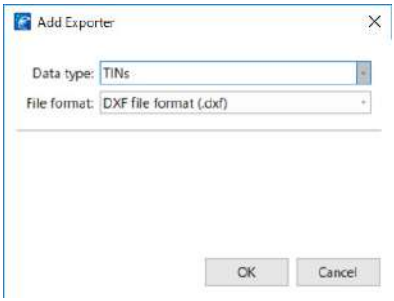
Export TIN Dialog

2. Set the destination to output the data.

   Set the folder to output the data in "Output path" input field.

3. Set the data format.

   Add items to be output to "Exporters" list. Add items by clicking [Add...] or edit the existing items by clicking [Edit...].

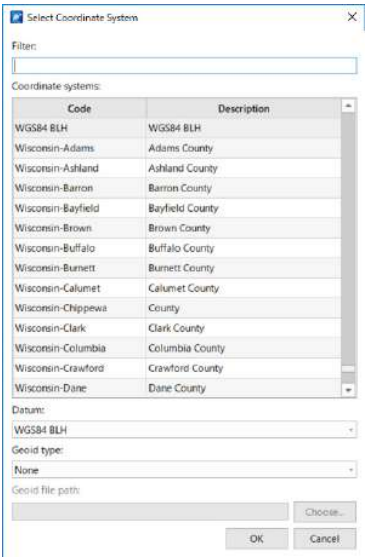


Add Exporter Dialog

Supported data formats: DXF, MAXML, PLY, TN3, LANDXML, DWG

4. Set the method for transforming coordinates.

   In case of TIN for which geodetic is referenced, the coordinate system to be output can be selected by selecting "Geodetic". If you click [Choose...], <Coordinate System Selection Dialog> will be displayed.



Coordinate System Selection Dialog

Filter : If you input a character string, the items of coordinate system list is narrowed down.
Coordinate systems: Click and select the target coordinate system.
Datum : If there are multiple datums, select the target datum.
Geoid type : To apply geoid, specify an already defined geoid or specify "Geoid file path" and select it from a file. If it is not necessary to apply it, specify "None".

5. Set the unit.

   Select distance unit and angular unit for the data to be output.

6. Execute the data output.

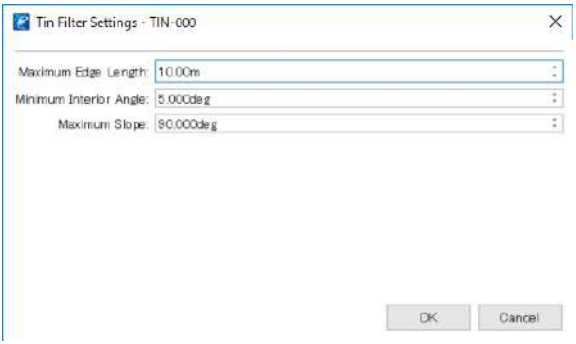
   If you click [OK], the data output will be executed.

Filter triangles

Filtering out unnecessary TINs meeting certain conditions.

1. Select a TIN item on <Workspace Window>, then right-click and select [Filter triangles] or click [Edit]-> [Filter triangles]  on the ribbon menu.

   <Tin Filter Settings Dialog> will be displayed.



Tin Filter Settings Dialog

2. Set some parameters on <TIN Filter Settings Dialog>.

   Maximum Edge Length : Remove triangles with side length exceeding this value.
   Minimum Interior Angle : Remove triangles with interior angle below this value.
   Maximum Slope : Remove triangles whose absolute slope exceeding this value.

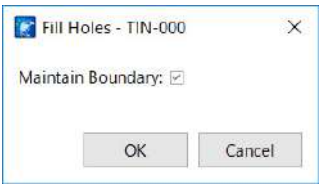
3. Click [OK] to apply "Filter triangles"

Fill Holes

Filling holes with new TIN.

1. Select a TIN item on <Workspace window>, then right-click and select [Fill Holes] or click [Edit]-> [Fill Holes]  on the ribbon menu.

   <Fill Holes Dialog> will be displayed.



Fill Holes Dialog

2. Set a parameter on <Fill Holes Dialog>.

   Maintain Boundary : Maintain the boundary of TIN area.
   In the case of the shape of TIN's boundary is concave polygon, you can prevent unnecessary TINs from being created around boundary.

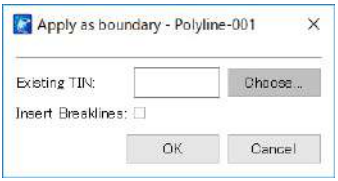
3. Click [OK].

   Fill Holes process will start.

Apply as boundary

1. Select a polyline item or polygon item in a model project on <Workspace Window>, then right-click and select [Apply as Boundary] or click [Edit]-> [Apply as Boundary]  on the ribbon menu.

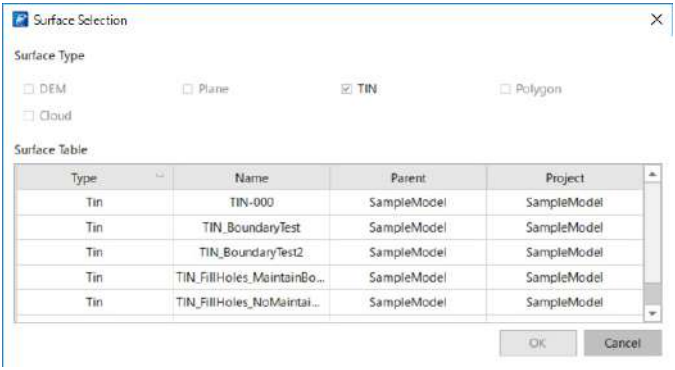
   <Apply as Boundary Dialog> will be displayed.



Apply as Boundary Dialog

2. At "Existing TIN", Click [Choose...]

   <Surface Selection Dialog> will be displayed.

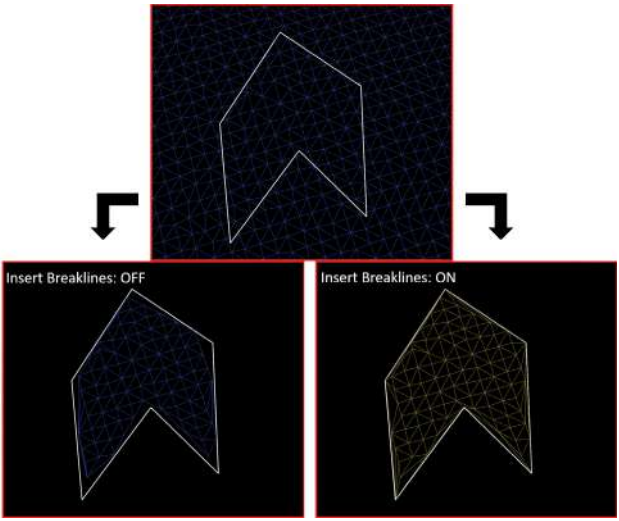


Surface Selection Dialog

3. Choose a surface from surface table, click [OK].

   Existing TIN is set.

4. When using the vertexes of a polyline or polygon as a part of TIN, please check "Insert Breaklines"



5. Click [OK].

DEM

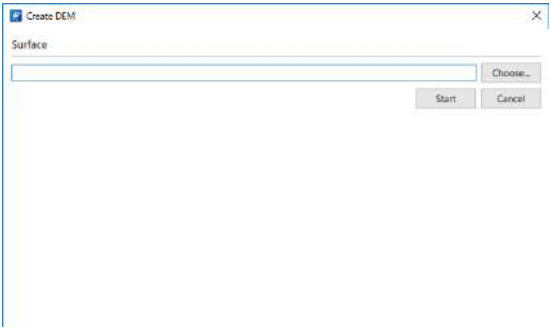
DEM (Digital Elevation Model) can be generated from point clouds, TIN, or DEM.

- It is impossible to create DEM in a model project whose coordinate system is ECEF.

Create DEM

1. Right-click DEMs folder in a model project on <Workspace Window> and then click [create DEM...] or click "Tool"->[DEM] on the menu.

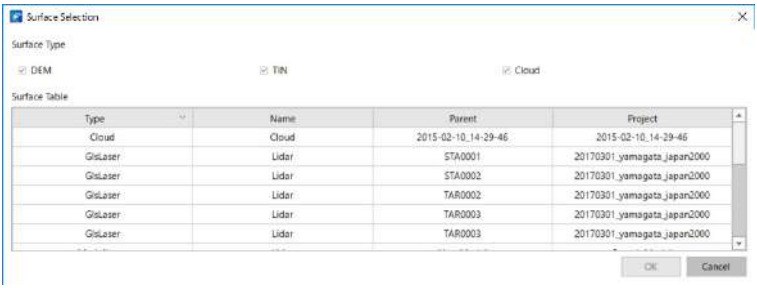
   <Create DEM Dialog> will be displayed.



Create DEM Dialog

2. Click [Choose] in "Surface".

   <Surface Selection dialog> will be displayed.

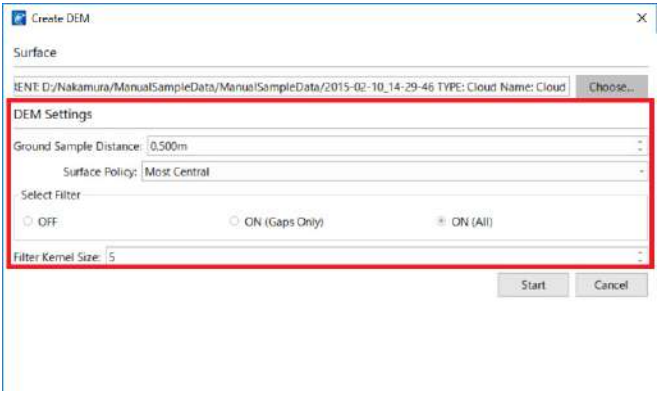


Surface Selection Dialog

3. Select a surface from Surface Table and click [OK].

   Here, surface is chosen, and "DEM Settings" will be displayed additionally on <Create DEM Dialog>.

4. Set the parameters in "DEM Settings".



Create DEM Dialog

Setting items	Description	
Filter	OFF	Doesn't apply the filter.
	ON (Gaps Only)	Applies filter only for the empty cells.
	ON (All)	Applies filter to all cells.
Filter Kernel Size	Specify the processing unit by the number of cells. Minimum value is 3.	

☞ For the details of each parameter, please refer to "Topographic resampling" in "2.6 Edit of Clouds".

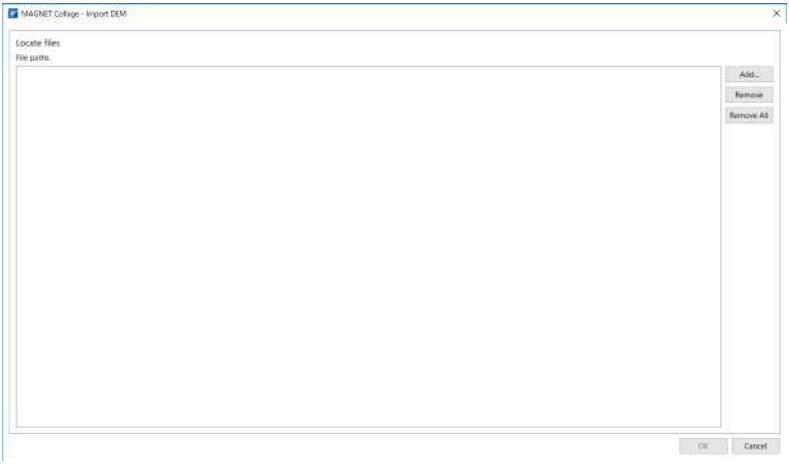
- In the case of selecting a something other than could as surface, the "Grid Sample Distance" will be only displayed.

5. Click [Start] to create DEM.

Import DEM

1. Right click DEMs folder in a model project and select "Import DEM..." or select "Import DEM"  in the "Edit" tab on the ribbon menu.

   <Import DEM Dialog> will be displayed.



Import DEM Dialog

2. Click [Add...] and specify the DEM to be loaded.

   ☞ Supported DEM data formats are as below:

Table 5: Supported DEM data formats

Data Type	Supported Data Format
DEM	DEM file format (*.tif *.asc *.txt)
	Geo TIFF File (*.tif)
	ESRI ASCII Grid File (*.asc *.txt)

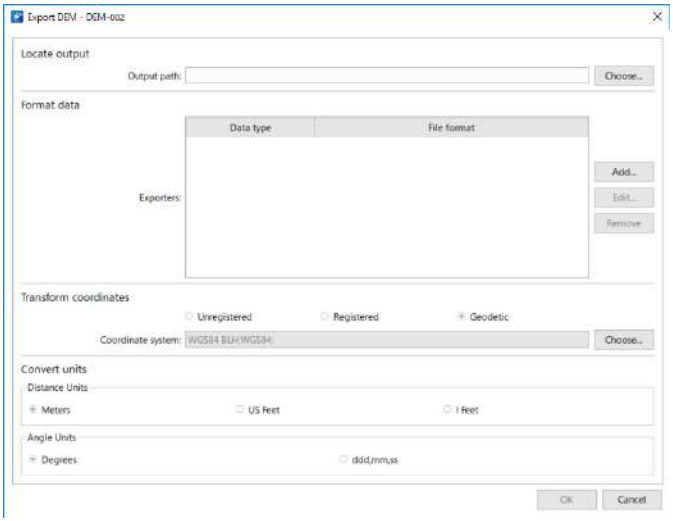
- Only supports DEM files which has correct Coordinate System information embedded in the file.

3. Click [OK] to import the DEM data.

Export DEM

1. Output the DEM to a file. Select the target DEM on <Workspace Window> and click "Export"  in "Import/Export" group in "Edit" tab on the menu.

   <Export DEM Dialog> will be displayed.



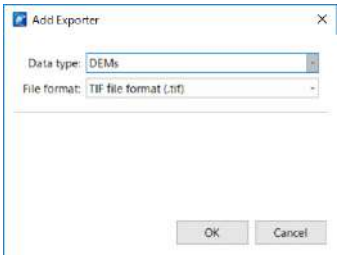
Export DEM Dialog

2. Set the destination to output the data.

   Set the folder to output the data in "Output path" input field.

3. Set the data format.

   Add items to be output to "Exporters" list. Add items by clicking [Add...] or edit the existing items by clicking [Edit...].

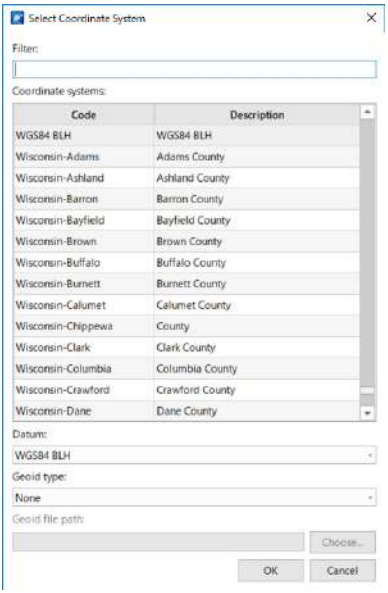


Add Export DEM Dialog

Supported data formats: ASC, TIF

4. Set the method for transforming coordinates.

   In case of DEM for which geodetic is referenced, the coordinate system to be output can be selected by selecting "Geodetic". If you click [Choose...], <Coordinate System Selection Dialog> will be displayed.



Coordinate System Selection Dialog

Filter	:	If you input a character string, the items of coordinate system list is narrowed down.
Coordinate systems	:	Click and select the target coordinate system.
Datum	:	If there are multiple datums, select the target datum.
Geoid type	:	To apply geoid, specify an already defined geoid or specify "Geoid file path" and select it from a file. If it is not necessary to apply it, specify "None".

5. Set the unit.

   Select distance unit and angular unit for the data to be output.

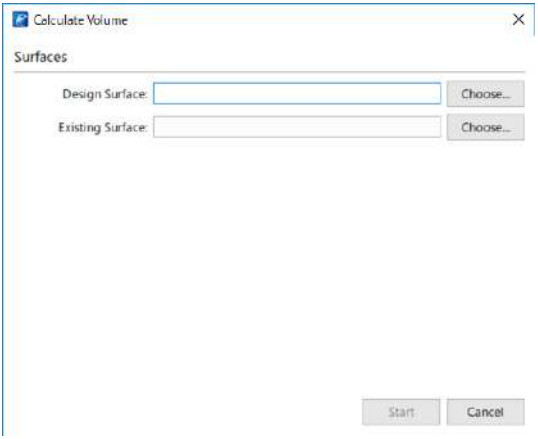
6. Execute the data output.

   If you click [OK], the data output will be executed.

■ Calculate Volume

1. Right click on the Volumes folder in a model project and select "Calculate Volume...", or select "Volume"  under the "Tools" tab.

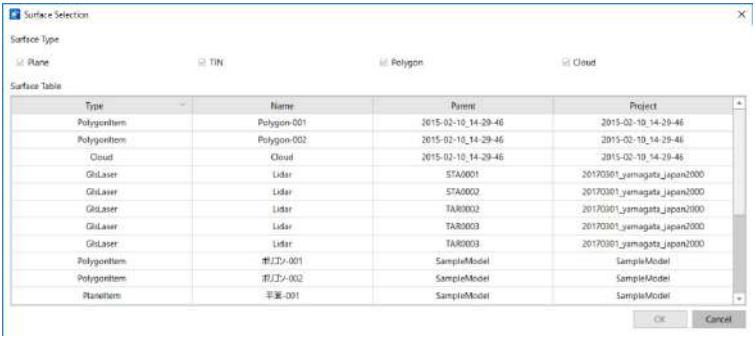
   <Calculate volumes Dialog> will be displayed.



Calculate Volumes Dialog

2. Click [Choose...] in "Design Surface", then select a surface and click [OK].

   Design surface is set.



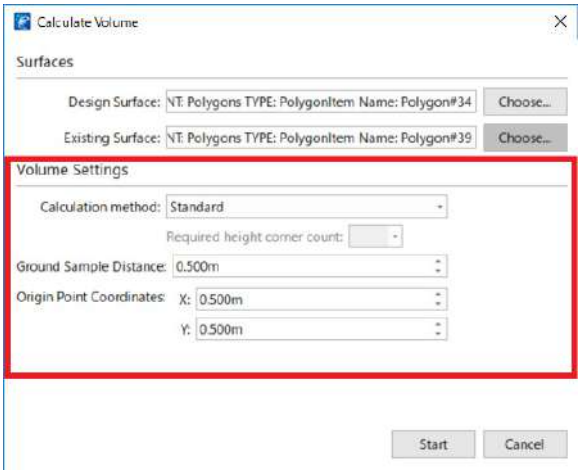
Surface Selection Dialog

3. Click [Choose...] in "Existing Surface", then select a surface and click [OK].

   Existing surface is set.

4. "Volume Settings" will be displayed additionally on the <Calculate Volumes Dialog>.

   Set some parameters in "Volume Settings"



- The required setting parameters vary depending on the surface types selected in "Design Surface" and "Existing Surface".

The setting parameters are as follows:

Common parameters

Calculation method : Standard/Custom

Standard : In the case of standard method, cells are divided into four from the size selected in "Ground Sample Distance" additionally. the volume of the only split cells which share a vertex included within the calculation range out of the four vertexes of the original cell is calculated.

Custom : Chose the minimum number of vertex of a cell needed to calculate volume. Only the cell whose vertexes are included within a calculation area exceeding the minimum number of vertex is calculated.

Required height corner count : Set the minimum number of vertexes of a cell included within the area of a design surface and an existing surface to calculate volume.

- In the case of "Custom" is selected, please select "Required height corner count"

Ground Sample Distance : Refer to "Topographic resampling" in "2.6 Edit of Clouds".

Origin Point Coordinate : Decide an origin of the grid.

Point clouds parameter

Surface Policy : Refer to "Topographic resampling" in "2.6 Edit of Clouds".

Plane parameter

Plane Setting : Bounded/Unbounded

Bounded : Calculate with the present boundary of a plane.

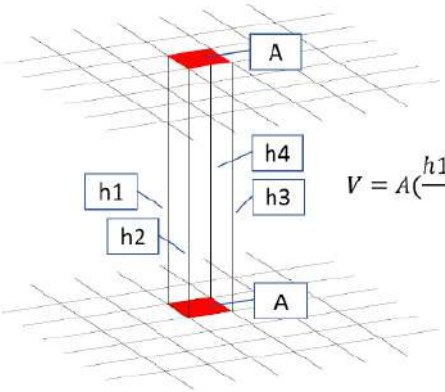
Unbounded : Calculate with considering the boundary of a plane as infinity.

5. When all parameters are set, then click [Start].

Volume calculation will start.

Note

- The method of calculating the volume of each cell

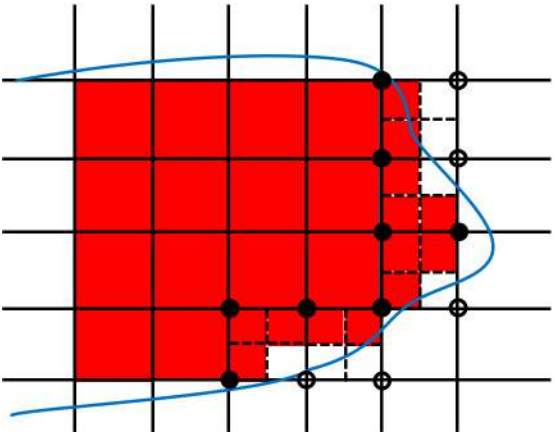


The base area A is calculated by squaring Ground Sample Distance. Then the elevation between the existing surface and the designed surface is calculated at the four corner of the cell, and the average elevation is calculated from the 4 corner elevations. the cell volume is calculated by multiplying the base area A by the average elevation.

- In the case of selecting calculation method as "Custom" and required height corner count as 3 or less, the average elevation is calculated from the elevations included within the calculation area.

Examples of Volume calculation

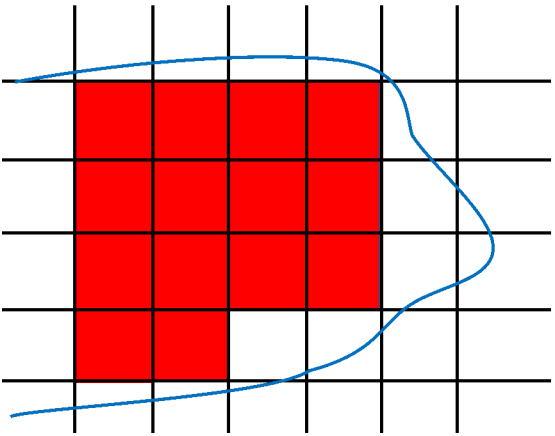
When setting a plane whose boundary is unbounded to the existing surface and the following surface (blue line) to the designed surface, the following figures are viewing the result of volume calculation just above.



Calculation Method: "Standard"

In the case of standard method, cells are divided into four from the size selected in "Ground Sample Distance" additionally (broken line). the volume of the only split cells which share a vertex (black dot) included within the calculation range out of the four vertexes of the original cell is calculated.

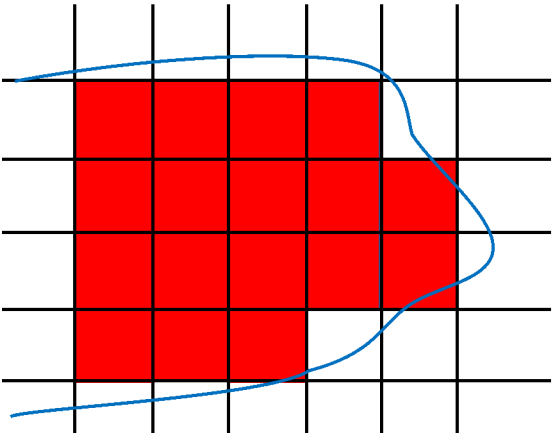
Next is the case of calculation method "Custom" and required height corner count is 4.



Calculation Method: "Custom" and Required Height Corner Count 4

The cell whose all vertexes are included within the calculation area is only calculated.

Last is the case of calculation method "Custom" and required height corner count is 3.



Calculation Method "Custom" and Required Height Corner Count 3

The cell whose more than 3 vertexes are included within the calculation area is only calculated.

Volume Report

1. Right-click a volume item on <Workspace Window>, then click "Volume Report" or click "Volume Report"  in the "Edit" tab on the ribbon menu.

   Volume report (TXT file format) will be displayed on an external text editor.

- When calculation method is standard, the half value of the ground sample distance set on < Calculate volumes dialog > will be written.

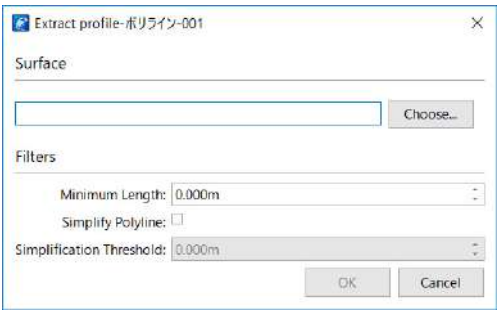
■ Cross Sections

Extract Profile

Profile can be extracted from a TIN surface along a selected polyline item.

1. Select a polyline item on <Workspace Window> and right-click on the selected polyline item, then select "Extract Profile...". or click "Profile"  in the "Edit" tab on the ribbon menu.

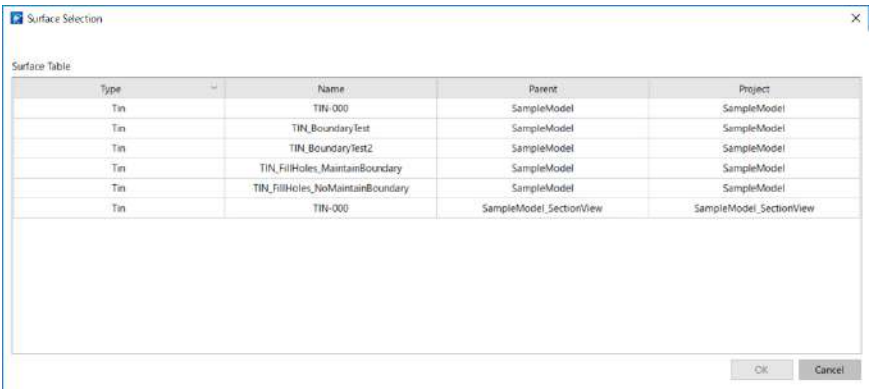
   <Extract Profile Dialog> will be displayed.



Extract Profile Dialog

2. Click [Choose...] of "Surface".

   <Surface Selection Dialog> will be displayed.



Surface Selection Dialog

3. Select a surface from the surface table, then click [OK].

4. Set filters.

Minimum Length	:	The minimum length for each profile, all profile with shorter length will be discarded.
Simplify Polyline	:	Option to simplify the extracted polyline.
Simplification Threshold	:	The maximum perpendicular distance from the polyline used to determine whether to include the vertex.

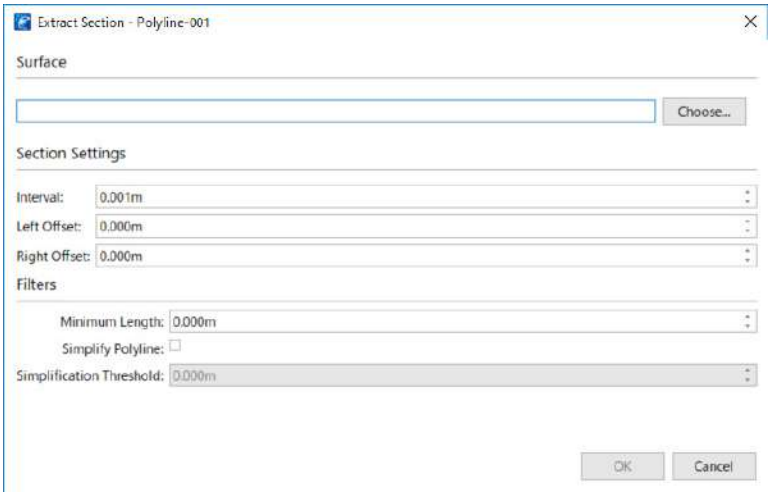
5. Click on [OK] to extract sections.

Extract Sections

Sections can be extracted from a TIN surface along a selected polyline item.

1. Right click a polyline item on <Workspace Window> and select "Extract Sections...", or click "Sections"  in the "Edit" tab on the ribbon menu.

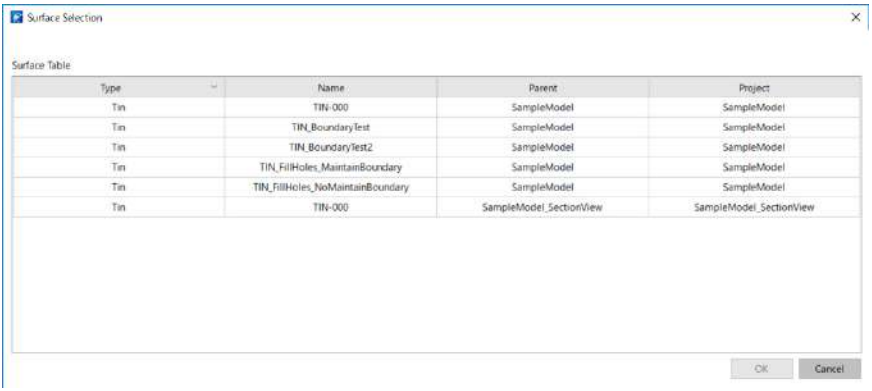
   <Extract Section Dialog> will be displayed.



Extract Section Dialog

2. Click [Choose...] of "Surface".

<Surface Selection Dialog> will be displayed.



Surface Selection Dialog

3. Select a surface from the surface table, then click [OK].

4. Set Section Settings.

Interval	:	Select the interval along polyline for the cross sections to be calculated.
Left offset	:	The maximum left perpendicular distance from the polyline which the cross section will be calculated.
Right offset	:	The maximum right perpendicular distance from the polyline which the cross section will be calculated.

5. Set Filter Settings.

Minimum Length	:	The minimum length for each section, all cross section with shorter length will be discarded.
Simplify Polyline	:	Option to simplify the extracted polyline.
Simplification Threshold	:	The maximum perpendicular distance from the polyline used to determine whether to include the vertex.

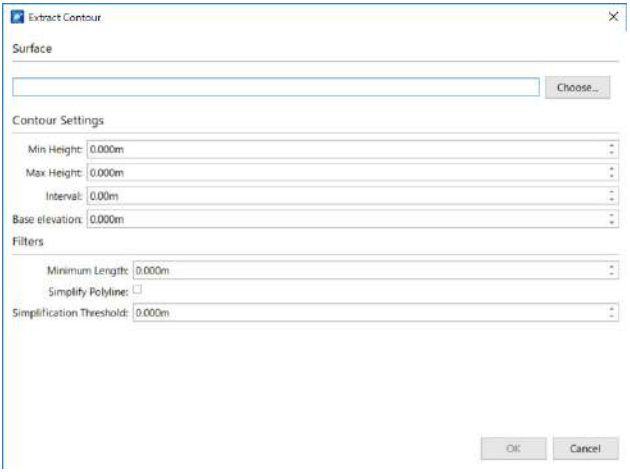
6. Click [OK] to extract sections.

Extract Contours

Contours could be extracted from a TIN surface.

1. Right click on "Contours" folder under Model Project and select "Extract Contours...,
   or select "Contours"  under the Tools tab.

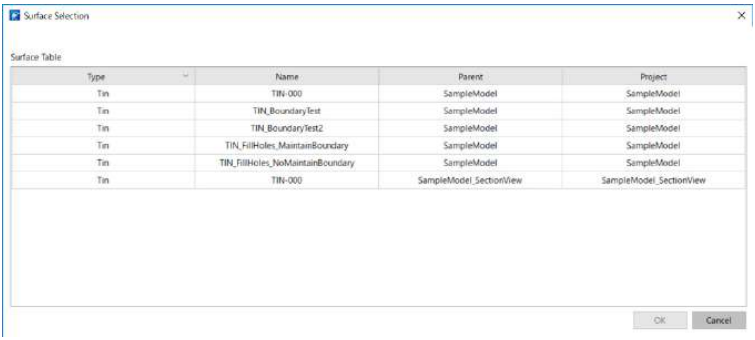
   <Extract Contour Dialog> will be displayed.



Extract Contour Dialog

2. Click [Choose...] of "Surface".

   <Surface Selection Dialog> will be displayed.



Surface Selection Dialog

3. Select a surface from the surface table, then click [OK].

4. Set Contour Settings

Min Height	:	The minimum height which the contour will be calculated.
Max Height	:	The maximum height which the contour will be calculated.
Interval	:	Select the interval for each contour to be calculated.
Base elevation	:	Select the base elevation of contours. Contours are created at a specific interval from the base elevation by lower than max height on the upper side and by higher than min height on the lower side.

5. Set Filter Settings

Minimum Length	:	The minimum length for each contour, all contours with shorter length will be discarded.
Simplify Polyline	:	Option to simplify the extracted polyline.
Simplification Threshold	:	The maximum perpendicular distance from the polyline used to determine whether to include the vertex.

6. Click [OK] to extract contours.

Export Cross Sections

Extracted cross sections could be exported as polylines to the same formats supported by CAD primitive export.

1. Right click on a profile item in a ProfilesSet in a ProfilesFolder (a section item in a SectionsSet in a SectionsFolder or a contour item in a ContoursSet in a Contours Folder) in a Model Project on the <Workspace Window> then select "Export Cross Sections...", or click "Export"  in the "Edit" tab on the ribbon menu.

   Follow the same procedure for exporting CAD primitives to export cross sections as a polyline.
   ☞ Refer to "Reading from a file and writing in a file" of "2.7 CAD Primitives".

Note

- It is possible to export all profile items (section items or contour items) by right clicking on a ProfilesSet (SectionsSet or ContoursSet).

Explode Cross Sections

Extracted cross sections could be copied to a CAD primitive polyline for further editing.

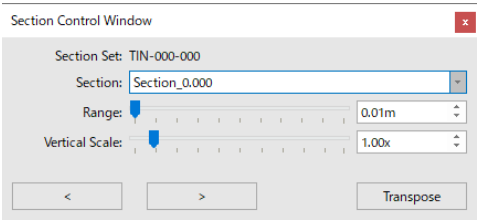
1. Right click on a profile item in a ProfilesSet in a ProfilesFolder (a section item in a SectionsSet in a SectionsFolder or a contour item in a ContoursSet in a Contours Folder) in a Model Project on the <Workspace Window> then select "Explode Cross Sections...", or click "Explode"  in the "Edit" tab on the ribbon menu.

   The selected item will be copied to the CAD primitive folder under the parent Model Project.

Section View

Showing a profile or a section along a section line. Cross section can be extracted from point clouds and TIN only.

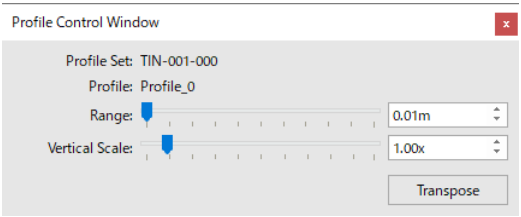
Window description



Section Control Window

Section Set	:	Showing the name of a surface whose section is extracted now.
Section	:	Showing the name of a section line along which a section is extracted. It is possible to select a section line from pull down list and also to change a section line by clicking [<], [>] located at the bottom of the window.
Range	:	Set a vertical distance from a section line. The point clouds or TIN within this range are selected and displayed as a cross section. It is possible to control the value using the slider or the spin box.
Vertical Scale	:	Change the vertical scale of the section view. The horizontal scale will not be changed. It is possible to control the value using the slider or the spin box.
[Transpose]	:	Transpose the cross section on the section view.

Window description



Profile Control Window

Profile Set	:	Showing the name of a surface whose profile is extracted now.
Profile	:	Showing the name of a section line along which a profile is extracted now.
Range	:	Set a vertical distance from a section line. The point clouds or TIN within this range are selected and displayed as a cross section. It is able to control the value using the slider or the spin box.
Vertical Scale	:	Change the vertical scale of the section view. The horizontal scale will not be changed. It is possible to control the value using the slider or the spin box.
[Transpose]	:	Transpose the cross section on the section view.

Operating procedure

1. Select a profile item in a ProfilesSet in a ProfilesFolder (or a section item in a SectionsSet in a SectionsFolder) in a Model Project on the <Workspace Window> and right-click on the item, then click [open section view] or double click the section line item.

   Or click [Edit] -> [Section view]  on the ribbon menu.

   <Section Control Window> (or <Profile Control Window>) will be displayed.
   <Main View> will be splited vertically (or horizontally), then section view will be displayed.

   Note

   • In the case of profile, when a section line has multiple vertexes, A red line will be displayed at the point according to the vertex on the section view. And a green line and orange line will be displayed at the start point and end point of a section line respectively on the section view.

   In the case of section, a red line will be displayed at the middle of a section line on the section view. And a green line and orange line will be displayed at the start point and end point of a section line respectively on the section view as profile.

   It is possible to change the position of the section view or hide it by using [Split View Vertically]  , [Split View Horizontally]  , [Hide Split View]  in the [View] tab on the ribbon menu.

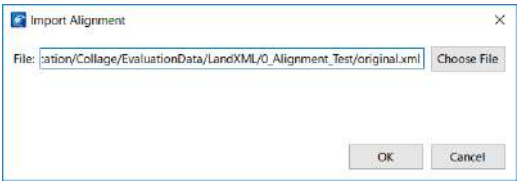
■ Alignment

Handle alignment data. It is possible to import alignments and extract sections from an alignment. Additionally, you can extract sections at the designed position in an imported alignment.

Import Alignment

1. Right click Alignments folder in a model project and select "Import Alignment... "

   <Import Alignment Dialog> will be displayed.

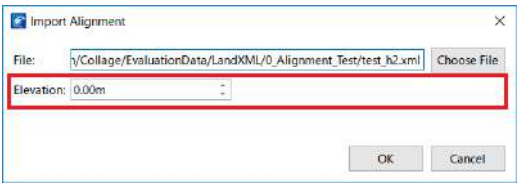


Import Alignment Dialog

2. Click [Choose File] in File, and choose an alignment data you want to import.

Data Type	File Format
Alignment	Land XML (.xml)
	Magnet XML (.mxl)

3. Set "Elevation" on the <Import Alignment Dialog>.



Import Alignment Dialog

Elevation : Set the elevation to the imported alignment item.

- The elevation will be displayed only when an imported alignment file has at least one alignment without vertical component.

4. Click [OK].

   Importing alignment will stat.

Extract profile

1. Select an alignment item on <Workspace Window> and right-click on the selected alignment item, then select "Extract Profile...". Or click "Profile"  in the "Edit" tab on the ribbon menu.

<Extract Profile Dialog> will be displayed.

☞ The subsequent operation is same as Extract Profile. Please refer to " Extract Profile".

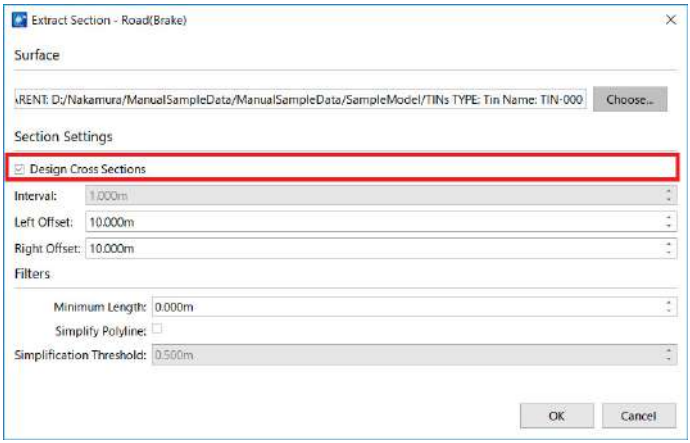
Extract section

1. Right click an alignment item on <Workspace Window> and select "Extract Sections...", or click "Sections"  in the "Edit" tab on the ribbon menu.

<Extract Section Dialog> will be displayed.

☞ The subsequence operation is same as Extract Sections besides being displayed "Design Cross Sections" checkbox on the <Extract Section Dialog> additionally. Please refer to "Extract Sections".

Design Cross Section:Using an interval designed in an alignment file previously.



Extract Section Dialog

■ Registration

The models in a model project can be registered.
Select the target model project on <Workspace Window> and click "Fuse" tab in the ribbon menu.

The registration by cloud-to-cloud and tie point, or georeferencing is available.
☞ For details on registration, refer to 3.4 Registration of Clouds.

■ Data copy between the projects

Coordinate systems take conditions as follows:

Source Coordinate system	Destination Coordinate system	Result
User-defined	User-defined	User-defined
User-defined	Geodetic	The data will be copied without coordinate transformations.
Geodetic	User-defined	The data will be copied without coordinate transformations.
Geodetic	Geodetic	The data will be copied after transforming to the destination coordinate system automatically. When copying a cloud data, the transformation will be applied to each point on the cloud. Source data must be performed geo-referencing.

2.7 CAD Primitives

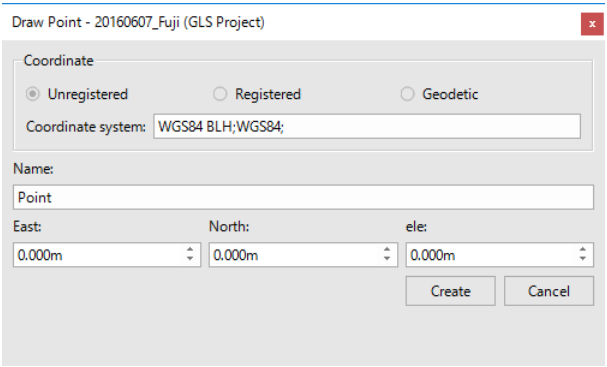
Elements of primitives can be registered in the project data. They are created by operating screens or read from a file. The registered primitives can be used as a pass point at adjustment calculation or a tie point at registration. Moreover, the created primitives can be output to a file.

■ Drawing of CAD primitives

Point

Select primitive item to which you want to add a point within the target point in <Workspace Window>

and click "Point" in "Tools" tab, <Point Drawing Window> will be displayed.



Point Drawing Window

Coordinate	:	"Unregistered", "Registered" and "Geodetic" coordinate system can be selected. For the "Registered" coordinate system, the coordinate system set on the system setting screen is reflected.
Name	:	A name can be set.
Coordinate value	:	A coordinate value can be input.
[Create] / [Cancel]	:	If you click [Create], the point will be added with the contents you input. If you click [Cancel], the point drawing mode will be quit.

A point can be also added by operating the moues. If you hold {Ctrl} key and click point of scan or clouds, the point will be added to the position. To quit the <Point Drawing Window>, press {Esc} key or hold {Ctrl} key and click the right mouse button or press . If you want to create a point at the position where there is

no point of scan or clouds, click "Interpolate" in "Tools" tab, and then carry out the work.

Distance

Select a primitive item to which you want to add distance in the target project on <Workspace Window> and click "Distance"  in "Tools" tab. The system will enter the distance drawing mode.

With the mouse operation, hold {Ctrl} key and click two points. Then, the distance between two points can be obtained. To quit the distance drawing mode, press {Esc} key or hold {Ctrl} key and click the right mouse button. If you want to set the position where there is no point of scan or clouds to start point or end point, click "Interpolate"  in "Tools" tab, and then carry out the work.

Distance Drawing Mode

Distance from two point primitives

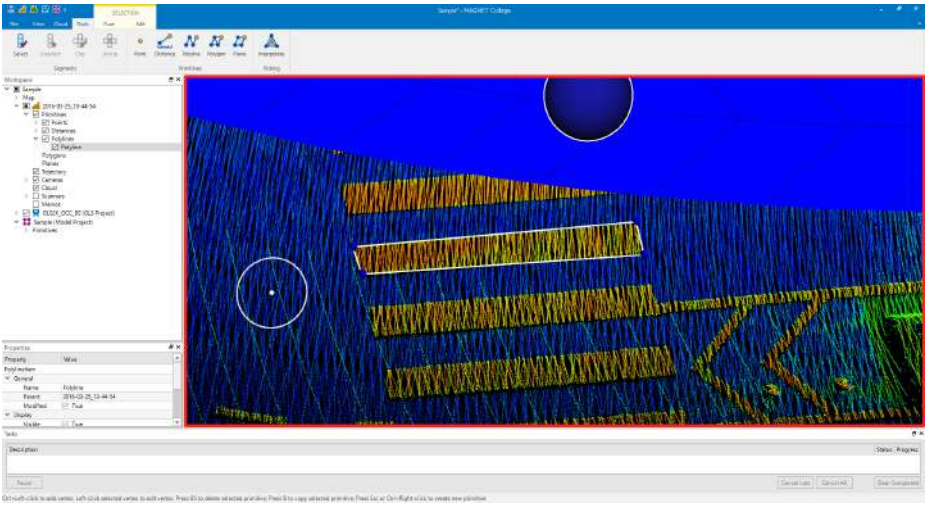
The distance between two point primitives can be calculated. Select two point primitives in the target project on <Workspace Window> and click "Distance"  in "Edit" tab.

Polyline

Select a primitive item to which you want to add a polyline in the target project in <Workspace Window> and click "Polyline"  in "Tools" tab. Then, the system will enter the polyline drawing mode.

With the mouse operation, hold {Ctrl} key and click a point. Then, a polyline setting that point to the vertex will be drawn. If you press {Esc} key or hold {Ctrl} key and click the right mouse button, the polyline will be fixed. To quit the polyline drawing mode, hold {Ctrl} key and double-click the right mouse button. To set the position where there is no point of scan or clouds to a vertex, click "Interpolate"  in "Tools" tab, and then carry out the work.

If you want to close a polyline, please check "IsClosed" in "Geometry" on the <Property Window>.



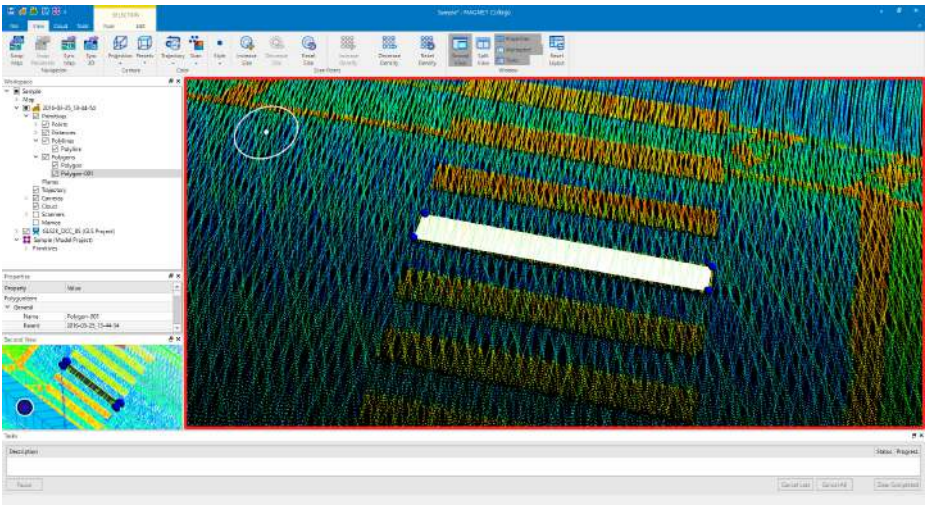
Polyline Drawing Mode

Polygon

Select a primitive item to which you want to add polygon in the target project in <Workspace Window> and click "Polygon"  in "Tools" tab. Then, the system will enter the polygon drawing mode.

With the mouse operation, hold {Ctrl} key and click a point. A polygon setting that point to a vertex will be drawn. If you press {Esc} key or hold {Ctrl} key and click the right mouse button, the polygon will be fixed. To quit the polygon drawing mode, hold {Ctrl} key and double-click the right mouse button.

If you want to set the position where there is no point of clouds to a vertex, click "Interpolate"  in "Tools" tab, and then carry out the work.



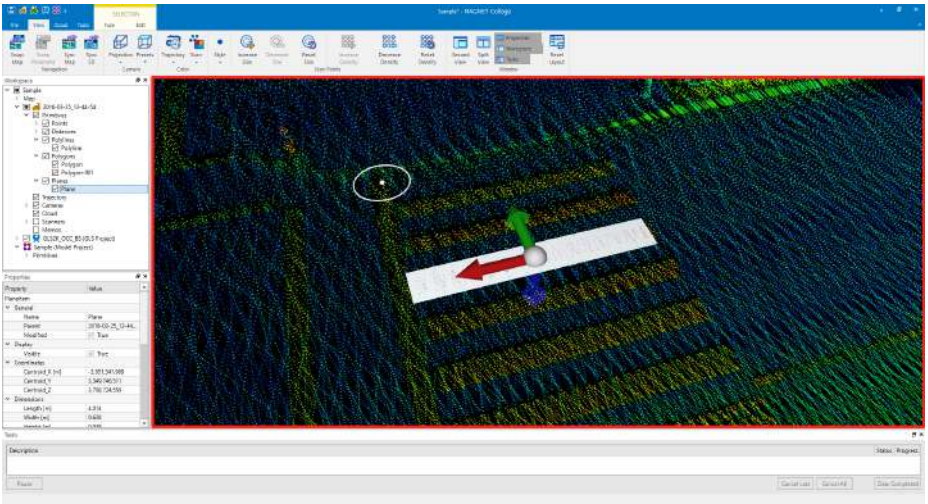
Polygon Drawing Mode

Plane

Display clouds on 3D view or map view.

Click "Rectangle"  or "Polygon"  in " Cloud" tab and select the target area.

Continuously, click "Plane"  in "Tools" tab.



Plane Drawing Mode

The details on the created plane surface will be displayed in <Property Window>.

Properties

Property	Value
PlaneItem	
General	
Name	Plane
Parent	2016-03-25_13-44-54
Modified	True
Display	
Visible	True
Coordinates	
Centroid_X [m]	-3,951,341.969
Centroid_Y	3,349,746.511
Centroid_Z	3,708,724.559
Dimensions	
Length [m]	4.014
Width [m]	0.638
Height [m]	0.049
Vectors	
FirstEigenVector_X	0.774
FirstEigenVector_Y	0.501
FirstEigenVector_Z	0.387
SecondEigenVector_X	0.090
SecondEigenVector_Y	-0.692
SecondEigenVector_Z	0.717
Normal_X	0.627
Normal_Y	-0.520
Normal_Z	-0.580
Statistics	
RMS [m]	0.011
MaximumDistance [m]	0.024
PointsCloserThanRms	77%
MeanDistance [m]	0.009
TotalNumberOfPoints	988
Area [m^2]	2.560
Volume [m^3]	0.124

Fit Plane to Picks

Create a fitting plane by three or more picked points on clouds or scans.

Click "Fit Plane to Picks"  in "Tools" tab. Next hold {Ctrl} key and click the left mouse button on a point of a cloud or scans. After picking enough points, click right mouse button with {Ctrl} key to add the plane to the workspace, or press {Esc} key to cancel the plane.

Elevated Plane

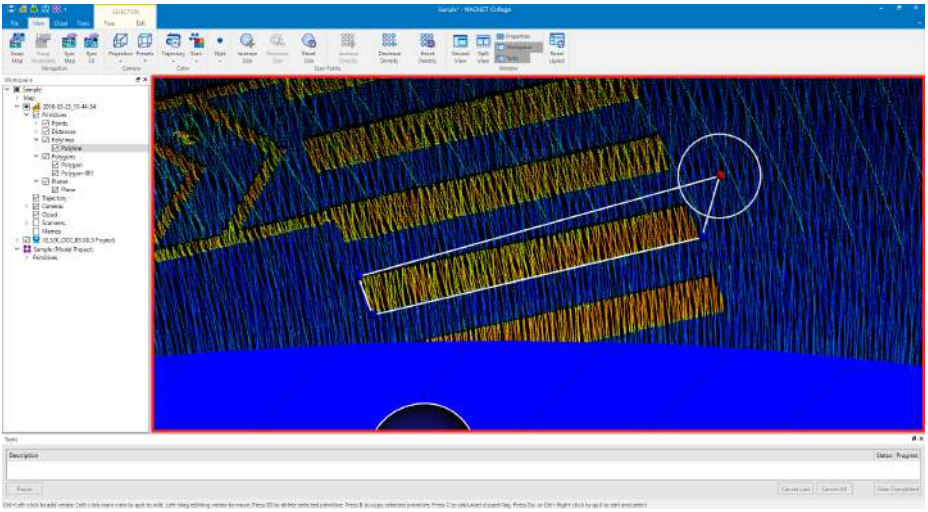
Create a plane at specified elevation. When the project was made with geodetic coordinate system, if geoid is applied, elevation will be specified, otherwise ellipsoidal height will be specified. When the project was made with user-defined coordinate system, local Z value will be specified.

Click "Elevated Plane"  in "Tools" tab. Input elevation dialog box will appear, the plane will be created at specified elevation.

■ Edit of CAD primitives

Edit of vertex

The created point, distance, polyline, and polygon can be edited by moving the vertex. If you select the target primitive in <Workspace Window>, the vertex will be displayed as a blue ball. If you place the mouse cursor on this blue ball, it will be displayed in yellow. If you click the yellow vertex, it will be displayed in red. The vertex displayed in red can be moved by dragging it by mouse. At this time, if you hold {Ctrl} key and click the right mouse button, a vertex can be inserted. A vertex that is displayed in red can be cleared by pressing {Backspace} key.



Copy and paste of polygon and polyline

Polygon and polyline can be copied and pasted.

1. Select the target polyline or polygon.
2. Click "Copy" 

in "Edit" tab.
3. Hold {Ctrl} key and click the right mouse button at the position you want to paste it.

   The copied item will be pasted.

■ Reading from a file and writing in a file

CAD primitives can be read from a file or written in a file.
Select the target primitive item in <Workspace Window>, and then click "Import"  or "Export"  in "Fuse" tab.

CAD Primitives Import Dialog

1. Set the location to import or export the file.

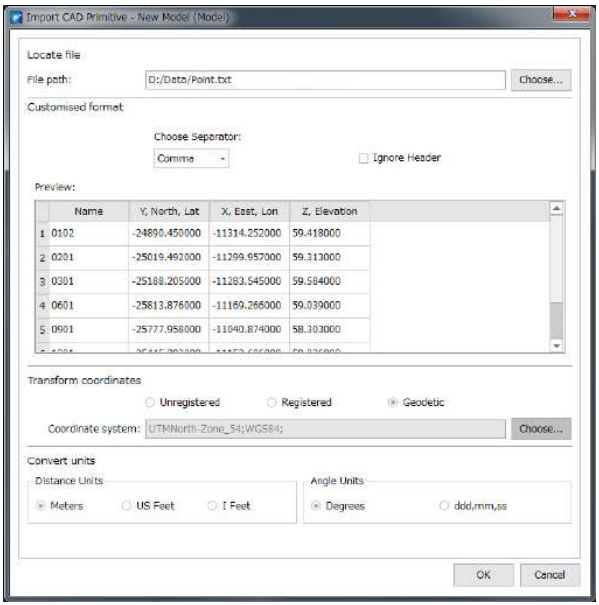
   Set the file to import or export in the "File path" input field.

   The file formats shown below are supported.

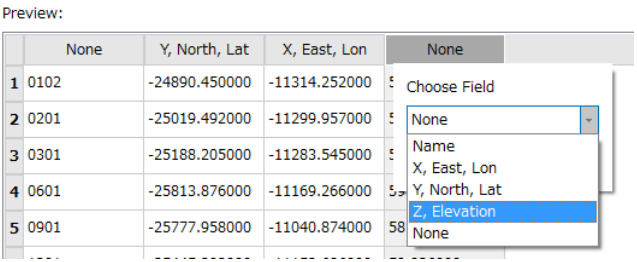
   AutoCAD 2000 Drawing (*.dwg)
   AutoCAD DXF (*.dxf)
   LANDXML (*.xml)
   MAGNETXML (*.mxl)
   ESRI Shape (*.shp)
   Points: name,n,e,z (*.csv)
   Points: name,e,n,z (*.csv)
   Choose your format from UI (*.txt)

   Custom text format:
   Select "Choose your format from UI (*.txt)" on <Select file to import Dialog> and click [Open].

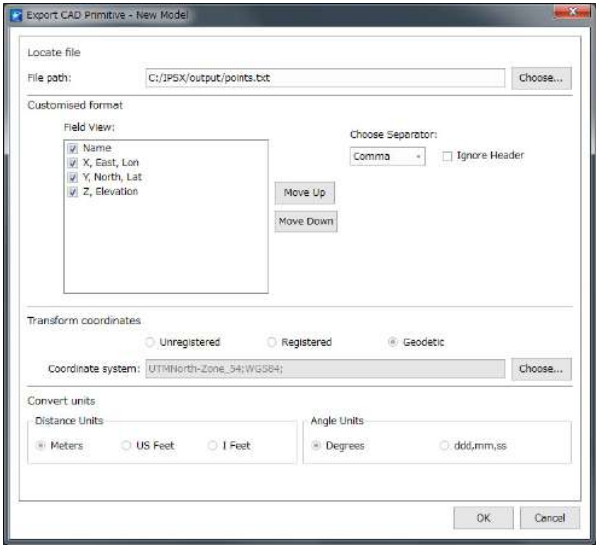
- Primitive type of distances and planes are not supported to export in any data format. Polylines are only supported in dxf, dwg, xml, mxl or shp, polygons are only supported in xml, mxl or shp.
- Supported LANDXML versions are 1.0, 1.2 or 2.0.
- The importing data in LANDXML or MAGNETXML can only process 3D coordinate system data.



Choose Separator	:	You can choose the filed separator from the character of Comma, Tab, Space, Semicolon or Custom. When you choose the custom character, you can specify an arbitrary character.
Ignore Header	:	When the file you want to import has a header line, this option should be checked to ignore the header.
Specify the field data	:	When clicking a column on the fixed row, <Choose Filed List> will appear. You can choose the appropriate field.



When you export data in text format, you can choose fields to be exported or field order.



Fields and order	:	The checked field items will be output in order from the top. The order can be swapped by clicking [Move Up] or [Move Down].
Choose Separator	:	You can choose the filed separator from the character of Comma, Tab, Space, Semicolon or Custom. When you choose the custom character, you can specify an arbitrary character.
Ignore Header	:	When this option is checked, the header line doesn't output to the file.

2. Set the method for transforming coordinates.

When importing CAD primitives:

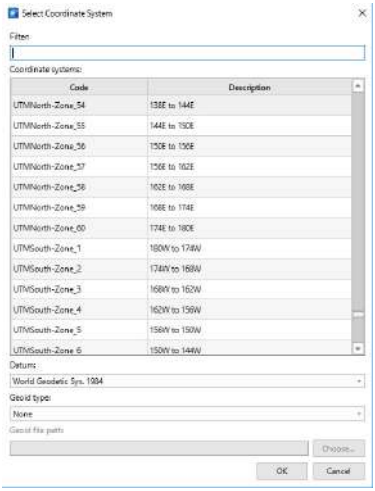
Coordinate System	Selected Item	Descriptions
Unregistered	- Station - Model	Source CAD primitive data will be imported as a station coordinate system which takes an instrument point as origin. Coordinate transformation is not performed during importing. Imported CAD primitives are added under the station item. In the case of a model item, it is same.
	- Station set - Model set	Source CAD primitive data will be imported as a station set coordinate system which was determined by registrations or geo-referencing for the station set. Coordinate transformation is not performed during importing. Imported CAD primitives are added under the station set item. In the case of a model set item, it is same.
	Project	Source CAD primitive data will be imported as a project coordinate system which was specified at creating the project. Coordinate transformation is not performed during importing. Imported CAD primitives are added under the project item.
Registered	- Station - Model	Source CAD primitive data will be imported as a registered coordinate system which was performed registrations for this station. CAD primitives will be stored under the station item after converting to the station coordinate system. In the case of a model item, it is same.
	- Station set - Model set	Source CAD primitive data will be imported as a registered coordinate system which was performed registrations for this station set. CAD primitives will be stored under the station set item after converting to the station set coordinate system. In the case of a model set item, it is same.
	Project	Source CAD primitive data will be imported as a project coordinate system which was specified at creating the project. Coordinate transformation is not performed during importing. Imported CAD primitives are added under the project item.

Geodetic	- Station - Model	Source CAD primitive data will be imported as a specified coordinate system. CAD primitives will be stored under the station item after converting to the station coordinate system. In the case of a model item, it is same.
	- Station set - Model set	Source CAD primitive data will be imported as a specified coordinate system. CAD primitives will be stored under the station set item after converting to the station set coordinate system. In the case of a model set item, it is same.
	Project	Source CAD primitive data will be imported as a specified coordinate system. CAD primitives will be stored under the project item, coordinate transformations will not be performed for the project item.

When exporting CAD primitives:

Coordinate System	Selected Item	Descriptions
Unregistered	- Station - Model	CAD primitives will be exported in the station coordinate system which takes an instrument point as origin. In the case of a model item, it is same.
	- Station set - Model set	CAD primitives will be exported in the station set coordinate system. In the case of a model item, it is same.
	Project	CAD primitives will be exported in the project coordinate system.
Registered	- Station - Model	CAD primitives will be transformed in the registered station coordinate system and exported to a file. In the case of a model item, it is same.
	- Station set - Model set	CAD primitives will be transformed in the registered station set coordinate system and exported to a file. In the case of a model set item, it is same.
	Project	CAD primitives will be exported in the project coordinate system.
Geodetic	- Station - Model	CAD primitives will be transformed to a specified coordinate system and exported to a file.
	- Station set - Model set	CAD primitives will be transformed to a specified coordinate system and exported to a file.
	Project	CAD primitives will be transformed to a specified coordinate system and exported to a file.

If you click [Choose...], <Coordinate System Selection Dialog> will be displayed.



Coordinate System Selection Dialog

Filter	:	If you input a character string, the items of coordinate system list is narrowed down.
Coordinate systems	:	Click and select the target coordinate system.
Datum	:	If there are multiple datums, select the target datum.
Geoid type	:	To apply geoid, specify an already defined geoid or specify "Geoid file path" and select it from a file. If it is not necessary to apply it, specify "None".

3. Set the unit.

   Select distance unit and angular unit for the data to be output.

4. Execute the import or export of files.

   If you click [OK], import or export the file will be executed.

2.8 Map Object

KML data, shape file, or ortho image can be allocated in the background of <Map View>.

■ KML Import

Select KML item under the map in <Workspace Window> and click "Add KML" 

in "Edit" tab and select a file to be read.

■ Reading of a shape file

Select SHP item under the map in <Workspace Window> and click "Add Shape file" 

in "Edit" tab and select a file to be read.

■ Reading of an ortho image

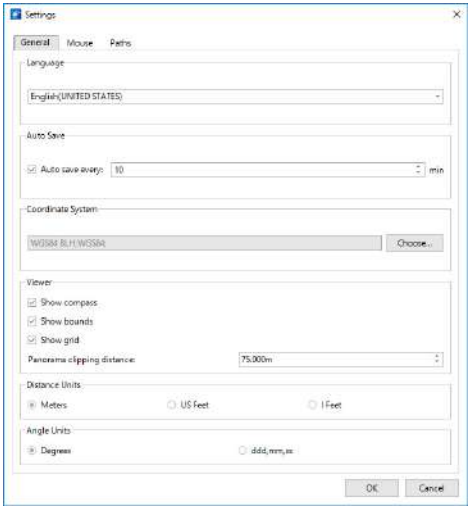
Select an ortho image under the map in <Workspace Window> and click "Add Orthophoto"  in "Edit" tab and select a file to be read. The ortho images in GeoTiff format are supported.

Map View Loaded Orthophotos

2.9 Application Setting

Here explains the setting items for the application.

■ General setting



Application Settings Dialog

Language

Select a display language. English or Japanese is selectable.

Auto Save

Set Auto-Save and Auto-Save interval. If this is checked, Auto-Save is applied to Workspace. Please note that Auto-Save will be temporarily suspended if the workspace has never been previously saved to a file.

Coordinate System

Set the standard coordinate system. It is applied to <Property Window> and trajectory information.

Viewer

Show compass	: Whether a compass is displayed on <3D View>
Show boundary	: Whether a boundary line is displayed when clouds are displayed
Show grid	: Whether grids are displayed on <Map View>
Panorama clipping distance	: Range of scan and clouds to be displayed on <Panorama View>

Distance Units

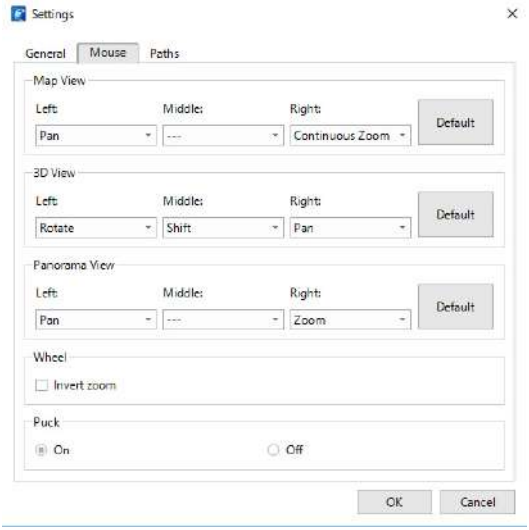
Set the standard distance unit. It is applied to <Property Window> and items of trajectory information.

Angle Units

Set the standard angular unit. It is applied to <Property Window> and items of trajectory information.

■ Mouse setting

Set how the mouse buttons move on <Map View>, <3D View>, and <Panorama View>.



Mouse Setting Screen

For right button, left button, and wheel button of mouse, following options can be set.

Map View	: translation, resizing, continuous resizing
3D View	: translation, shift, rotation
Panorama View	: translation, resizing

Wheel setting

Set the rotating direction of wheel at resizing.

Pack setting

Set whether a pack is displayed as mouse cursor on <3D View> and <Panorama View>.

■ Data path setting

Set the standard path of a data file. The data path can be set for the data shown below.

Work space, IP-S run data, RD-M run data, GLS project data, model project data, map data

2.10 Menu Commands

■ File tab

Workspace

Function	Description
New Workspace 	A work space is newly created.
Open Workspace... 	An existing work space is opened.
Open Recent Workspace 	A recently opened work space can be selected from the list and opened.
Save Workspace 	A work space is overwritten and saved.
Save Workspace As... 	A work space with a new name.
Close Workspace 	A work space is closed and a new work space status is created.

- When you create a new workspace, please save the workspace immediately. If you don't save the workspace as any name, auto-save function for workspace will not be enabled . (☞"2.9 Application Setting"-" Auto Save")

Data

Function		Description
IP-S3	Add Existing IP-S3 Run 	IP-S3 run data is added to a work space.
	Import IP-S2 Run 	IP-S2 run data is imported in a work space.
RD-M	Add Existing RD-M1 Run 	RD-M1 run data is added to a work space.

GLS	Add Existing GLS Project 	GLS project that is created by Collage is added to workspace. If you specify a GLS raw data directly, Collage will edit the raw data during GLS data processing. In this case, you can avoid a risk of the raw data modification by making a copy of the raw data to a local disk and adding the copy to workspace.
	Create New GLS Project 	GLS project is newly created and added to a workspace. In this case, a copy of GLS data is imported to GLS project.
Model	Add Existing Model Project 	Model project that is created by Collage is added to workspace.
	Create New Model Project 	Model project is newly created and added to a workspace.

Export

Function	Description
Export Orthophotos from clouds 	Export the orthophotos from clouds to a file

Help

Function	Description
About 	Confirms the version information of MAGNET Collage
Read Manual 	Displays the instruction manual of MAGNET Collage
Read RD-M Manual 	Displays the instruction manual of RD-M

- The instruction manual is provided in pdf format. To display the instruction manual, the software for displaying pdf document is separately required.

Settings

The <Application Settings Dialog> opens.

License

The <License Settings Dialog> opens.

■ View tab

Function	Description	
Swap Map 	Switches Map View and 3D View.	
Swap Panorama 	Switches Panorama View and 3D View.	
Sync Map 	Synchronizes Map View display range with a 3D View display range.	
Sync 3D 	Synchronizes 3D View display range with a Map View display range.	
Projection	Select a projection of a 3D View.	
	Perspective Projection 	Displays in perspective projection.
	Parallel Projection 	Displays in parallel projection.
Presets	Select a projection surface of a 3D View.	
	Top View 	Displays the top view.
	Bottom View 	Displays the bottom view.
	Front View 	Displays the front view.
	Back View 	Displays the back view.
	Right View 	Displays the right side view.
	Left View 	Displays the left side view.

Trajectory	Setting the display color of the trajectory	
	Velocity 	Objects are displayed in different colors depending on speed.
	Elevation 	Objects are displayed in different colors depending on height.
	Position Sigma 	Objects are displayed in different colors depending on position sigma.
	Flat 	Objects are displayed in a single color.
Scan	Setting the display color of the scan and the cloud	
	Intensity 	Objects are displayed in different colors depending on intensity.
	Grayscale 	Objects are displayed in gray scale.
	Image 	Color-coded point clouds are displayed.
	Elevation 	Objects are displayed in different colors depending on height.
	Flat 	Objects are displayed in a single color.
	Station 	Objects are displayed in different colors depending on station.
	Density 	Objects are displayed in different colors depending on density.

Style	Setting the point display style for scan and cloud	
	Point	Display using points. There is no difference in display size depending on distance.
	Circle	Display using circles. There is difference in display size depending on distance.
	Sphere	Display using spheres. Stereoscopic shade is given in addition to difference in size depending on distance.
Increase Size	The size of displayed point is enlarged.	
Decrease Size	The size of displayed point is minified.	
Reset Size	The size of displayed point is reset to the specified value.	
Increase Density	The displayed point density is increased.	
Decrease Density	The displayed point density is decreased.	
Reset Density	The displayed point density is reset to the specified value.	
Second View	Switch between show/hide <Second View>	
Split View	The view is split or restored.	
Properties	Switch between show/hide <Properties Window>	
Workspace	Switch between show/hide <Workspace Window>	

Tasks 	Switch between show/hide <Tasks Window>	
Reset Layout 	Returns the layout of main window to the initial state.	
DEM	Setting the display color of the DEM.	
	Elevation 	Objects are displayed in different colors depending on height.
	Flat 	Objects are displayed in single color.
TIN	Setting the display color of the TIN.	
	Elevation 	Objects are displayed in different colors depending on height.
	Flat 	Objects are displayed in single color.
Volume	Setting the display color of the Volume.	
	Interpolate 	Objects are displayed in gradient color depending on volume or
	Binary 	Objects are displayed in binary color.
Surface style	Setting the surface display style for DEM and TIN.	
	Flat 	Display using triangles.
	Smooth 	Display using triangles, but shadow is more smooth compared with
	Wireframe 	Display using wireframe.

■ Cloud tab

Function	Description
Polygon 	The selection mode of point clouds is set to/released from polygon selection mode.
All 	All displayed point clouds are selected.
Clear 	The selection of all selected areas is released.
Invert 	The selection is inverted. The selected area becomes unselected and unselected area becomes selected.
Replace 	The existing selected area is released and a new area is selected.
Add 	The existing selected area is left and a new selected area is added.
Subtract 	The section to be released from existing selected area is selected.
Intersect 	The section that intersects the existing selected area is left as selected area.
Hide 	The selected area is hidden.
Show 	The hidden area is shown.
Show All 	All hidden areas are shown.
Delete 	The selected area is deleted.
Restore 	The deleted area is restored to the standard status.
Restore All 	All deleted areas are restored to the standard status.

Normal		The clouds in standard status are displayed.
Hidden		The hidden clouds are displayed.
Deleted		The deleted clouds are displayed.
Reset		The display status is reset to the standard status.

■ Tools tab

Function	Description
Select 	Segments are enabled.
Unselect 	Segments are released.
Clip 	The selected segment is clipped.
Unclip 	The clip is released.
Point 	The mode is set to point drawing mode.
Distance 	The mode is set to distance drawing mode.
Polyline 	The mode is set to polyline drawing mode.
Polygon 	The mode is set to polygon drawing mode.
Plane 	The selected area is fitted by plane.
Fit Plane to Picks 	Create a fitting plane by three or more picked points on clouds or scans.
Elevated Plane 	Create a plane at specified elevation.
Interpolate 	Pointing of an area where there is no point by mouse cursor is enabled.

3. GLS DATA PROCESSING

This chapter describes processing of data captured by the TOPCON ground laser scanner system (GLS-1000 / GLS-1500 / GLS-2000).

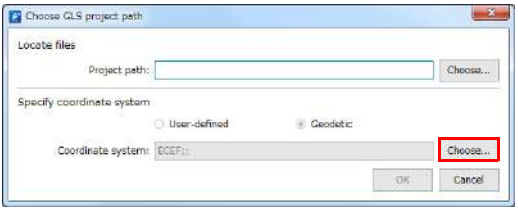
3.1 Data Import

When you add a GLS or Model project into the workspace, you have to specify the coordinate system of the project.

(1) Adding a new project

The coordinate system of the project will be selected from "User-defined" and "Geodetic" according to the coordinate system of the data.

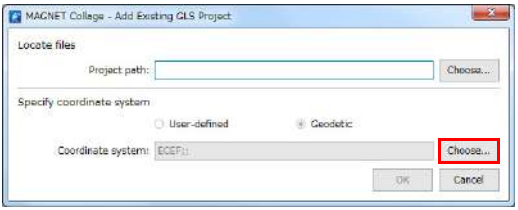
Coordinate system	Description
User-defined	This option is usually specified when the data will not be registered to geodetic coordinate systems such as indoors measurement data, structure measurement data. It will also be used for a construction coordinate system data.
Geodetic	This option is usually specified when the data will be registered to geodetic coordinate systems. It will be used when you perform registration with ground control points.



When you click [Choose...] of "Coordinate system", <Select Coordinate System Dialog> will be displayed. You can choose the target coordinate system on the dialog.

(2) Adding an existing project

The coordinate system of the project will be selected from "User-defined" and "Geodetic" according to the coordinate system of the data. Operations are same as (1).



■ Creation of new GLS project

Create new project folder on the drive for storing data.
There are the following two ways to create new project folder.

- Select "File" tab on the ribbon menu and "Data", then click "Create New GLS Project" 

.
- Select a workspace on <Workspace Window>, select "Fuse" tab on the ribbon menu and then click "Create GLS Project" 

.

1. Execute one of the above ways.
2. Select the project folder created in 1.
3. Select the GLS project added on the Workspace, click menu [Fuse] -> [Import Stations] 
.
4. Select the GLS measurement data (*.prj, *.sta) and open it.
   - When a selected file is *.prj, it will be imported to new station set that is created in workspace.
   - When a selected file is *.sta, it will be imported as a station.

■ Reading of existing GLS project

Add the GLS project to the Workspace.
There are the following three ways to add the GLS project.

- Select "File" tab on the ribbon menu and "Data", then click "Add Existing GLS Project..." 
.
- Select a workspace on <Workspace Window> and click "Add GLS" 
on the quick access.
- Select a workspace on <Workspace Window>, select "Fuse" tab on the ribbon menu and then click "Add GLS Project" 
.

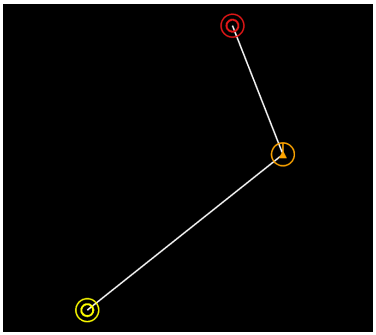
1. Execute one of the above ways.
2. Select the GLS project file (*.prj and others) and open it.

   There are two kind of GLS project data(*.prj).
   The first one is GLS raw data captured by GLS, the second one is the data processed by this application.
   When a selected file is GLS raw data, please handle raw data with care.

   - Processed data will be stored in GLS raw data folder. If it is not desired, please click menu [Fuse]->[Import Stations] and select *.prj file.

When data is imported, an occupation point, a backsight point and others are shown on the viewer as below.



- Occupation point

  The center of GLS mirror is displayed by an orange triangle.

  The occupation point on the ground is shown by a large orange circle, and the line drawn in the circle indicates azimuth 0.

- Backsight point

  A small red circle indicates the backsight point target. A large red circle indicates the backsight point target on the ground.

- GLS target

  A small yellow circle indicates the GLS target. A large yellow circle indicates GLS target on the ground.

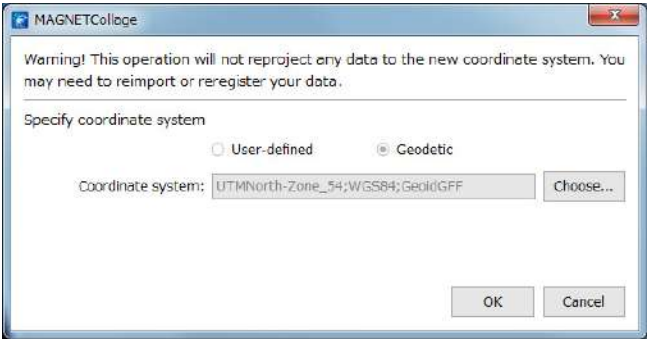
■ Resetting of project coordinate system

Coordinate system of the project can be reset by the following steps.

1. Click a project item on <Workspace Window>.

2. Click menu [Edit] -> [Reset Coordinate System] .

   <Reset Coordinate System Dialog> will appear.



3. Choose an appropriate coordinate system.

- This operation will not re-project any data to the new coordinate system. Captured scan data, targets or point coordinates will not be modified, nor existing point clouds.

3.2 Generation of Clouds and Panoramic Image

Select t... GLS project added on the Workspace, and click menu [Fuse] -> [Scans and Images] .

ProcessGlsScan Screen

■ Generation and coloring of clouds

1. Click [Start] of "Pose Scan".

   If "Limit Distance" is checked, the cloud will be filtered within the limits of the specified range between the station point and each measurement point.

   When the point cloud is colored simultaneously, click [Yes to All] in <Color Mapping Dialog>.
   ☞" Color Mapping Dialog"

■ Generation of panoramic image

1. Click [Start] of "Stitch Panoramas" to create the panoramic image of the station.

   • Please properly enter the distance to the measured target to the "Radius" of "Stitch Panoramas". The gap between the image and the point cloud by parallax can be corrected.

■ Noise removal

1. Click [Start] of "Filter Scan" to remove noise from created clouds.

• Select the strength of filter where noise removal is conducted from "Low" / "Middle" / "High" in "Level".

■ Smooth scan

Smooth cloud filter detects flat surfaces and brings noise points near to each flat surface.

Setting items	Description
Sigma	Sigma is the size of features we want to preserve. All details smaller than sigma will be considered noise and adjusted. Bigger sigma means big smoothing, zero sigma means no smoothing.
Feature preservation	Feature preservation controls the extent of edge detection. The edge sharpness will be increased from "Low"/"Medium"/"High" in order. Precise option is a little better for feature detection, Fast option is a few times faster especially for dense datasets and big Sigma.

3.3 Display of Image

The images photographed in the field at the time of measurement can be checked.

■ Gallery view

1. Select GLS Point Cloud / SCNxxxx / Image / Name of Image below the target Station item on <Workspace Window>.

2. Click menu [Edit] -> [Open Gallery]  .

   <Gallery View> will appear and the list of photographed images is displayed. And when the target image is double-clicked, it is enlarged on another window.

■ Panoramic image

1. Click [Fuse] on the menu, and conduct "Stitch Panoramas" in the processing of "Scans and Images".

   The image placeholder will be displayed at the location of the station.

2. Click the image placeholder.

   The panorama-projected sphere will be displayed.

3. Double-click the panorama sphere, or click menu [View] -> [Swap Panorama]  .

   The screen will be switched to Panorama display view.

Change of radius of projected sphere:

When panoramic image is recreated by changing the value of "Radius" in "Stitch Panoramas" item of <ProcessGlsScan Screen> and clicking [Start], the projection radius can be changed and the gap between the image and the point cloud by parallax can be corrected.

3.4 Registration of Clouds

When you capture data from two different scan positions, the data is not aligned. This is because the measurements the scanner takes are always recorded in the local coordinate system of the scanner. The scanner cannot determine that it was moved to a new position, so it does not align the data. This is the process to integrate the data obtained from scan positions different from that of registration into the same coordinate system. There are 5 different methods to register.

- Occupation and Backsight method
- Resection
- Target Registration and Georeferencing method
- Manual registration
- Cloud-to-Cloud

Please refer to "other functions" when needed. ☞ Refer to "Other functions"

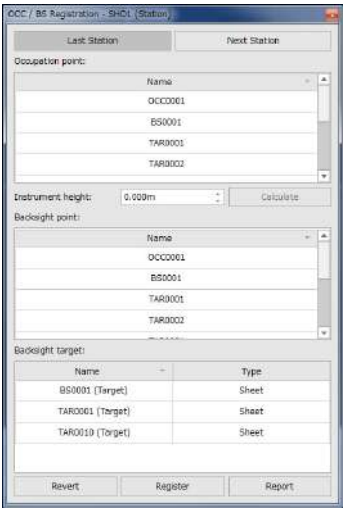
■ Occupation and Backsight

The laser scanner can be used to register from occupation point and backsight point.

1. Click the target station item on <Workspace Window>, then click menu [Fuse] -> [Occupation and Backsight] to display the <OCC / BS Registration Window>.

Note

- When coloring of the point cloud is set to "Station" , coloring is performed each station, so the operation is facilitated.



OCC / BS Registration Window

2. Select the point corresponding to the occupation point from the list of occupation point by double-clicking.

   The selected occupation point is highlighted in the table.
   If necessary, input the instrument height. If the point corresponding to the occupation point is not found, the point can be added from file or manually.

   If adding from file:

   ☞ For file input of basic CAD figures, please see "2.7 CAD Primitives"-"Reading from a file and writing in a file"

   If adding manually:

   1. Click Tree of Primitives just below GLS Project to open it.

   2. Select "Point", and click menu [Edit] -> [Point] .

   3. Click the proper location on the point cloud while pressing {Ctrl} key to create a point.

   * When the coordinate value needs to be fine tuned, edit the coordinate value on <Property Window>.

3. Select the point corresponding to the backsight point from the list of backsight point by double-clicking.

   The selected backsight point is highlighted in the table.
   If the point corresponding to the backsight point is not found, create the point in the same way as in the case of the occupation point.

4. Double-click the measurement data of the backsight point (measurement result of target scan) from the backsight target list to select the measurement data of the backsight point.

   The selected backsight point target is highlighted in the table.

   [Calculate] : Calculate instrument height by using occupation point, backsight point and backsight target. If you had measured rod height of backsight, you can calculate instrument height even when you had not measured instrument height at the site.
   *If you had not measured rod height, result will be negative. In this case 0.000 will be displayed.

5. Click [Register] to perform registration.

   • When registration is completed, never fail to proceed to the next station.

6. Click [Report] and check the registration result of each observation point.

   The residual error of X, Y, Z, H and V direction at each observation point and RMS are displayed.

7. Click [Last Station] or [Next Station] on <OCC / BS Registration Window> to select the next station.

   After selecting, perform from Procedure 2 to Procedure 5 in the same way.

8. Select the Station which you want to release on <Workspace Window>, and click menu [Fuse] -> [Clear] .

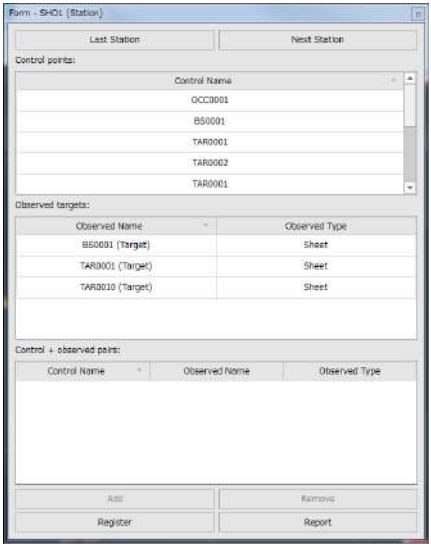
   The registration is released.
   It can be also released by selecting the Station, right-clicking and clicking [Clear Registration].

■ Resection

When laser scanner was placed with leveled condition, it is possible to match the scan position by using two or more control points assigned around the laser scanner and calculating the instrument position and direction.

1. Click the target Station item on Workspace, and click menu [Fuse] -> [Resection] .

   <Resection Registration Window> will appear.



Resection Registration Window

2. Select the target orientation point form "Control points" list by clicking it.

   If there is no control point in workspace, input the control point coordinates in file.

   ☞ For file input of basic CAD figures, please see "2.7 CAD Primitives"-"Reading from a file and writing in a file".

3. Select the observation point corresponding to the above orientation point from the "Observed targets" list by clicking it.
4. Click [Add] to add a pair of selected control point and observation point to "Control + observed pairs" list.

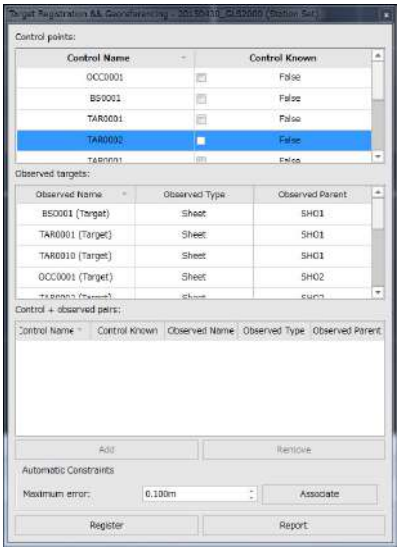
   When a pair is erased from the list, click [Remove].
5. When two or more sets of "Control + observed pairs" are registered and [Register] is clicked, backward intersection is executed.

■ Targets Registration and Georeferencing

This is the process to use the tie points of multiple scan positions and integrate data.
Georeferencing brings together data from different sources. it is an additional step you can take while registering your data to reorient the results of the registration into a global coordinate system, such as the State Plane Coordinate System (SPCS). For example, if you are using a combination of a laser scanner and another instrument, such as GPS or a total station, to obtain control information in the SPCS on the different targets you used during laser scanning, then you will want to reorient the results of your registration into that state plane.

1. Click the target Project item on Workspace, click menu [Fuse] -> [Targets and Georeferencing]  .

   <Targets Registration and Georeferencing Window> will appear.



Targets Registration and Georeferencing Window

2. Select the target orientation point from "Control points" list by clicking it.

   When a control point is used as a known point (global coordinates), set the checkbox status for known point to True. If it is set to False, it is handled as an unknown point and coordinates will be recalculated after registration.
   When integrating scan data by using tie points without establishing control points, it is not necessary to specify control points here.

3. Select the observation point corresponding to the above orientation point from "Observed targets" list by clicking it.

   It is possible to select multiple observation points. To select multiple observation points, select the points while pressing the {Ctrl} key.

   • Please note that registration is executed using the observation point which was selected first as the reference when multiple points are selected.

4. Click [Add] to add the pair of selected control point and observation point to the "Control + observed pairs" list.

   When integrating scan data using tie points without establishing control points, it is not necessary to specify the control points from the list of control points. In this case, a window to input the control points will open at this point. To use a known point, the coordinates can be manually input. To use a control point as an unknown point, input the coordinate values as (x, y, z) = (0.0, 0.0, 0.0).

5. Click [Register] to execute the registration.

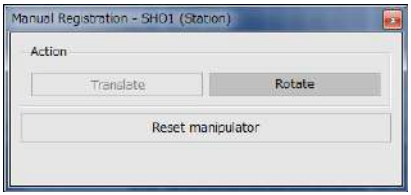
   Automatic Constraints: When [Associate] is clicked, observation points in observation point list will be automatically associated with corresponding points.
   Three or more sets of "Control + observed pairs" are required to perform Automatic Constraints.

■ Manual registration

It is possible to move the scan data in parallel direction or rotate it with mouse operation without use of targets and so forth.

1. Click the target Project item on Workspace, and click menu [Fuse] -> [Manual]  .

   <Manual Registration Window> will appear.



Manual Registration Window

2. Translate or rotate the point cloud.

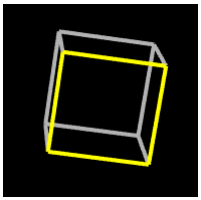
   Click [Translate] to place into clouds translation mode, and the point cloud can be translated by dragging the cube while pressing {Shift} key.
   Click [Rotate] to place into clouds rotation mode, and the point group can be rotated by dragging 3-axis rotating handle while pressing {Shift} key. Editable axis mode is selectable from X Axis, Y Axis, Z Axis and ALL Axes.

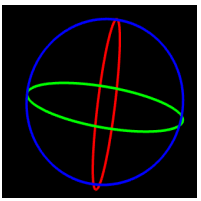
   Note

   • List of available shortcut keys are as follows.

Shift+Left-drag	:	Translate or Rotate
Shift+Left-double-click	:	Set dragger position only.
Shift+Ctrl+Left-double-click	:	Set dragger position and orientation.
Shift+Right-double-click	:	Match positions of two selected points in translate mode. Match orientation of two selected surfaces in rotate mode.

   • When the rotation of clouds is limited to the rotation around Z axis, the operation is facilitated.





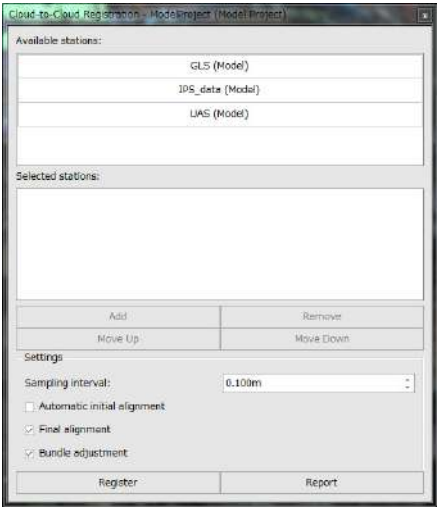
■ Cloud-to-Cloud

Two or more scan position data are integrated based on the 3D shape. It can be used without registration with tie points or occupation point/backsight point, or manual registration. It can also be used after executing registration with tie points or occupation point/backsight point or manual registration in order to further improve the integrated data.

- Please note the following points in registration by Cloud-to-Cloud:
  - There should be an overlapping area between the scan position data.
  - There should be sufficient "distinct and characteristic shapes" in the overlapping area.
  - Execute with the scan data position and specification roughly matched as initial condition.
  - Align by methods such as occupation point/backsight point method, resection method, and manual registration to roughly match the positions.

1. Click the target Project item on Workspace, and click menu [Fuse] -> [Cloud-to-Cloud] .

   < Cloud-to-Cloud Registration Window> will appear.



Cloud-to-Cloud Registration Window

2. Select the target Station from "Available stations" list by clicking it.

   Click [Add] to add to "Selected stations" list.
   A combination with {Shift} key and {Ctrl} key adds plural Stations collectively.

- Please note that registration is executed using the station which was selected first as the reference when multiple stations are selected.

3. Click [Register] to execute the registration.

Note

・As for Cloud-to-Cloud, overlapping can be conducted.

Setting items

Sampling interval : Sampling interval determines how much each scan is down-sampled prior to alignment. Clouds with fewer points align much faster. However, if too few points are used, small surface features will not participate in registration and the accuracy of results may suffer. It does not affect original data.
The default value is 10 [cm].

Note

- In general, sampling interval of 10 [cm] works well for a wide range of datasets. If scans are short range and extremely detailed, consider sampling interval of 5 [cm]. If scans cover a large area and have few small features, consider sampling interval of 20 [cm].

Automatic initial alignment:

Automatically adjust the initial point and direction of clouds before starting matching. This takes significant additional time and success is not guaranteed. The default is not checked.

Effects of the complexity of point clouds or amount of overlapping may show in some cases. If the results are not favorable, it is also possible to execute Cloud-to-Cloud after roughly matching the position by one of the registration methods and manually adjusting. In this case, it does not need to select.

Final alignment : Execute the processing of Cloud-to-Cloud. If this is unchecked, only "initial alignment" can be executed. The default is checked.

Bundle adjustment : When three or more scan positions are registered via Cloud-to-Cloud registration, bundle adjustment can reduce accumulation of registration errors. Although it takes some extra time to adjust registration results, this feature should be enabled for most datasets.

4. Click [Report].

<Report Window> will appear and the results of registration can be checked.

Distance between the points (RMS): Indicates the distance between the corresponding points of the clouds in overlapping the clouds. The points do not overlap completely. Though the value is not 0, it is an approximate guide.

Distance between the point and the face (RMS):

Indicates the distance between the face of the matched point cloud and the point of the matching one. If complete matching is done, the value is 0.

■ Complicated registration

If "Station Set" is used, the registration can be executed in a partially different manner.

Merit of use of station set

Suppose there are Station A, B and C and each of them has scan data.
As for Station A, geographical coordinates are given to the occupation point and the backsight point, but Station B and C do not have information for geodetic reference.
If B and C can be combined by Cloud to Cloud respectively after the geodetic reference of A, there is no problem. However, when "Cloud-to-Cloud" is implemented, a certain degree of overlapping of clouds is required. And also, complicated clouds of trees and others cause a large error, and the combination in "Cloud-to-Cloud" is limited on the grounds that it is undesirable.

Under the following conditions, registration can be implemented.

Condition	Station	Result
Condition 1	Station A	Geodetic reference is possible in OCC/BS.
Condition 2	Station A and B	The clouds are overlapped sufficiently, and Could to Cloud is possible.
Condition 3	Station A and C	The clouds are not overlapped sufficiently, and Could to Cloud is impossible. Tie point combination in Target Scan is also impossible.
Condition 4	Station B and C	The clouds are not overlapped sufficiently, and Could to Cloud is impossible. Tie point combination in Target Scan is possible.

In this case, if B+C and A can be combined in the Cloud-to-Cloud method after combining B and C in the tie point method, the whole registration and geodetic reference are possible.

In normal registration, the whole cannot be processed by mixing different methods, but the use of "Station Set" makes it possible.

1. Click the target Project item on Workspace, and click menu [Fuse] -> [Create Station Set] 

.

2. Newly read by clicking menu [Fuse] -> [Import Stations]  , or create the copy by dragging existing Station and dropping to created Station Set.

   An orientation point can be also added to Station Set by the following method.

   • Newly read by clicking menu [Fuse] -> [Import]  .

   • The copy can be created by dragging existing Primitives and dropping to created Station Set.

3. Register "Station Set"

   The registration of Station in Station Set can be executed with OCC/BS, Resection, Manual, Cloud to Cloud and Targets and Georeferencing in the same way as in the normal case.

4. Register "Station Set" and other Station

   The whole registration can be implemented with Manual, Cloud to Cloud and Targets and Georeferencing by processing on the supposition that Station Set is one Station.

■ Other functions

Create the occupation point or the point of target position

1. Select Station on Workspace, and click menu [Edit] -> [Create Point Primitive]  .

   The point primitive with occupation point coordinate is created.

2. Select "GLS Target", and click menu [Edit] -> [Create Point Primitive]  .

   The point primitive with coordinate values of the target is created.

Increase interval

When the geodetic coordinate is not input in the scan in the field, or after the registration is released, plural scan data is located around the origin with an overlap.

In this case, when you click menu [Fuse] -> [Spread]  , an occupation point is located at automatically and properly increased interval. Therefore, each occupation point is easily distinguished.

■ Report display of registration results

You can confirm the registration results of each station in the project.

When you click menu [Fuse] -> [Accuracy Report]  , the registration results will be displayed.

■ Release of registration

Select the Station you want to release on <Workspace Window>, and click menu [Fuse] -> [Clear].

Or select station, right-click and click [Clear Registration]  .

3.5 Showing Data on Map

Although GLS projects and model projects can be created in a projected coordinate system, the background map is displayed based on Earth-centered-Earth-fixed (ECEF) coordinate system. When a GLS project or model project is created in a projected coordinate system, you need to convert the data into ECEF to show clouds and CAD primitives on map.

■ Quick Georeference

To show data on map, you need to register the data onto a projected coordinate system properly, then need to copy the data into another model project of ECEF. In this case, each point of the cloud is converted precisely, but the process will take longer time.

"Quick Georeference" calculates rough transformation to show data on map quickly, so you can easily confirm rough alignment on map.
The result of quick georeference may have errors from several centimeters to tens of centimeters.

To apply quick georeference, the data of the project need to be registered onto a geodetic coordinate system.

When you created GLS project or model project with User-defined coordinate system, you cannot apply quick georeference.

When you created GLS project or model project with ECEF, the data in the project will appear on map immediately. So you don't need to use quick georeference.

■ Clearing Quick Georeference

You can clear quick georeference by selecting the project in workspace, then click [Clear Registration].
☞ "Release of registration"

3.6 Data Output

■ Output of orthophotos from clouds

In the function of Ortho output from clouds, output is conducted from all clouds which are displayed currently or scan. The color of Orthophotos is generated based on the color of scan on the application or the point cloud (reflection intensity/image/grayscale/height/flat/station).
Execute output according to the following procedure after selecting point cloud and scan which are output targets on Workspace.
And also, the function of Orthophotos output from clouds depends on the display mode of clouds. The point in clouds whose display mode is deleted or hidden status is not the target of output.
As a main use method, when Orthophotos output is done in the status where only road surface is displayed and other parts are hidden, Orthophotos focused on the road surface can be output.

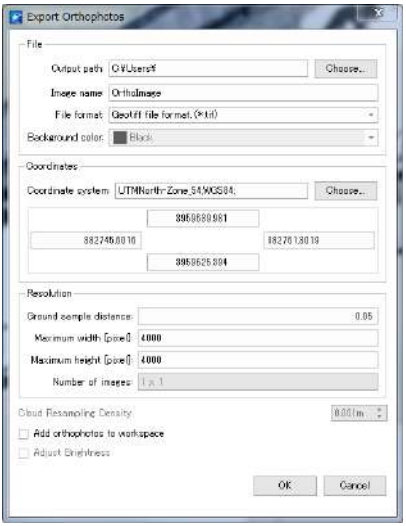
1. Click menu [Edit] -> [Export Orthophotos From Cloud]  .

   The mode will switch to map view mode and the output function will start. It will switch to rectangle selection mode.

2. Left-click on Map View, and select upper left and lower right of the output area.

   The output dialog will appear.

3. Edit the setting of output.



Export Orthophotos Dialog

Each setting item is as below.

Output path	:	Specify the output directory.
Image name	:	Specify orthophotos name.
File format	:	Selectable formats are as below. Geoff, Tiff + tifw, Jpeg + jpgw
Background color	:	Specify the background color of orthophotos. If output is done with Geoff and Tiff format, the background color is automatically transparent color.
Coordinate system	:	Select the output coordinate system. If you output with Geoff format, only UTM coordinate system can be selected.
Ground sample distance	:	Specify the length of one side of one picture cell.
Maximum width[pixel]	:	Specify the maximum width (pixel number) per one sheet of orthophotos. If orthophotos whose range exceeds the maximum width is output, it is divided into plural sheets.
Maximum height[pixel]	:	Specify the maximum height (pixel number) per one sheet of orthophotos. If orthophotos whose range exceeds the maximum height is output, it is divided into plural sheets.
Number of images	:	The number of output images is displayed.
Add orthophotos to workspace:		Specify if created orthophotos is automatically stored in Map folder in the application.

4. Start output by [OK].

■ Output of point clouds and CAD primitives

Execute export of point clouds and CAD primitives.

Output of GLS data

1. Select the Station which you want to output and click menu [Edit] ->[Export]  .

   <Export GLS Station Dialog> will appear.
2. Specify the folder of output destination by "Output path."
3. Register the type of data which is output to List of Output Items by "File format."
4. Click [Add...].

   <Output Items Add Dialog> will appear.
   Specify "Data type" and "File format" from the list.

Table 6: File Format

Data Type	File Format
GLS Point Clouds	PCD file format (.pcd)
	PTS file format (.pts)
	PTX file format (.ptx)
	CL3 file format (.cl3)
	E57 file format (.e57)
	RCS file format (.rcs)
	CLR file format (clr)
	LAS file format (.las)
	TXT file format (.txt)
	CL3+IJ+ALG file format (.cl3)
GLS Panorama	Jpeg file format (.jpg)

- The file extension of the import files must be lower case.

☞ When you export GLS data in text format, you can specify export items and the order. Please refer to "Output to a file" in "2.6 Edit of Clouds".

[Add...] : Add to List of "Output Items."
[Edit...] : Edit the item which is registered in List of "Output Items."
[Delete] : Delete the item which is registered in List of "Output Items."

5. Specify the coordinate system which is output by clicking "Transform Coordinate".

   ☞ Regarding "unregistered", "registered" and "projected", please refer to "2.7 CAD Primitives"-"Reading from a file and writing in a file".

   Click [Select...], and <Coordinate System Selection Dialog> will appear.
   Specify arbitrary coordinate system from the list. If the geoid height is corrected, specify "Geoid Type" and "Geoid File Path."

   The coordinate system list can be narrowed by specifying a search string in "Search String".
   The following coordinate system codes are specified in the Japanese coordinate system.

   Japan geodetic system: JAPAN-JAPAN_** (**: 01-19)

   World geodetic system: JAPAN2000-JAPAN_** (**: 01-19)

   The Geoid File which is selected for the first time conducts the transform to a unique file format (**.gff), and stores gff file in the storage place of selected geoid file. From the 2nd time, the converted gff file can be used.

6. Select Unit System from the radio button.
7. Click [OK] to execute the output of data.

Output of CAD primitive data

1. Select the Station which you want to output, and click menu [Fuse] -> [Export]  .

   < Export CAD Primitives Dialog> will appear.

2. Specify the Folder of output destination in "Locate file."

   The same as the output of GLS Data.

3. Specify the coordinate system which is output in "Transform coordinates"

   The same as the output of GLS data.

4. Select "Convert units" from the radio button.

   The same as the output of GLS data.

5. Click [OK] to execute the output of data.

   The same as the output of GLS data.

3.7 Data Edit

■ Slice and display of section

The point cloud can be sliced within the arbitrary range, and its section can be displayed.

- Slice
  Select GLS Scans / SCNxxxx below the target Station item on <Workspace Window>, click menu [Edit] -> [Slice]. Then, <Slice Window> will appear. Or select Polyline, and click menu [Edit] -[Slice]  .
  Click [Section View], and <Section Display Window> will appear on the split 3D view.
- Section
  Click [Section View] of <Slice Window>, and the section of slice will be displayed on the split 3D view.

■ Edit of point clouds

Selection of point clouds

Selection mode

Replace  : Return the existing selected area to unselected, and select new area.

Add  : While maintaining the existing selected area, add new selection area.

Subtract  : As for the existing selected area, change the specified area to unselected.

Intersect  : Select only the area which is included in both the existing selected area and newly specified area.

Display mode

Normal  : Display the point cloud in normal status.

Hidden  : Display the point cloud in hidden status.

Deleted  : Display the point cloud in deleted status.

Reset  : Change to the selection only in standard status.

Difference between hidden and deleted:

The "hidden" range is temporary hidden range, so once workspace is closed, hidden status is cleared, and when opened next, it returns to standard status. However, the "deleted" range is permanent deleted range, so even if workspace is closed, deleted status is maintained.

3.8 Integrating Data from Other Projects

With using model projects, you can integrate data from multiple projects. In this case, the coordinate system of the integrated data will be the coordinate system of the model project. When external data in the other coordinate system are added, the coordinate system is converted properly.
The integration of data is done as described in the table "Coordinate System and Condition in Data Copy between Different Projects".

■ Add the point cloud to model project

Creation of model project

Click menu [File] -> [Data] -> [Create New Model Project] 

, or select Root item on <Workspace Window> and click menu [Fuse] -> [Create Model Project] 

.

- Though Folder Selection Screen will appear, the folder which you select must be empty.

Creation of model

Select Model Project on <Workspace Window>, and click menu [Fuse] -> [Create Model] 
.

Add the point cloud to model

There are some conditions to copy the data between the projects.

(1) Coordinate systems and conditions

Source Coordinate system	Destination Coordinate system	Result
User-defined	User-defined	User-defined
User-defined	Geodetic	The data will be copied without coordinate transformations.
Geodetic	User-defined	The data will be copied without coordinate transformations.
Geodetic	Geodetic	The data will be copied after transforming to the destination coordinate system automatically. When copying a cloud data, the transformation will be applied to each point on the cloud. Source data must be performed geo-referencing.

(2) Drag and drop the CAD primitives

CAD primitives can be copied by drag and drop.

(3) Drag and drop the cloud

Clouds can be copied to a model by drag and drop.

(4) Create a cloud from selected regions on the viewer

Cloud can be copied to a model item by performing the create cloud command.

3.9 Menu Commands

■ Fuse tab

Function	Description
Scans and Images 	Execute processing of Clouds and panoramic image.
Clear 	Release the registration.
Manual 	Execute manual registration.
Occupation and Backsight 	Execute registration by occupation point and backsight point
Resection 	Execute registration by resection
Spread 	Display the data of each station at increased interval.
Cloud-to-Cloud 	Execute Cloud-to-Cloud.
Targets and Georeferencing 	Execute registration and geodetic reference by tie point.
Quick Georeferencing 	Project the data analyzed in the geodetic coordinate system on the map roughly.

Accuracy Report		Open Report of registration execution result.
Import Stations		Read the data of existing Station.
Create Station Set		Create new Station Set.
Import		Execute import of primitives.
Export		Execute output of primitives.

■ Edit tab

Function		Description
Show All		Display all hidden fields.
Hide All		Hide all displayed fields.
Jump To		Display the selected items on <Map View> and <3D View>.
Open Gallery		Display <Gallery View>.
Set Scope		Click one of Split Views and select arbitrary run data in <Work Space Window> and click "Set Scope". Then, two views can be displayed simultaneously.
Color Range		Set the displayed color of the trajectory
Distance		Create a distance item from two point primitives.
Remove Item		Remove the GLS project registered in the workspace. It does not mean deleting GLS project data from PC.

Delete		Delete the GLS station registered in the workspace. Selected GLS project will be deleted from PC.
Rename		Change the name.
Reset Coordinate System		Reset the coordinate system of the project.
Export		Execute export of point clouds.
Export Orthophotos From Cloud		Output orthophotos using the color information of the clouds. ☞ “Output of orthophotos from clouds”
Create Point Primitives		Primitives of selected station or target will be created.
Show in Explorer		Display the items that have been selected on the windows explorer.
Properties		Display <Property Window> of the selected item.
Table		Display sub-class properties of the selected item in a table format.

4. MMS DATA PROCESSING

This chapter describes the analysis of data which is obtained by Mobile Mapping System (MMS).

4.1 Data Import

■ Import IP-S3 data

There are the following four ways to import IP-S3 data.

- Select a workspace on <Workspace Window> and click "Add IPS"  on the quick access.
- Select a workspace on <Workspace Window> and select "[Add Existing IP-S Run...]" on the right-click.
- Select a workspace on <Workspace Window>, select "Fuse" tab on the ribbon menu and then click "Add IP-S Run"  .
- Select "File" tab on the ribbon menu and "Data", then click "Add Existing IP-S3 Run".

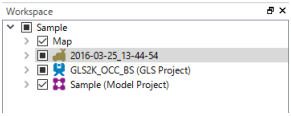
Workspace Window

1. Execute one of the above ways.

   <File Dialog> is displayed.

2. Select "raw.ipsx" file of the data folder to import and click [Open].

   If the import is terminated, IP-S3 run data will be added to the workspace.



Workspace Window

Note

- If the trajectory processing is not conducted when data is collected at Mobile Master Field, the trajectory displayed during collection is imposed.

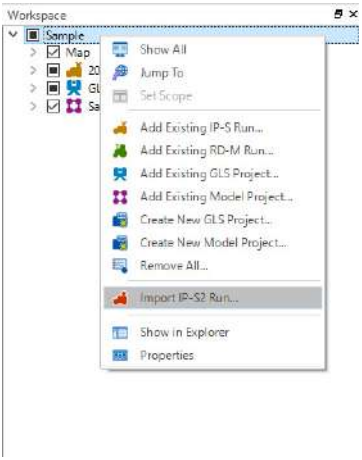
■ Import IP-S2 data

The system converts the exiting IP-S2 run data into an IP-S3 format to import IP-S2 data.

- You need to create an empty folder in advance to save the converted IP-S3 data.
  ☞ "Compatibility between applicable device and data"

You can start with the following three ways to import the IP-S2 data.

- Select a workspace on <Workspace Window> and select "Import IP-S2 Run..." on the right-click.
- Select a workspace on <Workspace Window>, select "Fuse" tab on the ribbon menu and then click "Import IP-S2 Run" .
- Select "File" tab on the ribbon menu and "Data", then click "Import IP-S2 Run".



Workspace Window

1. Execute one of the above ways.

   The following <IP-S2 Run Data Import Dialog > is displayed.

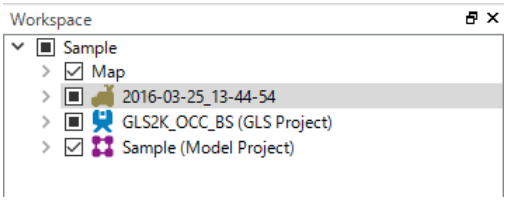
IP-S2 Run Data Import Dialog

2. Assign the IP-S2 run data folder to import from "IP-S2 run path" and the created IP-S3 run data folder to "IP-S3 run path", and click [OK].

   You can confirm the completion of the IP-S2 data import by following method.

   • "Status" on <Task Window> becomes "Completed" and "Progress" becomes "100%" after all the processing related to the IP-S2 data import has been through.

   Once the import has been completed, the IP-S2 run data will be added to the workspace.



Workspace Window

4.2 Data Processing

This chapter describes processing of run data captured by the IP-S3 system.

■ Processing flow

The following shows the data processing flow.

Step 1 : Vehicle trajectory estimation ☞ "Fuse vehicle trajectory"

Selecting processing method, setting and editing base station information, setting process parameters, performing estimations, and confirming results

Step 2 : Align multiple passes ☞ "Align multiple passes"

Pose scan, detect intersections and edit loops, performing Align multiple passes, and confirming results, Reverting

Step 3 : Align trajectory with GCPs
☞ "Adjust the trajectory by the ground control point(s)"

Pose scan, import GCPs, edit GCPs, Align trajectory with GCPs, and confirming results

Step 4 : Stitch Panoramic image
☞ "Stitching panoramic image and color mapping"

Stitch panoramic images, pose panoramic images, and color mapping

Step 5 : Cloud generation
☞ "Cloud generation"

Create cloud from scan

Step 6 : Filter cloud
☞ "Point cloud noise removal"

Filter cloud

Note

- The guideline of processing time for 1-hour of run data (urban area run, JPEG quality 75%, shooting distance interval 10m) is shown below.

Estimate trajectory	:	10’
Stitch panoramic images	:	80’
Pose panorama	:	1’
Pose scan (N=1)	:	35’
Colorize scans (N=1)	:	55’
Pose scan colorize scans (N=1)	:	60’
Create cloud (0.01, 0.01)	:	60’
Filter cloud	:	40’
Detect intersections	:	30’
Align multiple passes	:	50’
Trajectory Adjustment Using Ground Control Point(s)	:	10’

- The actual processing time varies depending on the data collection condition, etc.
- Please make sure your run data folder path doesn't include the dot character so that trajectory estimation can be processed normally.

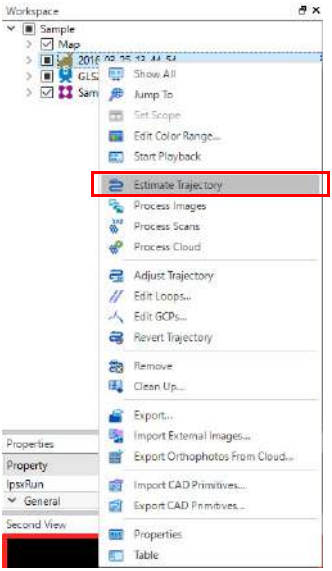
■ Fuse vehicle trajectory

The system estimates the trajectory based on GNSS, IMU and wheel encoder data of the run.

Select a fuse method

There are the following two ways to fuse the vehicle trajectory.

- Select a run data in which you want to fuse the vehicle trajectory on <Workspace Window> and select "Estimate Trajectory" on the right-click.

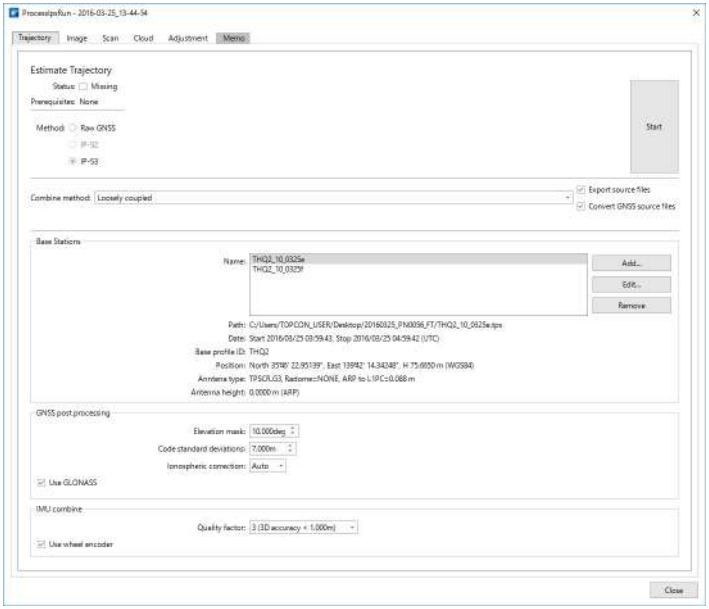


Workspace Window

- Select a run data in which you want to fuse the vehicle trajectory on <Workspace Window>, select "Fuse" tab on the ribbon menu and then click "Trajectory"  .

1. Execute one of the above ways.

"Trajectory" tab of <ProcessIpsRun Dialog> will be displayed.



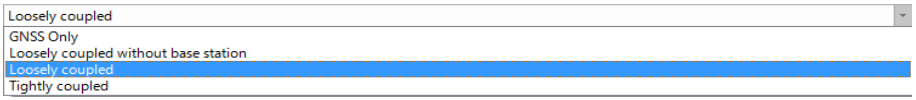
ProcessIpsRun Dialog (Trajectory tab)

2. Select "IP-S3" on "Method".

Note

- When you select "Raw GNSS" on "Method" and click [Start], the results of the GNSS single point positioning is imported. You cannot move to the next step with the results.
- When you import the IP-S2 trajectory whose analysis has been completed by Geoclean, select "IP-S2" and click [Start]. In this case, import the IP-S2 run data in advance.
  ☞ "Import IP-S2 data"

3. Select a trajectory analysis method on "Combine method".



Selection of an Analytical Method

There are the following four trajectory analysis methods.

- "GNSS Only"

  You can carry out the post-processing kinematic analysis together with the GNSS data of a base station. The trajectory may be lost in some places where the GNSS data is not available because this method uses the GNSS data only. You cannot move to the next step with the results.

- "Loosely coupled without base station"

  You can fuse the trajectory by combining vehicle single point positioning values and the IMU data, and the wheel encoder data when the GNSS data of a base station is not available. The data is not so accurate as the GNSS only as it uses the single point positioning, but the trajectory is not lost even when the GNSS data is not available. This method is used to understand an approximate trajectory using only the run data.

- "Loosely coupled"

  You can carry out trajectory analysis with the result of the post-processing kinematic analysis together with the GNSS data of a base station and a combination of the IMU data and the wheel encoder data. Producing high accuracy result, this method is recommended for the final trajectory analysis.

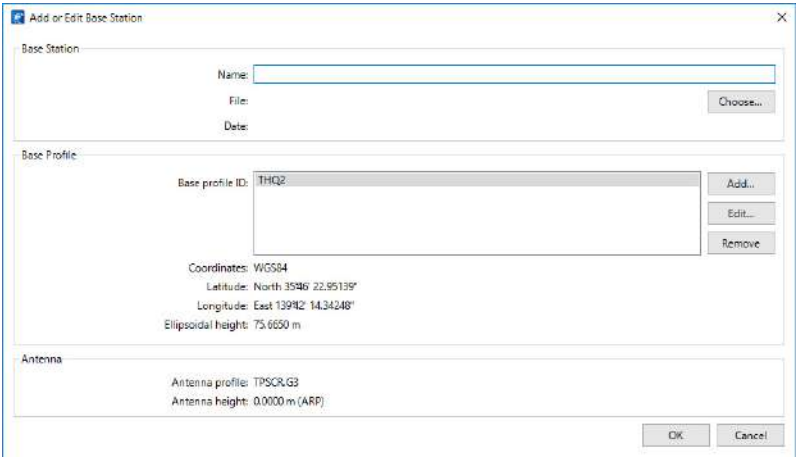
- "Tightly coupled"

  You can fuse the trajectory by the simultaneous combination of the GNSS dada of the base station and vehicle, the IMU data and the wheel encoder data. If the above "Loosely coupled" cannot be used for analyzing the trajectory, this method may improve the analysis.

Set-up and edit the base station information

1. Click [Add...] of "Base Stations" on <ProcessIpsRun Dialog>.

   <Add or Edit Base Station Dialog> is displayed.



Add or Edit Base Station Dialog

2. Click [Choose...] of "Base Station" on <Add or Edit Base Station Dialog>.

   <File Dialog> is displayed.

Note

- You can select a GNSS data format on the list in the right bottom of <File Dialog>. Select "Base station (*.tps)" for the TPS file and "RINEX (*.*o)" for the RINEX file.

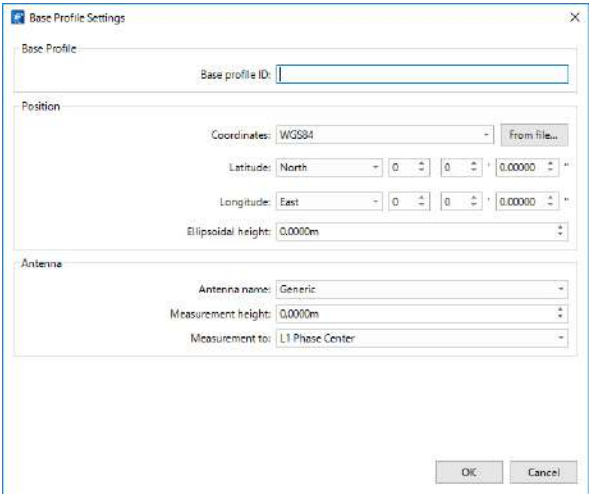
- The file title or file path of the base station cannot contain the character dot " . " in order to process estimate trajectory.

3. Select a base station GNSS data and click [Open].

   The assigned file name and "Date" are displayed.

4. Click [Add...] on <Add or Edit Base Station Dialog>.

   <Base Profile Setting Dialog> is displayed.

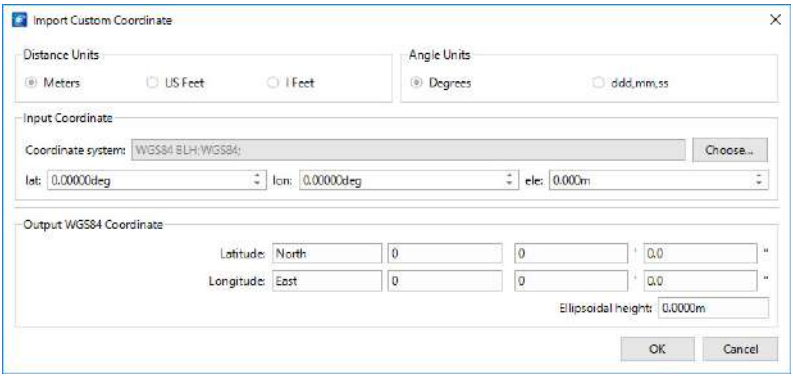


Base Profile Setting Dialog

5. Enter a base station name and a point name in "Base profile ID" of "Base Profile".
6. Select coordinates of the base station to enter on the "Coordinates" list and enter the coordinates.

   Note

   • If you select "Custom" on the coordinates list, <Import Custom Coordinate Dialog> is displayed. Select "Distance Units", "Angle Units" and "Coordinate system" and enter coordinates. When you click [OK], they are converted to the latitude, the longitude and the ellipsoidal height of WGS84 and reflected to the coordinates on <Base Profile Setting Dialog>.

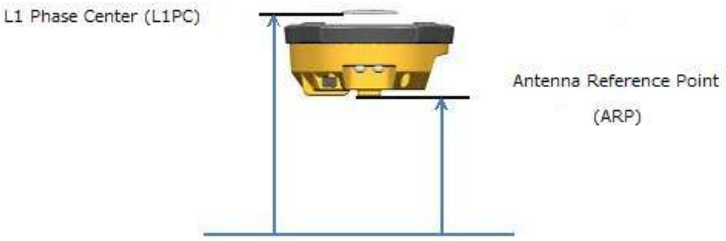


Import Custom Coordinate Dialog

7. Select an antenna name used in the base station on "Antenna name" list of "Antenna".

   • Make sure of entering the antenna name except the antenna name is unknown.

8. Enter "Measurement height".
9. Select "ARP" or "L1 Phase Center" on "Measurement to" list.



Types of Measurement Height (Example of TOPCON HiperV)

Note

• When you click [From file...] and assign the base station GNSS data file, the recorded approximate coordinates (single point positioning values) and antenna name are read out and reflected to <Add or Edit Base Station Dialog>.

10. Click [OK] to return to <Add or Edit Base Station Dialog>. The base profile entered here is added to the "Base profile ID" list on <Add or Edit Base Station Dialog>.
11. Select "Base profile ID" to use and click [OK] to return to <ProcessIpsRun Dialog>.

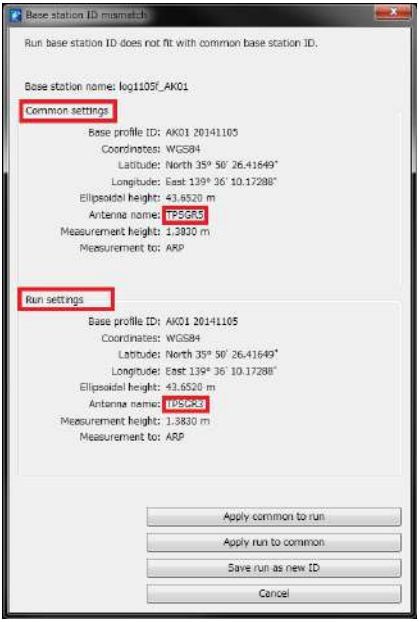
Note

• You can assign multiple GNSS data divided by time to the same "Base profile ID".

12. Repeat 1 to 12 when you register multiple base stations for the multi-base analysis.

Note

- Generally, the accuracy of the GNSS kinematic analysis tends to decrease as the mobile station becomes far from the base station. The multi base analysis is used to perform the GNSS kinematic analysis by installing multiple base stations along the route of the mobile station to prevent such accuracy deterioration.
- If you change "Base profile" of the run data which shares "Base profile ID" in the analysis, <Base Profile Mismatch Dialog> is displayed when you open <ProcessIpsRun Dialog>.



Base Profile Mismatch Dialog (Example of a Different Antenna Height)

You can select the following processing for the mismatched profile.

- [Apply common to run]
  Replace the run data settings with the shared settings on the same workspace and use them.
- [Apply run to common]
  Replace the shared settings on the same workspace with the run data settings and use them.
- [Save run as new ID]
  Register the run data settings as a new "[Base profile ID]" on the displayed <Base Profile Setting Dialog> and use them.
- [Cancel]
  Use the current run data settings.

Set-up the trajectory analysis parameter

These parameters are used for "GNSS only", "Loosely coupled" and "Tightly coupled" of "Method".
☞ "Fuse vehicle trajectory"

- Elevation mask
  You can assign the elevation mask of the GNSS satellite data between 0.1 and 90.0 degrees. Generally, the number of GNSS observation data is increased as the elevation mask is lowered (decreased), but it becomes susceptible to the influence of the multipath. When you heighten (increase) the elevation mask, it is less likely to be susceptible to the influence of the multipath, but the number of the available GNSS observation is decreased. The default value is 10.00 degrees.
- Code standard deviation
  You can assign the standard deviation of the assumed GNSS code observation value between 0.1 [m] and 99.9 [m]. Usually, 10.0 [m] for urban areas, 7.0 [m] for suburbs and 3.0 [m] for open spaces where almost no barriers are found (open sky) are adequate. The default value is 7.0 [m].
- Ionospheric correction
  You can assign the ionospheric correction from "Off", "On" and "Auto". The default is "Auto".

  Note
  - When you specify "On" for the multi base analysis where multiple base stations are used, the accuracy of the analytical results may improve.
- Use GLONASS
  Check when you use the observation data of the GLONASS satellite. If you uncheck it, the accuracy of the analytical results improves in rare cases. The default is checked.
- Use Wheel encoder
  Check when you use the wheel encoder data. The default is checked.
- Quality factor
  You can assign the quality factor of the post-processing kinematic solution used for updating the analytical results by the IMU and the encoder data when you have selected "Loosely coupled" on "Method". The default value is 3.

  The following shows standard accuracy of the quality factor and the post-processing kinematic solution.

Table 7: Number of quality factor and Guidance of 3D precision

Number	Guidance of 3D precision
1	<0.15[m]
2	<0.40[m]
3	<1.00[m]
4	<2.00[m]
5	<5.00[m]
6	<10.00[m]

When you assign 3 to "Quality factor", the analytical results by the IMU and the encoder data are to be updated by a post-processing kinematic solution with the 3D accuracy less than 1.00 [m].

The following shows a standard of the parameter adjustment at the observation environment of the run data.

Table 8: Guidance on parameter adjustment by observation environment

Observation / Parameter	Urban area	Suburb	Open sky
Elevation mask*	10°	10°	5° or greater
Code standard deviation*	10.0 m	7.0 m	3.0 m
Ionospheric correction	AUTO	AUTO	AUTO
GLONASS	Select	Select	Select
Quality factor *	3, 4	3	2, 3

Note

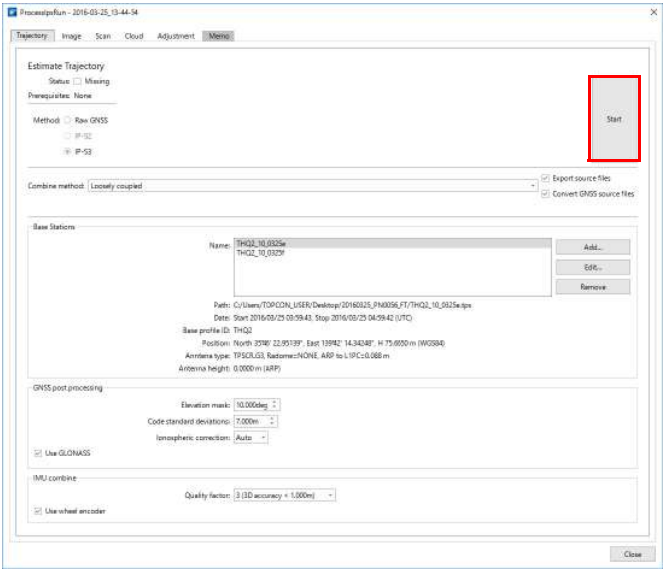
- It is recommended that you use the default parameter for the first analysis.
- It is recommended that you use the default parameters for items without * in the above table when you adjust parameters.

Execute the calculation

This section explains an example case of using "Loosely coupled" as "Combine method".

1. Set the base station information setting and editing and the trajectory analysis parameters, and click [Start] on <ProcessIpsRun Dialog>.

   "Status" changes from "Missing" to "Pending".



ProcessIpsRun Dialog (Trajectory tab)

Note

- Even if you click [Close] on <ProcessIpsRun Dialog> and close the dialog, the processing continues.

<Task Window> displays the progress of the trajectory analysis.

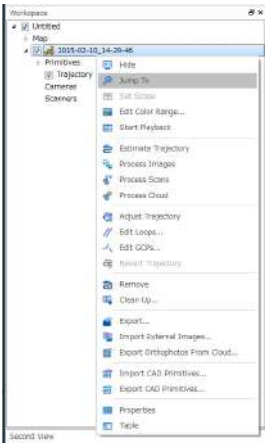
You can confirm the completion of the trajectory analysis when;

- "Status" on <Task Window> becomes "Completed" and "Progress" becomes "100%" after all the processing related to the trajectory analysis has been through.
- "Status" of "Estimate Trajectory" of "Trajectory" tab on <ProcessIpsRun Dialog> becomes "Current" and the box is checked.

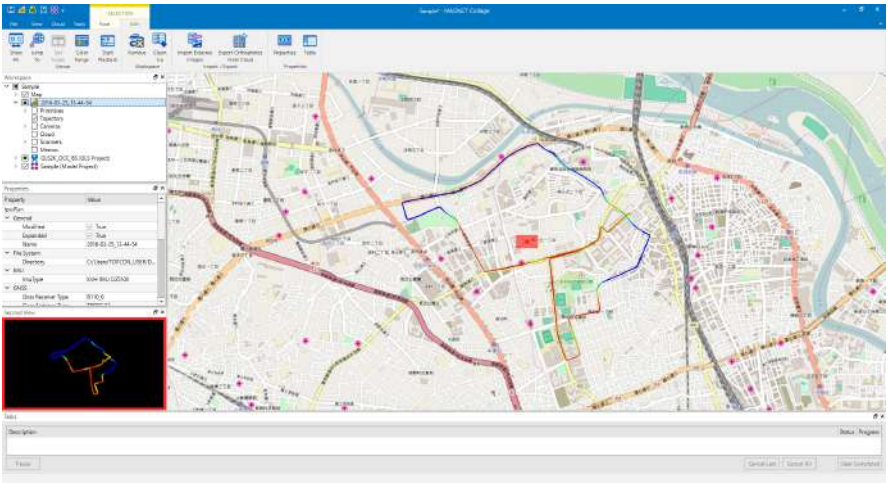
Note

- If you uncheck boxes of "Export source files" and "Convert GNSS source files" on the right side of "Method" list, the time spent for the trajectory analysis can be shortened from the second trajectory analysis.

2. Click [Close] on <ProcessIpsRun Dialog> to close the dialog.
3. There are the following two methods to display the analytical results.
   - Select a run data that you want to display the trajectory on <Workspace Window> and select "Jump To" on the right click.
   - Select a run data that you want to display the trajectory on <Workspace Window>, select "Edit" tab on the ribbon menu and then click "Jump To"  .



Workspace Window



Processing Results

☞ Refer to "2. BASIC OPERATION" for the operation of <Map View> and <3D View>.

Confirm results

This section explains how to confirm the accuracy of the trajectory.

1. Select "View" tab on the ribbon menu, click "Trajectory" and "Position Sigma" .

2. Move the mouse pointer on the trajectory on <Map View> or <3D View>.

   The standard deviation (unit [m]) of the point as well as the color bar on the right side of <View> are displayed. The trajectory is colored by the standard deviation.

3D View (Trajectory from processing results)

3. Confirm that the maximum value of the standard deviation is around 1.0 [m]. Also confirm that there are no separation of the height between the intersection runs of multiple times or trajectories of both ways, or no intermittent areas.

   Note

   • You can adjust the color of the color bar by designating the color and the range of the standard deviation. ☞ "4.5 Menu Commands"-"Edit tab"-"Set a range of trajectory color"

   (e.g.) This shows a trajectory display in <3D View> when the maximum value of the position sigma is 0.05m.

3D View (Trajectory from Processing Results)

■ Align multiple passes

This section describes pose scans based on the vehicle trajectory, and describes how to align multiple passes based on intersection detection.

Pose scan

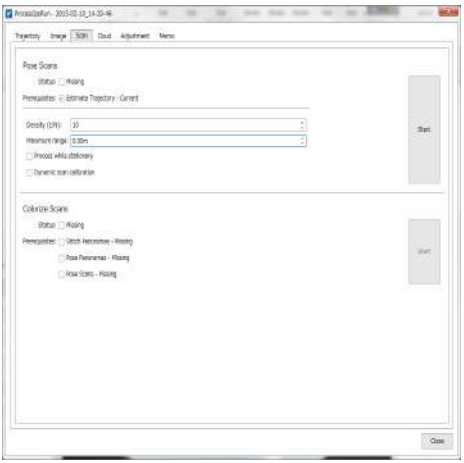
You can start with the following two ways to create scans.

- Select a run data in which the trajectory analysis has been completed on <Workspace Window> and select "Process Scans" on the right click.
- Select a run data in which the trajectory analysis has been completed on <Workspace Window>, select "Fuse" tab on the ribbon menu and then click "Scans" 

.

1. Execute one of the above ways.

   "Scan" tab on <ProcessIpsRun Dialog> is displayed. Confirm that "Estimate Trajectory - Current" is displayed on "Prerequisites" of "Pose scans" and the box is checked.



ProcessIpsRun Dialog (Scan tab)

   The processing time can be reduced by setting a value that thins out data in "Density (1/N)". Align multiple passes can be performed even when N is not 1. The following uses N = 10.
   If you also want to perform the processing on scan data from a stopped vehicle, select the "Process while stationary" check box.

   Specify the maximum distance of the scan data to be used in performing a pose scan in the "Max. matching distance". The default value is 0.0 [m].

   If you want to perform dynamic scan calibration, select the "Dynamic scan calibration" check box.

Note

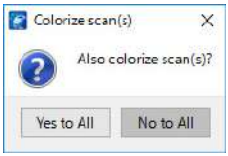
- In the "Pose scan" process, if the road surface determined to be flat among the run data, and if it has appropriate conditions to calculate the correction value of the scanner distance, the scanner distance correction value will be calculated.
  If the correction value is calculated even once, correction is made to the scanner distance value of the entire run data, and if it is calculated in multiple places, correction is made with a properly interpolated value.
  If the correction value is not calculated in the run data, no correction will be made.

2. Click [Start] of "Pose Scans".

"Status" changes from "Missing" to "Pending".

Note

- If the creation and allocation of the panoramas have been completed in advance, the following confirmation dialog is displayed. When you click "[Yes to All]", the scan data with color mapping is created.

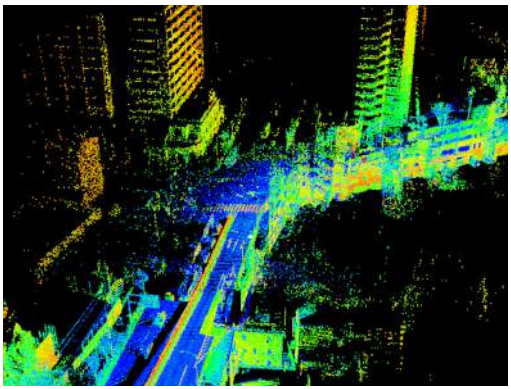


Color Mapping Dialog

You can confirm the completion of the creation of scans by one of the following methods.

- "Status" on <Task Window> becomes "Completed" and "Progress" becomes "100%" after all the processing related to the creation of scans has been through.
- "Status" of "Pose Scans" of "Scan" tab on <ProcessIpsRun Dialog> becomes "Current" and the box is checked.

Once the creation of scans is completed, the scans are displayed on both <Map View> and <3D View>.



Example of Pose Scans

Detect intersections of the trajectory and edit loops

1. Click "Adjustment" tab on <ProcessIpsRun Dialog>.

   Confirm that "Pose Scans - Current" will be displayed on "Prerequisites" of "Detect Intersections" and the box is checked.

2. Click [Start] of "Detect Intersections".

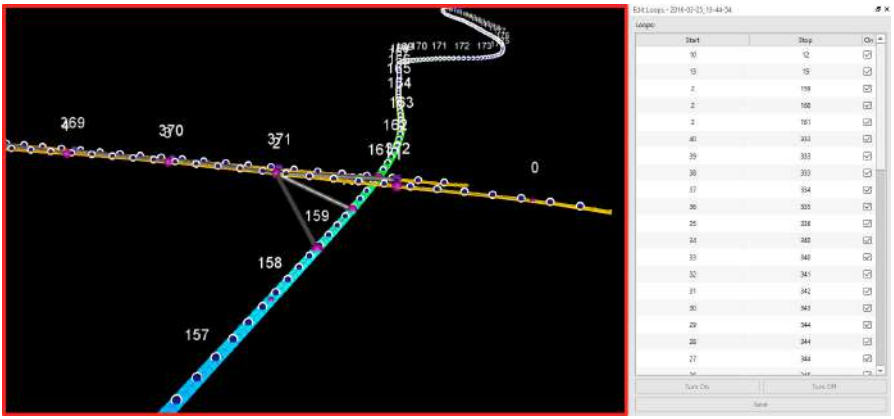
   You can confirm the completion of the intersection detection by one of the following methods.

   - "Status" on <Task Window> becomes "Completed" and "Progress" becomes "100%" after all the processing related to the intersection detection has been through.
   - "Status" of "Detect Intersections" of "Adjustment" tab on <ProcessIpsRun Dialog> becomes "Current" and the box is checked.

   You can start with the following two ways to edit loops.

   - Select a run data in which the intersection detection has been completed on <Workspace Window> and select "Edit Loops" on the right click.
   - Select a run data in which the intersection detection has been completed on <Workspace Window>, select "Fuse" tab on the ribbon menu and then click "Edit Loops"  .

   <Loop Edit Window> is displayed and red balls and gray pipes (hereinafter, pipes) are displayed on <Map View> and <3D View>.



3D View and Loop Edit Window

Each red ball has its own number, and the number corresponds to the number of "Start" or "Stop" on <Loop Edit Window>. A combination of one "Start" and one "Stop" is displayed as one pipe and indicates that scans obtained from the same intersection or both ways can be shared each other.

3. Confirm that pipes are not mis-connected between the upper road and the lower road at a grade separated intersection by referring to <Map View> or <3D View> and <Loop Edit Window>.

4. If you find a wrong pipe connection, select the pipe.

   Note

   • There are two ways to select a pipe, which are to click a corresponding pipe on <Map View> or <3D View> and to click a corresponding line of "Start" and "stop" on <Loop Edit Window>. The selected pipe is highlighted with red.

   The corresponding line of "Start" and "Stop" in <Loop Edit Window> of the selected pipe is selected.

5. Uncheck the box "On".

   Note

   • You can uncheck the box "On" by clicking [Turn Off] on <Loop Edit Window>. When you click [Turn On], the box is checked.

6. Click [Save] on <Loop Edit Window> when you complete the loop editing.

   Only a check-marked pipe on <Loop Edit Window> is the target of alignment of multiple passes.

Performing align multiple passes

1. Click "Adjustment" tab on <ProcessIpsRun Dialog>.

   Confirm that "Detect Intersections - Current" is displayed on "Prerequisites" of "Close Loops" and the box is checked.

   You can adjust the maximum gap between two scans by "Max. matching distance". The default value is 10 [m].

   Note

   • This setting is useful to avoid mismatching between two scans when there are two scans of the upper road and the tunnel that is under the road.

   When the "Enable error propagation" check box is selected, a correction amount for align multiple passes for a nonoverlapping part is evenly allocated from the overlapping parts before and after the nonoverlapping part. A trajectory that is smooth as a whole is obtained.
   When the check box is cleared, a correction amount for align multiple passes for a nonoverlapping part is decremented depending on the distance from an overlapping part.
   This check box is selected by default.

2. Click [Start] of "Close Loops".

   You can confirm the completion of the gap adjustment by one of the following methods.

   • "Status" on <Task Window> becomes "Completed" and "Progress" becomes "100%" after all the processing related to Close Loops has been through.
   • "Status" of "Close Loops" of "Adjustment" tab on <ProcessIpsRun Dialog> becomes "Current" and the box is checked.

Confirm results

1. Perform a pose scan again.

   ☞"4.2 Data Processing"-"Pose scan"

   Both the <Map View> and the <3D View> display the result of the align multiple passes processing. Zoom in the display to confirm that features (such as utility pole and sign) have no gaps in any part.

■ Reverting

If the results of the gap adjustment are unsatisfactory, you can recover the trajectory before Close Loops.

You can start with the following two ways to recover the original trajectory.

- Select a run data in which the gap adjustment has been completed on <Workspace Window> and select "Revert Trajectory" on the right-click.
- Select a run data in which the gap adjustment has been completed on <Workspace Window> and "Fuse" tab on the ribbon menu, and then click "Revert Trajectory" .

1. Execute one of the above ways.

   The confirmation dialog is displayed.

2. Click [OK].

- When you recover the original trajectory, you need to re-create scans and re-allocate panoramas to match the trajectory with the existing scan data and panoramas.

■ Adjust the trajectory by the ground control point(s)

You can adjust the trajectory by observing the GCP (GCP: Ground Control Point) whose correct coordinates are known on the scan and associating the trajectory to the known coordinate values.

1. Execute "Process Scans" based on the adjusted trajectory by Close Loops.

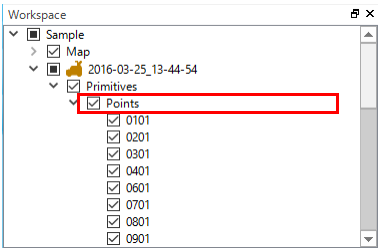
   ☞"4.2 Data Processing"-"Pose scan"

- Assign the N value of "Density (1/N)" without thinning down scans because the GCP needs to be observed correctly on the scan.

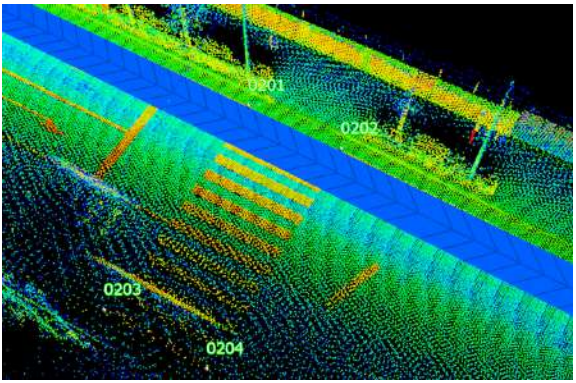
2. Import the coordinates of the GCP.

   ☞"4.5 Menu Commands" - "Fuse" tab - Import the CAD primitives

   The point name of the imported GCP on "Primitives" - "Points" under the run data on <Workspace Window>. The GCP location and the name are also displayed on <Map View> and <3D View>.



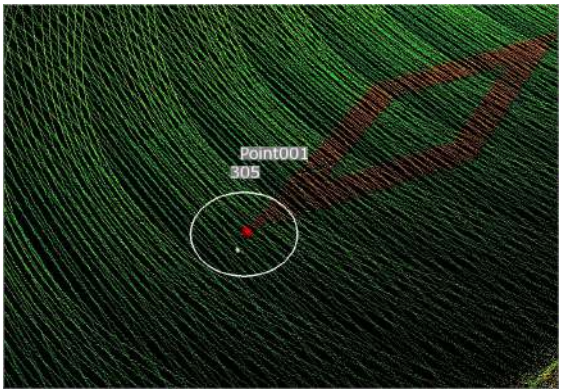
Workspace Window



Display of Point Names of Ground Control Points

3. Enable "Interpolate" of "Tools" tab on the ribbon menu to interpolate the coordinates in the space between scans.
4. You can Jump To the GCP by one of the following two methods.
   - Select a GCP to measure on "Primitives" - "Points" under the run data on <Workspace Window> and select "Jump To" on the right click.
   - Select a GCP to measure on "Primitives" - "Points" under the run data on <Workspace Window> and "Edit" tab on the ribbon menu, and then click "Jump To"  .
5. Observe the original characteristic point of the GCP on the scan by the pointer device.

☞"2.7 CAD Primitives"-"Drawing of CAD primitives"

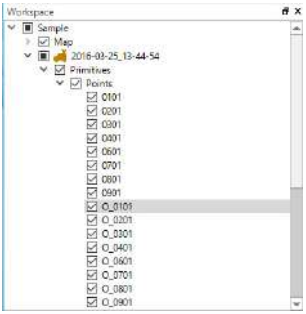


Example of measurement of GCP (measurement of apex of rhombus mark)

Note

- It is recommended to give a name identifiable as the GCP to the measured point.

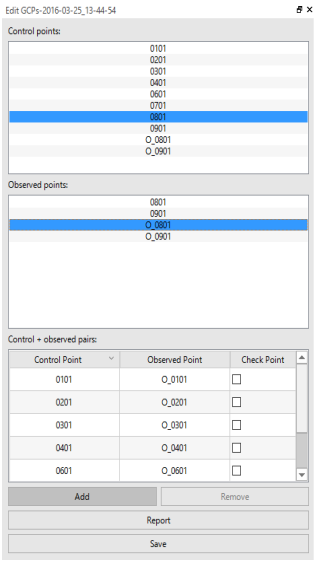
6. Perform the above 4 and 5 for each GCP.



Example with a Measurement Point Name

7. You can start with the following two ways to edit the GCP.
   - Select a run data in which the GCPs have been measured on <Workspace Window> and select "Edit GCPs..." on the right click.
   - Select a run data in which the GCPs have been measured on <Workspace Window>, select "Fuse" tab on the ribbon menu and then click "Edit GCPs"  .

   <Edit GCPs Window> is displayed.



Edit GCPs Window (GCP Correspondence Table)

8. Click a point name of a ground control point in the "Control points" list.

9. Click the observed point name corresponding to the point name of the above 8 from the "Observed points" list, and click [Add].

   The pair of the GCP and the observed point will be added to the registration in the "Control + observed pairs" list.

   Note

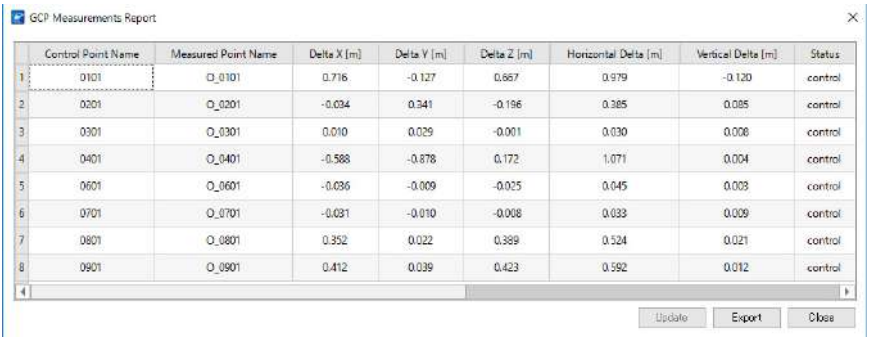
   • When you delete the registered pair of the GCP and the observed point, Click the pair on "Control + observed pairs" and then [Remove].

10. If you check "check point" of the registered pair, the pair is not used for the adjustment calculation, but is applicable to the comparison of errors in report display.

11. Perform the above 8 to 9 for each GCP and click [Save].

12. Click [Report] on <Edit GCPs Window>.

    <GCP Measurements Report Dialog> is displayed. You can check differences (coordinate, horizontal and vertical differences) between the current GCP and the observed point.



	Control Point Name	Measured Point Name	Delta X [m]	Delta Y [m]	Delta Z [m]	Horizontal Delta [m]	Vertical Delta [m]	Status
1	0101	O_0101	0.716	-0.127	0.667	0.979	-0.120	control
2	0201	O_0201	-0.034	0.341	-0.196	0.385	0.085	control
3	0301	O_0301	0.010	0.029	-0.001	0.030	0.008	control
4	0401	O_0401	-0.588	-0.878	0.172	1.071	0.004	control
5	0601	O_0601	-0.036	-0.009	-0.025	0.045	0.003	control
6	0701	O_0701	-0.031	-0.010	-0.008	0.033	0.009	control
7	0801	O_0801	0.352	0.022	0.389	0.524	0.021	control
8	0901	O_0901	0.412	0.039	0.423	0.592	0.012	control

GCP Measurements Report Dialog

• The "Horizontal Delta" and "Vertical Delta" are not exact but approximate.

• When you click [Export] on <GCP Measurements Report Dialog>, the contents of the displayed table are saved in the text file.

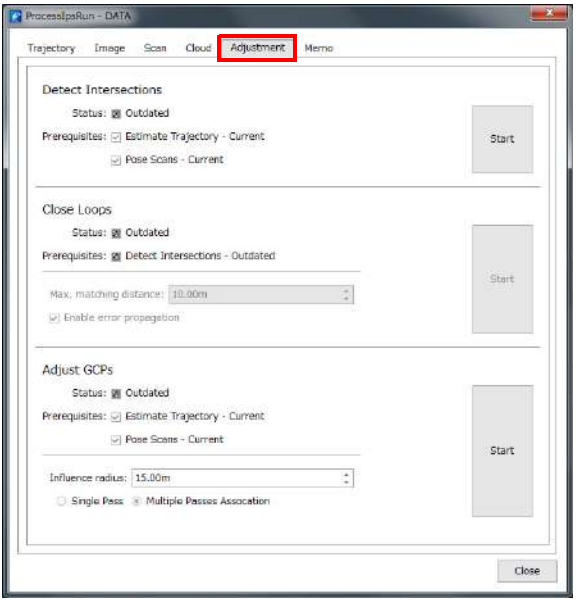
13. Click [Close] to close the dialog after you have confirmed the contents.

14. You can start with the following two ways to adjust the trajectory with the GCP.

    • Select a run data in which the GCP and the observed point have been associated on <Workspace Window> and select "Adjust Trajectory" on the right click.

    • Select a run data in which the GCP and the observed point have been associated on <Workspace Window>, select "Fuse" tab on the ribbon menu and then click "Adjustment"

"Adjustment" tab of <ProcessIpsRun Dialog> is displayed.



ProcessIpsRun Dialog (Adjustment Tab)

- Single Pass

  Each GCP correction can be used to correct each path. The GCP measurement must come from the scan associated to the pass, it is necessary to select and clip a segment on a trajectory pass to guarantee the association. Influence radius will apply to each GCP correction.

- Multiple Passes Association

  The same GCP correction can be used to correct multiple passes if these multiple passes are close enough to each other. The GCP correction will be applied to the passes within the influence radius from a GCP measurement point. The default radius length is 15m.

15. Click [Start] of "Adjust GCPs".

You can confirm the completion of the trajectory adjustment with the GCP by one of the following methods.

- "Status" on <Task Window> becomes "Completed" and "Progress" becomes "100%" after all the processing related to the trajectory adjustment with the GCP has been through.
- "Status" of "Adjust GCPs" of "Adjustment" tab on <ProcessIpsRun Dialog> becomes "Current" and the box is checked.

Confirming results

1. Execute "Process Scans" based on the adjusted trajectory with the GCP.

   ☞"4.2 Data Processing"-"Pose scan"

   • Assign the same value as that of the created scan (after the adjustment) when the GCP is observed to N value of "Density (1/N)".

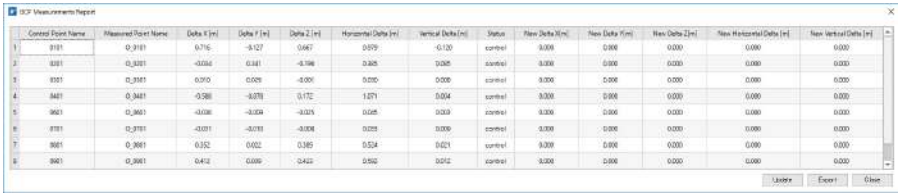
2. You can confirm the completion of the creation of scans by one of the following methods.

   • "Status" on <Task Window> becomes "Completed" and "Progress" becomes "100%" after all the processing related to the creation of scans has been through.
   • "Status" of "Pose Scans" of "Scan" tab on <ProcessIpsRun Dialog> becomes "Current" and the box is checked.

3. Open <Edit GCPs Window> by one of the following methods.

   • Select a run data in which the above scan has been completed on <Workspace Window> and select "Edit Loops..." on the right click.
   • Select a run data in which the above scan has been completed on <Workspace Window>, select "Fuse" tab on the ribbon menu and then click "Edit GCPs"  .

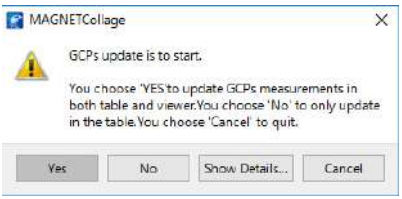
4. When <Edit GCPs Window> is displayed, click [Report] to display <GCP Measurements Report Dialog>.



GCP Measurements Report Dialog

5. Click [Update] on <GCP Measurements Report Dialog>.

   <Update Confirmation Dialog> is displayed.



Update Confirmation Dialog

[Yes] : It reflects the results of trajectory adjustment with the GCP to <GCP Measurements Report Dialog>, <Map View> and <3D View>. "_old" is added to the end of the name of the observed point before the adjustment, and that of after the adjustment is displayed with its original name.

[No] : It reflects the results of trajectory adjustment with the GCP to <GCP Measurements Report Dialog>. You can check the gap between the GCP and the observed point after the adjustment.

[Show Details...] : For the processing when [Yes] is clicked, the description is displayed.

[Cancel] : Cancels update

6. Click [Yes] or [No].

   The gap between the observed point and the GCP before the adjustment is displayed on the left side and after the adjustment is displayed on the right side of <GCP Measurements Report Dialog>.

GCP Measurements Report Dialog (Adjusted)

Reverting

If the results of the trajectory adjustment with the GCP are unsatisfactory, you can recover the trajectory before the trajectory adjustment with the GCP.
☞"4.2 Data Processing"-"Reverting"

■ Stitching panoramic image and color mapping

You can create panoramas from images of all-around camera and color scans using the color information of the panoramas (color mapping).

Stitching and pose estimation of panoramic image

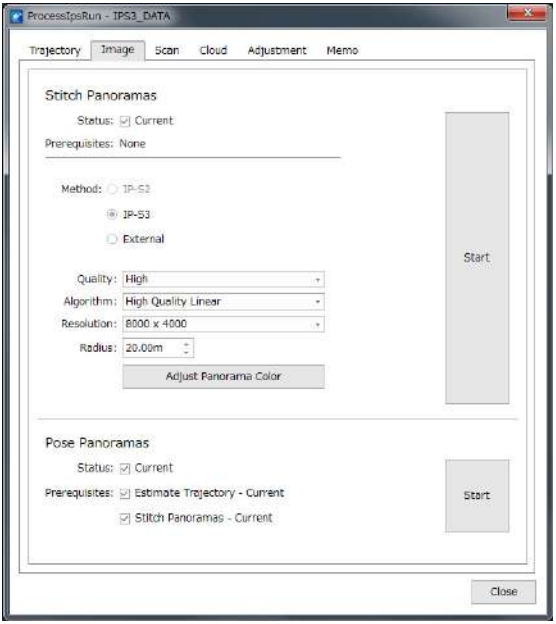
There are the following two ways to create panoramas.

- Select a run data in which the trajectory analysis or adjustment has been completed on <Workspace Window> and select "Process Images..." on the right click.
- Select a run data in which trajectory analysis or adjustment has been completed on <Workspace Window>, select "Fuse" tab on the ribbon menu, and then click "Images"  .

1. Execute one of the above ways.

   "Image" tab on <ProcessIpsRun Dialog> is displayed.

2. Select processing conditions in "Stitch Panoramas".



ProcessIpsRun Dialog (Image Tab)

Method : IP-S2 / IP-S3 / External

- When you have imported external panoramas, select "External".

Quality : High / Low

- When you select "High", you can acquire high quality panoramas, but the processing time tends to be long.

Algorithm : High Quality Linear / Bilinear / Downsampling

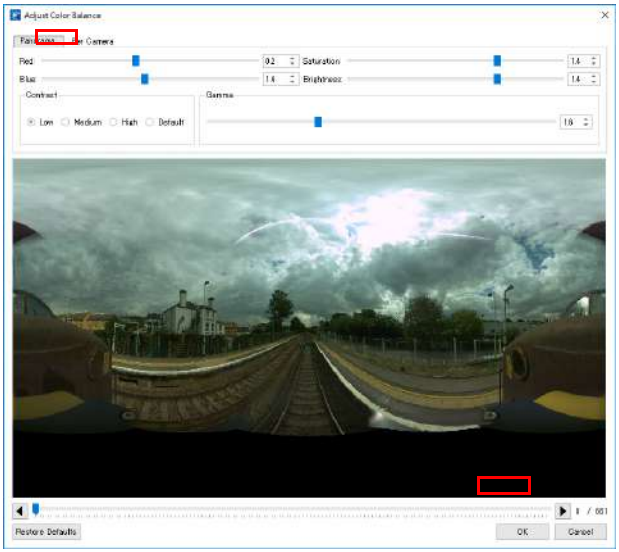
Resolution : 8000x4000 / 5400x2700 / 2048x1024

- When you select higher precision, the processing time tends to be longer.

Radius : Please properly enter the distance to the measured target to the "Radius" of "Stitch Panoramas". The gap between the image and the point cloud by parallax can be corrected.

3. Click [Adjust Panorama Color].

<Color Balance Adjustment Dialog> is displayed.
You can check panoramas taken during the run using the slider on the bottom of the dialog or ◀ / ▶.



Color Balance Adjustment Dialog (Panorama Tab)

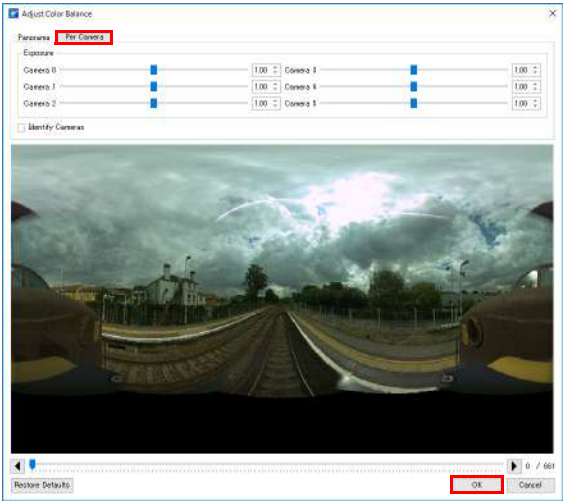
4. Select "Panorama" tab.

You can adjust the following items to the whole panorama.

Item	Adjustment Range
Red	-12.0 to 16.0
Blue	-12.0 to 16.0
Saturation	0.0 to 2.0
Brightness	0.0 to 2.0
Contrast	Low/Medium/High/Default
Gamma	0.5 to 4.0

Values displayed first on <Color Balance Adjustment Dialog> in each run data are recommendation and defaults. Click [Restore Defaults] to initialize all the items.

5. Select "Per Camera" tab.



Color Balance Adjustment Dialog (Per Camera Tab)

You can adjust the following items to each all-around camera.

Item	Adjustment Range	Initial Value
Exposure	0.80 to 1.20	1.0

When you check the box of "Identify Cameras", the camera numbers 0-5 are displayed on the image.

6. Click [OK] to close the dialog when you complete the image adjustment.
7. Click [Start] of "Stitch Panoramas" in the <ProcessIpsRun Dialog>.

   You can confirm the completion of the creation of panoramas by one of the following methods.

   - "Status" on <Task Window> becomes "Completed" and "Progress" becomes "100%" after all the processing related to the creation of panoramas has been through.
   - "Status" of "Stitch Panoramas" of "Image" tab on <ProcessIpsRun Dialog> becomes "Current" and the box is checked.

   Confirm that boxes of "Estimate Trajectory" and "Stitch Panoramas" of "Prerequisites" are checked in "Pose Panoramas".

8. Click [Start] of "Pose Panoramas" in the <ProcessIpsRun Dialog>.

   You can confirm the completion of "Pose panoramas" by one of the following methods.

   - "Status" on <Task Window> becomes "Completed" and "Progress" becomes "100%" after all the processing related to the allocation of panoramas has been through.
   - "Status" of "Pose Panoramas" of "Image" tab on <ProcessIpsRun Dialog> becomes "Current" and the box is checked.

Color mapping

You can start with the following two ways for color mapping of the scan data.

- Select a run data in which the allocation of panoramas has been completed on <Workspace Window> and select "Pose Scans" on the right click.
- Select a run data in which the allocation of panoramas has been completed on <Workspace Window>, select "Fuse" tab on the ribbon menu, and then click "Scans" 

.

1. Execute one of the above ways.

   "Scan" tab of <ProcessIpsRun Dialog> is displayed.

   Confirm that boxes of "Stitch Panoramas" and "Pose panoramas" of "Prerequisites" are checked in "Colorize Scans".

2. Click [Start] of "Colorize Scans".

   You can confirm the completion of "Colorize Scans" by one of the following methods.

   - "Status" on <Task Window> becomes "Completed" and "Progress" becomes "100%" after all the processing related to the color mapping of scans has been through.
   - "Status" of "Colorize Scans" of "Scan" tab on <ProcessIpsRun Dialog> becomes "Current" and the box is checked.

Before Color Mapping

After Color Mapping

■ Cloud generation

When you create point cloud data from the scan data, you can display large capacity data efficiently.

Note

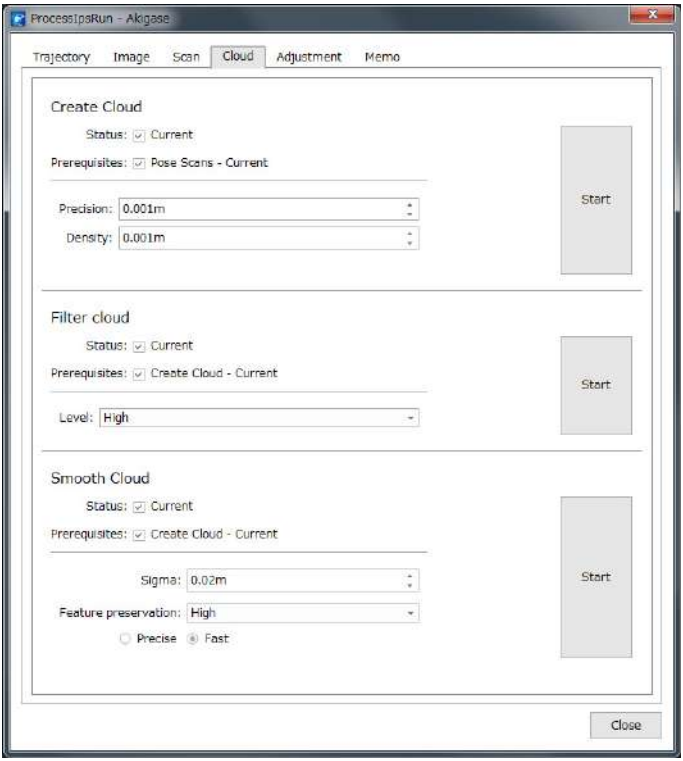
- It takes a lot of time to create the point cloud data. It is recommended to create the point cloud data after analyzing the vehicle trajectory and fixing the adjustment.

You can start with the following two ways to create the point clouds.

- Select a run data in which the allocation of scans has been completed on <Workspace Window> and select "Process Cloud..." on the right click.
- Select a run data where the allocation of scans has been completed on <Workspace Window>, select "Fuse" tab on the ribbon menu, and then click "Cloud"  .

1. Execute one of the above ways.

   "Cloud" tab of <ProcessIpsRun Dialog> is displayed.



ProcessIpsRun Dialog

2. Select the processing conditions in "Create Cloud".

   Precision : Precision can be specified.
   Density : Density can be specified.

3. Confirm that the box of "Pose Scans" of "Prerequisites" is checked in "Create Cloud".
4. Click [Start] of "Create Cloud"
5. You can confirm the completion of "Create Cloud" by one of the following methods.

   - "Status" on <Task Window> becomes "Completed" and "Progress" becomes "100%" after all the processing related to the creation of the point clouds has been through.
   - "Status" of "Create Cloud" of "Cloud" tab on <ProcessIpsRun Dialog> becomes "Current" and the box is checked.
   - After performing "Create Cloud", the "Status" of "Filter Cloud" and "Smooth Cloud" become "Current".

■ Point cloud noise removal

Noise may appear on the scan data due to the influence of the sunlight when the sunlight is strong during observing the run data. You can remove the noise from the run data in which the creation of the point clouds has been completed.

You can start with the following two ways to create the point clouds.

- Select a run data in which the creation of point clouds has been completed on <Workspace Window> and select "Process Cloud..." on the right click.
- Select a run data in which the allocation of scans has been completed on <Workspace Window>, select "Fuse" tab on the ribbon menu, and then click "Cloud"  .

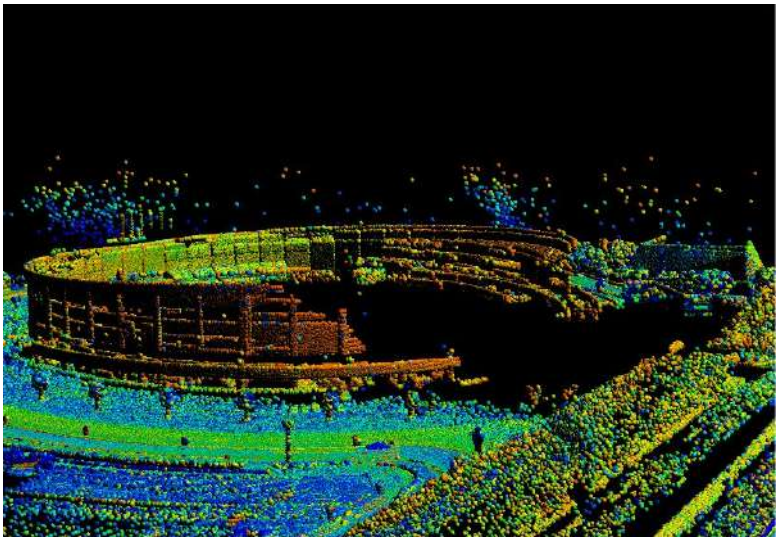
1. Execute one of the above ways.

   "Cloud" tab of <ProcessIpsRun Dialog> is displayed.

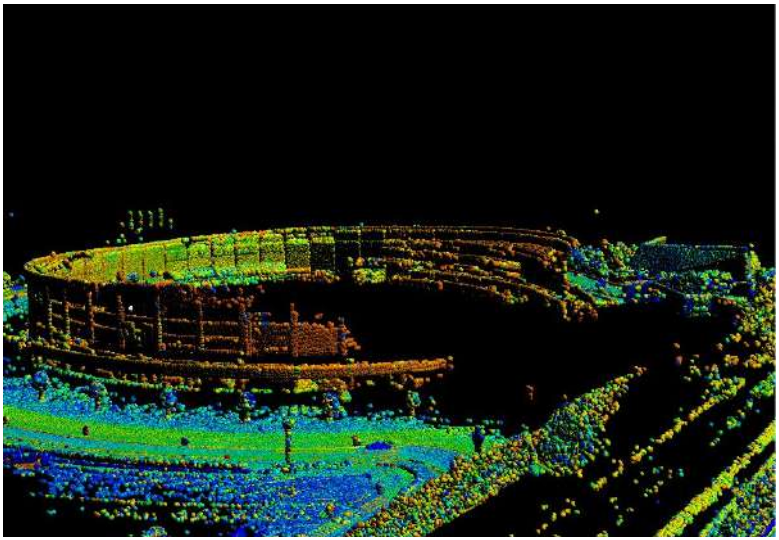
2. Confirm that the box of "Create Cloud" of "Prerequisites" is checked in "Filter cloud".
3. Assign the removal level of noise (Low / Medium / High) at "Level".
4. Click [Start] of "Filter Cloud".
5. You can confirm the completion of "Filter Cloud" by one of the following methods.

   - "Status" on <Task Window> becomes "Completed" and "Progress" becomes "100%" after all the processing related to the noise removal of the point clouds has been through.
   - "Status" of "Filter Cloud" of "Cloud" tab on <ProcessIpsRun Dialog> becomes "Current" and the box is checked.

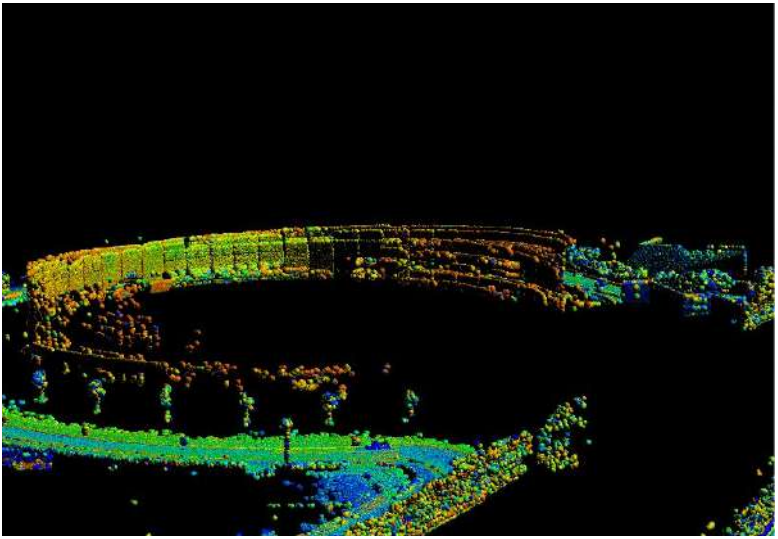
6. Display the point clouds on <Map View> or <3D View> and confirm that the noise removal of the point clouds is sufficient.



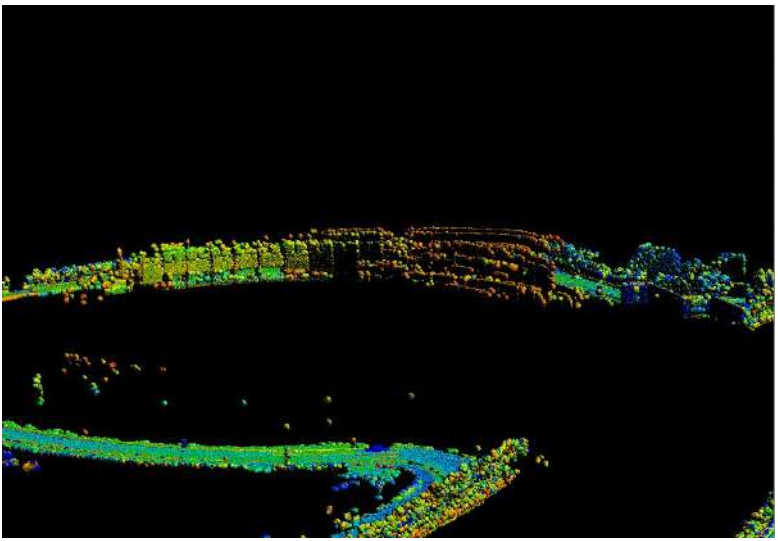
Example of Point Cloud (Before Noise Removal)



Example of Point Cloud (Noise Removal Level: Low)



Example of Point Cloud (Noise Removal Level: Medium)



Example of Point Cloud (Noise Removal Level: High)

■ Smooth Cloud

Smooth cloud filter detects flat surfaces and brings noise points near to each flat surface.

- "Smooth Cloud" will modify current cloud data. You have to perform "Create Cloud" again for restore the last state.

1. Set "Sigma" value.

   Sigma is the size of features we want to preserve. All details smaller than sigma will be considered noise and adjusted. Bigger sigma means big smoothing, zero sigma means no smoothing.

2. Set "Feature preservation" level.

   Feature preservation controls the extent of edge detection. The edge sharpness will be increased from Low/Medium/High in order.

3. Choose "Precise" or "Fast".

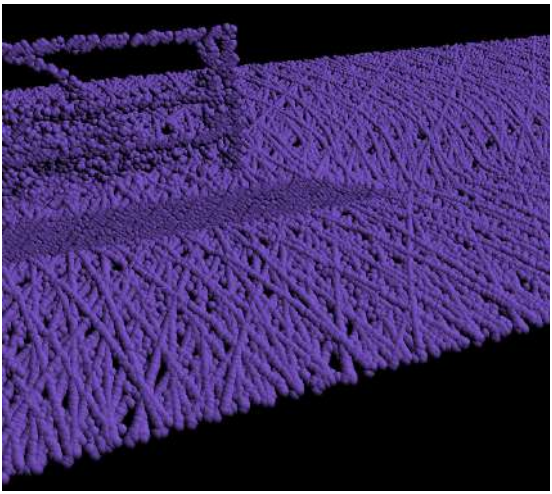
   Precise option is a little better for feature detection, Fast option is a few times faster especially for dense datasets and big Sigma.

4. Click the "Start" of "Smooth Cloud".

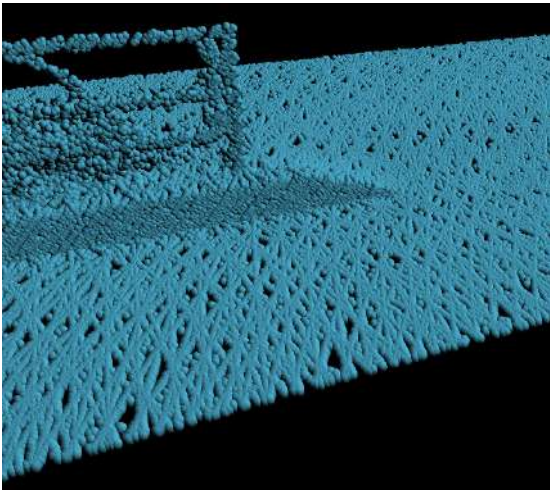
   You can confirm the completion of "Smooth Cloud" by one of the following methods.

   - "Status" on <Task Window> becomes "Completed" and "Progress" becomes "100%" after all the processing related to the smooth cloud has been through.
   - "Status" of Smooth Cloud of "Cloud" tab on <ProcessIpsRun Dialog> becomes "Current" and the box is checked.

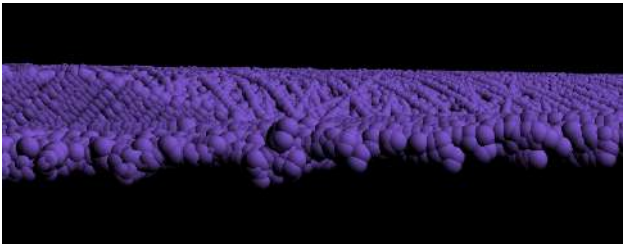
5. Display the point clouds on <Map View> or <3D View> and confirm that the smooth cloud is sufficient.



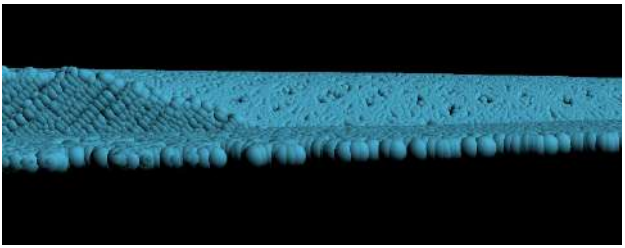
Example of point cloud (Before Smooth Cloud) - 1



Example of point cloud (After Smooth Cloud) - 1



Example of point cloud (Before Smooth Cloud) - 2



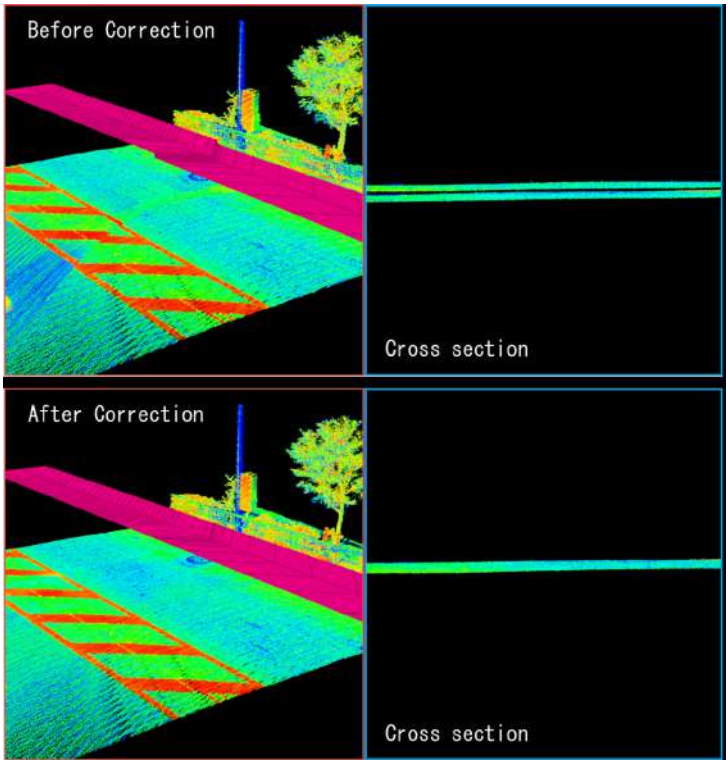
Example of point cloud (After Smooth Cloud) - 2

■ Trajectory jump correction

Correct the step (trajectory jump) of the trajectory caused by the error of the GNSS reception data while vehicle is stationary. With this function you can select the following two processes.

1. Process of simply smoothing trajectory jump
2. Combined process of smoothing trajectory jump and close loops (Auto Jump Correction)

If the trajectory jump is too high, smoothing the trajectory jump will cause the smoothed trajectory to tilt due to the step difference. By process 2., you can expect to correct the trajectory slope caused by smoothing the trajectory jump.



Comparison of Trajectory and Scan Before and After Trajectory Jump Correction

- In the case of process 2, it is necessary that the trajectory has both of an outward and a return route.

Work procedures

1. Select a run data in <Workspace Window>, right click on the run data and select "Correct trajectory jump", or select the ribbon menu "Analysis" tab and click

   "Trajectory jump correction"  .

   <Truck Jump Correction Window> is displayed on the right side of the screen.
   ☞"Trajectory jump correction"-"Explanation of trajectory jump correction window"

2. Set the "Minimum jump" and click [Detect].

   The detection result is displayed in the trajectory jump list.

3. Check the detected segment and select, modify or add a segment to execute trajectory jump correction.

   ☞For the method of selecting, modifying and adding a segment, please refer to "Trajectory jump correction"-"Explanation of trajectory jump correction window".

   • In the case of <<Combined process of smoothing trajectory jump and close loops>>, there are some restrictions on selection and editing of trajectory. ☞For details, please refer to (P.165).

   Hereafter, the work procedure differs depending on <<Process of simply smoothing trajectory jump>> and <<Combined process of smoothing trajectory jump and close loops>>. Please refer as necessary.

<<Process of simply smoothing trajectory jump>>

4. Turn Off the "Correct repeatedly" check box and click [Correct].

   The trajectory jump is smoothed. Close loops is not performed.

<<Combined process of smoothing trajectory jump and close loops>>

4. Turn On the "Correct repeatedly" check box and click [Correct].

   "Adjustment" tab of <ProcessIpsRun Dialog> will be displayed.
   ☞"Trajectory jump correction"-"Auto Trajectory Jump Correction"

5. Set the parameters related to the auto trajectory jump correction task, and click [Start].

   When the "Adjustment" tab of <ProcessIpsRun Dialog> is displayed, the default setting values are automatically set for the parameters related to the auto trajectory jump correction task. It is also possible to start processing without changing parameters.

   Auto trajectory jump correction starts.

You can confirm the completion of "Auto trajectory jump correction" by one of the following methods.

- Auto trajectory jump correction is completed, "Status" in <Task Window> is "Completed" and the progress is "100%".
- Auto trajectory jump correction is stopped by the "auto stop" function, the "Status" in <Task Window> is "completed" and the progress is "100%".

Note

- The current number of repetition is displayed in the description of <Task Window>.
- Auto trajectory jump correction task can be canceled during processing. If the task is canceled, the trajectory will revert to the result of the previous iteration. For example, if processing is canceled during the third iteration, the trajectory reverts to the result of the second iteration. You can see how many iterations were processed in the description of <Task Window>.

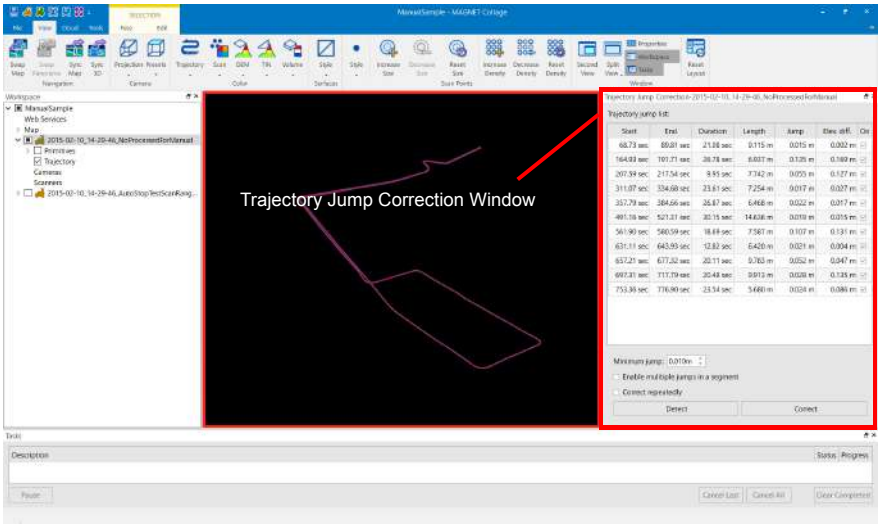
- If the "Auto stop" function is not used and the number of iteration is set too high, the resulting trajectory could start to deteriorate. It is recommended to use the default setting parameter.
- When the trajectory jump correction task is executed, the existing scan data is deleted.
- When the trajectory jump correction task is executed, the "Revert Trajectory..." operation is disabled. When returning the trajectory to the original, it is necessary to reprocess from "Estimate trajectory"
- While the trajectory jump correction task is being executed, operations on "Select", "Unselect", "Clip", and "Unclip" are prohibited for all run data.
- While the trajectory jump correction task is being executed, "Correct trajectory jumps" displayed when right clicking on traveling data in <Workspace Window>" and "Trajectory jump correction"

 in ribbon menu "Analysis" tab Is invalidated, and the trajectory jump correction task cannot be created.

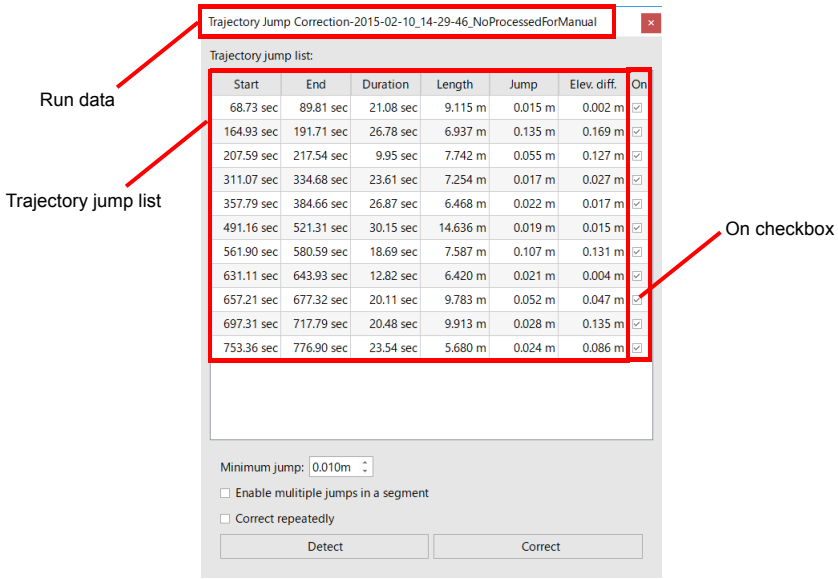
- "Edit loops" is not possible with auto trajectory jump correction. "Close loops" is done with all loops connected. However, by changing the "Max. matching distance" of the "Close Loops" parameter, it is possible to prevent unexpected adjustment of the trajectory.

Explanation of trajectory jump correction window

In <Trajectory Jump Correction Window>, it is possible to detect, edit, and select the trajectory segment.



Trajectory Jump Correction Window



Details of Trajectory Jump Correction Window

Run data : The name of the run data to be corrected is displayed.

Trajectory jump list : The selected segments are displayed.
The meanings of labels in each column of the list are as follows.

Label	Descriptions
Start	Cumulative running time at the start point of the selected segment
End	Cumulative running time at the end point of the selected segment
Duration	Time duration between Start and End
Length	The length of the selected segment
Jump	The height of the trajectory jump
Elev. diff.	The height difference between the start point and the end point of the selected segment
On	The checked segment is the correction target.

Minimum jump : Specify the minimum height difference to be detected as a trajectory jump, when [Detect] button is clicked.

Enable multiple jumps in a segment:

By turning on this, you can select multiple trajectory jumps in a segment.

Correct repeatedly : Choose the method of correction.

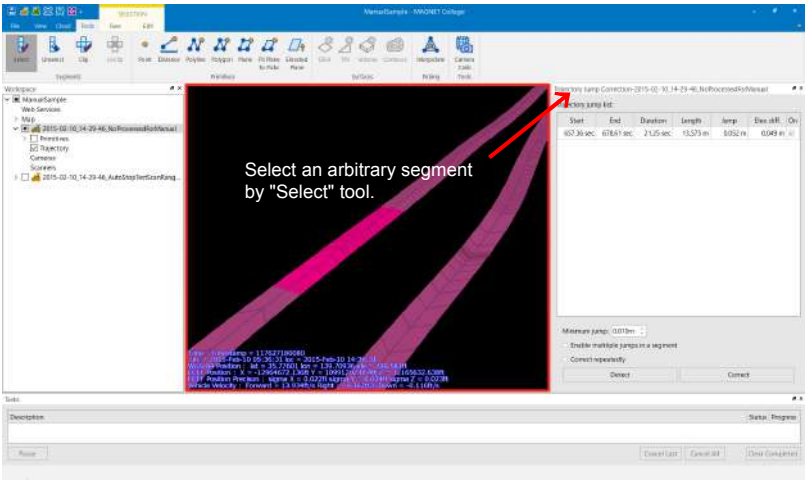
On: Process of simply smoothing trajectory jump

Off: Combined process of smoothing trajectory jump and close loops

[Detect] : Automatically detect trajectory jumps. The results will be displayed in list.

[Correct] : Trajectory correction is executed by the method selected in "Correct repeatedly" checkbox. Queue trajectory correction task if "Correct repeatedly" checkbox is off. Otherwise pops up the process dialog for further settings.

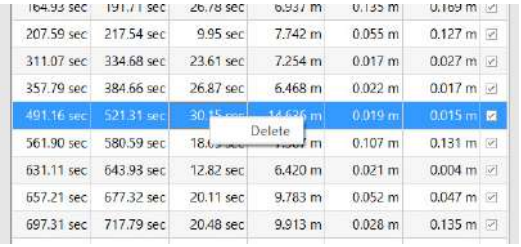
By selecting the "Select" tool in "Tools" tab of Ribbon menu, you can add an arbitrary segment to trajectory jump list by selecting an arbitrary segment.



Addition an Arbitrary Segment to Trajectory Jump List

By double clicking a trajectory jump list item, you can jump to the corresponding selected segment on the 3D view.

By right-clicking a trajectory jump list item and selecting "Delete", the item is removed from trajectory jump list and the corresponding selection is cleared on 3D view.

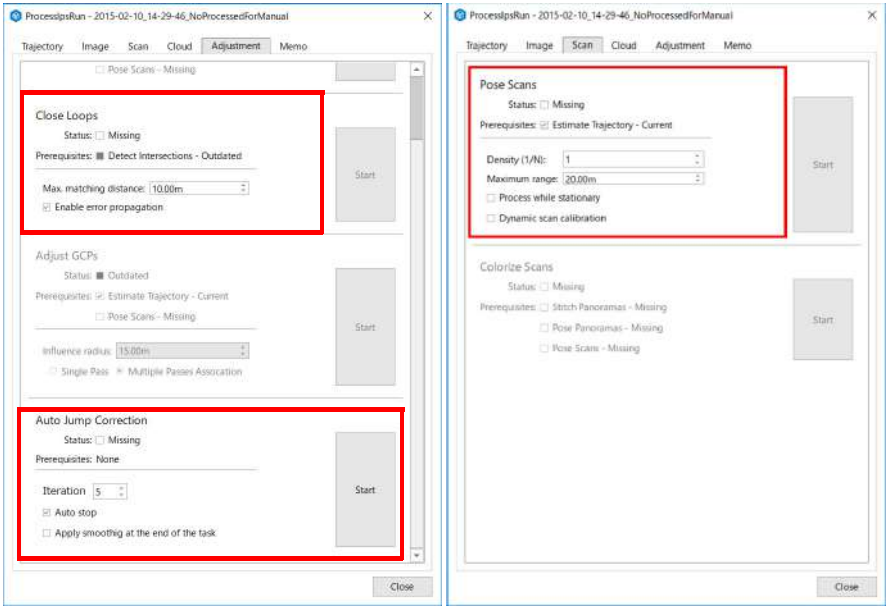


- When executing <<Combined Process of Smoothing Trajectory Jump and Close Loops>>, be sure to press [Detect] to select segments. Also, do not edit the detected segments or add new segments. If you want to choose segments to be processed, please use the delete function or the "On" check box.

Auto Trajectory Jump Correction

In the auto trajectory jump correction, the trajectory jump is gradually decreased while the trajectory is adjusted by repeating the trajectory jump smoothing and close loops process.

In the <ProcessIpsRun Dialog>, set the parameters of "Pose Scans" and "Close Loops" at the same time. However, the default setting values are automatically set when <ProcessIpsRun Dialog> is opened, so you can start processing immediately.



ProcessIpsRun Dialog (Left: "Adjustment" tab, Right: "Scan" tab)

Auto trajectory jump parameter

Iteration : The number of times to repeat trajectory jump smoothing and close loops. The default value is 5. The minimum number is 1 and the maximum number is 10.

Auto stop : When it is judged that the selected trajectory jumps have been sufficiently corrected, When enabled, the auto trajectory jump correction process will be stopped even the maximum iteration is not completed. The default is checked.

Apply smoothing at the end of the task:

After auto trajectory jump correction processing is completed, trajectory jump smoothing is performed. Normally, In the iterative correction process, close loops task is executed as the final task, but check this box On, after completing the iterative process, the trajectory jump correction is applied. The default is not checked.

☞ For the setting parameters in "4.2 Data Processing"-"Align multiple passes".

Note

- By turning off "Apply smoothing at the end of the task", you can reprocess the auto trajectory jump correction process when the correction condition by the "Auto stop" option is insufficient. At the time of reprocessing, please turn off the "Auto stop" option. Basically, reprocessing is not recommended.
- The following settings are applied when opening the task dialog.

Task	Parameter	Default value
Auto Jump Correction	Iteration	5 times
	Auto stop	On
	Apply smoothing at the end of the task	Off
Post Scans	Density (1/N)	1
	Maximum range	20m
	Process while stationary	Off
	Dynamic scan calibration	Off
Close Loops	Max. matching distance	10m
	Enable error propagation	On

■ IP-S2 run data registration

The following shows the processing flow for handling data collected by IP-S2 in MAGNET Collage.

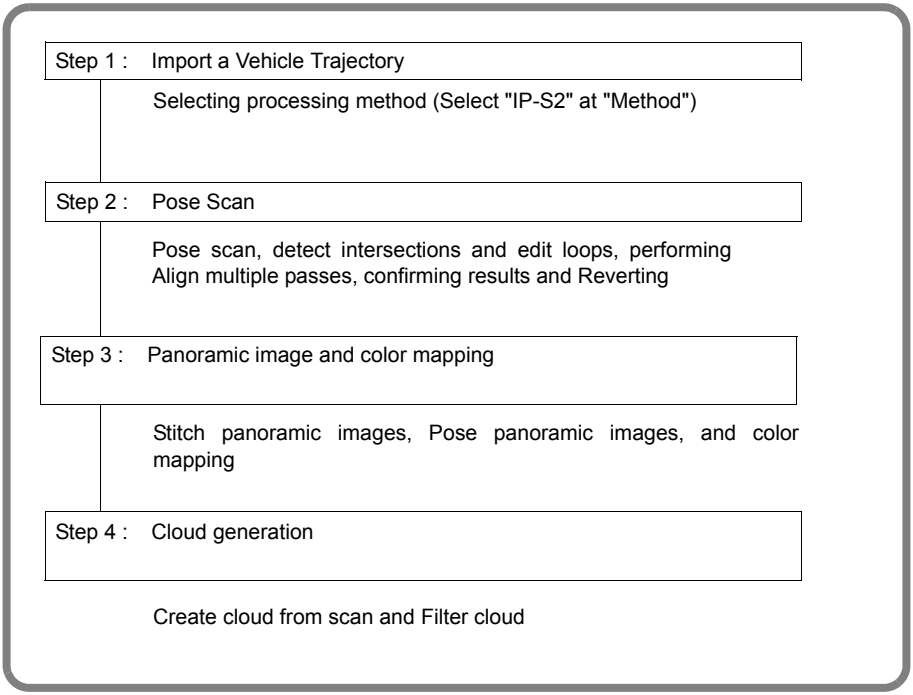
Note

- Import IP-S2 data in advance. ☞ "Import IP-S2 data"

For IP-S2 Compact+

You can import the analytical results of Geoclean and re-fuse the trajectory.

The flow of import process of the analytical results of Geoclean for is as follows:



The flow of re-analyzing process of the trajectory is as follows:

Step 1 : Import a Vehicle Trajectory

Selecting processing method (Select "IP-S3" at "Method"), setting and editing base station information, setting process parameters, performing estimations, and confirming results

Step 2 : Align multiple passes

Pose scan, detect intersections trajectory and edit loops, Align multiple passes, and confirming results

Step 3 : Align trajectory with GCPs

Pose scan, import GCPs, edit GCPs, Align trajectory with GCPs, and confirming results

Step 4 : Panoramic image and color mapping

Stitch panoramic images, Pose panoramic images, and color mapping

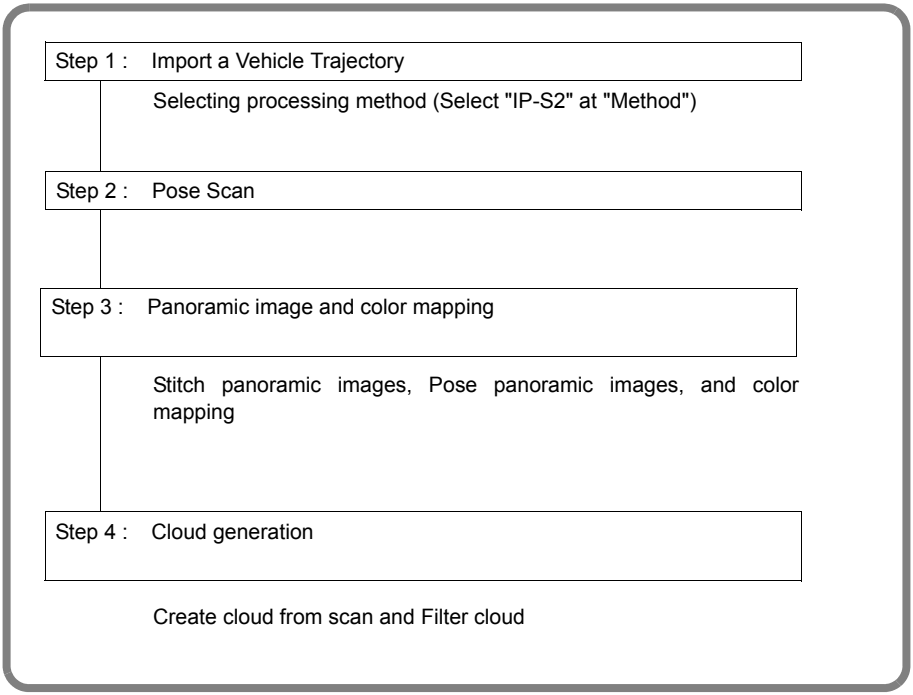
Step 5 : Cloud generation

Create cloud from scan and Filter cloud

For IP-S2

You can import the processing result of Geoclean, but cannot estimate the trajectory again.

The processing flow is as follows:



4.3 Data Editing

This section describes data editing methods after data processing.

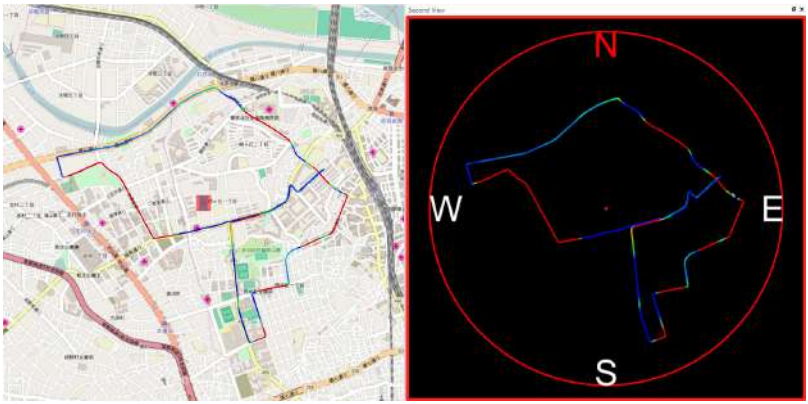
■ Jumping to trajectory location

You can Jump To the place of the imported run data and display the whole run trajectory.
There are two ways to Jump To the place of the trajectory.

- Select a run data whose trajectory you want to display on <Workspace Window> and select "Jump To" on the right click.
- Select a run data whose trajectory you want to display on <Workspace Window>, select "Edit" tab on the ribbon menu, and then click "Jump To"  .

1. Execute one of the above ways.

   You can Jump To the selected run data and the whole run trajectory data is displayed on <Map View> and <3D View>.



3D/Map View after Jumping

Note

- If <Panorama View> is displayed, it switches to <3D View> and the whole trajectory is displayed.

■ Specifying data editing segment

You can specify the segment to execute data editing. This allows you to process data in the necessary segment without any difficulty even with run data for long hours.

Specify an arbitrary segment to execute data editing.

1. Select "Tools" tab on the ribbon menu and click "Select Sections" .

2. Click end point of the section that you want to edit on <Map View> or <3D View>.

   The selected section is displayed in violet.

3. Click the other end point of arbitrary section on the trajectory.

   The selected section is displayed in violet.
   If you select multiple sections, repeat from the above 2.

Segment Selection

4. Select "Tools" tab of the ribbon menu and click "Clip" after you select sections.

The selected range is clipped.

Selected Segment after clipping

- The clipped range is reflected to the scan data as it is connected to the trajectory, but is not to the point clouds as it is not connected.

Scan after clipping

Cloud after clipping

Select "Tools" tab on the ribbon menu and click "Release Clip" to release the clip.

Select "Tools" tab on the ribbon menu and click "Release Sections" to release the whole section range.

■ Playback

The state of the observation in the run data can be played back (playback display).

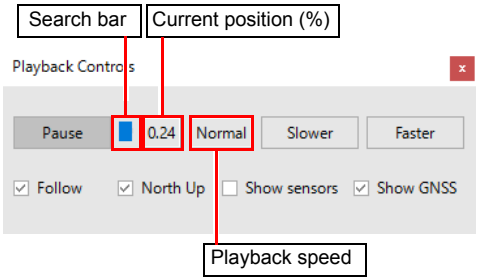
- The IP-S3 data can be played back after importing the run data or analyzing the trajectory.
- The IP-S2 data can be played back after analyzing the trajectory.

There are two ways to start playback.

- Select a run data on <Workspace Window> and select "Start Playback" on the right click.
- Select a run data on <Workspace Window>, select "Edit" on the ribbon menu, and then click "Start Playback"  .

1. Execute one of the above ways.

   The system displays <Playback Control Window> and start playback after creating time stamp file if no file necessary for the play back is available.



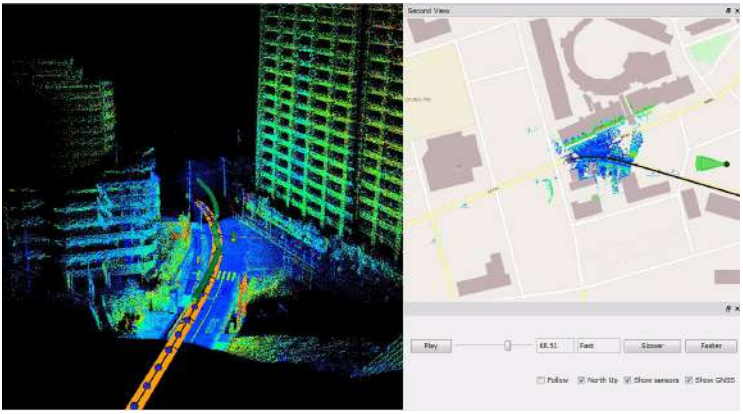
Playback Control Window

You can pause the playback with [Pause] and start it with [Play].
You can change the playback speed with [Slower] / [Faster].

When you check "Follow", the system performs playback while following the front of the trajectory.
When you check "North Up", the north direction becomes upper side of the screen on <Map View>. When you uncheck it, the running direction becomes upper side of the screen.

When you check "Show sensors", sensors  are displayed if the run data is IP-S3.

When you check "Show GNSS", the GNSS positioning  is displayed.

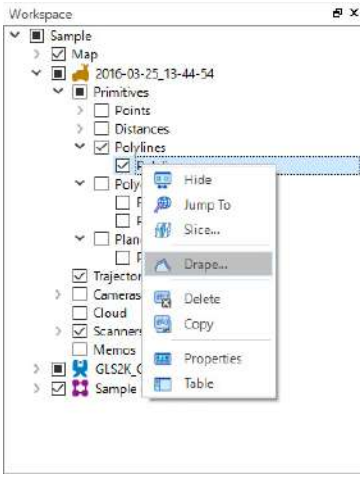


Playback

Drape

You can create a new polyline (drape) that complements the surface of the point clouds along the line assigned by the polyline after you create the point clouds.
Draw a polyline to drape.

1. Select a polyline to drape on <Workspace Window> and select "Drape..." on the right click.

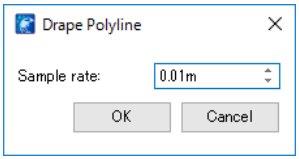


Workspace Window

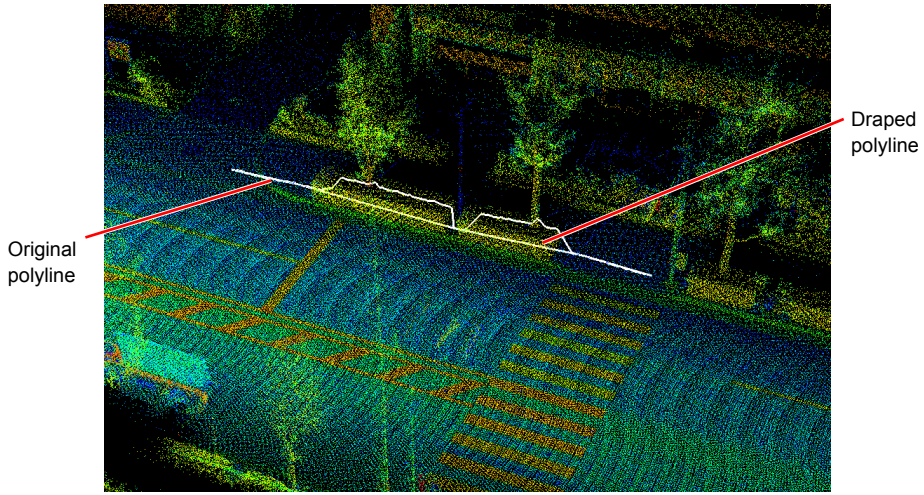
<Polyline Drape Dialog> is displayed.

2. Assign the distance between the apexes to create by the drape in "Sample rate" and click [OK].

   You can confirm the completion of polyline draping when "Status" on <Task Window> becomes "Completed" and "Progress" becomes "100%" after all the processing related to the drape has been through.

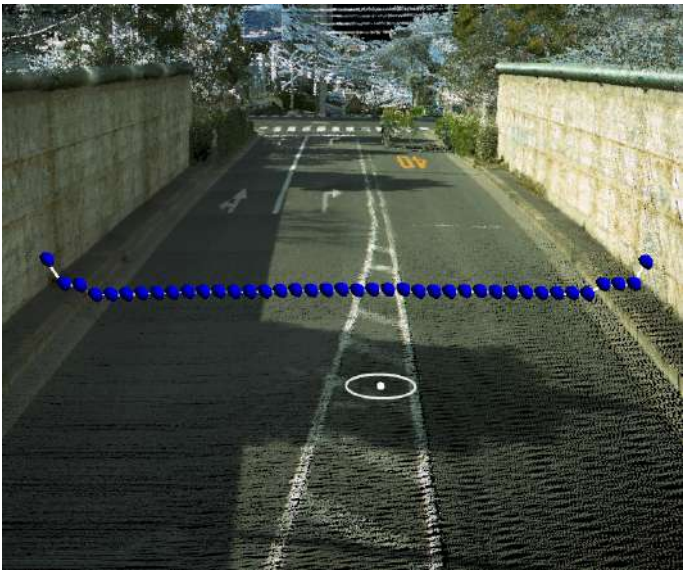


Polyline Drape Dialog



Drape Polyline and Original Polyline

Polyline before Drape



Polyline after Drape

4.4 Export Data

■ Run data file export

You can select a data type from the run data and export it in the selected file format.

Table 9: Data type/File format

Data Type	File Format
Cloud	LAS file format (.las)
	CL3 file format (.cl3)
	PCD file format (.pcd)
	RCS file format (.rcs)
	E57 file format (.e57)
	PTS file format (.pts)
	Optional text file format (.txt)
Panoramas	IP-S3 file format (.ipsx_img)
	Jpeg file format (.jpg)
	LandTrace file format (PosCam.csv)
	Text file format (.txt)
	CSV file format (.csv)
Raw GNSS	TPS file format (.tps)
	Text file format (.txt)
	CSV file format (.csv)
Scans	CL3 file format (.cl3)
	IP-S3 file format (.ipsx_scan)
	LAS file format (.las)
	Text file format (.txt)
	CSV file format (.csv)
	E57 file format (.e57)
	RCS file format (.rcs)
	PCD file format (.pcd)
	Optional text file format (.txt)
Trajectory	IP-S3 file format (.ipsx)
	KML file format (.kml)
	Text file format (.txt)
	CSV file format (.csv)
	SHP file format (.shp)

- The file extension of the import files must be lower case.

Note

- If you have selected the CSV file format (.csv) in "File format", you can select "Export" / "Skip" of "Header labels".
- If you have selected the text file format (.txt) in "File format", you can select "Export" / "Skip" of "Header labels" and a delimiter type of the data.
- When you export data in optional text file format, you can choose fields to be exported or field order. For details on data export, please refer to "2.6 Edit of Clouds" - "Output to a file".
- If you have selected the editing range of the run data in advance, the selected rang is to be exported. ☞ "4.3 Data Editing"-"Specifying data editing segment"

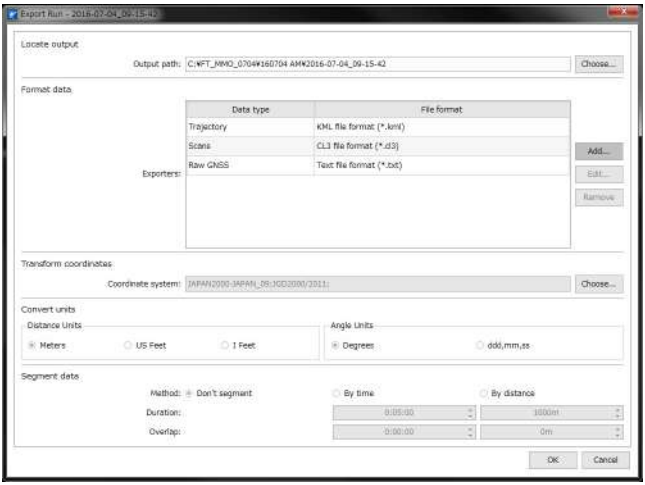
- You cannot select the editing range of the run data for the point cloud data (.las) (.cl3) and the GNSS data (.tps).

You can start with the following two ways to export the run data file.

- Select a run data on <Workspace Window> and select "Export..." on the right click.
- Select a run data on <Workspace Window>, select "Edit" tab on the ribbon menu, and then click "Export" .

1. Execute one of the above ways.

   The following <Run Data Export Dialog> is displayed.

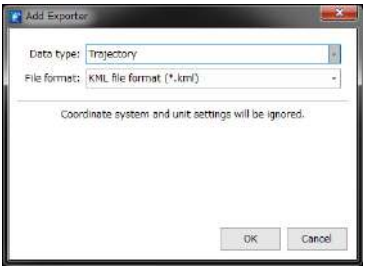


Run Data Export Dialog

2. Assign a folder to send the file to in "Output path".

3. Click [Add...].

   The following <Output Items Add Dialog> is displayed.



Output Items Add Dialog

4. Select a data to export in "Data type" and a file format in "File format" and then click [OK].

   If you add "[Items]", repeat from the above 3.

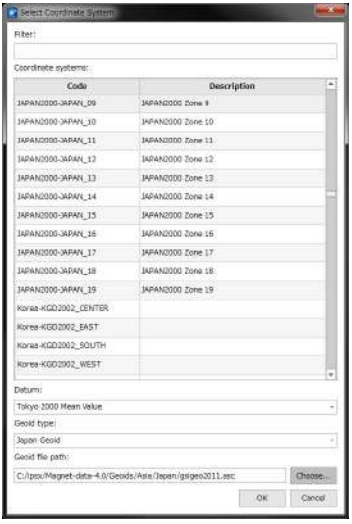
   Note

   • You cannot register a combination of "Data type" and "File format" in "Exporters" twice.

5. Click [Edit...] to edit the registered "Data type" and [Remove] to delete it.
6. Click [Choose...] of "Transform coordinates".

   The following <Coordinate System Selection Dialog> is displayed.

7. Select an arbitrary coordinate system on the list and click [OK].



Coordinate System Selection Dialog

Note

- You can narrow down the coordinate system list by specifying a filtering character string in "Filter".
- If geoid height correction is to be performed, specify "Geoid Type" and "Geoid file path".
- A geoid file that is selected for the first time is converted to an original file format (***.gff) and a gff file is saved in the location to save the selected geoid file. For the second selection or later, the converted gff file can be used.

8. Assign units of the distance and the angle in "Convert units".

- When you import the exported data in the ScanMaster, the system supports "Meters" only for the distance unit.
- ScanMaster and ImageMaster do not support the coordinate system "WGS84 BLH".

9. Assign a method in "Segment data" to divide the data into multiple files.

Method

By time	:	Divide the data by the assigned time and assign the time to duplicate. Time unit: (hour: minute: second)
By distance	:	Divide the data by the assigned distance and assign the distance to duplicate. Distance unit: meter

10. When you click [OK], the run data is exported.

- If you import the exported data in ImageMaster, you need to match the units of "Convert units" and ImageMaster.
  ☞ Refer to the user manuals of ImageMaster for setting the unit of ImageMaster.
- If you import the exported data in LandTrace-Evoluto MMS Edition,
  - cl3 type scan or cloud data (data that can be read at once is up to 3 million points)
  - jpg type panoramic data (only image sizes 2048x1024, 5400x2700)
  - LandTrace file type panoramic data

  Only [meter] is supported for the unit of data.

4.5 Menu Commands

■ Fuse tab

Function	Description
Trajectory 	Display "Trajectory" tab of <ProcessIpsRun Dialog>.
Images 	Display "Image" tab of <ProcessIpsRun Dialog>.
Scans 	Display "Scan" tab of <ProcessIpsRun Dialog>.
Cloud 	Display "Cloud" tab of <ProcessIpsRun Dialog>.
Adjustment 	Display "Adjustment" tab of <ProcessIpsRun Dialog>.
Edit Loops 	Display <Loop Edit Window>.
Edit GCPs 	Display <Edit GCPs Window>.
Revert Trajectory 	Recover the previous trajectory.
Import 	Import the CAD primitives from the file ☞ “Import CAD primitives”
Export 	Export the CAD primitives to the file ☞ “Export CAD primitives”

Import CAD primitives

Import the CAD primitives.

There are two ways to import CAD primitives.

- Select a run data from which you want to import CAD primitives on <Workspace Window> and select "Import CAD primitives..." on the right-click.
- Select a run data from which you want to import CAD primitives on <Workspace Window>, select "Fuse" tab on the ribbon menu, and then click "Import"  .

1. Execute one of the above ways.

   The following <Import CAD Primitives Dialog> is displayed.

2. Click [Choose...] in "Locate file".

Import CAD Primitives Dialog

The <File Dialog> is displayed.

3. Select a file to import.

Note

- Select a file format to import on the list in the right bottom of the file dialog. The following shows the data format that you can import.

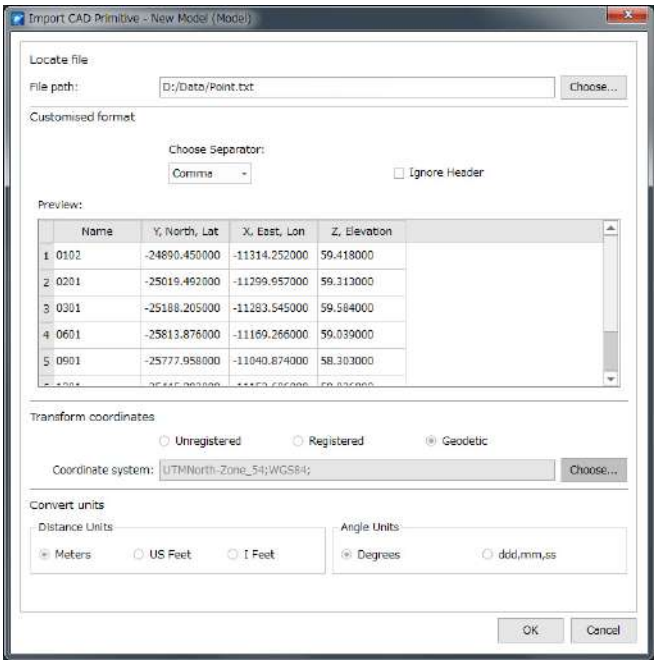
  AutoCAD 2000 Drawing (*.dwg)
  AutoCAD DXF (*.dxf)
  LANDXML (*.xml)
  MAGNETXML (*.mxl)
  ESRI Shape (*.shp)
  Points: name,n,e,z (*.csv)
  Points: name,e,n,z (*.csv)
  Choose your format from UI (*.txt)

- Object names cannot be imported in DWG and DXF formats.
- Up to 10 characters are supported for the object name of the SHP file.
- Polylines are only supported in dxf, dwg, xml, mxl or shp, polygons are only supported in xml, mxl or shp.
- Supported LANDXML versions are 1.0, 1.2 or 2.0.
- The importing data in LANDXML or MAGNETXML can only process 3D coordinate system data.
- CSV file must conform to the following format:
  (1) Enter values in order of Point Name, E, N, Z (or Point Name, N, E, Z) with separated by commas.
  (2) Please do not add header rows to a file.

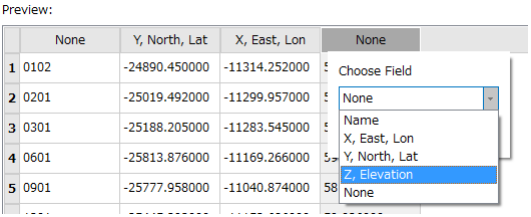
```
g01, 3917689.251, 330132.202, 321.154
g02, 3917676.326, 330127.672, 321.095
g03, 3917689.193, 330112.372, 321.110
g07, 3917726.219, 330106.073, 321.796
g08, 3917715.901, 330100.196, 321.761
g10, 3917698.849, 330121.684, 321.299
g11, 3917698.513, 330104.317, 321.524
```

Example of CSV File

When you import data in custom text format, select "Choose your format from UI" on the data type list.



Choose Separator	:	You can choose the filed separator from the character of Comma, Tab, Space, Semicolon or Custom. When you choose the custom character, you can specify an arbitrary character.
Ignore Header	:	When the file you want to import has a header line, this option should be checked to ignore the header.
Specify the field data	:	When clicking a column on the fixed row, <Choose Filed List> will appear. You can choose the appropriate field.



4. Click [Choose...] of "Transform coordinates".

   <Coordinate System Selection Dialog> is displayed.

5. Assign "Coordinate systems", "Datum", "Geoid type" and "Geoid file path" of the file to import, and then click [OK].

6. Assign units of the file to import in "Distance units" and "Angle units".

7. Click [OK].

   The CAD primitive is imported.

Export CAD primitives

Export the CAD primitives.

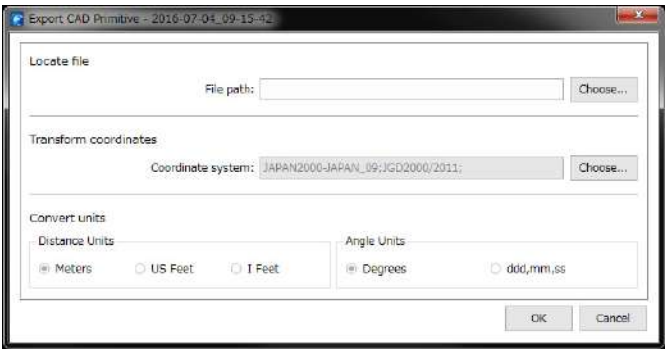
There are two ways to export CAD primitives.

- Select a run data to which you want to export the CAD primitive on <Workspace Window> and select "Export CAD primitives..." on the right-click.
- Select a run data to which you want to export the CAD primitive on <Workspace Window>, select "Fuse" tab on the ribbon menu, and then click "Export"  .

1. Execute one of the above ways.

   The following <Export CAD Primitives Dialog> is displayed.

2. Click [Choose...] of "Locate file".



Export CAD Primitives Dialog

The <File Dialog> is displayed.

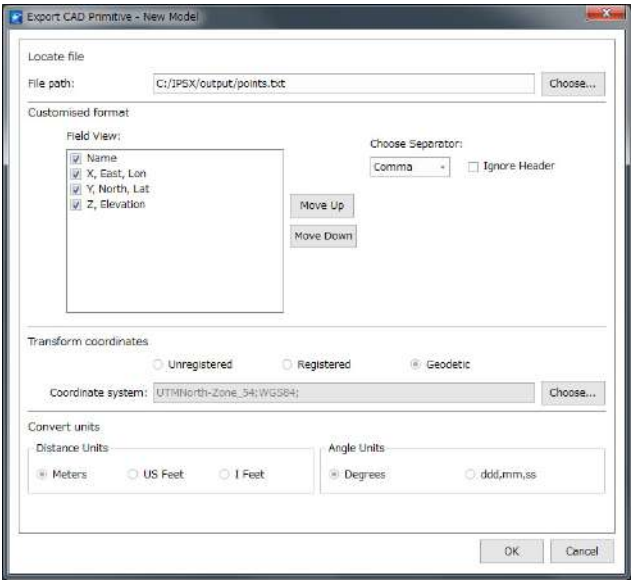
3. Select a file to export.

Note

- Select a file format to export on the list in the right bottom of the file dialog.
  The following shows the data format that you can export.
  - AutoCAD 2000 Drawing (*.dwg)
  - AutoCAD DXF (*.dxf)
  - LANDXML (*.xml)
  - MAGNETXML (*.mxl)
  - ESRI Shape (*.shp)
  - Points: name,n,e,z (*.csv)
  - Points: name,e,n,z (*.csv)
  - Choose your format from UI (*.txt)

- Up to 10 characters are supported for the object name of the SHP file.
- Polylines are only supported in dxf, dwg, xml, mxl or shp, polygons are only supported in xml, mxl or shp.
- Supported LANDXML versions are 1.0, 1.2 or 2.0.

When you export data in text format, you can choose fields to be exported or field order.



Fields and order	:	The checked field items will be exported in order from the top. The order can be swapped by clicking [Move Up] or [Move Down].
Choose Separator	:	You can choose the filed separator from the character of Comma, Tab, Space, Semicolon or Custom. When you choose the custom character, you can specify an arbitrary character.
Ignore Header	:	When this option is checked, the header line doesn't output to the file.

4. Click [Choose...] of "Transform coordinates".

   <Coordinate System Selection Dialog> is displayed.

5. Assign "Coordinate systems", "Datum", "Geoid type" and "Geoid file path" of the file to export, and then click [OK].
6. Assign units of the file to export in "Distance units" and "Angle units".
7. Click [OK].

   The CAD primitive is exported.

■ Edit tab

Function	Description
Show All 	Display all hidden fields.
Hide All 	Hide all displayed fields.
Jump To 	Display the selected items on <Map View> and <3D View>.
Slice 	Scan or clouds can be displayed by cutting it as an arbitrary cross-section surface.
Distance 	Create a distance item from two point primitives.
Set Scope 	Click one of Split Views and select arbitrary run data in <Work Space Window> and click "Set Scope". Then, two views can be displayed simultaneously.
Color Range 	Set the displayed color of the trajectory.
Playback 	Play the observation in the run data.
Remove 	Remove the run data registered in the workspace.
Clean Up 	Delete the analytical results. ☞"Delete the analytical results"
Import External Images 	Import external panoramas.
Export 	Output the run data (panoramas, GNSS observed values, trajectories, point clouds and scans) in file.
Export Orthophotos From Cloud 	Output of orthophotos using the color information of the clouds.
Show in Explorer 	Display the items that have been selected on the windows explorer.
Properties 	Display <Property Window> of the selected item.

Table		Display sub-class properties of the selected item in a table format.

Set a range of trajectory color

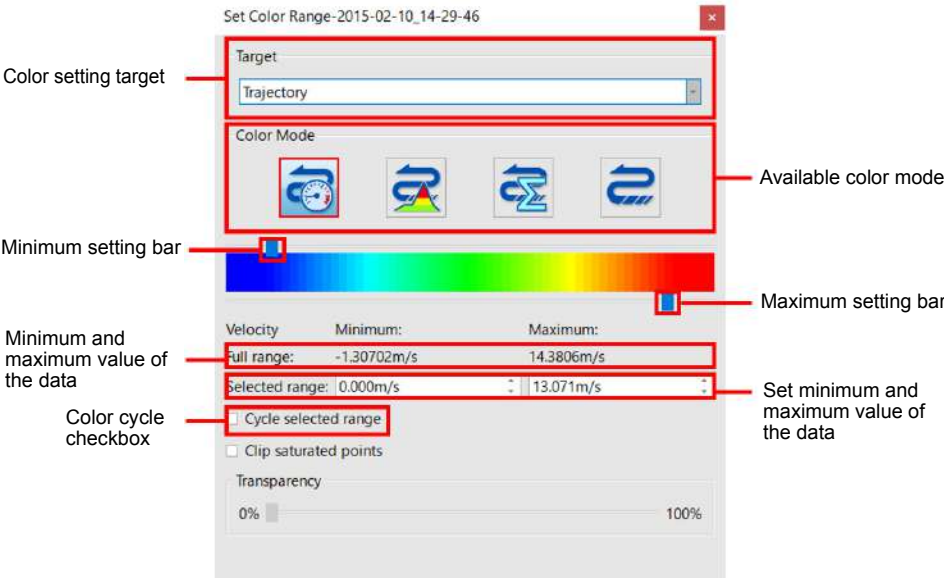
This section explains how to adjust the display color of the trajectory.

There are two ways to display <Set Color Range Window>.

- Select a run data on <Workspace Window> and select "Edit Color Range..." on the right click.
- Select a run data on <Workspace Window>, select "Edit" tab on the ribbon menu and then click "Color Range"  .

1. Execute one of the above ways.

   The following <Set Color Range Window> is displayed.



Set Color Range Window

You can select attributes that you want to edit by clicking "Velocity"  / "Elevation"  / "Sigma"  in "Trajectory Color".

Note

• When you click "Flat"  , the trajectory is displayed in a single color.

2. You can select a color range by the following two methods.

• Adjust the numeric range in "Selected range".
• Adjust the numeric range with the sliders located the top and the bottom of the color bar.

Note

• When you check "Cycle selected range", the color in the selected range can repeatedly be displayed.

3D view (Example that the trajectory color is assigned with "Velocity")

Remove run data

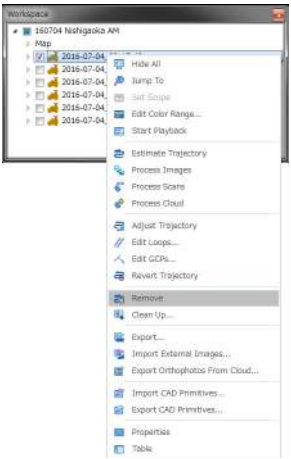
Remove the run data imported in the workspace.

Note

- In this operation, the registration of the run data is released form the work space, but the run data itself is not deleted. Use the explorer to delete the run data itself.
- When you re-import the removed run data on the workspace, the data is imported in previous status.

There are the following two ways to remove the run data.

- Select a run data to delete on <Workspace Window> and select "Remove Run Data" on the right click.
- Select a run data to delete on <Workspace Window>, select "Edit" tab on the ribbon menu, and then click "Remove"  .



Workspace Window

1. Execute one of the above ways.
2. When the <Warning Dialog> is displayed, click [OK].

   The run data will be removed.

Warning Dialog

Delete the analytical results

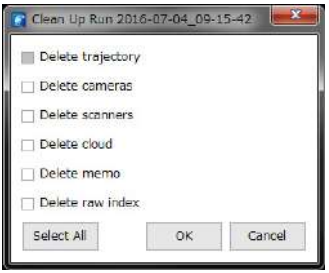
Delete a part or the whole of the results acquired from the run data analysis.

There are two ways to delete the analytical results.

- Select a run data on <Workspace Window> and select "Clean up..." on the right click.
- Select a run data on <Workspace Window>, select "Edit" tab on the ribbon menu, and then click "Clean Up"  .

1. Execute one of the above ways.

   The < Clean Up Run Dialog> is displayed.



Clean Up Run Dialog

Check applicable boxes if you want to delete a part of the analytical results.
Click [Select All] if you want to delete all of them, and click [OK].

You can confirm the completion of delete of the analytical results when "Status" on <Task Window> becomes "Completed" and "Progress" becomes "100%".

Import external images

You can import external image data such as images processed by image processing software.

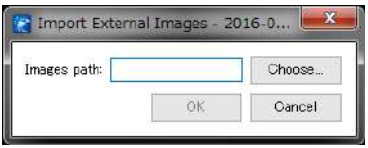
There are the following two ways to import external image data.

- Select a run data from which you want to import images on <Workspace Window> and select "Import External Images..." on the right-click.
- Select a run data from which you want to import images on <Workspace Window> and select "Edit" tab on the ribbon menu, and then click "Import External Images"  .

1. Execute one of the above ways.

   The following <External Image Import Dialog> is displayed.

2. Click [Choose...] and select a folder where images are saved.



External Image Import Dialog

- You should have saved panoramas of the run data (JPEG format) in the assigned folder.
- The image data file name should be like
  ladybug_panoramic_000000.jpg
  ladybug_panoramic_000001.jpg
  Add a frame number and save.

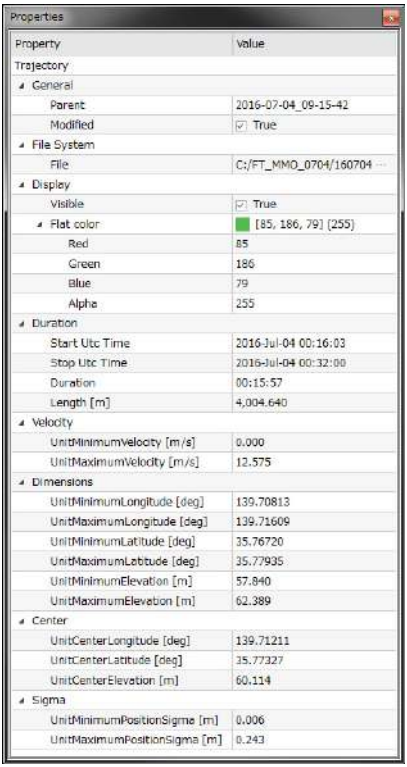
3. Click [OK] on <External Image Import Dialog>.

   The import of the image data starts.

Display properties

<Property Window> displays detailed information related to the selected item on <Workspace Window>.
An example that "[Trajectory]" is selected is as follows.

General	:	If a trajectory is changed, "Modified" is check-marked.
File System	:	Displays the file path.
Display	:	Indicates display/hide. You can switch display and hide by checking and unchecking it.
Duration	:	Displays the start/stop time of the run data (UTC), run time, distance (m).
Velocity	:	Displays the minimum and maximum velocities (m/s).
Dimensions	:	Displays the minimum and maximum latitudes (deg), longitudes (deg), and elevations (m).
Center	:	Displays the center values of the latitudes, longitudes, and elevations.
Sigma	:	Displays the minimum and maximum position and velocity sigmas.



Property Window

Display property table

There are the following two ways to display property table.

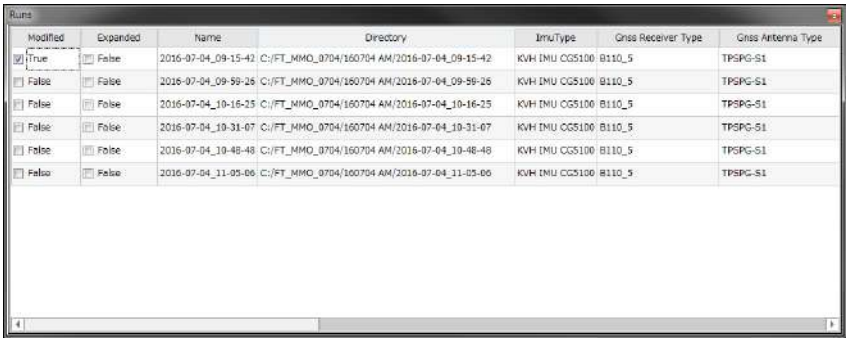
- Select a run data on <Workspace Window> and select "Table" on the right click.
- Select a run data on <Workspace Window>, select "Edit" tab on the ribbon menu and then click "Table"  .

1. Execute one of the above ways.

   The <Run Data Table> is displayed.

   Note

   • When you select multiple run data, a list of properties are displayed on <Run Data Table>.



Property Display of Multiple Run Data

5. EDITING POINT CLOUDS

This chapter describes the edition of the point clouds generated from each sensor. Integration of the point clouds output from sensors is done in model project.

5.1 Data Import

Prepare "Model" or "Model Set" for each cloud to be integrated, and load the data.
"Model Project", "Model" and "Model Set" each has a coordinate system. Load the data in the same coordinate system onto the same "Model". Correct the slightly misaligned clouds by reading them in other model and using any method shown in "5.2 Integration of Data from Different Projects".

■ Preparation of a new model project

1. Create an empty folder in the directory to save data.

   • Carry out this processing on Explorer.

2. Click "New model project" from menu [File] -> [Data], or select the root item in workspace and click "Create a model project" from "Fuse" menu.

■ Addition to an existing model project

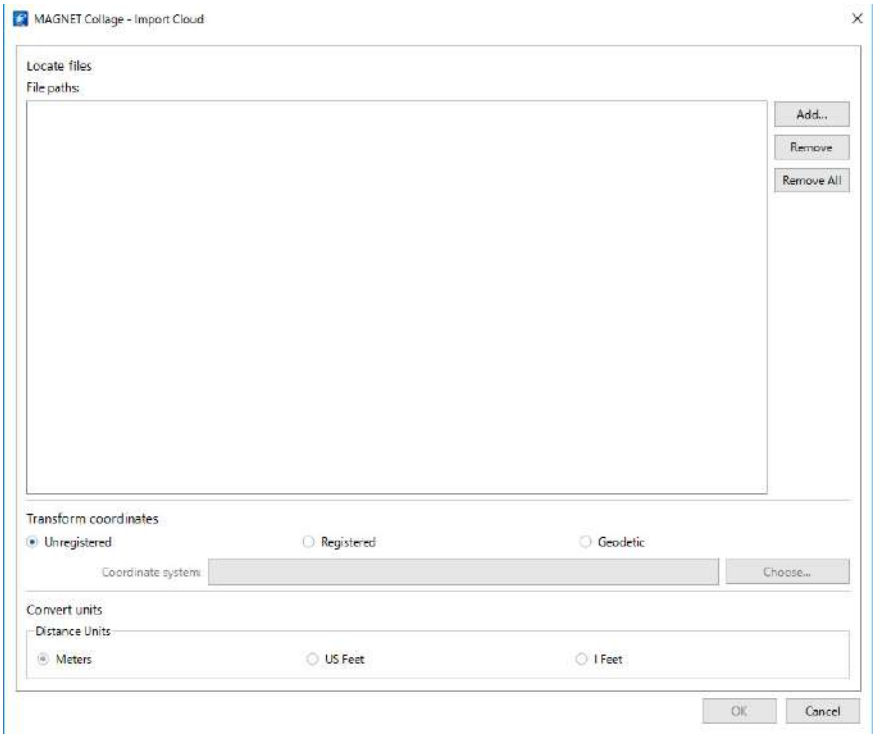
Add the model project to the Workspace.
There are the following three ways to add the model project.

- Select "File" tab on the ribbon menu and "Data", then click "Add Existing Model Project..." .
- Select a workspace on <Workspace Window> and click "Add model" on the quick access.
- Select a workspace on <Workspace Window>, select "Fuse" tab on the ribbon menu and then click "Add a model project" .

1. Execute one of the above ways.
2. Select a model project file (*.mpj) and click [Open].

■ Loading the clouds data from file

1. Click the right mouse button on "Model" node for which you want to load clouds on the workspace or click the menu [Fuse] -> [Import Cloud]  while "Model" node is selected.
   <Clouds Import Dialog> is displayed.



Clouds Import Dialog

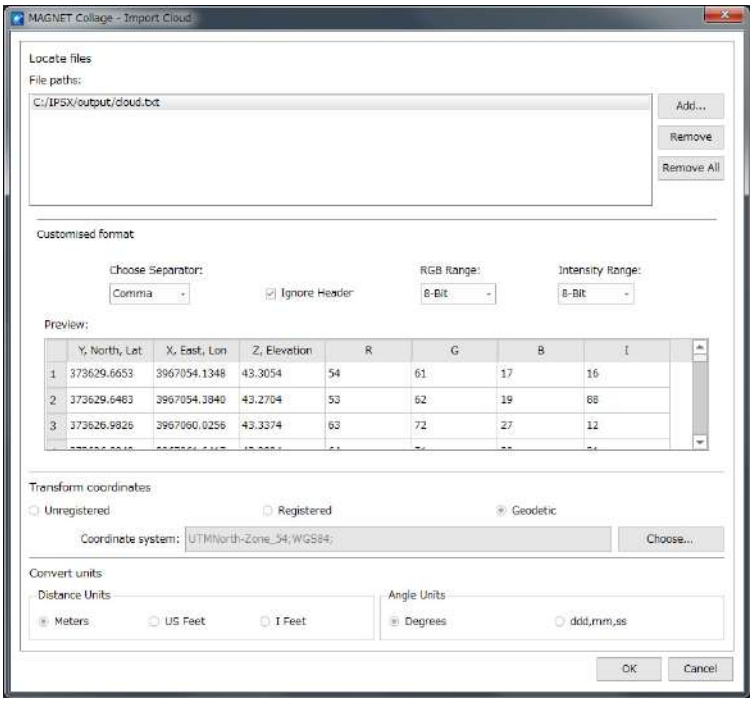
2. Click [Add...] and specify the point clouds to be loaded.

☞ The point clouds formats that can be loaded are shown below. If necessary, set up the coordinate system.

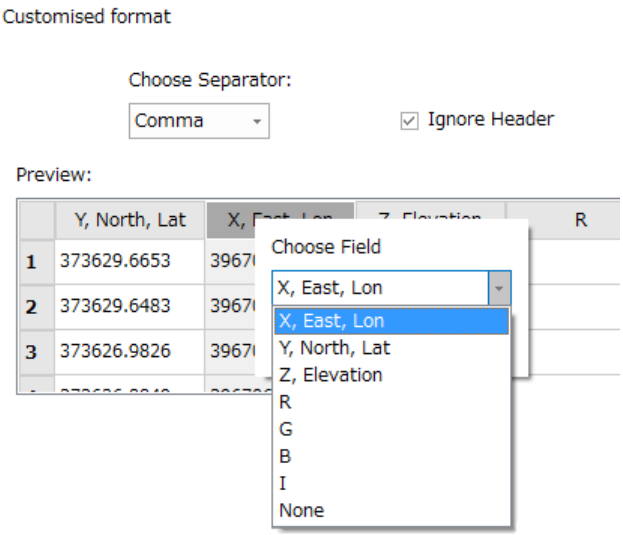
Data Type	File Format
Point Clouds	PCD file format (.pcd)
	PTS file format (.pts)
	PTX file format (.ptx)
	CL3 file format (.cl3)
	E57 file format (.e57)
	FLS file format (.fls)
	CLR file format (clr)
	LAS file format (.las)
	TXT file format (.txt)

- The file extension of the import files must be lower case.

When you chose a text format file on the file selection dialog, you can specify fields to be imported by clicking the text format file in the list of file paths.



Choose Separator	:	You can choose the filed separator from the character of Comma, Tab, Space, Semicolon or Custom. When you choose the custom character, you can specify an arbitrary character.
Ignore Header	:	When the file you want to import has a header line, this option should be checked to ignore the header.
RGB Range	:	You can choose the range of RGB from the options of 0.0~1.0, 8 bit (0~255) or 16 bit (0~65535).
Intensity Range	:	You can choose the range of intensity from the options of 0.0~1.0, 8 bit (0~255) or 16 bit (0~65535).
Specify the field data	:	When clicking a column on the fixed row, <Choose Filed List> will appear. You can choose the appropriate field.



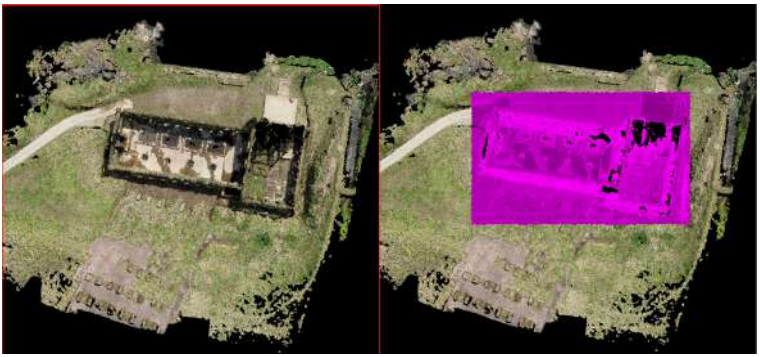
3. When setup is completed, click [OK].

   Loading of clouds data is started.

■ Loading point clouds data from another project

1. Select menu [Cloud] -> [Rectangle]  or [Polygon]  and select the point clouds on the 3D view.

   The color of the selected point changes and only the point whose color is highlighted will be copied.



2. Select the "Model" node in the "Model project" to be added on the workspace and click menu [Fuse] -> [Create Cloud].

   The point clouds will be added to the selected model.

5.2 Integration of Data from Different Projects

There are three different methods to integrate the point clouds from different sensors, which are "Manual" , "Cloud-to-Cloud" and "Targets and Georeferencing" .

Each of these methods is described below:

- The order of integration is important when integrating point clouds. Determine the order in advance so that the other point clouds are integrated with the point clouds with highest accuracy as the base.

Note

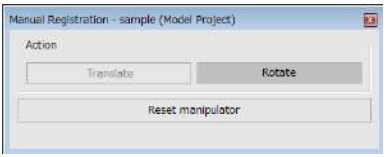
- This process becomes easier by setting the point clouds color to flat.

  Select "Flat" from menu [View] -> [Scan] .

■ Manual integration

1. Click the <Model> node to change the coordinates on workspace and click [Fuse] -> [Manual] on the menu.

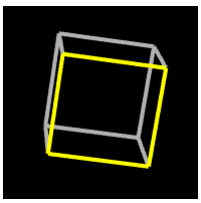
   <Manual Registration Dialog> is displayed.

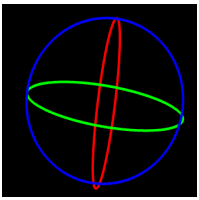


Manual Registration Dialog

2. Select [Translate] or [Rotate].
3. Double-click the center of movement while pressing the {shift} key to display a sphere or a cube.

   Sphere or cube is displayed.
   In case of [Translate], a cube is displayed. In case of [Rotate], a sphere is displayed.





4. Drag the sphere or cube while pressing the {shift} key.

   The coordinate system that changed to yellow when red, green or blue indicating the coordinate system is clicked rotates for the sphere. For the cube, it moves to the direction of the plane whose color has changed to yellow.
   Execute rotation or Translate to match the base model.

5. Close the <Manual Registration Dialog>.

   The processing is terminated.

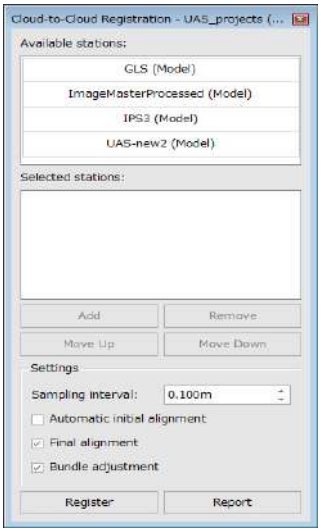
■ Cloud-to-Cloud

Point clouds are integrated automatically. This function can be used if a rough position is correct.

- If there is a large shift between the point clouds to be integrated, execute this function after implementing "Manual integration" or "Target Registration and Georeferencing".

1. Select the <Model Project> to execute Cloud-to-Cloud and click menu [Fuse] -> [Cloud-to-Cloud] .

   <Cloud-to-Cloud Registration Window> will open.



Cloud-to-Cloud Registration Window

2. Select the "Model" node name displayed in "Available stations" or "Selected stations" and execute [Add] or [Remove].

Note

- The model indicated on top will be the base model. Move the model you want to set up as the base to the top by clicking [Move Up].

3. Set up the items on the window and click [Register].
4. After the processing of <Task Window> is completed, click menu [Fuse] -> [Accuracy Report]  .

   The registration results can be checked.

■ Target Registration and Georeferencing

Here explains the method of registration at a target, which is created for each model at control point.
Create a common point (tie point) between models to be registered.
Prepare the common point at characteristic parts of the point clouds (places where the location can be identified such as a corner of a building or the end of a line), or on a marker for which control point survey has been conducted.

1. Select the model node to prepare the common point on <Workspace Window> and click [Tools] -> [Point]  on the menu to left-click while pressing the {ctrl} key on the <3D View>.
2. Execute the process in the above 1 for other models to be registered and prepare the common point.

   Note

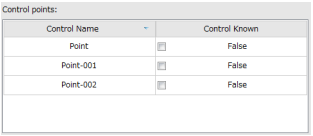
   - The work can be carried out easily by using "Split View" mode.
     ☞"Split view"
3. Select the <Model Project> node on <Workspace Window> when the models are to be registered for the control point, and then select menu [Fuse] -> [Import]  .
   < Import CAD Primitives Dialog >is displayed.
4. Specify the settings and execute loading the control point.

   ☞"Reading from a file and writing in a file"
5. Select the "Model Project" node which includes the model to be registered on <Workspace Window> and click menu [Fuse] -> [Targets and Georeferencing].

<Target Registration and Georeferencing Window> is displayed.

Target Registration and Georeferencing Window

6. Select the control point on Control points.



7. Select more than one point of each model corresponding to the selected control point while pressing {Ctrl} key from "Observed targets" and click [Add].

   The point will be added to the field "Control + observed pairs".

   • Add to the pairs of known point and observation point for at least three control points. Accuracy will be higher as the number of added point is larger.

8. Click [Register] on <Target Registration and Georeferencing Window>.

   Processing will be executed, and the <Registration Success Dialog > will appear when completed. The processing results are displayed by clicking the [Report] on the <Target Registration and Georeferencing Window>.

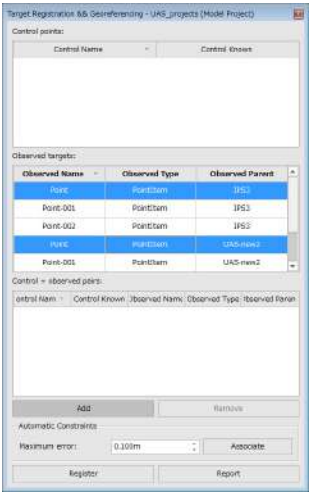
■ Target Registration and Georeferencing (without control point)

Here explains the registration method of common points (tie points) set between models.

☞ The procedure for creating tie points is same as “Target Registration and Georeferencing”. See 1 to 3 above.

4. Select the "Model Project" node which includes the model to register on <Workspace Window>, and click menu [Fuse] -> [Targets and Georeferencing].

   <Target Registration and Georeferencing Window> is displayed.



5. Select multiple points at the same place prepared in each model from <Observation points> while pressing the {ctrl} key, and click [Add].

   The points will be added to the field "<Control + observed pairs>".

   • Add to the pairs of known point and observation point for at least three control points. Accuracy will be higher as the number of added point is larger. A provisional point is added to "Known points" in this case.

6. Click [Register] on the <Target Registration and Georeferencing Window>.

   Processing will be executed, and the <Registration success dialog> will appear when completed. The processing results are displayed by clicking the [Report] on the <Target Registration and Georeferencing Window>.

■ Complicated registration

☞ For details on complicated registration, refer to 3.4 Registration of Clouds-"Complicated registration".
Chapter 4.4 of the station set has the same function as the "Model Set" of the model project and the station has the same function as the "Model" of the model project.

■ Cancelling registration

1. Select the <Model Project> or <Model> node on the workspace and click [Fuse] -> [Cancel] on the menu.

   The coordinate system only for the selected node will return to the initial state.

2. To check whether the registration is released or not, right-click the node and click [Property].

   The [Transformation] item will be "0."

5.3 Menu Commands

■ Fuse tab

Function	Description
Cloud 	Adjust the density and precision of created cloud.
Cancel 	Cancels the model registration and restores the initial state. It can be used for "Model Project" or "Model" node on <Workspace Window>.
Manual 	"Model Project" or "Model" can be manually moved in parallel direction or rotated.
Leave interval 	Assigns the data for each model with arbitrary interval.
Cloud-to-Cloud 	Matching of point clouds is executed. Point clouds in the models selected on < Cloud-to-Cloud Registration Window> will be registered.
Targets and Georeferencing 	A common point among the models is prepared, and registration is executed so that the common point would come to the same position. In model,
Quick Geo Reference 	Project the data analyzed in the geodetic coordinate system on the map roughly.
Accuracy report 	Indicates the accuracy of the model registration results. The names of the registered models "Cloud Name", rate of overlap between point clouds "Overlap", and the amount of shift between points when registration is executed using tie points "Point to Point RMS" are displayed.
Import Models 	Loads an existing model project.
Create Model Set 	Prepared a new model set.
Create Model 	Prepares a new model.

Import 	Loads points, distance, polyline, polygon, or planes into a basic figure.
Export 	Outputs points, distance, polyline, polygon, or planes. The output format can be selected from AutoCAD 2000 Drawing(*.dwg), AudoCAD DXF(*.dxf), ESRI Shape(*.shp), and Points name,n,e,z(*.csv).

■ Edit tab

Function	Description
Show All 	Switches between view/hide for all the ite. ms in the project in which the node selected on Workspace is registered.
Hide All 	The node selected on <Workspace Window> is displayed on screen.
Jump To 	Display the selected items on <Map View> and <3D View>.
Set Scope 	Click one of Split Views and select arbitrary run data in <Work Space Window> and click "Set Scope". Then, two views can be displayed simultaneously.
Color Range 	Display color can be edited. ☞ "Set a range of trajectory color"
Distance 	Create a distance item from two point primitives.
Remove Item Workspace 	Remove the Model project registered in the workspace. It does not mean deleting Model project data from PC.
Delete 	Delete the Model station registered in the workspace. Selected Model project will be deleted from PC.
Change Name 	Changes the name of the node. It can be used for model nodes.
Export 	Execute export of point clouds.

Export Orthophotos From Cloud 	Generates the ortho for the point clouds that is displayed.
Show in Explorer 	Display <Property Window> of the selected item.
Properties 	Display sub-class properties of the selected item in a table format.
Table 	Display sub-class properties of the selected item in a table format.

6. COLLAGE WEB

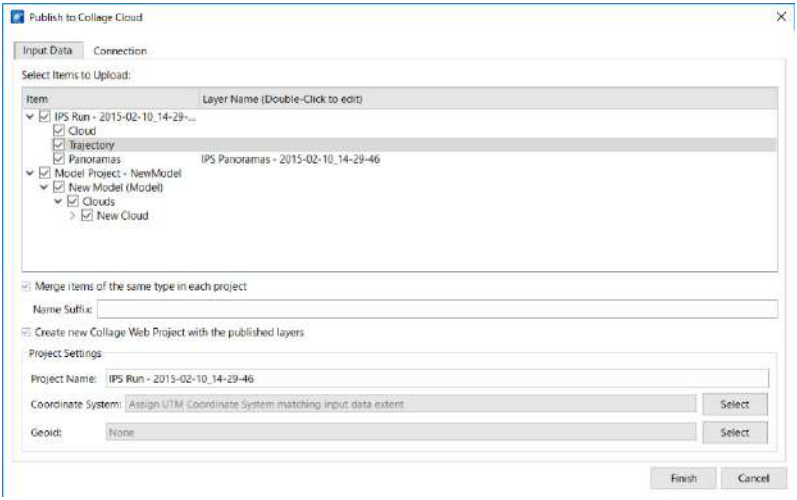
This chapter describes two MAGNET Collage's functions for Collage Web : uploading data from MAGNET Collage to Collage Web and reading Collage Web's data to see it in MAGNET Collage viewer.

6.1 Data Upload from MAGNET Collage

1. Right-click some item in <Workspace Window> and then click "Publish to Collage Web...", or select some item in <Workspace Window> and then click "Publish"  in "Edit" tab.

   The <Publish to Collage Web> is displayed.

2. Check the data you want to upload.



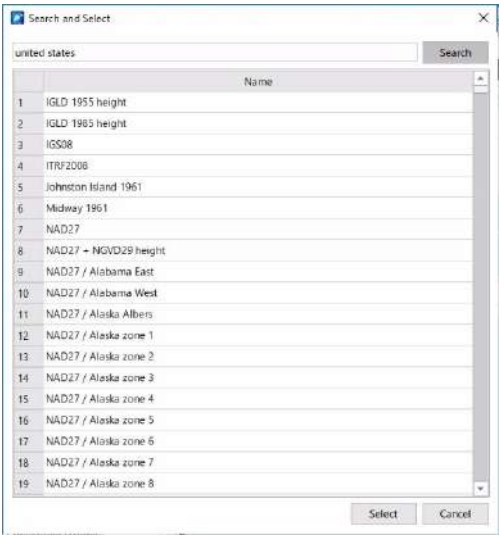
Publish to Collage Web Dialog (Input Data tab)

If "Merge items of the same type in each project" is checked, same type data in each project of MAGNET Collage will be merged into one layer and uploaded.
A layer name suffix can be set for all the merged layers.

If "Create new Collage Web Project with the published layers" is checked, new Collage Web project which contains the uploaded layers will be created.
The Coordinate System and Geoid can be manually set for the project. Setting a custom Coordinate System overrides the automatic calculation that is done normally.

Set the coordinate system for the project to click [Select] by the right side of coordinate system text box.

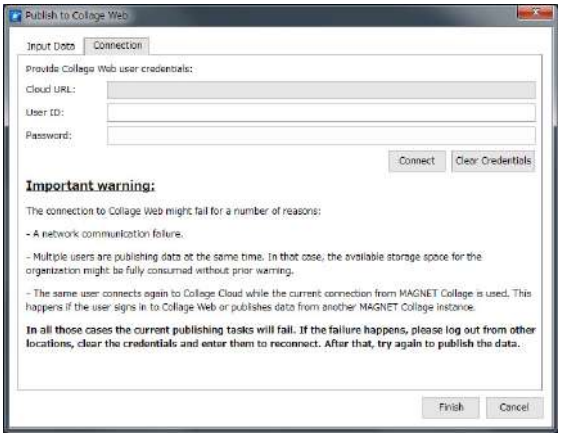
<Search and select dialog> will be displayed.



- If the search bar is empty, coordinate system list isn't displayed.
- At least a whole word is needed for search.
- The keywords are coordinate system, ESPG code, country and states.

Input a keyword to the search bar and click the [Search]. Coordinate system list will be displayed. Choose a coordinate system and click [Select].

3. Input your "User ID" and "Password" in "Connection" tab and click [Connect].



Publish to Collage Web Dialog (Connection tab)

- The connection to Collage Web might fail for a number of reasons:
  - A network communication failure. When you communicate over public Wi-Fi or mobile data connection, the network can becomes unstable, in that case the connection to Collage Web might be disconnected.
  - Multiple users are publishing data at the same time. In that case, the available storage space for the organization might be fully consumed without prior warning.
  - The same user connects again to Collage Cloud while the current connection from MAGNET Collage is used. This happens if the user signs in to Collage Web or publishes data from another MAGNET Collage instance.

  In all those cases the current publishing tasks will fail. If the failure happens, please log out from other locations, clear the credentials and enter them to reconnect. After that, try again to publish the data.

4. After login successfully done, click [Finish] to start uploading.

   The "User ID" and "Password" fields will be cleared by clicking [Clear Credentials].

   These are data types in each system you can upload.

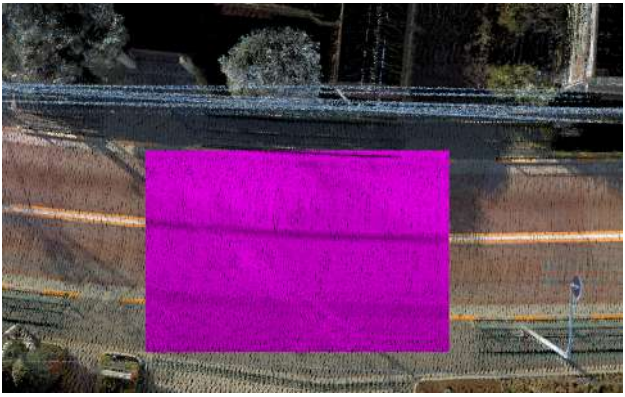
System	Data
IP-S	Point Cloud, Panorama Image, Trajectory, Primitives
GLS	Point Cloud, Panorama Image, Primitives
Model	Point Cloud, Primitives

   - If your previous upload was not completed, you will be asked if you want to try to finish the previous upload or start over the processing, creating a new layer in the server.

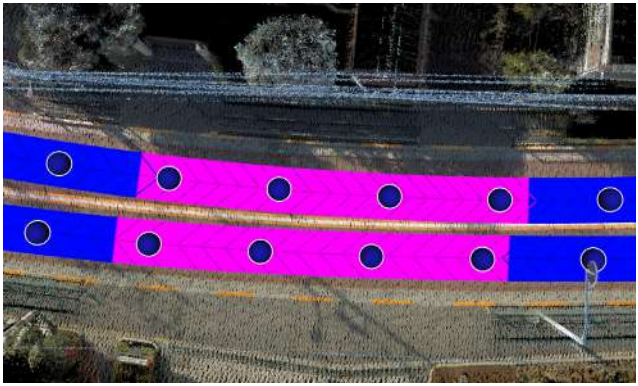
6.2 Advice before Uploading the Data

1. Limit the data to be exported.

   For point clouds, the Cloud Selection tools allow to choose a portion of the data in the Map or 3D views. ☞ "2.6 Edit of Clouds"-"Selection and edit of clouds"



   For panoramas and trajectories, a segment or multiple segments of the trajectory can be selected by using the Select tools in the Tools tab.
   ☞ "4.3 Data Editing"-"Specifying data editing segment"



   The publication of point clouds, panoramas and trajectories will be automatically limited to the selected points and trajectory segments.

2. Simplifying the point clouds.

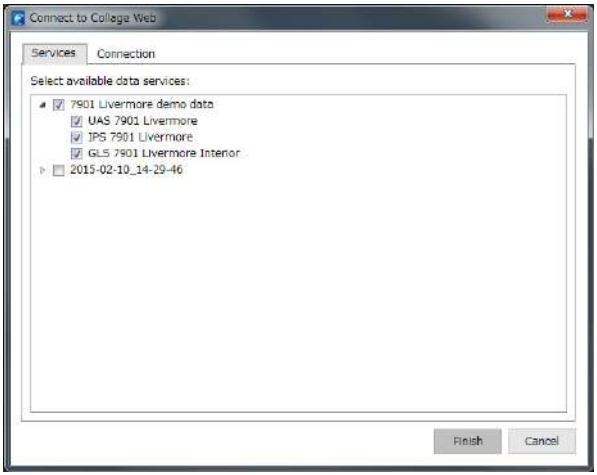
   To reduce the storage and streaming consumed, the user may publish point clouds with reduced density. The appropriate density will depend on the target application: feature extraction, cross section extraction, visual inspection, etc.
   ☞ "2.6 Edit of Clouds"-"Creating clouds and adjustments"

6.3 Open from Collage Web

1. Right-click "Web Services" in <Workspace Window> and then click "Open from Collage Web...", or select some item in <Workspace Window> and then click "Add"  in "Fuse" tab.

   The <Connect to Collage Web> is displayed.

2. Check the data you want to read from Collage Web.

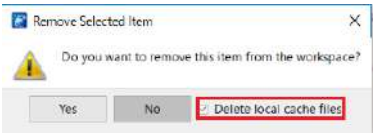


Connect to Collage Web Dialog (Services tab)

3. Input "User ID" and "Password" in "Connection" tab and click [Finish] to start reading data.

Note

- The files downloaded from the point clouds added to the Web Services are stored in a local cache folder. The location of those files can be accessed using the Show Selected Item in the Explorer tool. The files are stored to improve the performance of the visualization and also to avoid downloading them repeatedly. They are reused every time the same Web Service is added to a workspace. You can delete these files manually if you don't need them anymore.
  You can also delete these items and cache files at the same time deleting them on <Remove Selected Item Dialog> with checking "Delete local cache files" check box.



7. APPENDIX

7.1 Product Comparison Table

Category	Group	Command	Free	Subscription License			
			Viewer	Canvas Kit	Scan Kit	Mobile Kit	Scan & Mobile Kit
File	Workspace	New	✓	✓	✓	✓	✓
		Open	✓	✓	✓	✓	✓
		Open Recent	✓	✓	✓	✓	✓
		Save	✓	✓	✓	✓	✓
		Save As	✓	✓	✓	✓	✓
		Close	✓	✓	✓	✓	✓
	Data	Add Existing IP-S3 Run	✓	✓	✓	✓	✓
		Import IP-S2 Run				✓	✓
		Add Existing RD-M1 Run	✓	✓	✓	✓	✓
		Add Existing GLS Project	✓	✓	✓	✓	✓
		Create New GLS Project			✓		✓
		Add Existing Model Project	✓	✓	✓	✓	✓
		Create New Model Project		✓	✓	✓	✓
	Export	Export Orthophotos From Cloud		✓	✓	✓	✓
	Help	About	✓	✓	✓	✓	✓
		Read Manual	✓	✓	✓	✓	✓
		Read RD-M Manual	✓	✓	✓	✓	✓
	Settings	Settings	✓	✓	✓	✓	✓
	License	License	✓	✓	✓	✓	✓

Category	Group	Command	Free	Subscription License			
			Viewer	Canvas Kit	Scan Kit	Mobile Kit	Scan & Mobile Kit
View	Navigation	Swap Map	✓	✓	✓	✓	✓
		Swap Panorama	✓	✓	✓	✓	✓
		Sync Map	✓	✓	✓	✓	✓
		Sync 3D	✓	✓	✓	✓	✓
	Camera Projection	Perspective	✓	✓	✓	✓	✓
		Parallel	✓	✓	✓	✓	✓
	Camera Presets	TOP	✓	✓	✓	✓	✓
		Bottom	✓	✓	✓	✓	✓
		Front	✓	✓	✓	✓	✓
		Back	✓	✓	✓	✓	✓
		Right	✓	✓	✓	✓	✓
		Left	✓	✓	✓	✓	✓
	Trajectory Color	Velocity	✓	✓	✓	✓	✓
		Elevation	✓	✓	✓	✓	✓
		Position Sigma	✓	✓	✓	✓	✓
		Flat	✓	✓	✓	✓	✓
	Scan Color	Intensity	✓	✓	✓	✓	✓
		Grayscale	✓	✓	✓	✓	✓
		Image	✓	✓	✓	✓	✓
		Elevation	✓	✓	✓	✓	✓
		Flat	✓	✓	✓	✓	✓
		Station	✓	✓	✓	✓	✓
		Density	✓	✓	✓	✓	✓
	DEM	Elevation	✓	✓	✓	✓	✓
		Flat	✓	✓	✓	✓	✓
	TIN	Elevation	✓	✓	✓	✓	✓
		Flat	✓	✓	✓	✓	✓
	Volume	Interpolate	✓	✓	✓	✓	✓
		Binary	✓	✓	✓	✓	✓
	Surfaces Style	Flat	✓	✓	✓	✓	✓
		Smooth	✓	✓	✓	✓	✓
		Wireframe	✓	✓	✓	✓	✓
	Render Style	Point	✓	✓	✓	✓	✓
		Circle	✓	✓	✓	✓	✓
		Sphere	✓	✓	✓	✓	✓

Category	Group	Command		Free	Subscription License			
				Viewer	Canvas Kit	Scan Kit	Mobile Kit	Scan & Mobile Kit
View	Scan Points	Increase Size		✓	✓	✓	✓	✓
		Decrease Size		✓	✓	✓	✓	✓
		Reset Size		✓	✓	✓	✓	✓
		Increase Density		✓	✓	✓	✓	✓
		Decrease Density		✓	✓	✓	✓	✓
		Reset Density		✓	✓	✓	✓	✓
	Window	Second View		✓	✓	✓	✓	✓
		Split View	Split Horizontally	✓	✓	✓	✓	✓
			Split Vertically	✓	✓	✓	✓	✓
			Hide Split View	✓	✓	✓	✓	✓
		Properties		✓	✓	✓	✓	✓
		Workspace		✓	✓	✓	✓	✓
		Tasks		✓	✓	✓	✓	✓
		Reset Layout		✓	✓	✓	✓	✓
Cloud	Selection	Rectangle		✓	✓	✓	✓	✓
		Polygon		✓	✓	✓	✓	✓
		All		✓	✓	✓	✓	✓
		Clear		✓	✓	✓	✓	✓
		Invert		✓	✓	✓	✓	✓
	Modify Selection	Replace		✓	✓	✓	✓	✓
		Add		✓	✓	✓	✓	✓
		Subtract		✓	✓	✓	✓	✓
		Intersect		✓	✓	✓	✓	✓
	Classification	Hide		✓	✓	✓	✓	✓
		Show/Hide		✓	✓	✓	✓	✓
		Show All/Hide All		✓	✓	✓	✓	✓
		Delete			✓	✓	✓	✓
		Restore			✓	✓	✓	✓
		Restore All			✓	✓	✓	✓
	Display	Normal		✓	✓	✓	✓	✓
		Hidden		✓	✓	✓	✓	✓
		Deleted		✓	✓	✓	✓	✓
		Reset		✓	✓	✓	✓	✓

Category	Group	Command	Free	Subscription License			
			Viewer	Canvas Kit	Scan Kit	Mobile Kit	Scan & Mobile Kit
Tools	Segments	Select		✓	✓	✓	✓
		Unselect		✓	✓	✓	✓
		Clip		✓	✓	✓	✓
		Unclip		✓	✓	✓	✓
	Primitives	Point	✓	✓	✓	✓	✓
		Distance	✓	✓	✓	✓	✓
		Polyline	✓	✓	✓	✓	✓
		Polygon	✓	✓	✓	✓	✓
		Plane		✓	✓	✓	✓
		Fit Plane to Picks		✓	✓	✓	✓
		Elevated Plane		✓	✓	✓	✓
	Surfaces	DEM		✓	✓	✓	✓
		TIN		✓	✓	✓	✓
		Volume		✓	✓	✓	✓
		Contours		✓	✓	✓	✓
	Picking	Interpolate	✓	✓	✓	✓	✓
Fuse (Workspace)	Workspace	Add Existing IP-S3 Run	✓	✓	✓	✓	✓
		Add Existing RD-M1 Run	✓	✓	✓	✓	✓
		Add Existing GLS Project	✓	✓	✓	✓	✓
		Add Existing Model Project	✓	✓	✓	✓	✓
		Create New GLS Project			✓		✓
		Create New Model Project		✓	✓	✓	✓
	Import / Export	Import IP-S2 Run				✓	✓
Fuse (IP-S)	Process	Trajectory				✓	✓
		Images				✓	✓
		Scans				✓	✓
		Cloud				✓	✓
	Adjustment	Adjustment				✓	✓
		Edit Loops				✓	✓
		Edit GCPs				✓	✓
		Revert Trajectory				✓	✓
		Trajectory Jump Correction				✓	✓
	Primitives	Import	✓	✓	✓	✓	✓
		Export		✓	✓	✓	✓

Category	Group	Command	Free	Subscription License			
			Viewer	Canvas Kit	Scan Kit	Mobile Kit	Scan & Mobile Kit
Fuse (GLS)	Process	Scans and Images			✓		✓
	Registration	Clear			✓		✓
		Manual			✓		✓
		Spread			✓		✓
		Cloud-to-Cloud			✓		✓
		Targets and Georeferencing			✓		✓
		Accuracy Report			✓		✓
		Occupation and Backsight			✓		✓
		Resection			✓		✓
		Quick Georeferencing			✓		✓
	Workspace	Import Stations			✓		✓
		Create Station Set			✓		✓
	Primitives	Import	✓	✓	✓	✓	✓
		Export		✓	✓	✓	✓
Fuse (Model)	Process	Clouds		✓	✓	✓	✓
	Registration	Clear		✓	✓	✓	✓
		Manual		✓	✓	✓	✓
		Spread		✓	✓	✓	✓
		Cloud-to-Cloud		✓	✓	✓	✓
		Targets and Georeferencing		✓	✓	✓	✓
		Accuracy Report		✓	✓	✓	✓
		Automatic Georeferencing		✓	✓	✓	✓
	Workspace	Import Models		✓	✓	✓	✓
		Create Model Set		✓	✓	✓	✓
		Create Model		✓	✓	✓	✓
		Create Cloud		✓	✓	✓	✓
	Import / Export	Import Cloud		✓	✓	✓	✓
	Primitives	Import	✓	✓	✓	✓	✓
		Export		✓	✓	✓	✓
Fuse (Collage Web)	Collage Web	Add	✓	✓	✓	✓	✓
Edit (Workspace)	Viewer	Show/Hide	✓	✓	✓	✓	✓
		Show All/Hide All	✓	✓	✓	✓	✓
		Jump To	✓	✓	✓	✓	✓
		Set Scope	✓	✓	✓	✓	✓
	Workspace	Remove All	✓	✓	✓	✓	✓
	Properties	Show in Explorer	✓	✓	✓	✓	✓
		Properties	✓	✓	✓	✓	✓

Category	Group	Command	Free	Subscription License			
			Viewer	Canvas Kit	Scan Kit	Mobile Kit	Scan & Mobile Kit
Edit (Map)	Viewer	Jump To	✓	✓	✓	✓	✓
	Properties	Show in Explorer	✓	✓	✓	✓	✓
		Properties	✓	✓	✓	✓	✓
Edit (KMLs)	Viewer	Jump To	✓	✓	✓	✓	✓
	Workspace	Remove All	✓	✓	✓	✓	✓
		Add KML	✓	✓	✓	✓	✓
	Properties	Properties	✓	✓	✓	✓	✓
		Table	✓	✓	✓	✓	✓
Edit (Shapefiles)	Viewer	Jump To	✓	✓	✓	✓	✓
	Workspace	Remove All	✓	✓	✓	✓	✓
		Add Shapefile	✓	✓	✓	✓	✓
	Properties	Properties	✓	✓	✓	✓	✓
		Table	✓	✓	✓	✓	✓
Edit (Orthophotos)	Viewer	Jump To	✓	✓	✓	✓	✓
	Workspace	Remove All	✓	✓	✓	✓	✓
		Add Orthophoto	✓	✓	✓	✓	✓
	Properties	Properties	✓	✓	✓	✓	✓
		Table	✓	✓	✓	✓	✓

Category	Group	Command	Free	Subscription License			
			Viewer	Canvas Kit	Scan Kit	Mobile Kit	Scan & Mobile Kit
Edit (IP-S)	Viewer	Show/Hide	✓	✓	✓	✓	✓
		Show All/Hide All	✓	✓	✓	✓	✓
		Jump To	✓	✓	✓	✓	✓
		Set Scope	✓	✓	✓	✓	✓
		Color Range	✓	✓	✓	✓	✓
		Playback	✓	✓	✓	✓	✓
		Slice	✓	✓	✓	✓	✓
	Workspace	Remove	✓	✓	✓	✓	✓
		Clean Up				✓	✓
		Delete	✓	✓	✓	✓	✓
		Delete All	✓	✓	✓	✓	✓
	Import / Export	Export				✓	✓
		Import External Images				✓	✓
		Export Orthophotos From Cloud		✓	✓	✓	✓
	Primitives	Point	✓	✓	✓	✓	✓
		Distance	✓	✓	✓	✓	✓
		Polyline	✓	✓	✓	✓	✓
		Polygon	✓	✓	✓	✓	✓
		Plane		✓	✓	✓	✓
		Fit Plane to Picks		✓	✓	✓	✓
		Elevated Plane		✓	✓	✓	✓
	Collage Web	Upload	✓	✓	✓	✓	✓
	Properties	Show in Explorer	✓	✓	✓	✓	✓
		Properties	✓	✓	✓	✓	✓
		Table	✓	✓	✓	✓	✓

Category	Group	Command	Free	Subscription License			
			Viewer	Canvas Kit	Scan Kit	Mobile Kit	Scan & Mobile Kit
Edit (GLS)	Viewer	Show/Hide	✓	✓	✓	✓	✓
		Show All/Hide All	✓	✓	✓	✓	✓
		Jump To	✓	✓	✓	✓	✓
		Set Scope	✓	✓	✓	✓	✓
		Color Range	✓	✓	✓	✓	✓
		Slice	✓	✓	✓	✓	✓
	Registration	Reset Coordinate System			✓		✓
	Workspace	Remove Item	✓	✓	✓	✓	✓
		Delete	✓	✓	✓	✓	✓
		Delete All	✓	✓	✓	✓	✓
		Rename	✓	✓	✓	✓	✓
	Import / Export	Export		✓	✓	✓	✓
		Export Orthophotos From Cloud		✓	✓	✓	✓
	Primitives	Point	✓	✓	✓	✓	✓
		Distance	✓	✓	✓	✓	✓
		Polyline	✓	✓	✓	✓	✓
		Polygon	✓	✓	✓	✓	✓
		Plane		✓	✓	✓	✓
		Fit Plane to Picks		✓	✓	✓	✓
		Elevated Plane		✓	✓	✓	✓
		Create Point		✓	✓	✓	✓
	Collage Web	Upload	✓	✓	✓	✓	✓
	Properties	Show in Explorer	✓	✓	✓	✓	✓
		Properties	✓	✓	✓	✓	✓
		Table	✓	✓	✓	✓	✓
Edit (Model)	Viewer	Show/Hide	✓	✓	✓	✓	✓
		Show All/Hide All	✓	✓	✓	✓	✓
		Jump To	✓	✓	✓	✓	✓
		Set Scope	✓	✓	✓	✓	✓
		Color Range	✓	✓	✓	✓	✓
		Slice	✓	✓	✓	✓	✓
		Section View	✓	✓	✓	✓	✓
	Registration	Reset Coordinate System		✓	✓	✓	✓
	Workspace	Remove Item	✓	✓	✓	✓	✓
		Clean Up		✓	✓	✓	✓
		Delete	✓	✓	✓	✓	✓
		Delete All	✓	✓	✓	✓	✓
		Rename	✓	✓	✓	✓	✓

Category	Group	Command	Free	Subscription License			
			Viewer	Canvas Kit	Scan Kit	Mobile Kit	Scan & Mobile Kit
Edit (Model)	Import / Export	Export		✓	✓	✓	✓
		Export Orthophotos From Cloud		✓	✓	✓	✓
		Import Alignment		✓	✓	✓	✓
		Import TIN		✓	✓	✓	✓
		Export TIN		✓	✓	✓	✓
		Import DEM		✓	✓	✓	✓
		Export DEM		✓	✓	✓	✓
		Export Contours		✓	✓	✓	✓
		Export Profile		✓	✓	✓	✓
		Export Cross Sections		✓	✓	✓	✓
	Primitives	Point	✓	✓	✓	✓	✓
		Distance	✓	✓	✓	✓	✓
		Polyline	✓	✓	✓	✓	✓
		Polygon	✓	✓	✓	✓	✓
		Plane		✓	✓	✓	✓
		Fit Plane to Picks		✓	✓	✓	✓
		Elevated Plane		✓	✓	✓	✓
		Explode Cross Section		✓	✓	✓	✓
		Apply As Boundary		✓	✓	✓	✓
	Surfaces	Filter Triangles		✓	✓	✓	✓
		Fill Holes		✓	✓	✓	✓
	Process	Calculate Point Cloud Density		✓	✓	✓	✓
	Collage Web	Upload	✓	✓	✓	✓	✓
	Properties	Show in Explorer	✓	✓	✓	✓	✓
		Properties	✓	✓	✓	✓	✓
		Table	✓	✓	✓	✓	✓
Edit (Collage Web)	Viewer	Jump To	✓	✓	✓	✓	✓
	Properties	Properties	✓	✓	✓	✓	✓
		Table	✓	✓	✓	✓	✓

TOPCON CORPORATION (Manufacturer)

75-1 Hasunuma-cho, Itabashi-ku, Tokyo 174-8580, Japan **https://www.topcon.co.jp**

Please see the following website for contact addresses.

GLOBAL GATEWAY http://global.topcon.com

©2017 TOPCON CORPORATION
ALL RIGHTS RESERVED