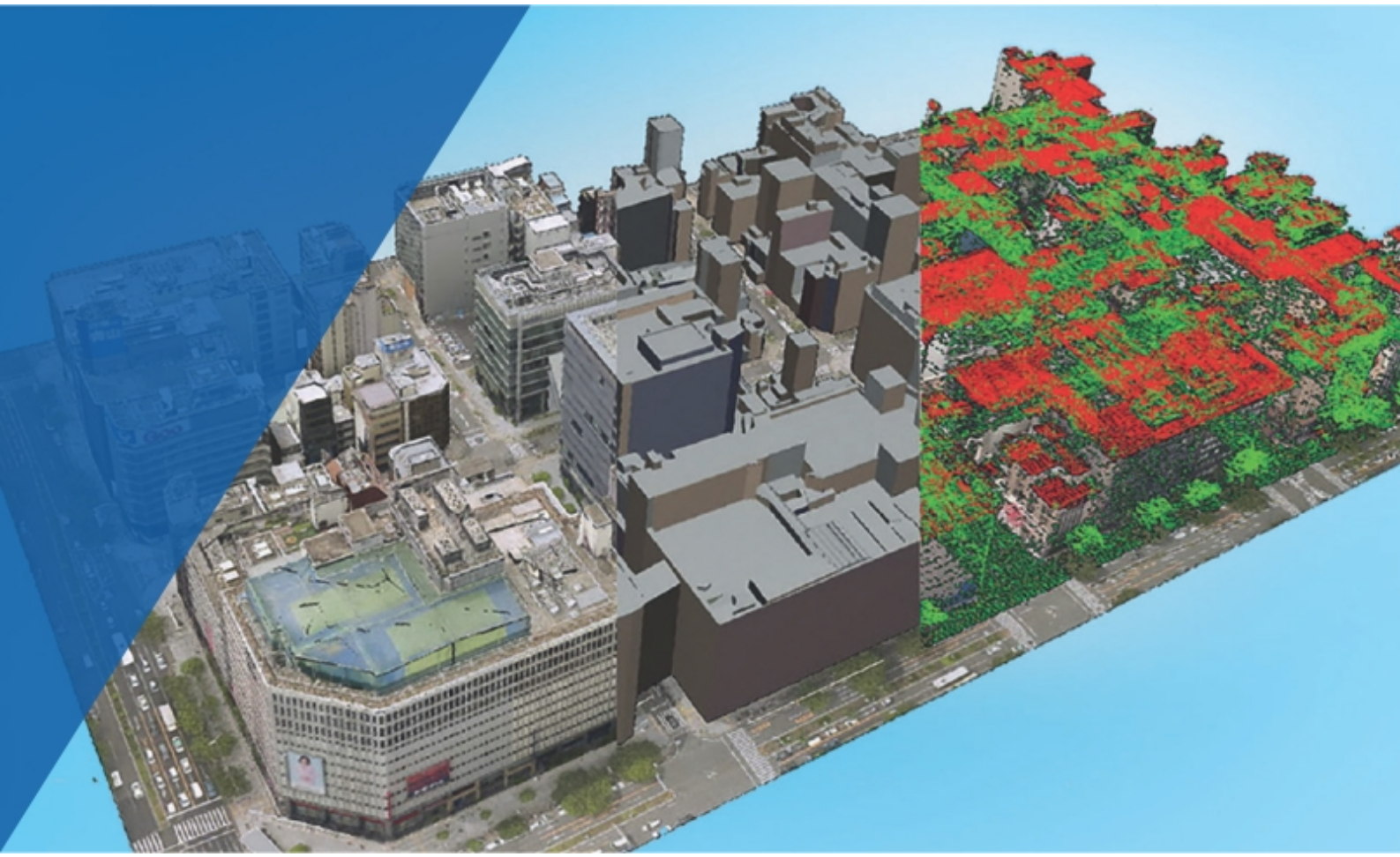
TerraScan USER GUIDE

64-bit version



Document created in August, 2020

Terra solid

POINT CLOUD INTELLIGENCE

Contents

Contents

Contents

Contents

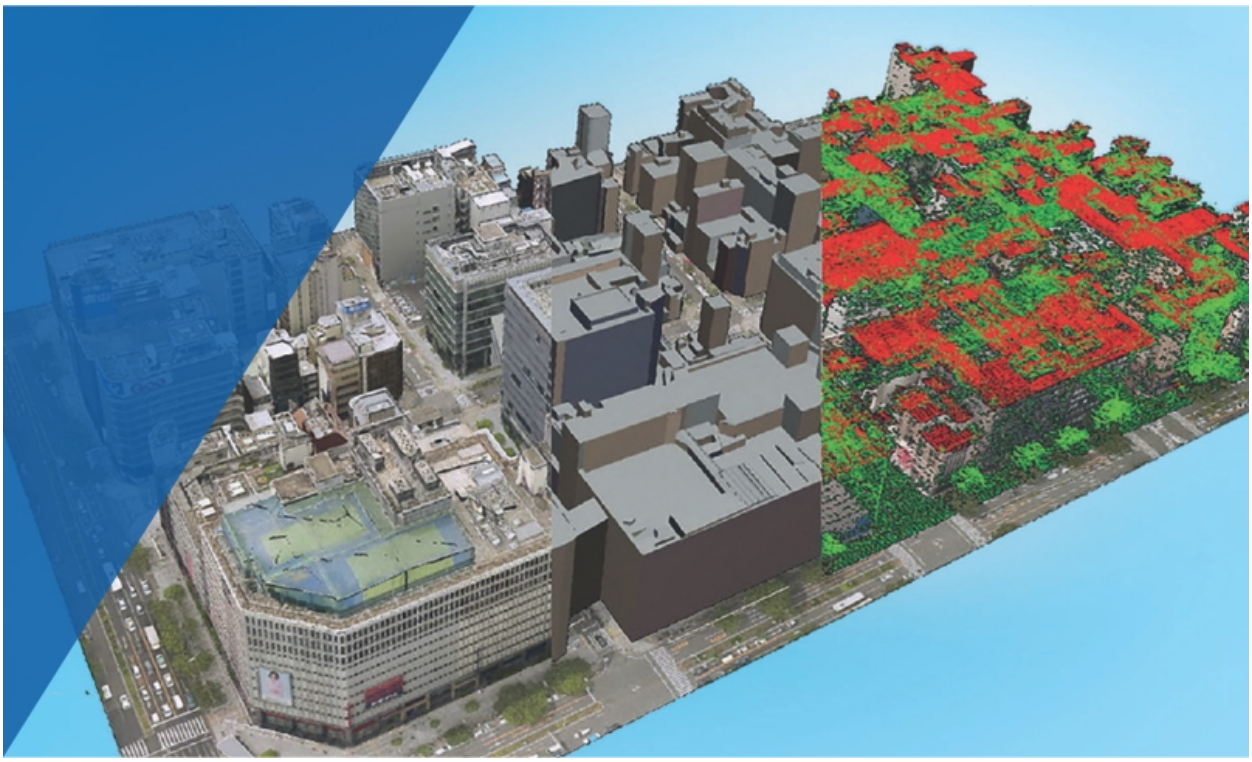
Contents

Contents

Contents

TERRASCAN USER GUIDE

64-bit TerraScan



Version 18.08.2020

© 2020 Terrasolid Ltd

Printed: August 2020 in Finland.

Trademarks

About this User Guide

This document serves as a user's guide for the 64-bit versions of TerraScan. The entry-level version, TerraScan Lite, is functionally a subset of the full version, TerraScan. TerraScan UAV is aiming to users who process point clouds collected with UAS (Unmanned Airborne Systems, also referred to systems carried by Drones). Tools available in TerraScan, TerraScan UAV, and/or TerraScan Lite work identically in all versions. Tools that are not available in TerraScan Lite are marked as "*Not Lite*" in the documentation. Tools that are not available in TerraScan UAV are marked with "*Not UAV*" in the documentation.

The PDF version of the user guide is created in order to provide an offline version of the online webhelp. It shall be updated together with the webhelp. Some parts of the webhelp may be left out on purpose in the PDF document. In case of inconsistency, the online webhelp is the primary source of information. The user is responsible for keeping his/her offline version updated.

Document conventions

The following conventions and symbols appear in this guide:

- **Data click** - click on the data mouse button, usually the left button on a right-hand mouse.
- **Reset click** - click on the reset mouse button, usually the right button on a right-hand mouse.
- < > - angle brackets are used to refer to keyboard keys, for example, <Enter>.
- *Command* - type a command in the Spaccels window of Spatix or the key-in line of MicroStation and then press <Enter>.
- OR - alternate procedures or steps in a procedure.
- C:\TERRA64 - paths to directories of files on a hard disk are written with capital letters.
- ***To do*** - the beginning of a workflow is introduced with bold-italic letters.
- When no distinction between Spatix and MicroStation versions is necessary, this document refers to the CAD environment simply as "CAD platform".

Notes and hints are highlighted in light blue boxes.

Spatix documentation

The User Guide for Spatix is available as preliminary release.

Terrasolid software runs on top of Spatix. The functionality of Terrasolid software is the same in Spatix and on top of Bentley products whenever possible. Any differences are clearly mentioned in the User Guide.

MicroStation documentation

This user guide is written under the assumption that the reader knows how to use the basic MicroStation features. You should refer to any documentation of MicroStation whenever you need information about tools and functionality of the CAD platform.

Terrasolid software runs on top of the full version of Bentley MicroStation or some other CAD products of Bentley, such as PowerDraft. Compatible Bentley products are listed on Terrasolid's webpage. The CAD platform causes no difference in functionality of Terrasolid software. Therefore, only the term "MicroStation" is used when referring to any Bentley product.

Terminology

Spatix and MicroStation often use different terms for referring to the same thing. Long-time users of MicroStation are used to the terminology of the Bentley products. New Spatix users without MicroStation knowledge only get to know the Spatix terminology.

To keep the text of the User Guide simple, only one term is used (normally the MicroStation term) if no specific separation of MicroStation and Spatix terminology is necessary. The following table provides an overview of the terminology of the two CAD platforms:

MICROSTATION	SPATIX	REMARK
Level	Layer	also Level list, Level manager, Active level
Select Element	Choose Element	selection tool
Line string	Polyline	element type
Shape	Polygon	element type
Complex element • Create Complex Chain • Create Complex Shape	Big element • Construct Big Element / Big line • Construct Big Element / Big polygon	element type • tool • tool
Cell	Symbol	element type, also Cell library
Fence (= selected shape)	= selected polygon	many TerraScan functions consider a selected polygon as fence

MICROSTATION	SPATIX	REMARK
Key-in command	Spaccel (**Sp**atix **accel**erate**s**)	typed command to call a function, also key-in window, key-in line

TerraScan

Introduction

TerraScan is a dedicated software solution for processing laser scanning point clouds. It can easily handle millions of points as all routines are tweaked for optimum performance.

Its versatile tools prove useful for a number of application fields, such as transmission lines, flood plains, proposed highways, stock piles, forest areas, city models, road and railroad surveying, and much more.

The application reads points from binary files or text files. It provides tools to:

- view the points three-dimensionally
- define your own point classes such as ground, vegetation, buildings or wires
- organize huge point clouds in projects
- manage trajectory information
- classify points using automatic filter routines
- classify points interactively
- digitize features by snapping to laser points
- detect and vectorize object features, such as buildings, powerline wires and towers, overhead wires, road breaklines, rails
- analyze object conditions, such as road surfaces, lines-of-sight, clearance areas, danger objects for roads, rails and wires, change detection
- create colored point clouds
- export colored raster images
- output classified points into text or binary files
- and much more

TerraScan is integrated with the CAD platform. This CAD environment provides a large number of useful tools and capabilities in the areas of view manipulation, visualization, vector placement and labeling. A basic understanding of the CAD platform usage is required in order to be productive with TerraScan.

TerraScan Lite

TerraScan Lite is a light version of TerraScan and provides a subset of the functionality of the full version. It can be used to view point clouds, setup projects, work on loaded points and classify points manually. It provides all the tools for manual 3D building model editing.

TerraScan Lite does not include automatic classification routines, building vectorization and road analysis tools.

Hardware and software requirements

TerraScan is built on top of a CAD platform, such as Spatix or Bentley MicroStation. You must have a computer system capable of running any compatible CAD platform.

Terra applications run parallel on Spatix and on MicroStation. Only one installation of Terra applications is needed and files are shared by the CAD platforms, such as license and settings files.

To run TerraScan, you must have the following:

- quad-core processor or better, good processor frequency
- 8 GB RAM minimum, 16 GB RAM or more recommended
- 1024*768 resolution display or better
- SSD hard disc or other storage device with fast access speed is recommended
- Windows x64 version 7 or later
- Any of the compatible CAD platforms:

GISware Integro, purchased by Terrasolid	Bentley
• Spatix	• MicroStation CONNECT Edition • PowerDraft CONNECT Edition • OpenCities Map PowerView CONNECT Edition • OpenCities Map CONNECT Edition • OpenCities Map Enterprise CONNECT Edition • ContextCapture Editor CONNECT Edition • OpenRoads Designer

Installation of TerraScan requires about 2 MB of free hard disk space.

Installation

Terrasolid applications may be delivered as a zip file or on a USB-Stick. The installation package of Terrasolid applications for Spatix includes the setup for Spatix itself as well. Therefore, you can install Spatix and Terrasolid software in one step.

A **zip package** contains the software - it does not include the User Guides. This is the normal delivery method of the software if you download it from the Terrasolid webserver.

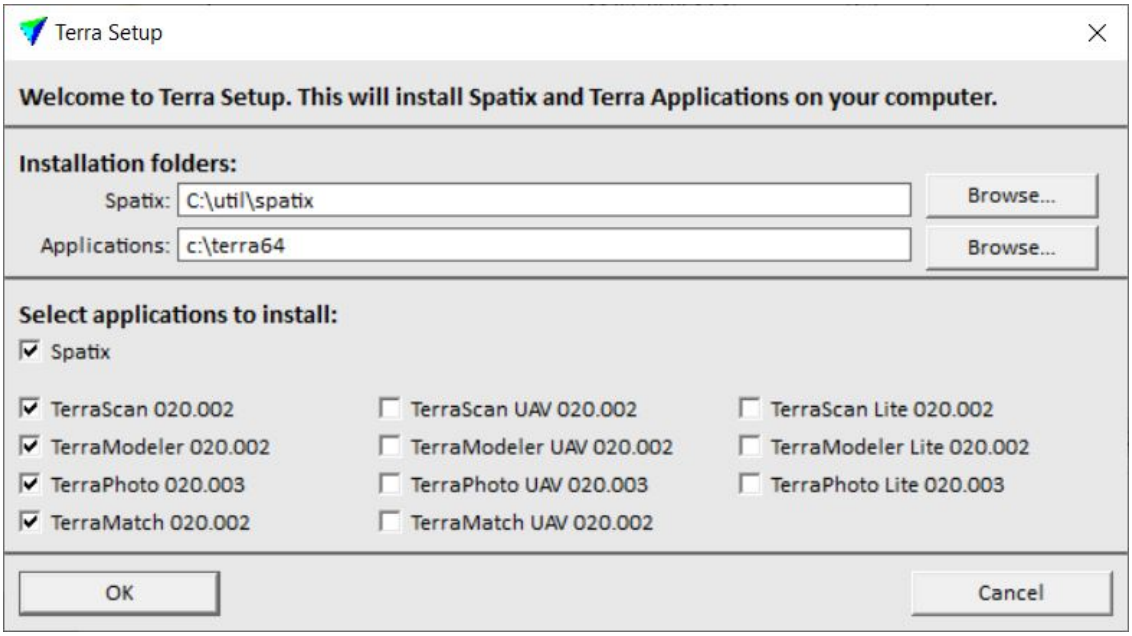
A **USB-Stick** may include the User Guides in PDF format in addition to the installation files. The USB-Stick may further include versions for multiple environments. You choose the version which corresponds to your operating system and MicroStation version. You install Terrasolid software from an USB-Stick probably only if you participate in a training event.

To install TerraScan from a zip file together or on top of Spatix:

1. Unpack the zip archive with any zip file manager.

2. Start **SETUP.EXE** which is part of the zip archive. You must have administrator permissions in order to run setup successfully.

   The installation program tries to determine where Spatix has been installed and opens the **Terra Setup** dialog:



3. Check and possibly change the installation folder of Spatix. Click on the **Browse** button in order to select a new installation folder for Spatix. The folder is created automatically, if it does not exist.

4. Define the directory where to install TerraScan and maybe other Terra applications.

   The default path is C:\TERRA64. You can change this to another location. The specified directory is created automatically, if it does not exist. Install all Terrasolid applications into one folder, such as C:/TERRA64.

5. Select all Terrasolid applications that you want to install.

   Select either the full version, the UAV version or the Lite version of an application. The versions do not run parallel on the same CAD platform.

6. Click OK to start the installation.

   A message is displayed when the installation is finished.

To install TerraScan from a zip file on top of MicroStation:

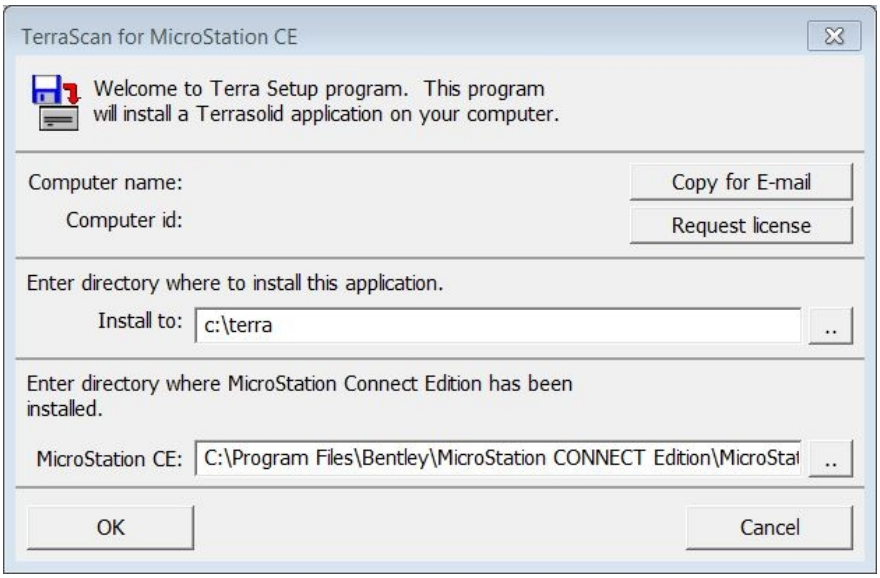
1. Unpack the zip archive with any zip file manager.

2. Start **SETUP.EXE** which is part of the zip archive.

   This may open a dialog confirming the execution of SETUP.EXE and/or prompting for the administrator password.

The installation program needs to know where the CAD platform (Spatix or MicroStation) has been installed. It automatically searches all local hard disks to find the CAD platform installation directory.

The installation dialog opens. The dialog is the same for all Terra applications:



3. Define the directory where to install TerraScan.

   The default path is C:\TERRA. You may change this to another location. The specified directory is created automatically if it does not exist.

4. Check the MicroStation CE directory. Replace the path if the correct location was not found automatically.

5. Click OK to start the installation.

   A message is displayed when the installation is finished.

The installation folder contains a README.TXT file which explains the installation of the software in batch mode. The allows to install several Terrasolid applications in one step.

To install TerraScan from USB-Stick on top of MicroStation:

1. Insert the USB-Stick.

2. Locate the correct directory which corresponds to your computer configuration.

3. Start **SETUP.EXE** from that directory.

   The installation program tries to determine where MicroStation has been installed and opens the **Terra Setup** dialog.

4. Define the directory where to install TerraScan and maybe other Terra applications.

   The default path is C:\TERRA. You can change this to another location. The specified directory is created automatically, if it does not exist.

5. Check the **MicroStation** directory. Replace the path if the correct location was not found

automatically.

You can use the **Scan** button to automatically search the hard disk for the MicroStation installation. Alternatively, you can use the **Browse** button to locate the MicroStation installation folder yourself.

6. Click OK to continue.

   This opens another **Terra Setup** dialog.

7. Select the TerraScan **for MicroStation** item in the dialog.

   You may select all applications for which you have installation files.

8. Click OK to start the installation.

   A message is displayed when the installation is finished.

Starting TerraScan

TerraScan is an application that runs on top of Spatix (Ix App) or MicroStation (MDL Application).

To start TerraScan in Spatix:

1. Select **Execute** command from the **Ix Apps** menu in Spatix.

   The **Choose Ix app to execute** dialog opens, a standard Windows dialog to open a file.

2. Browse to the \APP folder of the Terra applications installation directory.

   By default, the path is C:\TERRA64\APP.

3. Select the **tscan.ix** file.

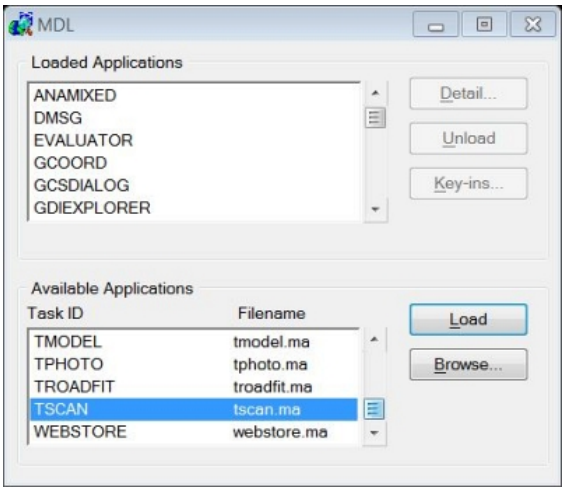
   You may select other applications as well.

4. Click **Open** in order to start all selected applications.

To start TerraScan in MicroStation:

1. Select **MDL Applications** command from the **Utilities** ribbon in MicroStation.

   The **MDL dialog** opens:



2. In the **Available Applications** list, select TSCAN.

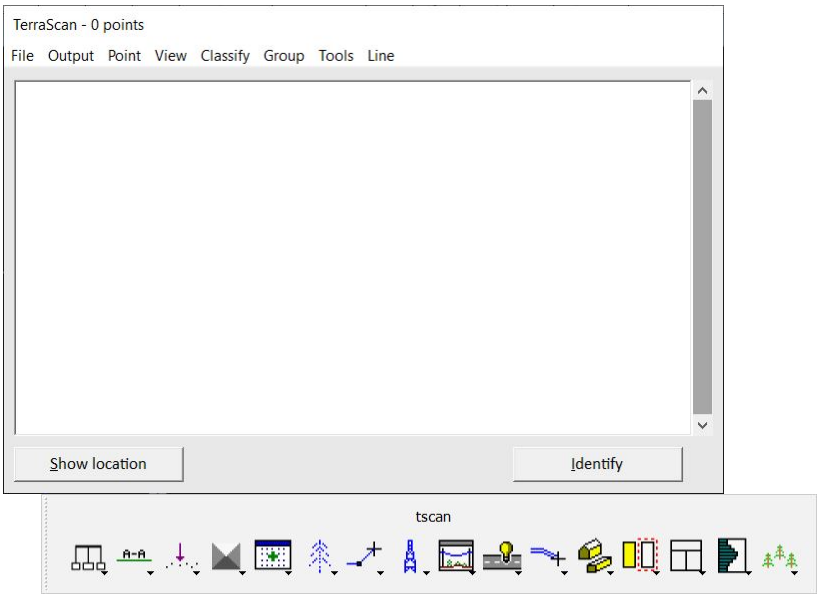
3. Click the **Load** button.

   OR

1. Key in mdl load tscan.

The **Available Applications** list shows all MDL applications that MicroStation is able to locate. MicroStation searches for MDL applications in the directories listed in **MS_MDLAPPS** configuration variable. If MicroStation can not find TSCAN.MA, you should check the variable in the **Configuration Variable** dialog of MicroStation. Make sure the directory path of the TSCAN.MA file is included in the variable values. See also Installation Directories and Configuration Variables for more information.

When the application is loaded, it opens the TerraScan window and TerraScan toolbox according to the settings for loading TerraScan:



If the **TerraScan window** is accidentally closed, it can be re-opened with the key-in command:

scan app mainwin

If the **TerraScan toolbox** is accidentally closed, it can be re-opened with the **Main toolbox** command from the Toolboxes submenu in the **Tools** pulldown menu or with the key-in command:

scan app main

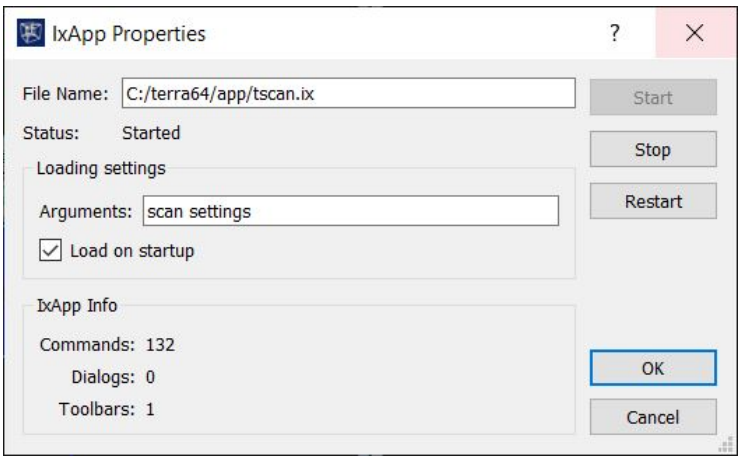
Unloading TerraScan

TerraScan is unloaded automatically when you exit Spatix or MicroStation. Sometimes you may want to unload the application while continuing to work with the CAD platform. This frees up the memory reserved by TerraScan.

To unload TerraScan in Spatix:

1. Select **tscan.ix** command from the **Ix Apps** menu in Spatix.

   The **IxApp Properties** dialog opens:



2. Click on the **Stop** button.

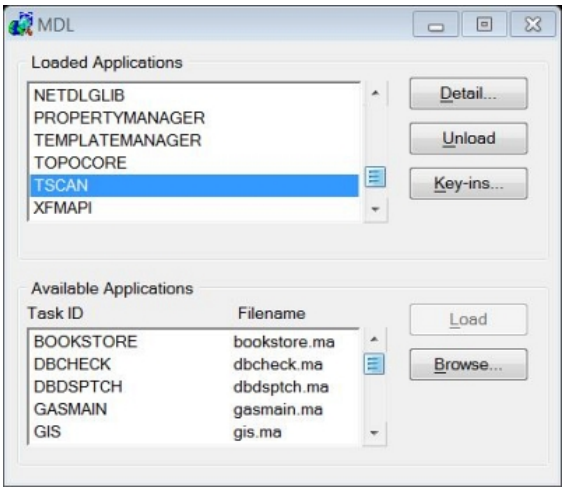
   This unloads TerraScan, closes the **TerraScan** window and toolbox and updates the Status and IxApp Info in the **IxApp Properties** dialog.

3. Close the dialog with **OK** or **Cancel**.

To unload TerraScan in MicroStation:

1. Select **MDL Applications** command from the **Utilities** ribbon in MicroStation.

   The **MDL dialog** opens:



2. In the **Loaded Applications** list, select TSCAN.

3. Click on the **Unload** button.

   OR

1. Key in mdl unload tscan.

   This unloads the application and frees the memory allocated for it.

TerraScan Settings

Settings control the way how tools and commands of TerraScan work. They are organized in logical categories. The Settings dialog is opened by the Settings tool.

SETTINGS FOLDER / CATEGORY	SETTINGS CATEGORY
Building vectorization / Editing tools	Alignment reports
Building vectorization / Levels	Block naming formulas
Building vectorization / Model	City Databases
Component fitting / Colors	Classify Fence tool
Component fitting / Levels	Collection shapes
Component fitting / Operation	Color mixures
Component fitting / Profile	Default coordinate setup
Component fitting / Weights and styles	Default line qualities
Coordinate transformations / Built-in projection systems	Elevation labels
Coordinate transformations / Transformations	Loaded points
Coordinate transformations / US State Planes	Object library
Coordinate transformations / User projection systems	Operation
File formats / Default storage format	Point display
File formats / File name extensions	Rail section templates
File formats / LAS formats	Road section parameters
File formats / Leica formats	Scanner systems
File formats / Optech formats	Scanner waveform profiles
File formats / User point formats	Section templates
File formats / User trajectory formats	Slave computers
Powerlines / Active line	Snapping
Powerlines / Profile layouts	Street View images
Powerlines / Tower functions	Target objects
Powerlines / Tower statuses	Trajectory accuracies
Powerlines / Tower types	Travel View tool
	Tree species
	Tree visualization types

SETTINGS FOLDER / CATEGORY	SETTINGS CATEGORY
	Undo and backup
	User vegetation indexes

Alignment reports

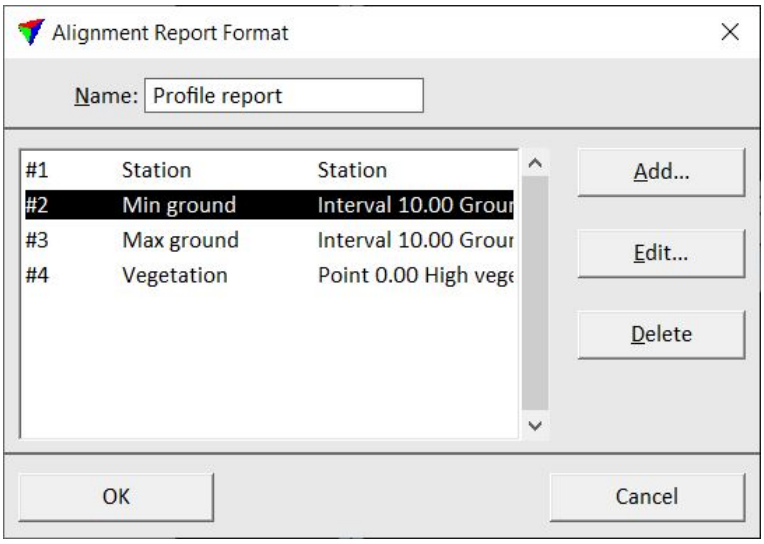
Alignment reports category contains a list of alignment report formats. The formats are used by the Output alignment report command. The report format defines what information is included in the output report along an alignment element. It consists of a descriptive name and a list of columns.

To define a new alignment report format:

1. Open the **Alignments reports** category.

2. Click **Add** in the **Settings** dialog.

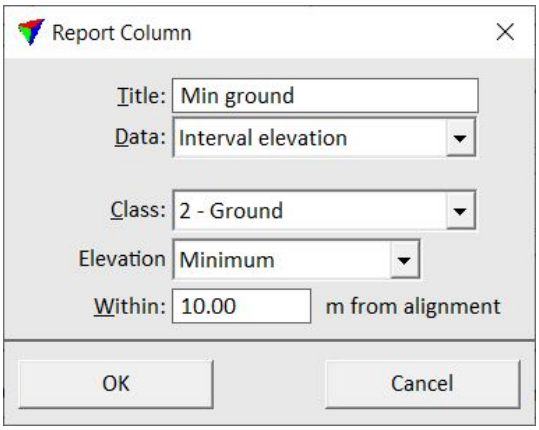
   The **Alignment Report Format** dialog opens:



3. Type a **Name** for the report format.

4. Click **Add** in the **Alignment Report Format** dialog in order to add a new column definition to the report.

   The **Report Column** dialog opens:



5. Define settings and click OK.

6. Repeat steps 4 and 5 for all columns you want to add to the report format.

7. Click OK in the **Alignment Report Format** dialog.

8. Close the **Settings** dialog in order to save the alignment report settings for TerraScan.

SETTING	EFFECT
Title	Title of the report column.
Data	Content of the report column: • **Alignment station** - station value along an alignment element. • **Alignment easting** - easting coordinate at stations along the alignment element. • **Alignment northing** - northing coordinate at stations along the alignment element. • **Alignment elevation** - elevation coordinate at stations along the alignment element. • **Interval elevation** - minimum or maximum **Elevation** of points in a given **Class**. A rectangular search area is defined by the offset left and right of the alignment station given in the **Within** field. The interval step size along the alignment element given at report output time. • **Point elevation** - closest, minimum, average, or maximum **Elevation** value from points in a given **Class** inside a circular area. The center of the circular area is at the given **Offset** from the alignment station, the size is determined by the **Radius** value. • **Surface elevation** - elevation value of a surface model. This requires at least one active surface model in TerraModeler. The elevation value is computed from the selected

SETTING	EFFECT
	Surface type at the XY location of the alignment station plus the given **Offset** value. • **Column difference** - computes the difference between two other **Columns** of the report. Select the number of the columns in the selection lists of the dialog. • **Alert** - writes an asterisk character (*) in the report if the difference between two columns is bigger or smaller than a given limit. Select the number of the columns and the comparison sign in the selection lists of the dialog. Define the limit value in the text field.
Offset	An offset value in the dialog refers to the distance from the alignment element. The selected **Data** value is determined from the location defined by the alignment station and the offset. A negative offset is left, a positive offset right of the alignment element.

Alignment report formats are stored in a configuration file ALREPFMT.INF in the TerraScan installation folder. You can copy this file to other computers in order to make alignment report formats available on them.

Block naming formulas

Block naming formulas category shows a list of naming conventions that can be used to create block names for TerraScan projects. The naming formulas are available in Add by boundaries dialog which can be opened from the Block pulldown menu of the TerraScan **Project** window.

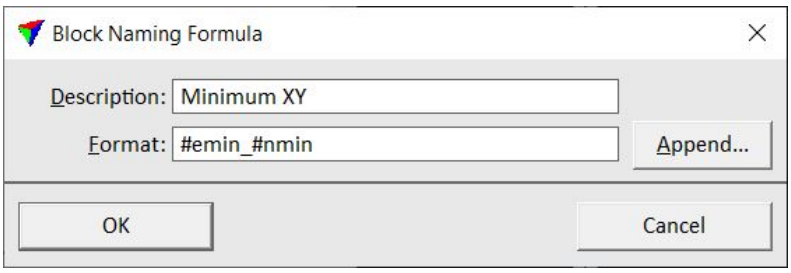
You can **Add**, **Edit**, and **Delete** block naming formulas by using the corresponding buttons in the **Settings** dialog.

To create a new block naming formula:

1. Open the **Block naming formulas** category.

2. Click **Add** in the **Settings** dialog.

   The **Block Naming Formula** dialog opens:



3. Define settings and click OK.

4. Close the **Settings** dialog in order to save the block naming formula settings for TerraScan.

SETTING	EFFECT
Description	Description of the formula.
Format	Format of the formula. This may include any free text and variable definitions. The resulting text string is probably used naming files, so it should follow naming conventions for storing files.
Append	Opens the **Append field** dialog. The dialog contains a list of variables which can be added to a formula: • **minimum | maximum easting | northing** - minimum or maximum values of easting or northing corner coordinates of a block boundary. • **block size** - block height or width value, whatever is larger. • **block number** - number according to the selection order of block boundaries.

Block naming formulas are stored in a configuration file BLOCKNAMING.INF in the TerraScan installation folder. You can copy this file to other computers in order to make block naming formulas available on them.

Building vectorization / Editing tools

Editing tools category in **Buildings vectorization** folder defines the default tool of the Building Edges toolbox.

SETTING	EFFECT
Start 'Modify Edge' as default tool	If on, the Modify Edge tool is activated by default when other building model modification tools are reset.

Building vectorization / Levels

Levels category in **Building vectorization** folder sets levels which are used for drawing building models into the CAD file. These settings are used for automatic building vectorization and the Check Building Models tool.

SETTING	EFFECT
Models to check	Levels for **Roof**, **Walls**, and **Base** polygons if a model is marked for checking. This is the status after automatic detection.
Active model	Levels for **Roof**, **Walls**, and **Base** polygons if a model is active. This is the case if the Check Building Models tool is started and a model is selected in the list of vector models.
Approved models	Levels for **Roof**, **Walls**, and **Base** polygons if a model has been approved. This is the status after the model has been checked and approved with the Check Building Models tool.

Building vectorization / Model

Model category in **Building vectorization** folder defines design settings for drawing automatically created building models in the CAD file. See Chapter 3D Building Models for a detailed description of building vectorization options and tools in TerraScan.

SETTING	EFFECT
Average elevations	Vertices in a building model are set to the same elevation if they are closer to each other than the given tolerance value. As a result, there is no elevation jump in a building model even if there is no exact intersection line but the elevation difference of vertices is smaller than the tolerance. In addition, roof patches may not be exactly planar shapes. To enforce exact planarity for roof patches, set this value to 0.0. This may lead to vertices that are not exactly at the same XYZ location anymore although they are vertices of intersection lines.
Walls start	Distance between the ground level and the start of walls below the ground. Defines the elevation of the base polygon vertices.
Search ground	Distance from a wall location within which ground is seached for placing the base of walls.
Roof thickness	Distance between upper and lower level of the roofs. If the value is 0.0, the roof planes are represented by single polygons.

SETTING	EFFECT
Roof	Default color of roof polygons. Uses the active color table of the CAD file.
Roof sides	Default color of roof side polygons. Uses the active color table of the CAD file. This is ignored if **Roof thickness** is set to 0.0.
Walls	Default color of wall polygons. Uses the active color table of the CAD file.

City databases

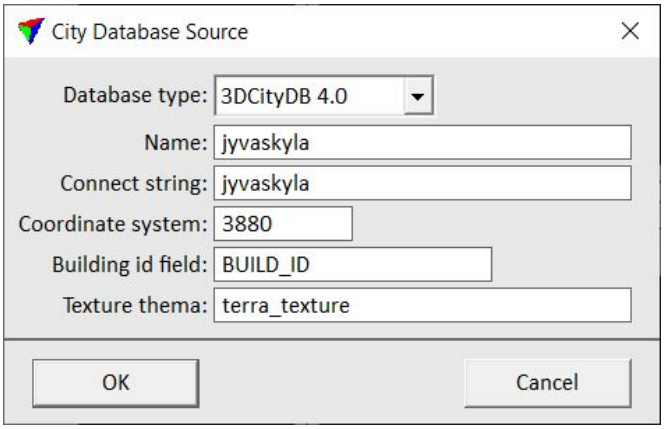
City databases category shows a list of database connections that can be used to write, read, or delete 3D building models to/from 3D City Databases. The connections are available in the dialogs of the Write Buildings to Database, Read Buildings from Database, Delete Database Buildings tools in the Building toolbox of TerraScan.

You can **Add**, **Edit**, and **Delete** database connections by using the corresponding buttons in the **Settings** dialog.

To create a new database connection:

1. Open the **City databases** category.
2. Click **Add** in the **Settings** dialog.

   The **City Database Source** dialog opens:



3. Define settings and click OK.
4. Close the **Settings** dialog in order to save the database connection settings for TerraScan.

SETTING	EFFECT
Database type	Version of the 3D City Database: **3DCityDB 3.0** or **3DCityDB 4.0**.
Name	Name of the database source for internal use in TerraScan. This name appears in the selection list of the tool dialogs.
Connect string	Name of the database to which the software connects.
Coordinate system	EPSG number of the coordinate system used in the database. The EPSG number identifies each coordinate system in a unique way.
Building id field	Name of the ID field in the database. The ID identifies each building in the database in a unique way.
Texture thema	Appearance/Theme name for textures in the 3D City Database. This is applied when building models with textures are written to the database. The name also appears, for example in selection lists, when other software exports/imports the building models from the database.

City database connections are stored in a configuration file CITY_DATABASES.INF in the TerraScan installation folder. You can copy this file to other computers in order to make database connections available on them, provided that the same database can be connected on another computer.

Classify Fence tool

Classify Fence tool category determines the symbology of a fence displayed by the Classify Fence tool.

SETTING	EFFECT
Color	Color of the fence. Uses the active color table of the CAD file.
Weight	Line weight of the fence. Uses standard line weights.
Style	Line style of the fence. Uses standard line styles.

Collection shapes

Collection shapes category shows a list of collection shape types. Collection shapes can be used to group laser points. Typical collection shape type examples are building, road, or tree. The actual grouping is done by placing collection shapes with the Place Collection Shape tool. The collection shape type determines what kind of an object the polygon encloses as well as the level and the symbology of the polygon.

Collection shapes can be further used to output laser points into separate files according to the collection or group they belong to. See Output collections command from the Tools pulldown menu of the TerraScan **Project** window.

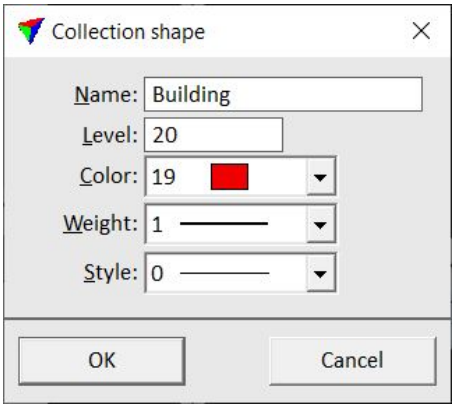
You can **Add**, **Edit**, and **Delete** collection shape types by using the corresponding buttons in the **Settings** dialog.

To create a new collection shape type:

1. Open the **Collection shapes** category.

2. Click **Add** in the **Settings** dialog.

   The **Collection shape** dialog opens:



3. Define settings and click OK.

4. Close the **Settings** dialog in order to save the collection shape settings for TerraScan.

SETTING	EFFECT
Name	Name of the collection shape type.
Level	Number of the level on which collections shapes are placed.
Color	Color of collections shapes. Uses the active color table of the CAD file.
Weight	Line weight of collection shapes. Uses standard line weights.

SETTING	EFFECT
Style	Line style of collection shapes. Uses standard line styles.

Collection shapes are stored in a configuration file COLLECTION_SHAPES.INF in the TerraScan installation folder. You can copy this file to other computers in order to make collection shape definitions available on them.

Color mixtures

Color mixtures category shows a list of color mixtures. A color mixture can be used to assign color values to laser points. For some object types, such as trees, color mixtures look more naturally compared with constant colors or colors extracted from images. This is especially true for point clouds collected by ground-based mobile mapping systems.

Color mixtures are defined by using the Hue Saturation Value (HSV) color model. Hue defines the color value on a 360-degree color circle ranging from Red (0 deg) via Yellow (60 deg), Green (120 deg), Cyan (180 deg), Blue (240 deg), Magenta (300 deg) back to Red. Saturation defines the intensity or purity of the color. A smaller saturation sets the color closer to a gray shade. Value defines the lightness or darkness of the color. A smaller value sets the color closer to black.

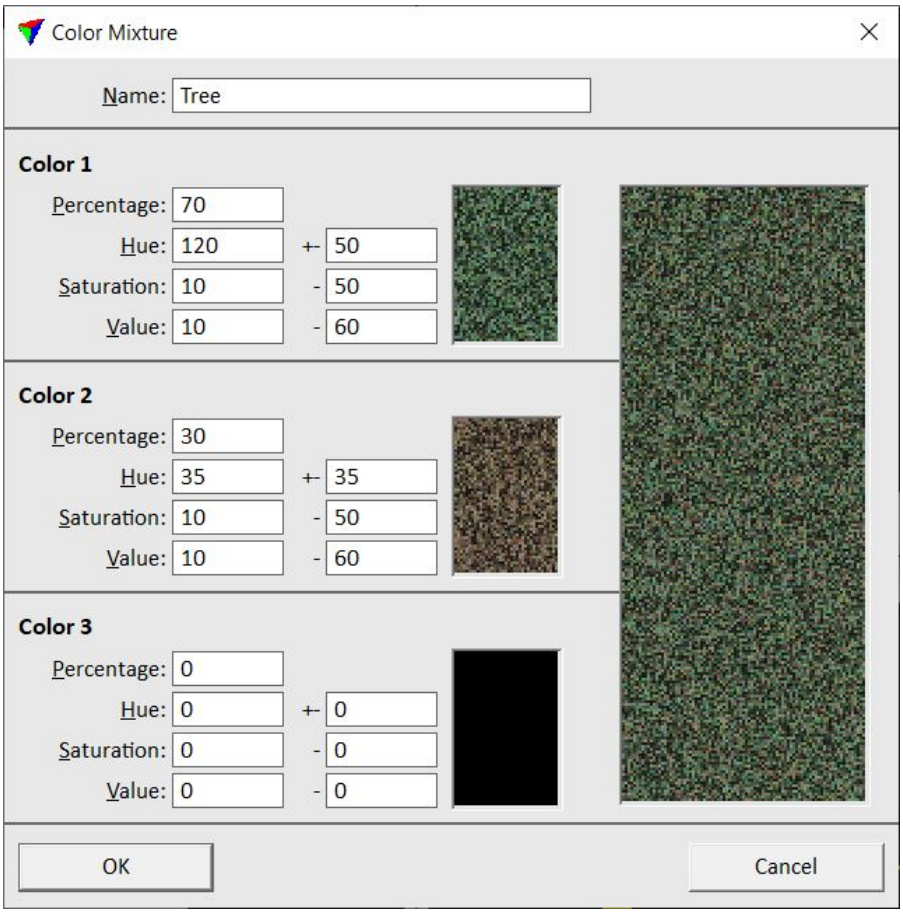
Color mixtures are used by the Assign color to points command for loaded points and by the Assign color macro action.

You can **Add**, **Edit**, and **Delete** color mixtures by using the corresponding buttons in the **Settings** dialog.

To create a new color mixture:

1. Open the **Color mixtures** category.
2. Click **Add** in the **Settings** dialog.

   The **Color Mixture** dialog opens:



3. Define settings and click OK.

4. Close the **Settings** dialog in order to save the color mixture settings for TerraScan.

SETTING	EFFECT
Name	Name of the color mixture.
Percentage	Percentage value for using a color in the mixture.
Hue	Hue of a color and its variation. Values can vary from 0 to 359, the variation can range from 0 to 180.
Saturation	Saturation range of a color. Values can vary from 0 to 180.
Value	Value range of a color. Values can vary from 0 to 180.

Color mixtures are stored in a configuration file COLOR_MIXTURES.INF in the TerraScan installation folder. You can copy this file to other computers in order to make color mixture definitions available on them.

Component fitting / Colors

Colors category in **Component fitting** folder defines colors for drawing the different elements of geometry component fitting. All color values are given in RGB color space. You can define a color by typing the RGB values in the text fields of the dialog or by clicking on the >> Button next to the text fields. The button opens the standard **Color** dialog of Windows and lets you select a color.

See Chapter Fit Geometry Components for more information about the topic.

SETTING	EFFECT
Arc	Color of fitted arc elements.
Clothoid	Color of fitted clothoid elements.
Line	Color of fitted line elements.
Hilite	Color of fitted highlighted elements.
Temporary	Color of fitted temporary elements.
Horizontal	Color of horizontal alignment elements.
Vertical	Color of vertical alignment elements.
Label	Color of label elements.
3D shape	Color of the final 3D shape.

Component fitting / Levels

Levels category in **Component fitting** folder sets levels used for drawing geometry component fitting elements into the CAD file. Define a level by typing the level number in the text fields of the dialog.

See Chapter Fit Geometry Components for more information about the topic.

SETTING	EFFECT
Horizontal levels	
Surveyed alignment	Level for drawing the surveyed horizontal alignment elements.
Horizontal components	Level for drawing the fitted horizontal component elements.
Point weights	Level for drawing the point weight values for horizontal elements.
Vertical levels	
Profile frame	Level for drawing the frame of the profile that displays the vertical geometry fitting elements.

SETTING	EFFECT
Surveyed alignment	Level for drawing the surveyed vertical alignment elements.
Vertical components	Level for drawing the fitted vertical component elements.
Point weights	Level for drawing the point weight values for vertical elements.
3D result	
3D shape	Level for drawing the final 3D shape.

Component fitting / Operation

Operation category in **Component fitting** folder defines whether results of the geometry component fitting process are saved automatically in the CAD file or not. See Chapter Fit Geometry Components for more information about the topic.

SETTING	EFFECT
Save geometry automatically	If on, results of the geometry fitting process are saved automatically.

Component fitting / Profile

Profile category in **Component fitting** folder defines the layout of the profile that is used for representing the geometry component fitting elements. See Chapter Fit Geometry Components for more information about the topic.

SETTING	EFFECT
Horizontal scale	Scale factor for horizontal elements.
Vertical scale	Scale factor for vertical elements.
Elevation grid	Step size for elevation values.
Relative margin	Size of a margin around the profile.
Station values	Number of decimals for station value labels.
Component values	Number of decimals for geometry component value labels.
Text size	Size of text elements. Given in millimeters plotted on paper.
Font	Font type of text elements. Uses a list of fonts available in the CAD file.

Component fitting / Weights and styles

Weights and styles category in **Component fitting** folder sets the line weights and styles for drawing geometry component fitting elements. It uses standard line weights and styles. See Chapter Fit Geometry Components for more information about the topic.

SETTING	EFFECT
Line weight	Line weight of component fitting elements.
Hilite weight	Line weight of highlighted elements.
Hilite style	Line style of highlighted elements.
Preview weight	Line weight of preview elements.
Preview style	Line style of preview elements.

Coordinate transformations / Built-in projection systems

Builtin projection systems category in **Coordinate transformations** folder defines what projection systems are available for transformation. This effects lists in dialogs for transforming from WGS84 longitude and latitude coordinates to planar coordinate systems or from one to another projection system. A projection system or set of projection systems is available if the corresponding option in the list of **target systems** is switched on.

Supported target systems are listed in the following table:

Belgium LB72/BEREF2003	Netherlands RD/NAP 2008
Belgium LB72/NLGEO2018	Netherlands RD/NAP 2018
Deutsche Bahn GK	Norway EUREF89 NTM
Czech/Slovak S-JTSK	South Africa
Finnish KKJ	Swedish RT90
Finnish ETRS-TM35FIN and ETRS-GK	Swedish SWEREF99
Northern Ireland OSGM02	UK National Grid OSTM02
Northern Ireland OSGM15	UK National Grid OSTM15
Northern Ireland OSGM02	UTM WGS North
Northern Ireland OSGM15	UTM WGS South
Japan	

Coordinate transformations / Transformations

Transformations category in **Coordinate transformations** folder contains a list of coordinate transformations which can be used to transform the position of laser data, trajectories, and other data.

You can **Add**, **Edit**, and **Delete** transformation by using the corresponding buttons in the Settings dialog. The **Copy** button copies the selected transformation to the clipboard. With the **Paste** button you can paste a transformation from the clipboard. The **Derive** button can be used for Deriving a transformation from a set of control point pairs.

Several types of coordinate transformations are supported:

- Linear transformation
- *Equation transformation - discarded feature in x64 software*
- Known points transformation
- Xy multiply transformation
- 3D translate & rotate transformation
- 3D Affine transformation
- Projection change transformation

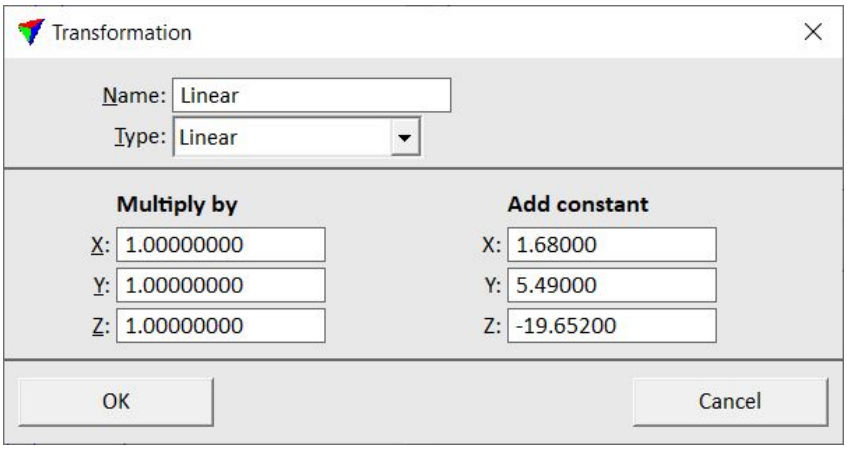
To define a new transformation:

1. Open the **Transformations** category in the **Coordinate transformations** folder.
2. Click **Add** in the **Settings** dialog.

   This opens the **Transformation** dialog.
3. Type a **Name** for the transformation and select a transformation **Type**. Define the other settings depending on the transformation type.
4. Close the **Settings** dialog in order to save the modified settings for TerraScan.

Linear transformation

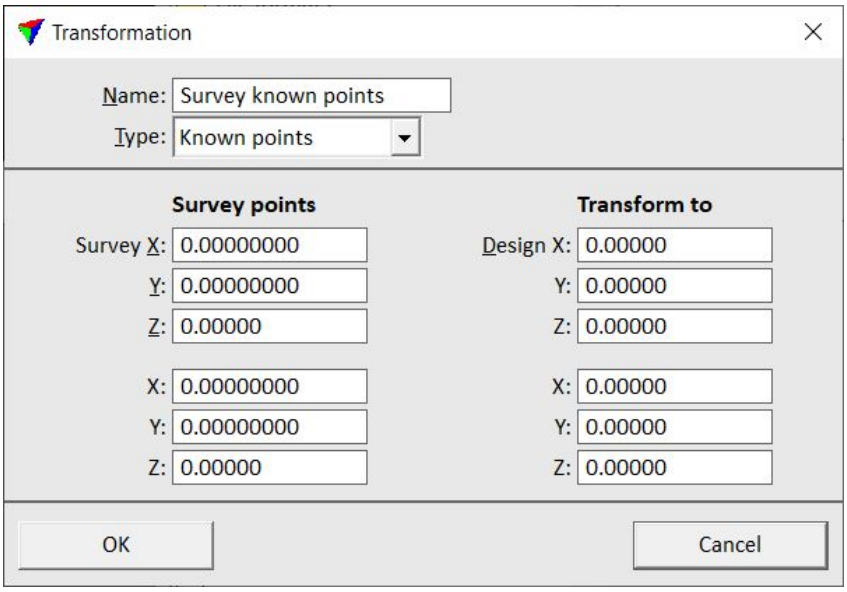
Linear transformation scales and/or translates coordinate values. You can assign a coefficient and a constant offset for each coordinate axis. The target coordinates are computed by multiplying the original coordinates with the given coefficient and by adding a given constant value.



SETTING	EFFECT
Multiply by - X	Coefficient for multiplying the easting coordinate.
Multiply by - Y	Coefficient for multiplying the northing coordinate.
Multiply by - Z	Coefficient for multiplying the elevation coordinate.
Add constant - X	Value to add to the easting coordinate.
Add constant - Y	Value to add to the northing coordinate.
Add constant - Z	Value to add to the elevation coordinate.

Known points transformation

Known points transformation lets you specify the coordinates of two known points in the original coordinate system (survey coordinates) and their respective coordinates in the target system (CAD file coordinates).



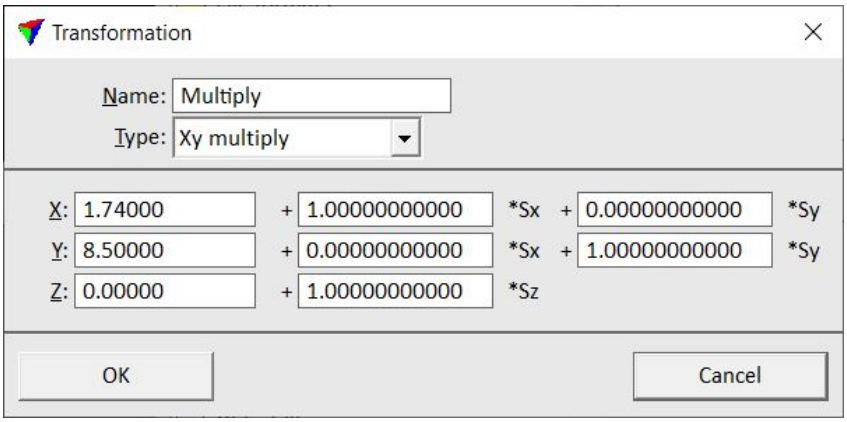
SETTING	EFFECT
Survey X, Y, Z	First known point in the original coordinate system.
X, Y, Z	Second known point in the original coordinate system.
Design X, Y, Z	First known point in the target coordinate system.
X, Y, Z	Second known point in the target coordinate system.

Xy multiply transformation

Xy multiply applies a transformation using equations:

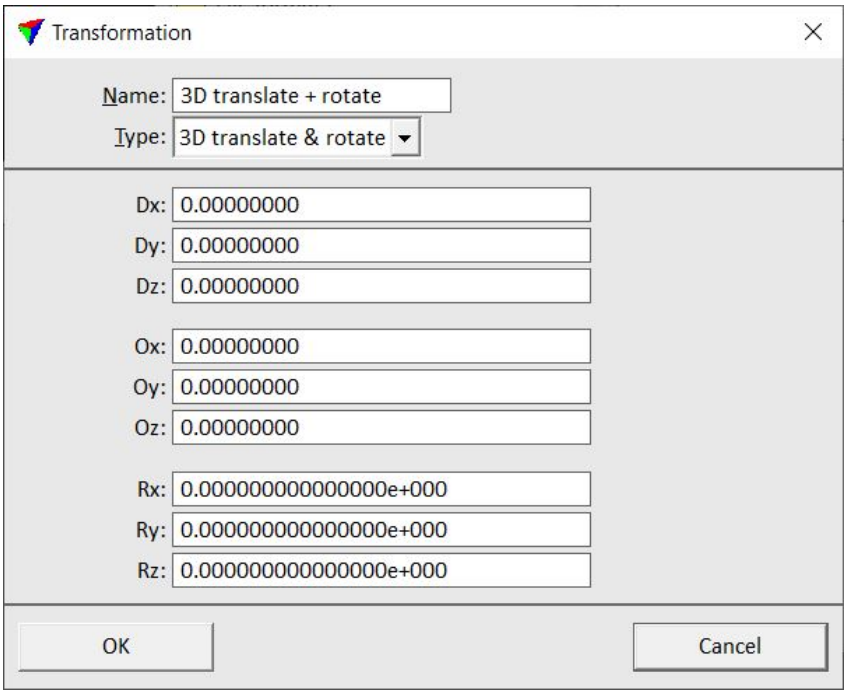
```
NewX = dx + a * Sx + b * Sy
NewY = dy + c * Sx + d * Sy
NewZ = dz + e * Sz
```

where dx, dy, dz, a, b, c, d, and e are constant parameters of the transformation and Sx, Sy, Sz are the original (survey) coordinates. This is often used as 2D Helmert type of transformation.



3D translate & rotate transformation

3D translate & rotate applies a three dimensional translation and rotation to coordinates.



SETTING	EFFECT
Dx, Dy, Dz	Values to add to X, Y, Z coordinates.
Ox, Oy, Oz	X, Y, Z coordinates of the rotation center point.
Rx, Ry, Rz	Rotation angle in radians around X, Y, Z axes.

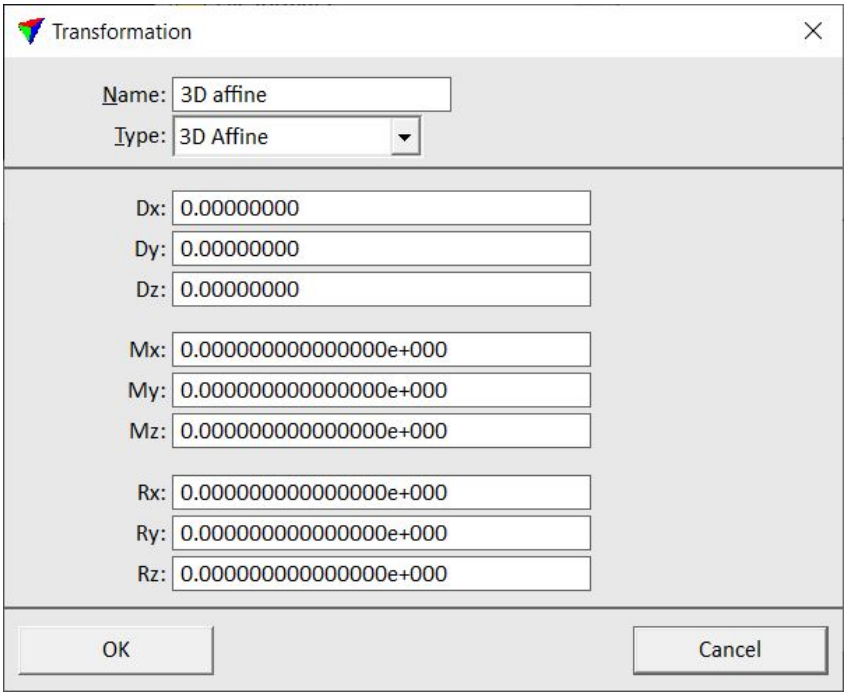
3D Affine transformation

3D Affine applies separate translation, rotation and scaling for each coordinate axis. The transformation is defined by equations:

```
NewX = dx + (1.0 + mx) * X + rz * Y - ry * Z
NewY = dy + (1.0 + my) * Y - rz * X + rx * Z
```

```
NewZ = dz + (1.0 + mz) * Z + ry * X - rx * Y
```

where dx, dy, dz, mx, my, mz, rz, ry, and rz are constant parameters of the transformation and X, Y, Z are the original coordinates.



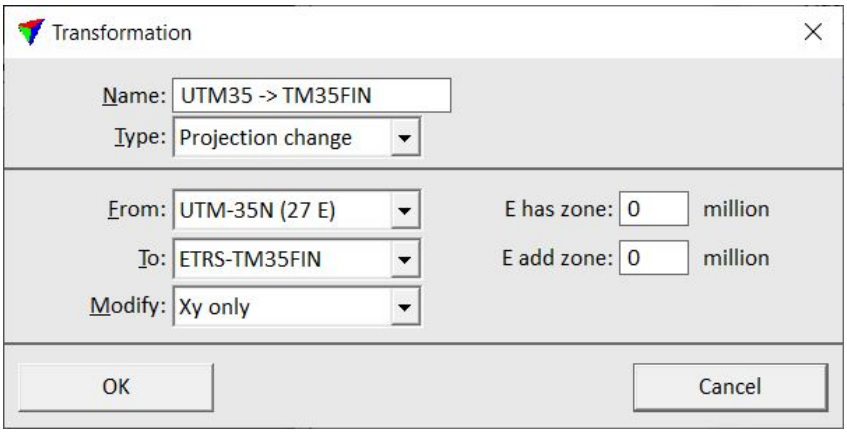
SETTING	EFFECT
Dx, Dy, Dz	Values to add to X, Y, Z coordinates (translation).
Mx, My, Mz	Factors to scale the data along the X, Y, Z axes.
Rx, Ry, Rz	Rotation angle in radians around X, Y, Z axes.

Projection change transformation

Projection change transforms coordinates from one projection system to another. The software transforms the X, Y, Z coordinates from the source projection system back into WGS84 geocentric X, Y, Z and then computes the transformation into the target projection system.

All projections systems that are active in Coordinate transformations / Built-in projection systems, Coordinate transformations / US State Planes, or defined in Coordinate transformations / User projection systems are available for a projection change transformation.

If you already applied a geoid correction, you should run a reverse geoid correction to the data set before using a projection change transformation. This is essential in cases where the source and the target systems use different ellipsoids or datums. A geoid correction or a reverse geoid correction is only applied automatically if the UK National Grid system is used in the transformation.



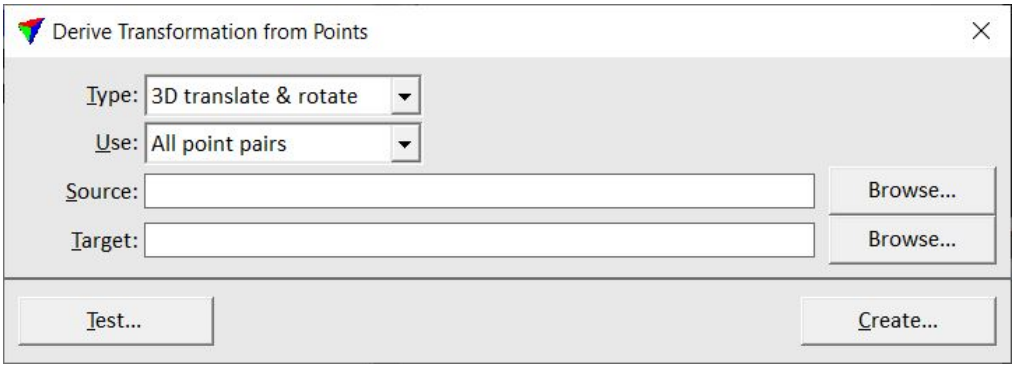
SETTING	EFFECT
From	Source projection system.
To	Target projection system.
Modify	Coordinate values to modify: • **Xyz** - modifies all coordinates. • **Xy only** - no changes to elevation values.

You can copy transformations from one Terra application to another. Select the transformation in the **Settings** dialog and click on the **Copy** button to copy the definition to the clipboard. Click on the **Paste** button in the other Terra application to paste the definition.

Deriving a transformation

You can also derive transformation parameter values from point pairs. This requires that identical control points (point pairs) are available in source and target coordinate values. The points must be stored in text files. The number of required control point pairs depends on the transformation type.

To derive a transformation, click on the **Derive** button in the **Settings** dialog. This opens the **Derive transformation from points** dialog:



SETTING	EFFECT
Type	Type of the derived transformation:

	• **2D transformation** - parameter values for a 2D Helmert transformation are derived. • **3D translate & 2D rotate** - parameter values for a 3D translation and a 2D rotation transformation are derived. • **3D translate & rotate** - parameter values for a 3D translation and rotation transformation are derived. • **7 parameter affine** - parameter values for a 3D affine transformation (7 parameters) are derived. • **9 parameter affine** - parameter values for a 3D affine transformation (9 parameters) are derived.
Use	Points used for deriving the transformation: • **All point pairs** - uses all control point pairs. • **Inside source fence only** - points inside a fence in the source coordinate system are used. • **Inside target fence only** - points inside a fence in the target coordinate system are used.
Source	Text file that contains the point pair coordinates in the source system.
Target	Text file that contains the point pair coordinates in the target system.

The transformation derivation can be tested by using the **Test** button. This computes the parameter values and displays the result in a report window. The report includes residuals which determine how accurate the transformation works.

To create the transformation, click on the **Create** button. This opens the **Transformation** dialog that displays the derived parameter values. Type a **Name** for the transformation and click OK in order to add the transformation to the list in the **Settings** dialog.

Coordinate transformations / US State Planes

US State Planes category in **Coordinate transformations** folder contains a list of US State Plane projection systems using NAD83 datum. Check the toggle box of those state plane systems you want to use.

You can view the parameters of a system by using the **View** button. In case you need to change the parameters of a built-in US State Plane definition, you can use the **Copy** button to copy/paste the system into **Coordinate transformations / User projection systems**.

Coordinate transformations / User projection systems

User projection systems category in **Coordinate transformations** folder contains a list of user defined projection systems. You can define your projection system based on **Transverse Mercator / Gauss-Krueger**, **Lambert conic conformal** or **Hotine oblique mercator** projection.

A projection system definition can be divided into three distinct parts:

- **Ellipsoid** - defined by **Semi-major axis** and **Inverse flattening**.
- **Datum** - defined by seven parameter Bursa/Wolfe transformation.
- **Projection** - defined by the projection type, true origin, false origin, scale factor at the central meridian, and distance unit.

The list of user projection system displays a toggle box for each row. The toggle box indicates whether a projection system is active or not. Only active projection systems can be selected when applying a transformation. To activate or deactivate a projection system, place a data click inside its toggle box in the list.

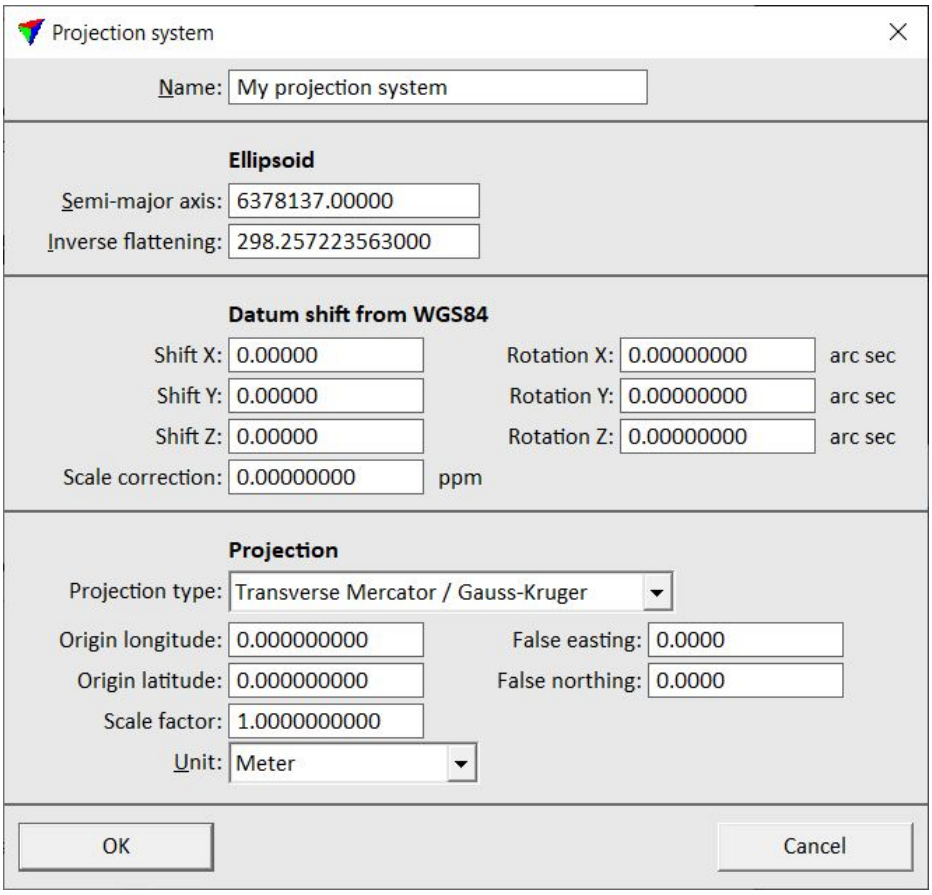
You can **Add**, **Edit**, and **Delete** user projection systems by using the corresponding buttons in the **Settings** dialog. The **Copy** button copies the selected projections system definition to the clipboard. With the **Paste** button you can paste a projection system definition from the clipboard.

To define a new projection system:

1. Open the **User projection systems** category in the **Coordinate transformations** folder.

2. Click **Add** in the **Settings** dialog.

   This opens the **Projection system** dialog:



3. Define settings and click OK.

4. Activate the projection system.

5. Close the **Settings** dialog in order to save the modified settings for TerraScan.

SETTING	EFFECT
Name	Descriptive name for the projection system.
Semi-major axis	Semi-major axis of the target ellipsoid.
Inverse flattening	Inverse flattening of the target ellipsoid.
Shift X	Datum X shift from WGS84 to the target system in meter.
Shift Y	Datum Y shift from WGS84 to the target system in meter.
Shift Z	Datum Z shift from WGS84 to the target system in meter.
Rotation X	Datum rotation around the X axis in arc seconds.
Rotation Y	Datum rotation around the Y axis in arc seconds.

SETTING	EFFECT
Rotation Z	Datum rotation around the Z axis in arc seconds.
Scale correction	Datum scale correction as parts per million. The actual scale factor is computed as 1.0 + (0.000001 * ScaleFactor).
Projection type	Type of the projection system: **Transverse Mercator/Gauss-Kruger**, **Lambert conic conformal**, or **Hotine oblique mercator**.
Origin longitude	Longitude of the true origin in decimal degrees.
Origin latitude	Latitude of the true origin in decimal degrees.
False easting	Map coordinate easting of the true origin.
False northing	Map coordinate northing of the true origin.
Scale factor	Scale factor on the central meridian.
Unit	Distance unit: **Meter**, **International foot**, **US Survey Foot,** or **International yard**.

You can copy user projection systems from one Terra application to another. Select the system in the Settings dialog and click on the **Copy** button to copy the definition to the clipboard. Click on the **Paste** button in the other Terra application to paste the definition. You can also paste the definition in a text editor in order to save it into a text file.

Default coordinate setup

Default coordinate setup category defines the default values for the coordinate setup of Terra applications. The default values can be changed by using the Define Coordinate Setup tool.

SETTING	EFFECT
Resolution	Default integer steps per master unit in a CAD file used by Terra applications. This effects the number of decimals stored for coordinate values of points in TerraScan. Example: Resolution 100 per meter stores coordinates with 2 decimals, which is centimeter accuracy.
Easting	Default easting coordinate of the CAD file origin.
Northing	Default northing coordinate of the CAD file origin.
Elevation	Default elevation coordinate of the CAD file origin.

Default flightline qualities

Default flightline qualities category defines quality tags for flightlines. These quality settings are used by Cut overlap command in case trajectories have not been imported. The flightline numbers refer to the line number attribute that can be stored for points in TerraScan.

A value range of 0 - 65535 refers to all possible line numbers. A value of -1 refers to no line number.

SETTING	EFFECT
Bad	Line number range with **Bad** quality tag.
Poor	Line number range with **Poor** quality tag.
Normal	Line number range with **Normal** quality tag. By default, this is the quality tag for all lines.
Good	Line number range with **Good** quality tag.
Excellent	Line number range with **Excellent** quality tag.

Elevation labels

Elevation labels category defines the format of elevation values drawn as text elements. The settings are used when points are drawn into the CAD file using the Write to design file command and if the points are drawn as **Elevation labels** which is set in the Define Classes dialog.

SETTING	EFFECT
Accuracy	Number of decimals to display for elevation values.
Display plus	If on, positive elevation values start with a plus (+) sign.
Display minus	If on, negative elevation values start with a minus (-) sign.

File formats / Default storage format

Default storage format category in **File formats** folder defines what binary format is the default storage format for laser data and what GPS time format is the default format for storing time stamps. The formats are used by default for new projects in the **Project information** dialog opened by New project command.

SETTING	EFFECT
Format	Default format for laser data: **FastBinary**, **LAS 1.0**, **LAS 1.1**, **LAS 1.2**, **LAS 1.4**, **Scan binary 16 bit line**, or **Scan binary 8 bit lines**.
Time type	Default format for storing time stamps of laser points: **GPS seconds-of-week** or **GPS standard time**

File formats / File name extensions

File name extensions category in **File formats** folder defines default file extensions for various file formats. These extensions are expected for reading files in the corresponding formats and used as default extensions when you output points from TerraScan. However, you can also read files with different extensions as long as the format of the file is correct. You can also define other extensions for each output step.

SETTING	EFFECT
East North Z	Extension for plain XYZ text files. Default is xyz.
Code East North Z	Extension for text files containing point class and coordinates. Default is txt.
TerraScan binary	Extension for 8-bit/16-bit binary files in TerraScan format. Default is bin.
FastBinary	Extension for files in TerraScan FastBinary format. Default is fbi.
LAS binary	Extension for binary files in LAS format. Default is las.

File formats / LAS formats

LAS formats category in **File formats** folder defines the bit depth of color values in LAS files. The setting can be used to read LAS files with incorrectly stored color values.

SETTING	EFFECT
Bit depth	Bit depth of color values in the LAS file: • **Correct 16 bits** - color values are stored as 16 bit values. This is the correct value according to the LAS standard format definition. • **Low 12 bits** - color values are stored as 12 bit values. • **Low 8 bits** - color values are stored as 8 bit values.

SETTING	EFFECT
Check bit depth in 'Read points'	If on, the software checks color bit depth when reading LAS files. If all color values are between 0-255 (8 bit values), it automatically scales the color values to proper 16 bit values.

Set the value to **Low 8 bits** or **Low 12 bits** if your read or import LAS files with incorrect color values. Set the value to **Correct 16 bits** in order to store color values correctly. Switch on the **Check bit depth in 'Read points'** option in order to enable automatic scaling of color values.

File formats / Leica formats

Leica formats category in **File formats** folder defines rules how to interpret intensity values coming from specific Leica file formats.

SETTING	EFFECT
Read	Reading intensity of Leica LDI files: **Raw intensity** or **Normalized intensity**.

File formats / Optech formats

Optech formats category in **File formats** folder defines rules how to interpret data coming from specific Optech file formats.

SETTING	EFFECT
Scale intensity	Factor for scaling intensity values.
Use as last echo	Defines which value is used as last echo: **First xyz**, **Second xyz**, or **Lower xyz**.
Ignore first echoes	If on, TerraScan filters out first echoes from Optech xyzxyzii type files based on the elevation difference of the first and last echo.
Less than	First echos less than the given elevation difference above the corresponding last echo are ignored. This is only active if **Ignore first echoes** is switched on.

File formats / User point formats

User point formats category in **File formats** folder contains a list of user-defined point formats. You can define your own formats which can be used for the input or output of point data. The software can read any text files where each row contains the information of one point and the point attributes

are organized in columns (fields). The file format definition determines what fields are included for each point and what is the order of the fields.

The text file formats may contain delimited fields or fixed length fields. The delimiter can be comma, space, tabulator, or semicolon. A fixed length fields is defined by constant column widths and positions in each row.

You can **Add**, **Edit**, and **Delete** point formats by using the corresponding buttons in the **Settings** dialog. The **Copy** button creates an identical copy of a selected format definition. The **Move up** and **Move down** buttons change the order of formats in the list.

The following attributes can be imported/exported from/to text files:

- **No field** - no field defined in the text file.
- **Ignore** - the column in the text file is ignored. This applies only for importing text files into TerraScan.
- **Easting**, **Northing**, **Elevation** - XYZ coordinates.
- **Longitude**, **Latitude** - position in degrees, minutes, and seconds.
- **Class** - class number.
- **Code** - class code. Can be defined in the Define classes dialog of TerraScan.
- **Echo type** - echo type as text string. Possible values are 'First', 'Last', 'Intermediate', and 'Only'.
- **Index** - unique number for each point.
- **Intensity** - intensity value as integer.
- **Line** - line number.
- **Time** - time stamp.
- **Collection** - number of collection shape. Only used with Output collections command for projects.
- **Group** - group number. Can be assigned automatically to point groups by using the Assign groups command or macro action, or manually by using the Create point group tool.
- **Surface dz** - difference between a point and a TerraModeler surface. This requires that a surface model is loaded into TerraModeler.
- **Red 8|16**, **Green 8|16**, **Blue 8|16** - RGB color values. The number determines the color depth which can be either 8-bit or 16-bit. Color values can be assigned to points by using the Extract color from images command, or the Assign color to points command or macro action.
- **Echo number** - echo number as number.
- **Number of echos** - total number of echos at the position of a point.
- **Mirror angle** - scan angle in degrees. Values must range between -128 to +127.
- **Scanner** - scanner number.
- **Image** - image number. An image number can be computed and stored for points if the corresponding setting in the Extract color from images command is set.
- **Distance** - distance value computed with the Compute distance command or macro action.
- **Echo length** - echo length value. The value can be extracted from waveform information in TerraScan using the Extract echo properties command. It can also be provided by system-specific point formats, such as *Riegl Extra Bytes* in LAS files.
- **Parameter** - parameter value. The value may be set from an attribute of a system-specific point format, such as *Riegl Extra Bytes* in LAS files.
- **Dimension** - dimension attribute computed with Compute normal vector command or macro action.

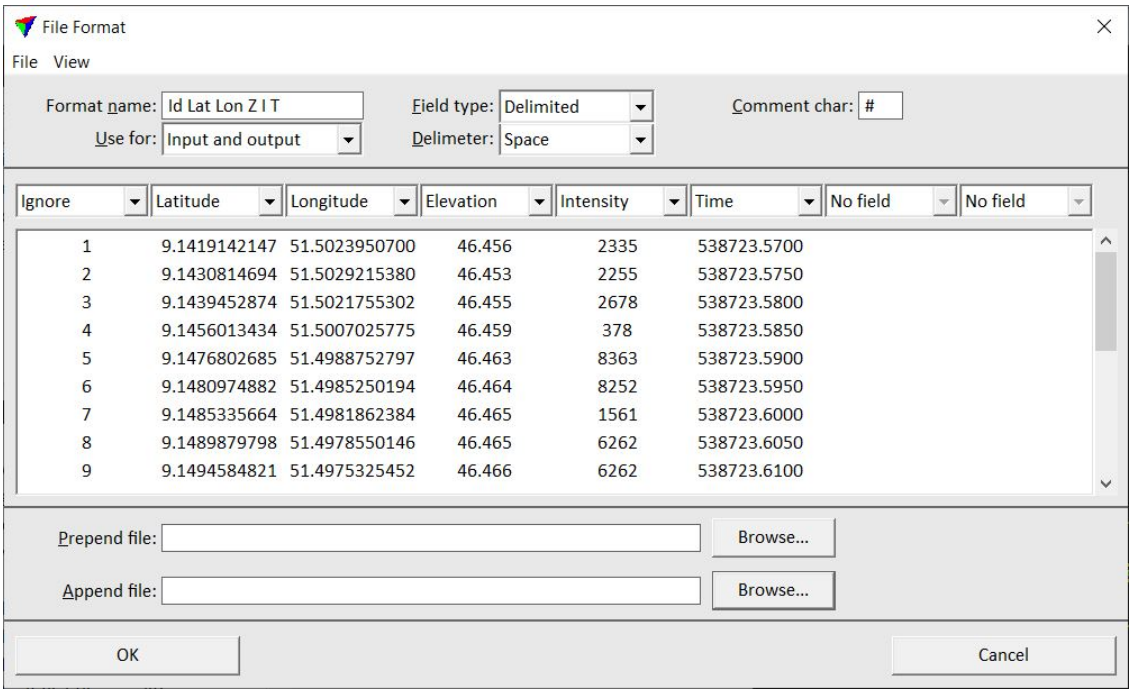
- **Normal X**, **Normal Y**, **Normal Z** - XYZ normal vector components computed with Compute normal vector command or macro action.

To define a new point format:

1. Open the **User point formats** category in the **File formats** folder.

2. Click **Add** in the **Settings** dialog.

   The **File Format** dialog opens:



3. (Optional) Select **Load example** command from the **File** pulldown menu of the **File Format** dialog. Select an example file of the format you want to define.

   This reads the first lines of the text file and shows its content in the field list. The software also tries to detect the **Field type** and the **Delimiter**.

4. If required, change the number of fields that are available in the dialog by using the commands from **View** pulldown menu. You can choose between 8, 10 or 15 fields.

5. Type a **Format name** and define the other settings.

6. Select the correct attribute for each field you want to import.

7. Click OK to the **File Format** dialog.

8. Close the **Settings** dialog in order to save the modified settings for TerraScan.

SETTING	EFFECT
Format name	Descriptive name of the new format.
Use for	Defines the usage of the format:

SETTING	EFFECT
	• **Input only** - files of this format can be loaded into TerraScan using the Read points command or the Load Airborne Points tool. • **Output only** - files of this format can be saved into new text files using the Save points As command or the Output points, Output by line macro actions. • **Input and output** - files of this format can be loaded into TerraScan and saved into new text files.
Field type	Defines fields are separated in the text file: **Delimited** or **Fixed length**.
Delimiter	Delimiter character used in text files: **Space**, **Tabulator**, **Comma**, or **Semicolon**. This is only active if **Field type** is set to **Delimited**.
Comment char	Character that introduces comment lines in the text file. Lines beginning with this character are ignored when points are read from a text file.
Prepend file	Location of a text file from which the content is added at the beginning of an output file. This is not active if **Use** for is set to **Input only**.
Append file	Location of a text file from which the content is added at the end of an output file. This is not active if **Use** for is set to **Input only**.

There are some text file format already implemented in TerraScan. See Supported file formats for a list of implemented file formats. User point formats are stored in a configuration file OUTFMT.INF in the TerraScan installation folder. You can copy this file to other computers in order to make point file formats available on them.

File formats / User trajectory formats

User trajectory formats category in **File formats** folder contains a list of user-defined trajectory formats. You can define your own file formats which can be used when reading in trajectory information from text files.

For the definition of trajectory formats, the same steps and settings apply as for point formats described above in File formats / User point formats. The differences in usage and attributes are listed in the table below.

SETTING	EFFECT
Use for	Defines the usage of the format:

SETTING	EFFECT
	• **Input only** - files of this format can be loaded into TerraScan using the Import files command from the Manage Trajectories dialog. • **Output only** - files of this format can be saved into new text files using the Output positions command from the Manage Trajectories dialog. • **Input and output** - files of this format can be loaded into TerraScan and saved into text files.
No field	Selection of what trajectory position attribute is stored in the field: • **No field** - no field defined in the text file. • **Ignore** - the column in the text file is ignored. • **Time** - time stamp. • **Easting**, **Northing**, **Elevation** - XYZ coordinates. • **Longitude**, **Latitude** - position in degrees, minutes, and seconds. • **Heading**, **Roll**, **Pitch** - orientation angles. • **X|Y|Z accuracy** - accuracy estimates for XYZ position values. • **Heading|Roll|Pitch accuracy** - accuracy estimates for orientation angle values.

Trajectory formats are stored in a configuration file TRAJFMT.INF in the TerraScan installation folder. You can copy this file to other computers in order to make trajectory formats available on them.

Loaded points

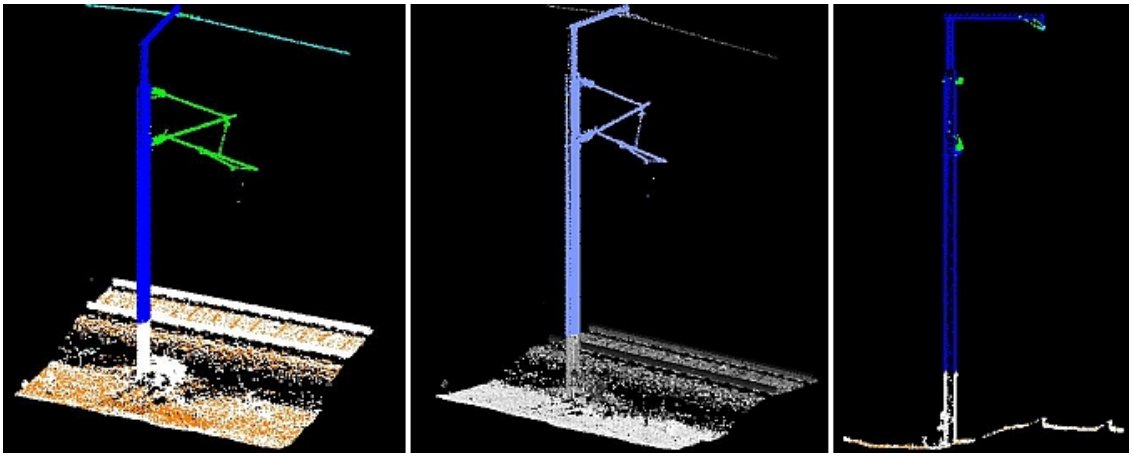
Loaded points category defines the symbology for highlighting selected points in TerraScan. It is used, for example, if the location of several selected points is shown by using the **Show location** button in the TerraScan window.

SETTING	EFFECT
Hilite color	Color for the rectangles drawn around selected points. Uses the active color table of the CAD file.
Hilite weight	Line weight for the rectangles drawn around selected points. Uses standard line weights.

Object library

Object library category shows a list of object definitions that can be used for automatic object detection. Currently, poles are the objects that can be detected automatically from very dense point clouds, such as mobile scanning point clouds. Find poles tool makes use of the object library.

Object detection in TerraScan relies on the computation of distance from ground and groups of points with a certain shape. Therefore, it is required to compute distance and assign groups in a point cloud in order to define an object in the **Settings** and to detect objects. For object definition, it is further required to classify the static parts of a sample object in order to separate them from the variable parts of the object. Distance computation, group assignment, and classification can be done using the menu commands for loaded points in the **TerraScan** window, manual classification tools, or macro actions.



Sample point cloud for object definition. Classification of static (blue)/variable parts (green) shown on the left and group shown in the center image.

The right image shows a vertical cross section of the sample point cloud for object definition. The arm of the pole points in the direction of the alignment, such as the trajectory line.

The object detection can be supported by an alignment element, a line element drawn in the CAD file. A trajectory line drawn in the CAD file can be used as alignment. The object definition defines whether an object is located left or right of the alignment or, with other words, whether the scanner system was driven on the left side or right side of the object. It is recommended to use an alignment in order to speed up object detection significantly.

In the object detection process, the software classifies points on objects into a given class. The object definition determines the target class.

(MicroStation only) In addition, it may place cell elements in the CAD file. A cell element can be created using MicroStation tools for cell management. Cells are stored in cell libraries. A cell library must be attached to the CAD file before cells from the library can be used. The object definition determines the name of the cell that is placed for a certain object type.

The object items of the library are stored in text files in the TerraScan installation folder, for example C:\TERRA64\TSCAN\OBJECT_LIBRARY. The text file stores the points belonging to the group in the sample point cloud using object coordinates. The origin of the object group is defined by the base point of an object and set to XYZ = 0,0,0 in the object coordinate system. The X axis points in cross

section direction of the object (to the right in a cross section view), the Y axis is perpendicular to the X axis (away from the viewer in a cross section view), and the Z axis points upwards (up in a cross section view).

You can **Add**, **Edit**, and **Delete** objects by using the corresponding buttons in the **Settings** dialog. The delete action deletes the object definition from the settings but it does not delete the object text file from the object library folder.

To add a new object item to the library:

1. Open the **Object library** category

2. Choose a sample object in the point cloud. You may create a separate file from the sample.

3. (Optional) Make sure that the sample represents an "ideal" object, for example, a vertical pole. You may rotate the sample point cloud of a leaning pole by using the Rotate points button in the **Settings** dialog.

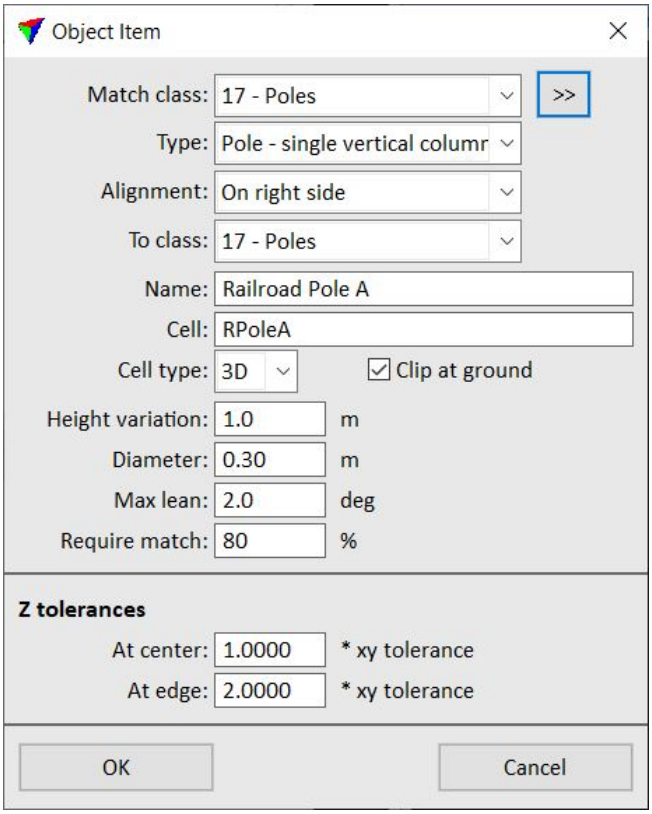
4. (Optional) Create a cell element by using MicroStation tools for cell management. (*Not Spatix*)

If distance computation and grouping is not yet done for the point cloud, it must be done for the sample before an object can be defined.

5. Draw an accurate vertical cross section of the sample object. If the object has a horizontal cross arm, the centerline of the section should follow the direction of the cross arm accurately.

6. Click **Add** in the **Settings** dialog.

   The **Object Item** dialog opens:



7. Define settings and click OK.

8. Define the object base point with a data click in the cross section view.

   The base point defines the origin of the object in an object coordinate system. You may snap to a point at ground level in order to define the base point.

9. Define the group which defines the object with another data click close to points of the group.

   This adds the object to the library.

10. Close the **Settings** dialog in order to save the object library settings for TerraScan.

SETTING	EFFECT
Match class	Point class used for defining the sample object. The class should contain point from the static parts of an object. This is only available when a new object is added. The setting is not available when an existing object item is edited.
Type	Type of the object: • **Pole - single vertical column** - pole object with a single column and maybe horizontal parts. • **Pole - complex** - pole object of a more complex structure.
Alignment	Location of an object relative to an alignment element: • **Not defined** - no location relative to an alignment is defined. • **On left side** - the alignment is located left of the sample object. • **On right side** - the alignment is located right of the sample object.
To class	Point class into which points of groups of this object type are classified.
Name	Name of the object item. Defines the name of the text file created for this object item.
Cell	Name of the cell element that is placed for the object item. The same name must be used for the cell element in a MicroStation cell library.
Clip at ground	If on, the cell element is clipped at ground level. Parts under ground are removed from the cell element.
Height variation	Maximum variation of the height of the object.
Diameter	Approximate diameter of the pole column. This is only available if **Type** is set to **Pole - single vertical column**.

SETTING	EFFECT
Max lean	Maximum angle by which an object may be off from vertical.
Require match	Determines how well an object must match the sample object definition in order to be detected as this object type.
At center	The two **Z tolerances** values determine the elevation variation of points on vertical parts relative to horizontal parts of an object in the point cloud compared to the sample object point cloud. In very dense MLS point clouds, points on the edge of objects, such as vertical arms of street lamps, railway posts, etc. vary more than points on the central vertical part of the object. Thus, **At edge** value should be larger than **At center** value, for example 1.0 vs. 3.0. In less dense ALS point clouds, horizontal parts of objects, such as powerline tower arms, are represented more confidently and with less variation than vertical parts. Thus, **At center** value should be larger than **At edge** value, for example 3.0 vs. 2.0.
At edge	

Object definitions are stored in a configuration file OBJECT_LIBRARY.INF in the TerraScan installation folder. The sample objects in the library are stored in folder \OBJECT_LIBRARY as text files with the extension .XYZ. You can copy these files to other computers in order to make object definitions available on them.

Rotate points

Rotate points can be used to rotate a sample point cloud in a section view. This may be required, for example, to get a sample pole object vertical.

To rotate points:

1. Draw a vertical cross section of the object. The vertical part of the object should be clearly visible in the section. If the object has a horizontal cross arm, the centerline of the section should follow the direction of the arm accurately.

2. Select **Rotate points** button in the **Object library** category of the **Settings**.

3. Define a first base point with a data click, for example close to the lower end and on the left side of a pole.

   The software shows a horizontal line at the mouse pointer location which acts as a helping line for defining the base of a vertical object.

4. Define a second base point with another data click on the right side of a pole.

The software shows a vertical helping line which extends dynamically with the mouse pointer movement.

5. Define a first top point with a data click, for example close to the upper end and on the left side of a pole.

   The software shows a horizontal line at the mouse pointer location which acts as a helping line for defining the top of a vertical object.

6. Define a second top point with another data click on the right side of a pole.

   This rotates the point cloud according to the four points defined with data clicks.

7. Use the Cut section tool in order to create another vertical section of the pole that is perpendicular to the previous section.

8. Repeat steps 2 to 6 in order to rotate the pole in the second section view.

Create a MicroStation cell element

Not Spatix

A MicroStation cell element can be placed by TerraScan for representing an object. The cell may be a simple 2D symbol or a more complex 3D structure. The cell origin should be the same as the origin/base point of the TerraScan object group. For objects standing on the ground, the origin should be on the ground.

MicroStation cell elements are stored in cell libraries. A cell library is stored in files with the extension .CEL. In order to create, modify, or use a cell element, the cell library must be attached to the CAD file.

Example workflow for creating a cell element:

1. Draw a vertical cross section of the object. If the object has a horizontal cross arm, the centerline of the section should follow the direction of the arm accurately.

   A line element drawn in the center of the cross arm and following its direction may be used as helping line for creating accurate cross section views and for making the section view creation repeatable.

2. (Optional) Draw another helping line that starts from the center base point of the object. For a single-column pole object, the base point is the center of the column at ground elevation. Use, for example, Mouse point adjustment tool in order to place the helping line start point exactly at ground elevation.

3. Digitize the object using MicroStation tools. The object may be represented by lines, shapes, or any other MicroStation element type. Several elements of different types may form a cell. Snap to the start point of the helping line created in step 2 in order to place the base point of the cell on the ground. You may use any drawing aids of MicroStation for digitizing the elements of the cell.

4. Extend the base point of the object below the ground. For example, if the height of an object may vary by around 1.5 m, extend the base point to about 2 m below ground. You may use, for example, the **Extend Line** tool of MicroStation in order to extend a line element to a specified distance below ground.

5. Select **Define Cell Origin** tool of MicroStation. Snap to the start point of the helping line created in step 2 and place the origin of the cell with a data click. The cell origin should be located on the base point of the object at ground elevation.

6. Select all elements that form the cell for the object.

   OR

6. Select **Place Fence** tool of MicroStation. Set **Fence Type** to be **Block** or **Shape**, and **Fence Mode** to be **Inside**. Place a fence that includes completely all elements that form the cell for the object. Do not include the helping lines.

7. Open the **Cell Library** window of MicroStation.

8. Select **New** command from the **File** pulldown menu in order to create a new cell library. Define a location and name for storing the library.

   OR

8. Select **Attach file** command from the **File** pulldown menu in order to attach an existing cell library. Browse to the storage location and select the library file.

9. Select the **Create** button from the **Cell Library** window. The button is only active if a cell origin has been placed and if elements are selected or inside a fence (steps 5 and 6).

10. Define a name and description of the cell. The cell name is the link between the TerraScan object item and the cell element. Select cell type **Graphic** and click **Create**.

Operation

Operation category defines actions performed when TerraScan is loaded.

SETTING	EFFECT
Open Main window	If on, the TerraScan window is opened.
Open Main tool box	If on, the TerraScan toolbox is opened.
Close AccuDraw	If on, the **MicroStation AccuDraw** is closed. This has no effect in Spatix.
Set AccuSnap off	If on, the **MicroStation AccuSnap** is deactivated. This has no effect in Spatix.
Maximum	Maximum amount of threads used for TerraScan processing. Normally, you should set this to the number of processor cores of your computer or a bit higher.

Point display

Point display category determines how points in TerraScan are drawn on the screen relative to other CAD file or Terra Application elements. It also defines default display settings as well as various additional settings related to more specific point display in TerraScan.

SETTING	EFFECT
Display method	
Order	Order of drawing points on the screen relative to other elements, such as vector data in the CAD file, images in TerraPhoto, surface models in TerraModeler: • **Before vectors** - point are drawn first on the screen, before other elements. This means, they are displayed behind other elements in CAD file views. • **After vectors** - points are drawn last on the screen, after other elements. This means, they are displayed in front of other elements in CAD file views.
Default display mode settings - The values can be changed in the Display mode dialog.	
Weight	Default size of points on the screen. Can be defined by point class or uses standard line weights.
Speed	Default speed for point display: • **Fast - sparse points** - amount of displayed points depends on the zoom factor. If you zoom out, only a subset of points is drawn. This is the recommended setting for displaying a larger amount of points. • **Normal** - more points are drawn. The software decides based on the density of the point cloud. In sparse data sets, this already draws all points. • **Slow - all points** - all points are drawn at every zoom level. This may slow down the display speed for a large amount of points.
Borders	Default setting for enhanced depth display of the point cloud. The setting determines how much space on the screen is covered by black borders that separate foreground points from background points. An enhanced depth perception can be achieved by displaying points with **Borders > 0**. See also the description of the setting in the Display mode dialog.

SETTING	EFFECT
Coloring schemes - The same action can be performed for the corresponding coloring modes in the Display mode dialog.	
Fit automatically	If on, color schemes for displaying points by elevation and intensity are fitted automatically to the corresponding values of points when they are loaded into TerraScan. If off, the software keeps the elevation and intensity values of the previous data set for the color schemes.
Background in camera views	
Draw sky if 'Use depth' is on	If on, the background of camera views is drawn as "sky color scheme" (blue to black color gradient). This effects views for which the **Borders** setting is > 0 in the Display mode dialog. It may result in a more realistic point cloud visualization, especially for mobile ground-based data sets.
Group coloring mode - Refers to the Group coloring mode in the Display mode dialog.	
Display non-grouped points	If on, points that are not assigned to any group are displayed with the color selected in the color field. Click on the color field in order to select the display color. Uses the active color table of the CAD file.
Vegetation index limits - Refers to the Vegetation index coloring mode in the Display mode dialog.	
Normalized difference	Threshold value for displaying points by Vegetation index / Normalized difference method. The default value is 0.0, difference values may range from -1 to +1.
Visual band difference	Threshold value for displaying points by Vegetation index / Visual band difference method. The default value is 0.05, difference values may range from -1 to +1.

Powerlines / Active line

Active line category in **Powerlines** folder defines settings for the display of an active powerline line string. The settings effect the display of a line string element after it has been selected by the Activate Powerline tool.

SETTING	EFFECT
Hilite	Parts of the line string that are highlighted: **No hilite**, **Vertices**, or **Line segments**.

SETTING	EFFECT
Color	Color of a highlighted line string. Uses the active color table of the CAD file.
Weight	Line weight of a highlighted line string. Uses standard line weights.
Style	Line style of a highlighted line string. Uses standard line styles.

Powerlines / Profile layouts

Profile layouts category in **Powerlines** folder contains a list of user defined profile layouts. Each layout definition contains a list of data rows that appear below the profile. Currently, the software doesn't use profile layouts in any tool or command.

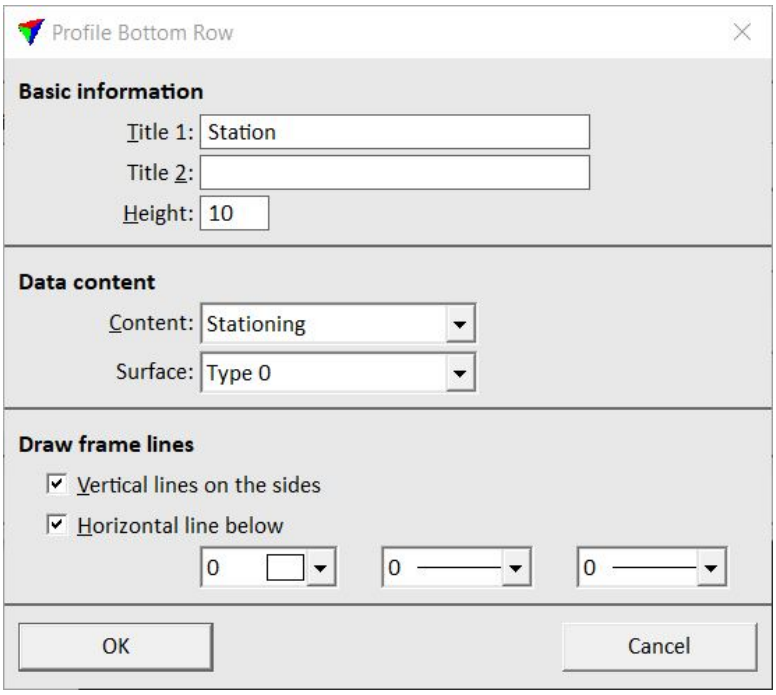
You can **Add**, **Edit**, and **Delete** profile layouts by using the corresponding buttons in the **Settings** dialog. The **Copy** button creates an identical copy of a selected layout definition. You can **Add**, **Edit**, and **Delete** bottom rows to a profile layout by using the corresponding buttons in the **Profile layout** dialog.

To define a new profile layout:

1. Open the **Profile layouts** category in the **Powerlines** folder.
2. Click **Add** in the **Settings** dialog.

   The **Profile layout** dialog opens.
3. Type a **Name** for the profile layout.
4. Click **Add** in the **Profiles layout** dialog in order to add a new data row that is displayed below a profile.

   The **Profile Bottom Row** dialog opens:



5. Define **Basic information** settings.

6. Select an auto-text option for the **Content** list as well as additional settings depending on the content selection. Choose **Other** as **Content** if nothing of the list entries fit to your data.

7. Select settings for **Draw frame lines**.

8. Click OK in the **Profile Bottom Row** dialog.

9. Add more data rows if necessary.

10. Click OK to the **Profile layout** dialog.

11. Close the **Settings** dialog in order to save the profile layout settings for TerraScan.

SETTING	EFFECT
Title 1	Text used as first line of a title in the bottom row.
Title 2	Text used as second line of a title in the bottom row.
Height	Height of the bottom row. Given in millimeters on paper.
Content	Defines the type of information displayed in the bottom row: • **Stationing** - stations along the alignment element of the profile. • **Surface elevations** - elevations of surfaces of the given **Surface type**. This refers to surfaces loaded in TerraModeler.

SETTING	EFFECT
	• **Tower base z** - elevation of a tower base point. • **Tower turn angle** • **Tower span** • **Crossing object stationing** - station along the alignment element where another object crosses the powerline. • **Other** - space reservation for any other content that can be added manually.
Vertical lines on the sides	If on, vertical lines are drawn on the left and right side of the bottom row.
Horizontal line below	If on, a horizontal line is drawn below the bottom row using the selected color, line weight and style.

Powerlines / Tower functions

Tower functions category in **Powerlines** folder contains a list of different functions for powerline towers. Typical function examples are suspension towers, tension towers, and dead-end towers.

You can **Add**, **Edit**, and **Delete** tower functions by using the corresponding buttons in the **Settings** dialog.

A tower function is defined by an **Abbreviation** and a **Description** which can be typed in the fields of the **Tower Function** dialog.

The tower function is applied when the tower model is placed using the Place Tower tool. The information can be included in a report created by the Export Powerline tool.

Tower functions are stored in a configuration file TOWER_FUNCTIONS.INF in the TerraScan installation folder. You can copy this file to other computers in order to make tower functions available on them.

Powerlines / Tower statuses

Tower statuses category in **Powerlines** folder contains a list of different statuses for powerline towers. Status examples may be existing, planned, broken, etc.

You can **Add**, **Edit**, and **Delete** tower statuses by using the corresponding buttons in the **Settings** dialog.

A tower status is defined by an **Abbreviation** and a **Description** which can be typed in the fields of the **Tower status** dialog.

The tower status is applied when the tower model is placed using the Place Tower tool. The information can be included in a report created by the Export Powerline tool.

Tower statuses are stored in a configuration file TOWER_STATUSES.INF in the TerraScan installation folder. You can copy this file to other computers in order to make tower statuses available on them.

Powerlines / Tower types

Tower types category in **Powerlines** folder contains a list of user-defined tower types. The tower type determines the general design of a tower, including the number, position and length of cross arms and attachments.

The values defined for the tower height as well as position and length of cross arms and attachments do not have to be exact if towers are placed with the Place Tower tool using no template. For placing towers using templates the accuracy of the values determine how well a template fits to the real towers design.

It is recommended to enter a text in the **Description** fields of **Tower Type** and **Tower Type Cross Arm** dialogs, because the editing tools for powerline processing refer to this field. The other descriptive information is mainly used in reports. See Chapter Powerlines for more information about powerline processing.

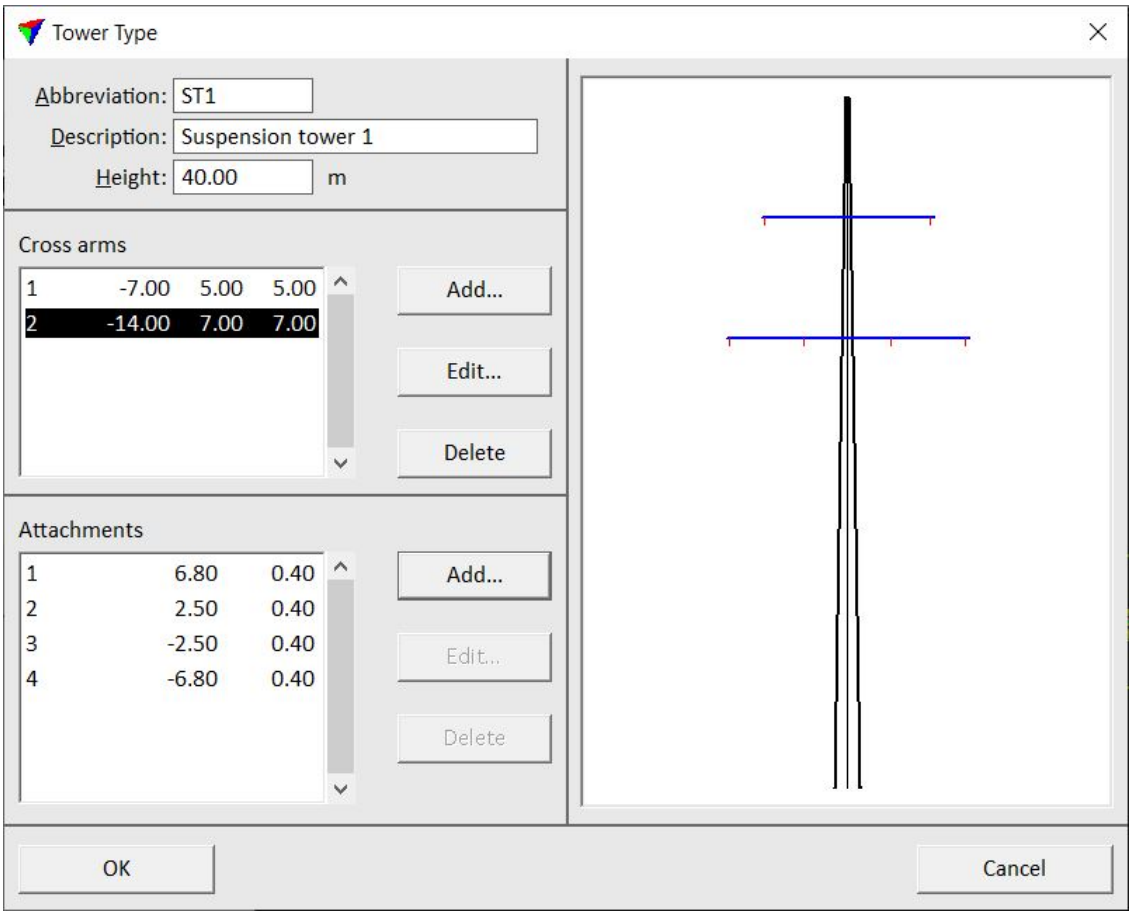
You can **Add**, **Edit**, and **Delete** tower types by using the corresponding buttons in the **Settings** dialog. You can **Add**, **Edit**, and **Delete** cross arms and attachments for a tower type by using the corresponding buttons in the **Tower type** dialog.

To add a new tower type:

1. Open the **Tower types** category in the **Powerlines** folder.

2. Click **Add** in the **Settings** dialog.

   The **Tower Type** dialog opens:

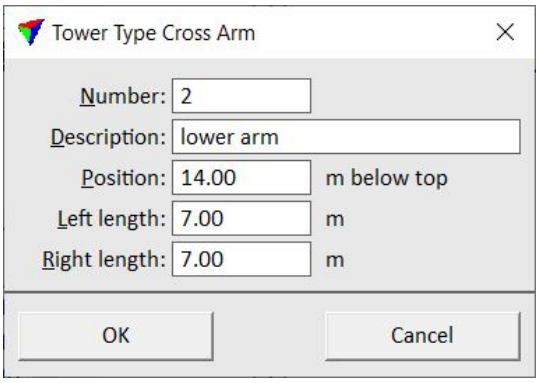


3. Define **Abbreviation**, **Description** and **Height** values.

SETTING	EFFECT
Abbreviation	Abbreviation of the tower type.
Description	Description of the tower type.
Height	Height of a tower.
Cross arms	List of cross arms for this tower type. Use buttons next to the list to **Add**, **Edit**, and **Delete** cross arms.
Attachments	List of attachments per cross arm. Select a cross arm and use buttons next to the list to **Add**, **Edit**, and **Delete** cross arms.

4. Click **Add** in the **Tower Type** dialog in order to add a cross arm.

   This opens the **Tower Type Cross Arm** dialog:



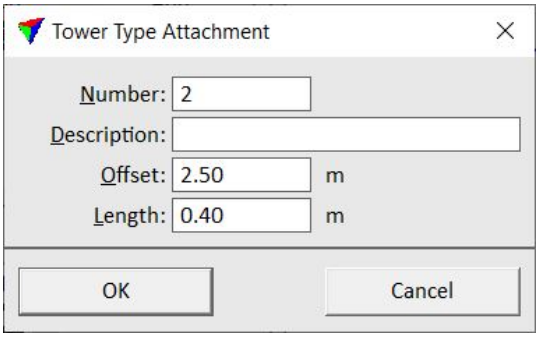
5. Define settings and click OK.

SETTING	EFFECT
Number	Number of the cross arm. Increases automatically for each new arm added to the tower type.
Description	Description of the cross arm.
Position	Position of the cross arm relative to the top of the tower.
Left length	Length of the cross arm to the left side of the tower.
Right length	Length of the cross arm to the right side of the tower.

6. Repeat steps 4 and 5 for all cross arms that belong to this tower type.

7. Select a cross arm and click **Add** in the **Tower Type** dialog in order to add an attachment to the selected cross arm.

   This opens the **Tower Type Attachment** dialog:



8. Define settings and click OK.

SETTING	EFFECT
Number	Number of the attachment. Increases automatically for each new attachment added

SETTING	EFFECT
	to a cross arm.
Description	Description of the attachment.
Offset	Position of the attachment along the cross arm relative to the tower center. A positive offset creates an attachment right of the tower center, a negative offset left of the tower center.
Length	Length of the attachment.

9. Repeat steps 7 and 8 for all attachments per cross arm and all cross arms of the tower type.

10. Click OK to the **Tower type** dialog.

11. Close the **Settings** dialog in order to save the tower type settings for TerraScan.

Tower types are stored in a configuration file TOWER_TYPES.INF in the TerraScan installation folder. You can copy this file to other computers in order to make tower types available on them.

Rail section templates

Rail section templates category shows a list of rail section definitions that can be used for automatic rail vectorization using the Find Rails tool.

A drawing of the rail section can be used to define the geometry of a rail section template. Both rails of the rail section must be drawn into a CAD file using the correct measures. It is an advantage to draw them in a way that the center point of the rail section is at the CAD file origin (coordinates XY = 0,0). As an alternative, you can also define a rail section by typing the start and end point coordinates of section lines in an input dialog.

You can **Add**, **Edit**, and **Delete** rail section templates by using the corresponding buttons in the **Settings** dialog. You can **Add**, **Edit**, and **Delete** parts of a rail section templates by using the corresponding buttons in the **Rail section template** dialog.

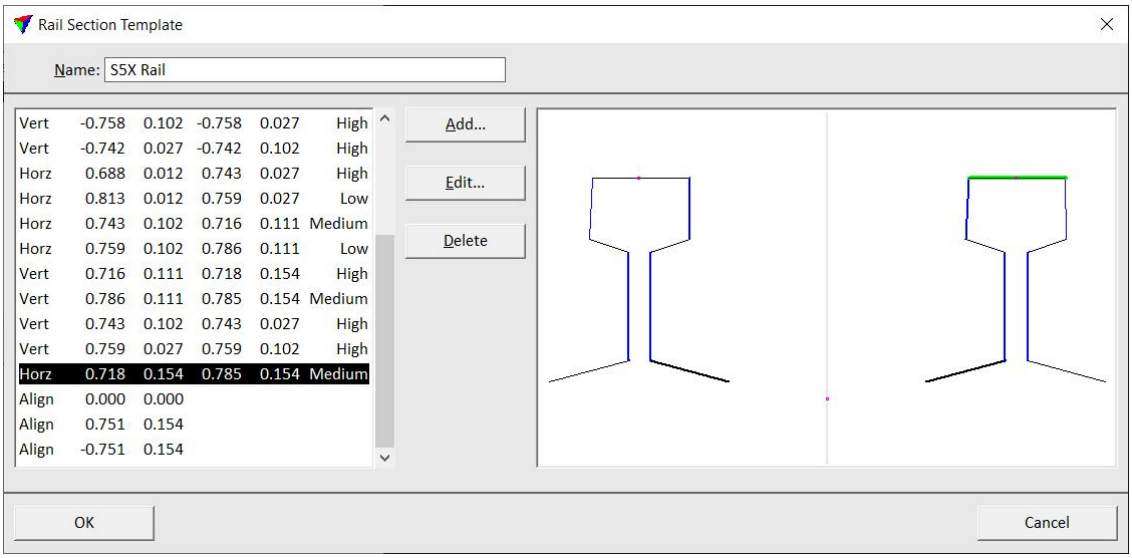
To add a new rail section:

1. (Optional) Draw the rail section into a CAD file and select it.

2. Open the **Rail section templates** category.

3. Click **Add** in the **Settings** dialog.

   The **Rail Section Template** dialog opens:



If a section drawing has been selected, the section definition is shown in the dialog.

The dialog shows the list of rail section elements on the left side. A preview of the rail section is shown on the right side. In the preview, horizontal elements are drawn in blue, vertical elements in black, void elements in red. Elements of high weight are drawn with a thicker line, medium and low weight elements with thinner lines. Alignment positions are marked with a small red dot. An element selected in the list is highlighted in green in the preview.

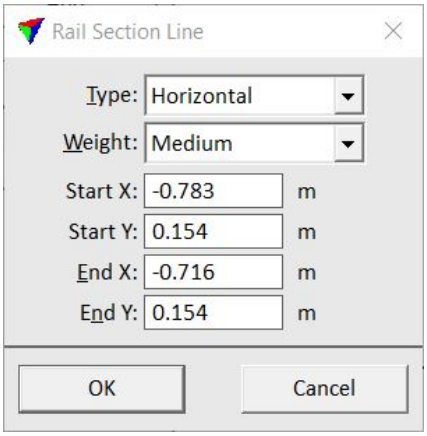
4. Type a **Name** for the rail section.

5. Click **Add** in the **Rail Section Template** dialog in order to add a new element to the section.

   OR

6. Select a line in the list of section elements and click **Edit** in the **Rail Section Template** dialog in order to edit an existing element of the section.

   The **Rail Section Line** dialog opens:



7. Define settings and click OK.

8. Add/Edit more elements of the section if necessary.

9. Click OK in the **Rail Section Template** dialog.

10. Close the **Settings** dialog in order to save the rail section template settings for TerraScan.

SETTING	EFFECT
Type	Type of the rail section line: • **Horizontal** - horizontal line as part of the rail section. Used to find the Z location of a rail in automatic rail detection. • **Vertical** - vertical line as part of the rail section. Used to find the XY location of a rail in automatic rail detection. • **Void** - line that indicates a location without laser data close to rails. • **Alignment** - location of linear vector elements that are drawn in automatic rail vectorization.
Weight	Weight of a rail section line: **Low**, **Medium**, or **High**. A line with higher weight takes priority over lines with lower weights in automatic rail detection. This is only active if **Type** is not set to **Alignment**.
Start X | Y End X | Y	Start and end point coordinates of a rail section line. Given in the rail section's coordinate system. The origin of the system (0,0) should be in the center of the rail section. This is only active if **Type** is not set to **Alignment**.
Position X | Y	Location of an alignment element. Given in rail section's coordinates. This is only active if **Type** is set to **Alignment**.

Rail section templates are stored in a configuration file RAIL_SECTIONS.INF in the TerraScan installation folder. You can copy this file to other computers in order to make rail section templates available on them.

Road section parameters

Road section parameters category defines level, color, text size, and unit settings for drawing road section parameter elements into the CAD file. The settings effect the display of elements that have been detected with the macro action Compute section parameters and drawn into the CAD file using the Read / Section parameters command. The description of the macro action contains a detailed explanation of the different road section parameters.

SETTING	EFFECT
Level	CAD file level on which the different road section parameters are drawn.
Color	Color of the different road section parameters. Users the active color table of the CAD file.

SETTING	EFFECT
Text size	Size of text elements that are drawn for the different road section parameters.
Unit	Unit for expressing the different road section parameter values. Slopes can be expressed in **Degree** or **Percentage**, other parameters can be expressed in **Master units** of the CAD file or in **Millimeters** on paper.

Scanner systems

Scanner systems category shows a list of scanner system configurations. A scanner system can include several scanners where each scanner has its own lever arm definition. The system definition also defines the misalignment between the IMU and the scanner system.

In addition, each scanner in a system can contain a link to a waveform profile. See Scanner waveform profile for more information.

The scanner system number must be unique. It is used to establish a link between a scanner system and trajectory files.

You can **Add**, **Edit**, and **Delete** scanner systems by using the corresponding buttons in the **Settings** dialog. You can **Add**, **Edit**, and **Delete** scanners by using the corresponding buttons in the **Scanner System** dialog.

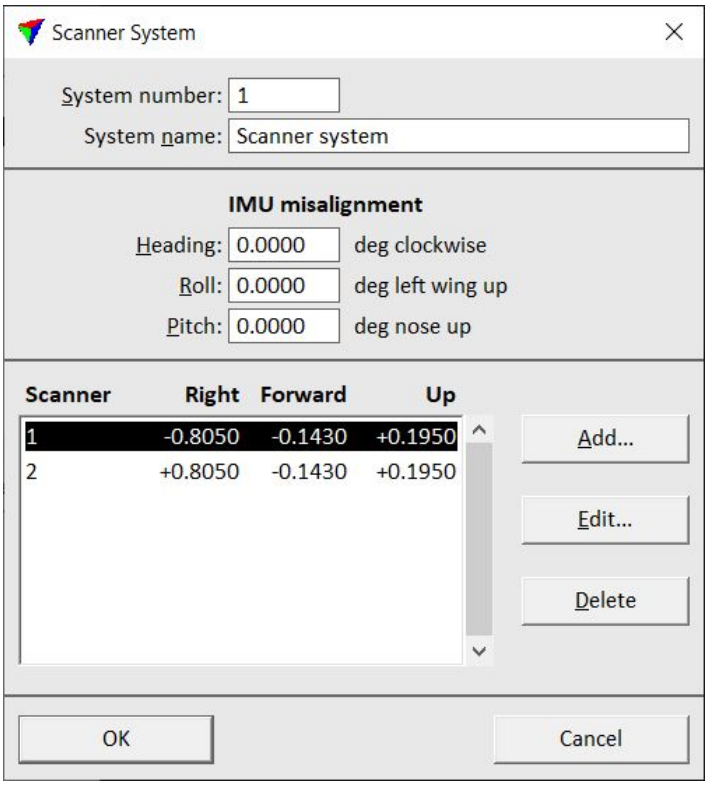
The lever arm of a scanner describes the distance between the IMU and the scanner. It is expressed as X, Y, and Z components of a vector. The direction of the three vector components is as follows:

- **X** - positive values to the right, negative to the left.
- **Y** - positive values forward, negative backward.
- **Z** - positive values up, negative down.

To add a new scanner system:

1. Open the **Scanner systems** category.
2. Click **Add** in the **Settings** dialog.

   The **Scanner System** dialog opens:

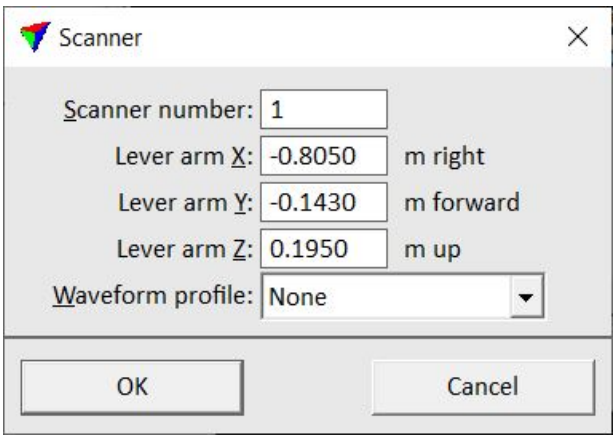


3. Type a **Scanner number** and a **Scanner name**.

4. If necessary, define values for the misalignment angles **Heading**, **Roll**, and **Pitch** between the scanner system and the IMU.

5. Click **Add** in the **Scanner System** dialog in order to add a new scanner.

   The **Scanner dialog** opens:



6. Define settings and click OK.

7. Repeat steps 5 and 6 for all scanners that belong to the system.

8. Click OK in the **Scanner System** dialog.

9. Close the **Settings** dialog in order to save the scanner system settings for TerraScan.

SETTING	EFFECT
Scanner number	Number of the scanner. Must be unique within a system.
Lever arm X | Y | Z	X, Y, and Z component of the lever arm vector between IMU and the scanner.
Waveform profile	Waveform profile linked to the scanner.

Scanner systems are stored in a configuration file SCANNER_SYSTEMS.INF in the TerraScan installation folder. You can copy this file to other computers in order to make scanner system definitions available on them.

Scanner waveform profiles

Scanner waveform profiles category shows a list of scanner waveform profile definitions. Scanner waveform profiles provide the reference or standard waveform shape that is typical for a scanner. They are required for waveform processing tasks, such as the extraction of echo properties or the extraction of additional points.

The waveform profile can be created from loaded laser points and trajectory information. The trajectory must include the link to the waveform file. The waveform profile can be best extracted from laser points on open ground, preferable hard surface, where a wider range of intensity values are represented. The sample points should not be too close to line edges.

You can **Add**, **Edit**, and **Delete** waveform profiles by using the corresponding buttons in the **Settings** dialog. You can also **Copy** a profile definition and view it in a text editor or **Paste** a profile definition from a text editor in TerraScan.

To create a new scanner waveform profile:

1. Use Manage Trajectories tool and commands from the **Trajectories** dialog in order to import and manage trajectories, and to link the trajectories with the waveform files.

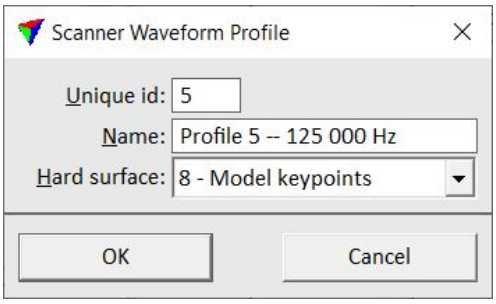
2. Load points into TerraScan.

3. Classify points in sample areas that are suited for creating the scanner waveform profile.

4. Open the **Scanner waveform profiles** category.

5. Click **Add** in the **Settings** dialog.

   The **Scanner Waveform Profile** dialog opens:



6. Define settings and click OK.

   The software extracts the waveform profile.

7. Close the Settings dialog in order to save the modified settings for TerraScan.

SETTING	EFFECT
Unique id	Number of the waveform profile. Must be unique in the list of scanner waveform profiles.
Name	Name of the waveform profile.
Hard surface	Laser point class that the software uses to extract the waveform profile. Uses the active class definitions in TerraScan.

Scanner waveform profiles are stored in a configuration file WAVEFORM_PROFILES.INF in the TerraScan installation folder. You can copy this file to other computers in order to make scanner waveform profiles available on them.

Section templates

Not Spatix

Section templates category shows a list of cross section templates. Section templates may represent, for example, clearance areas or tunnel sections. They can be used for the classification of point clouds with the By section template routine.

A section template is defined by the outline of the section and it's origin point. The outline must be drawn into a CAD file top view using correct measures. If the section is used in a classification step, it is aligned to an element by using the origin point.

You can **Add**, **Edit**, and **Delete** section templates by using the corresponding buttons in the **Settings** dialog.

To create a new section template:

1. Draw the outline of the cross section as closed shape in a CAD file top view.

2. (Optional) Draw a point at the position of the alignment element relative to the cross section shape.

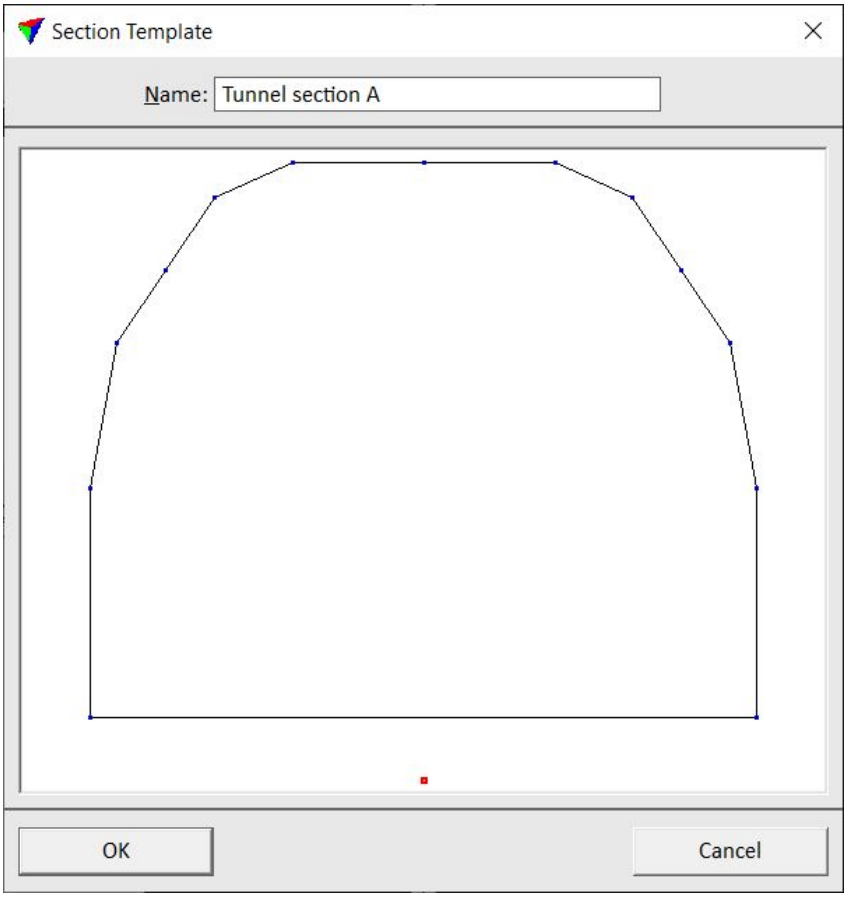
3. Open the **Section templates** category.

4. Select the cross section shape drawn in step 1.

5. Click **Add** in the **Settings** dialog.

6. Define the origin point of the section. If you created an element that represents the origin point, snap to the element in order to define the correct location.

   The **Section Template** dialog opens:



7. Type a **Name** for the section template.

8. Click OK in the **Tunnel Sections** dialog.

9. Close the **Settings** dialog in order to save the modified settings for TerraScan.

Section templates are stored in a configuration file SECTION_TEMPLATES.INF in the TerraScan installation folder. You can copy this file to other computers in order to make section templates available on them.

Slave computers

Slave computers category shows a list of computers that may participate in TerraSlave computation tasks. By default, the list shows only the local computer but you can add any network computer that you want to use for processing TerraSlave tasks.

A detailed description of TerraSlave is provided in the TerraSlave User Guide.

You can **Add**, **Edit**, and **Delete** computers by using the corresponding buttons in the **Settings** dialog.

To add a new computer:

1. Open the **Slave computers** category.

2. Click **Add** in the **Settings** dialog.

   The **Slave Computers** dialog opens:

3. Define settings and click OK.

4. Close the **Settings** dialog in order to save the slave computer settings for TerraScan.

SETTING	EFFECT
Computer	Name of the computer
Slave folder	Installation directory of TerraSlave on the **Computer**. The default local path is C:\TERRA64\TSLAVE. Use UNC paths for distant slave computers.
Max instances	Maximum amount of processor cores used for TerraSlave processing tasks.

Slave computers are stored in a configuration file SLAVE_COMPUTERS.INF in the TerraScan installation folder. You can copy this file to other computers in order to make slave computer definitions available on them.

Snapping

Snapping category determines whether the capability of snapping to loaded points is switched on or off. If **Snap to loaded points** is switched on, you can snap to laser points in the same way as to CAD elements by using the **Tentative** mouse button. This may be helpful, for example, for dynamically rotating a view around a location in the point cloud or for digitizing vector data based on the point cloud.

Street View images

Street View images category defines display settings for Google Street View© images opened by the Show Street View tool. In addition, the key for accessing the Street View© images is set.

SETTING	EFFECT
Access key	Hard-coded access key for accessing Street View© images. The key can be changed to a company's or person's own access key.
Image width	Width of the Street View© image in the browser window. Given in pixels.
Image height	Height of the Street View© image in the browser window. Given in pixels.
Field of view	Field of view angle. Given in degree.
Pitch	Angle off from horizontal viewing direction. Given in degree.

Target objects

Target objects category shows a list of target object definitions. Target objects are normally used for matching point clouds of static terrestrial laser scanners. Supported shape primitives of target objects include ball, cone, and pyramid.

A target object may represent the location of a control point for which the coordinate values are known.

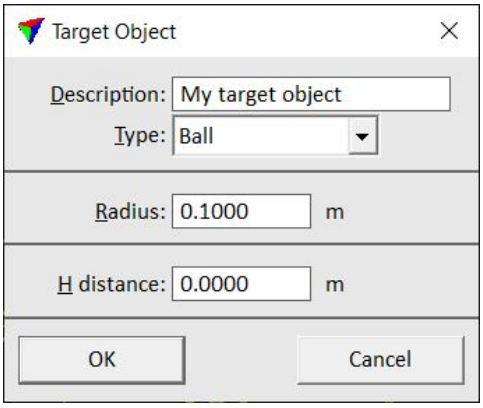
You can **Add**, **Edit**, and **Delete** target objects by using the corresponding buttons in the **Settings** dialog.

To add a new target object:

1. Open the **Target objects** category.

2. Click **Add** in the **Settings** dialog.

The **Target Object** dialog opens:



3. Define settings and click OK.

4. Close the **Settings** dialog in order to save the target object settings for TerraScan.

SETTING	EFFECT
Description	Description of the target object.
Type	Shape primitives: **Ball**, **Cone**, or **Pyramid**.
Radius	Radius of the ball or cone.
Depth	Depth of a cone or pyramid.
Width	Width of a pyramid.
Height	Height of a pyramid.
A distance	Distance from the target object's center point to the known control point location along the xy line from scanner to object.
B distance	Distance from the target object's center point to the known control point location perpendicular to xy line from scanner to object.
H distance	Elevation difference from the target object's center point to known control point location.

Target objects are stored in a configuration file TARGETS.INF in the TerraScan installation folder. You can copy this file to other computers in order to make target object definitions available on them.

Trajectory accuracies

Trajectory accuracies category lets you specify minimum accuracy estimates to apply when importing trajectories.

SETTING	EFFECT
Enforce minimum accuracies during import	If on, minimum accuracy estimates are applied when trajectories are imported into TerraScan.
Xy	Minimum accuracy estimate for horizontal coordinate values.
Z	Minimum accuracy estimate for vertical coordinate.
Heading	Minimum accuracy estimate for heading values values.
Roll/pitch	Minimum accuracy estimate for roll and pitch values.

Travel View tool

Travel View tool category defines basic camera and view settings for the Travel View tool.

SETTING	EFFECT
Camera	Camera viewing angle. Given in degree.
Front clipping	Distance in front of a camera view within which content is clipped off.
Back clipping	Distance after which content in a camera view is clipped off in the background.

Tree species

Tree species category shows a list of tree species. The tree type definitions are used in tree cell placement with the Create Tree Cells tool.

You can **Add**, **Edit**, **Sort**, and **Delete** tree species by using the corresponding buttons in the **Settings** dialog.

There are two example tree species defined in the default setup of TerraScan. They use cells of a cell library also provided with the TerraScan installation. The library is stored in the \CELL folder of the Terra applications installation directory, for example C:\TERRA64\CELL\KARTTALI.CEL.

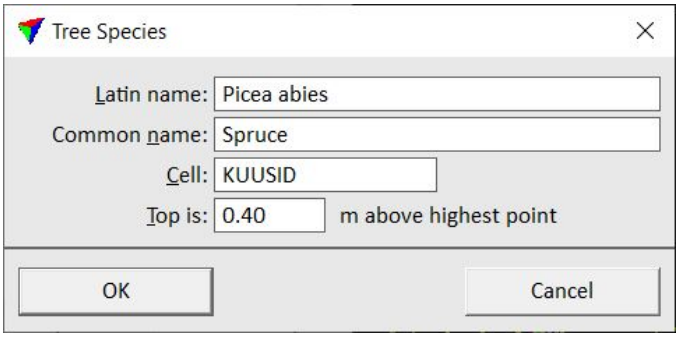
Any settings related to MicroStation cell elements do not have any effect in Spatix.

To create a new tree species:

1. Open the **Tree species** category.

4. Click **Add** in the **Settings** dialog.

The **Tree Species** dialog opens:



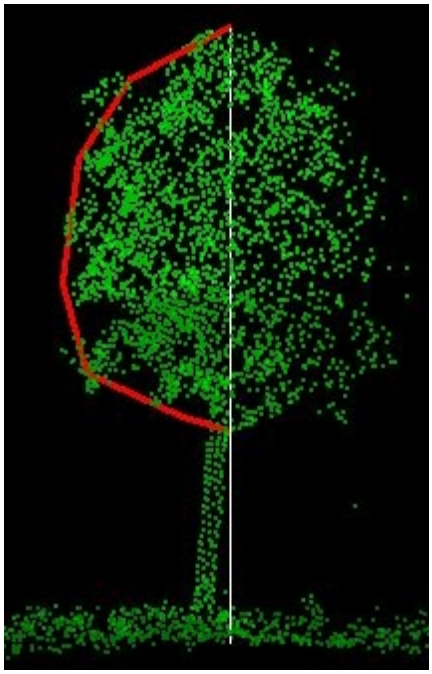
5. Define settings and click OK.

6. Close the **Settings** dialog in order to save the tree species settings for TerraScan.

SETTING	EFFECT
Latin name	Latin (scientific) name of the tree species.
Common name	Name of the tree species as it is commonly used.
Cell	Name of the cell used for this tree species. The name must correspond to the cell name used in a MicroStation cell library.
Top is	Distance between highest hit on the tree in the point cloud and the top of the tree cell.

Tree types are stored in a configuration file TREE_SPECIES.INF in the TerraScan installation folder. You can copy this file to other computers in order to make tree type definitions available on them.

Tree visualization types



Tree visualization types category shows a list of tree types. A tree type is defined by the shape of the tree crown cross section and additional parameters. The tree type definitions are used in automatic tree detection from laser points with the Detect trees command.

You can **Add**, **Edit**, and **Delete** tree types by using the corresponding buttons in the **Settings** dialog.

Any settings related to MicroStation cell elements do not have any effect in Spatix.

To create a new tree type:

1. Draw an outline of one half of a tree crown cross section as a line string in a CAD file front view.

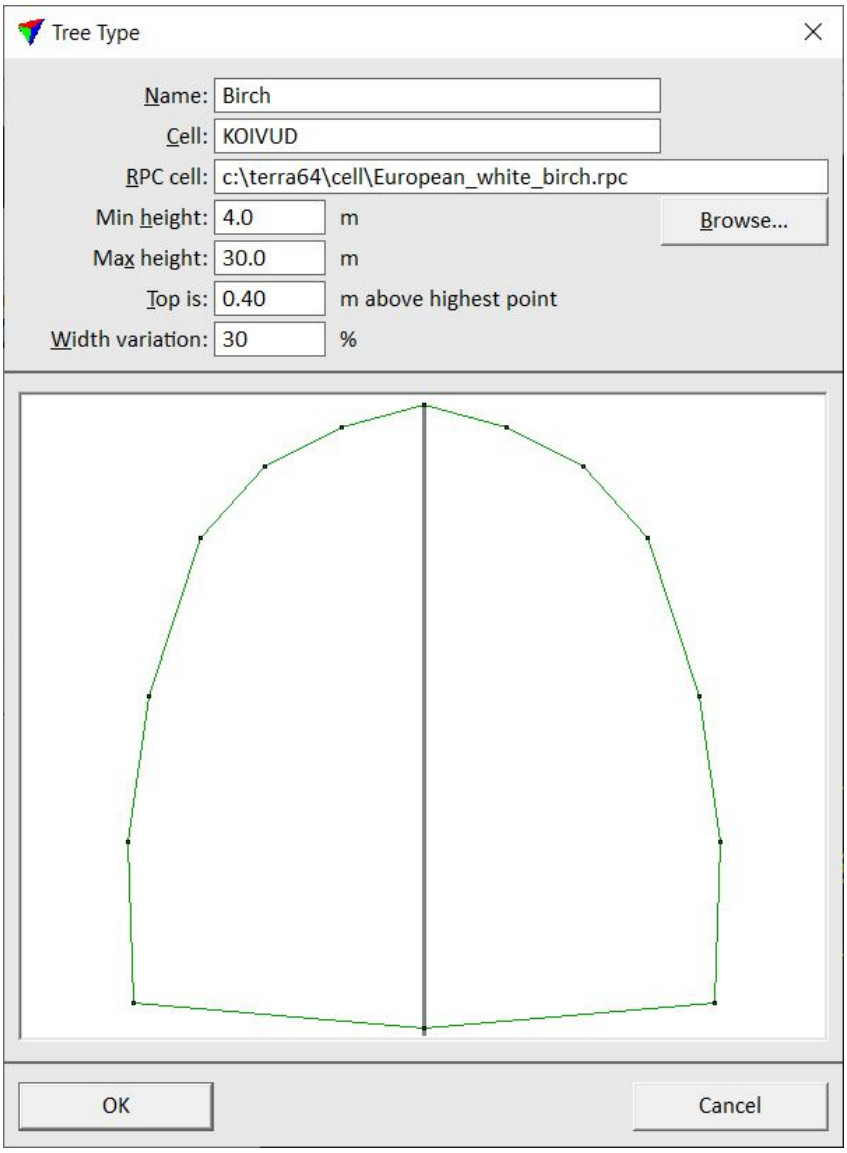
You may use a cross section of a tree in laser points as background for digitizing the tree crown shape. The image on the right shows the approximate tree section centerline in white (used as helping line) and the outline in red.

2. Select the outline.

3. Open the **Tree types** category.

4. Click **Add** in the **Settings** dialog.

   The **Tree Type** dialog opens:



5. Define settings and click OK.

6. Close the **Settings** dialog in order to save the tree type settings for TerraScan.

SETTING	EFFECT
Name	Name of the tree type.

SETTING	EFFECT
Cell	Name of a MicroStation cell that is drawn into the CAD file when trees are detected.
RPC cell	Name and location of a RPC cell file that is used in rendered views to replace detected trees.
Min height	Minimum height of a tree.
Max height	Maximum height of a tree.
Top is	Distance between highest hit on the tree in laser points and the tree top of the tree cells.
Width variation	Variation of tree crown width for the tree type. Given in percent.

Tree types are stored in a configuration file TREE_TYPES.INF in the TerraScan installation folder. You can copy this file to other computers in order to make tree type definitions available on them.

Undo and backup

Undo and backup category defines how much memory the application allocates for its undo buffer. The setting effects the amount of steps that can be undone with Undo or From list commands. Further, the category lets you define settings for saving an automatic backup copy of loaded points.

SETTING	EFFECT
Buffer	Amount of memory allocated for undo actions. The recommended value range is 16 - 64 MB.
Save loaded points	If on, TerraScan saves a backup copy of loaded points automatically in regular time intervals as long as the points are modified.
Every	Time interval for saving loaded points automatically.
Folder	Storage directory for the backup copy of the loaded data.

User vegetation indexes

User vegetation indexes category shows a list of definitions for vegetation that can be computed from color channels.

TerraScan implements the two most common vegetation index formulas: the Normalized Difference Vegetation Index and the Visual Band Vegetation Index. These indexes assume the channel order Red Green Blue (+ Near-Infrared). If color values are assigned to a point cloud using this common channel

order, no additional settings are required and the implemented display option and classification tool can be used.

The **User vegetation indexes** category lets you define additional computation rules for vegetation. This may be necessary, if the order of color channels is different, or if other channels than RGB and NIR are involved. The settings dialog provides a number of pre-defined formulas for channel computation. The user can choose from these formulas and define the channel assignment. The user vegetation indexes are then also available for displaying points by vegetation index and for classifying loaded points.

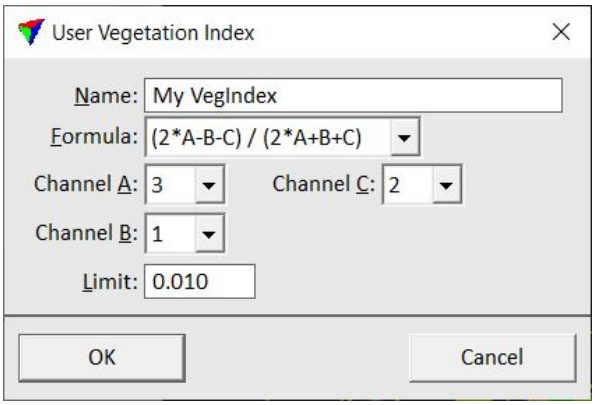
You can **Add**, **Edit**, and **Delete** vegetation indexes by using the corresponding buttons in the **Settings** dialog.

To add a new target object:

1. Open the **User vegetation indexes** category.

2. Click **Add** in the **Settings** dialog.

   The **User Vegetation Index** dialog opens:



3. Define settings and click OK.

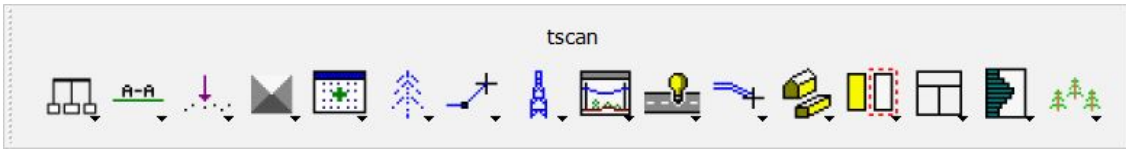
4. Close the **Settings** dialog in order to save the vegetation index settings for TerraScan.

SETTING	EFFECT
Name	Name of the vegetation index.
Formula	List of pre-defined formulas for channel computation.
Channel A | B | C	Number of the color channel used in the selected **Formula**.
Limit	Threshold value for displaying points by Vegetation index using the selected **Formula**.

User vegetation indexes are stored in a configuration file VEGETATION_INDEXES.INF in the TerraScan installation folder. You can copy this file to other computers in order to make vegetation index definitions available on them.

TerraScan Toolbox

The **TerraScan toolbox** is opened when TerraScan is loaded and if the **Open Main tool box** option in the Operation category in TerraScan **Settings** is switched on.



The toolbox can be re-sized by dragging its lower right corner with the mouse pointer *(MicroStation only)*. It can be docked to the edges of the CAD platform user interface.

There are 16 toolbars included in the **TerraScan** toolbox. A toolbar can be displayed by keeping the data mouse button pressed for 1-2 seconds on a tool icon in the **TerraScan** toolbox. The toolbar pops up and any tool can be selected with a data click. A toolbar can also be extracted from the toolbox by selecting **Open as ToolBox** (MicroStation) or **As Toolbar** (Spatix) command from the toolbar pop-up or by selecting the toolbar name from the Toolboxes submenu in the **Tools** pulldown menu.

If the **TerraScan** toolbox is accidentally closed, it can be re-opened with the **Main tool box** command from the Toolboxes submenu in the **Tools** pulldown menu or with the key-in command:

scan app main

3D Building Models

TerraScan provides a set of tools for building vectorization based on airborne point cloud (ALS or photogrammetric) data. The 3D vector models are created fully-automatically but for higher accuracy, they can be modified manually with dedicated tools. These tools ensure that the topology of a building model is preserved and allow fast and easy editing. In addition, TerraScan may be used for creating or updating building models stored in 3D City Databases.

According to the common way for describing building models, the models of TerraScan are at level-of-detail (LOD) 2. In LOD 2, roof shapes and the overall structure of roofs are represented but walls are just plain vertical polygons.

The automatic vectorization is based on classified points of the ground and on building roofs. Building footprints can be used in the vectorization process for placing walls or roof edges. Image data loaded in TerraPhoto supports the automatic vectorization of buildings. For manual editing, images in camera views improve the result essentially, because edges of roofs, roof structures, and smaller details may not be detectable accurately in the point cloud data.

The automatic building vectorization runs on loaded laser points using the Vectorize Buildings tool or for a TerraScan project using the Vectorize buildings macro action.

The workflow for automatic building vectorization can be outlined as follows:

1. Classify ground points using the Ground classification routine.

2. Classify high points which may be hits on building roofs using the By distance classification routine. This classification also includes points from high vegetation and other high objects.

3. Classify points on building roofs using the Buildings classification routine.

4. (Optional) If images are available, load a mission and an image list into TerraPhoto. The camera parameters of the mission and the image list should be adjusted in order to provide accurately positioned images.

6. Create vector models of buildings using the Vectorize Buildings tool for loaded points or run a macro including the Vectorize buildings macro action on a TerraScan project.

7. Review and improve building models with the help of the Check Building Models tool and tools in the Building Patches toolbox and the Building Edges toolbox.

The quality of the automatic building vectorization depends on the quality of the laser data processing that is done in preparation of the vectorization, but also on the point density of the data. A higher point density results in more accurate models. The following number may serve as a guideline for estimating the possible results of the automatic vectorization:

- Low density < 2 points / m² - good models of large buildings, more problems with small buildings, loss of small details and roof structures
- Medium density 2-10 points / m² - good models
- High density > 10 points / m² - accurate models with details and roof structures

As alternative to point cloud data, TerraScan can also utilize line elements for the creation of 3D building models. The line elements must represent different types of roof edges, such as outer edges, internal edges along elevation jumps, and intersection lines, and they must form a closed line work for each building. From the line network, the Construct Roof Polygons tool tries to create closed polygons which represent roof planes. Finally, the Create Buildings from Polygons can be used to create the 3D vector models from the roof polygons.

The major advantage of this building vectorization approach is the automatic production of 3D building models for large areas in a comparatively short time. The process can also model complex roofs that are non-planar and contain a lot of detailed roof patches. The tools for improving the result of the automatic process are versatile and make the manual work fast and simple.

A disadvantage of the vectorization process is that it fully relies on the quality of the source data, which is either laser data or a line work for building roofs. If, for example, laser data is missing on parts of a building roof, there is no way to create at least an approximate building model based on the represented roof parts.

3D City Database

TerraScan also provides tools for writing, reading, updating and deleting building models stored in 3D City Databases. The 3D City Database is a free geo database to store, represent, and manage virtual 3D city models on top of a standard spatial relational database. The database schema implements the CityGML standard for storing geometry and semantic information of urban objects. More information, instructions for 3DCityDB setup and examples are available on the official internet pages of 3D City Database.

You may update building models stored in a 3DCityDB or write new building models in a database. A typical update workflow can be outlined as follows:

1. Read building models stored in a 3DCityDB.

2. Review and improve building models with the help of the Check Building Models tool and tools in the Building Patches toolbox and the Building Edges toolbox.

3. If necessary, delete models of demolished buildings from the database.

4. If necessary, create models for new buildings using the Vectorize Buildings tool for loaded points or run a macro including the Vectorize buildings macro action on a TerraScan project.

5. Write modified models to the database using the method **Replace by Id** or **Replace by centroid**.

6. Write new models to the database using the method **Add to database**.

In this scenario, the CAD file acts only as temporary working environment. The building models are only stored in the database.

Buildings toolbox

The tools in the **Buildings** toolbox are used to create 3D building models automatically from point clouds, roof lines or roof polygons, and to check these building models.

TO	USE TOOL	
Vectorize buildings from point cloud		Vectorize Buildings
Construct polygons from roof lines		Construct Roof Polygons
Create building models from roof polygons		Create Buildings from Polygons
Merge small polygons to larger footprint polygons		Merge Footprint Polygons
Draw relevant roof lines into the CAD file		Draw Roof Lines
Check building models one at a time		Check Building Models
Write buildings to a 3D City database		Write Buildings to Database

TO	USE TOOL	
Read buildings from a 3D City database		Read Buildings from Database
Delete buildings from a 3D City database		Delete Database Buildings

At the moment, cells and thus, the tools of the **Buildings** toolbox do only work in MicroStation. There is not yet any corresponding element type in Spatix.

Check Building Models

Not Spatix

Check Building Models tool can be used to check automatically created building models in an organized way. It is intended to be used after building vectorization by Vectorize Buildings tool, Vectorize Buildings macro action, or Create Buildings from Polygons tool. The tool includes a validation option for checking the planarity of roof planes and/or the match between vector models and building footprint polygons.

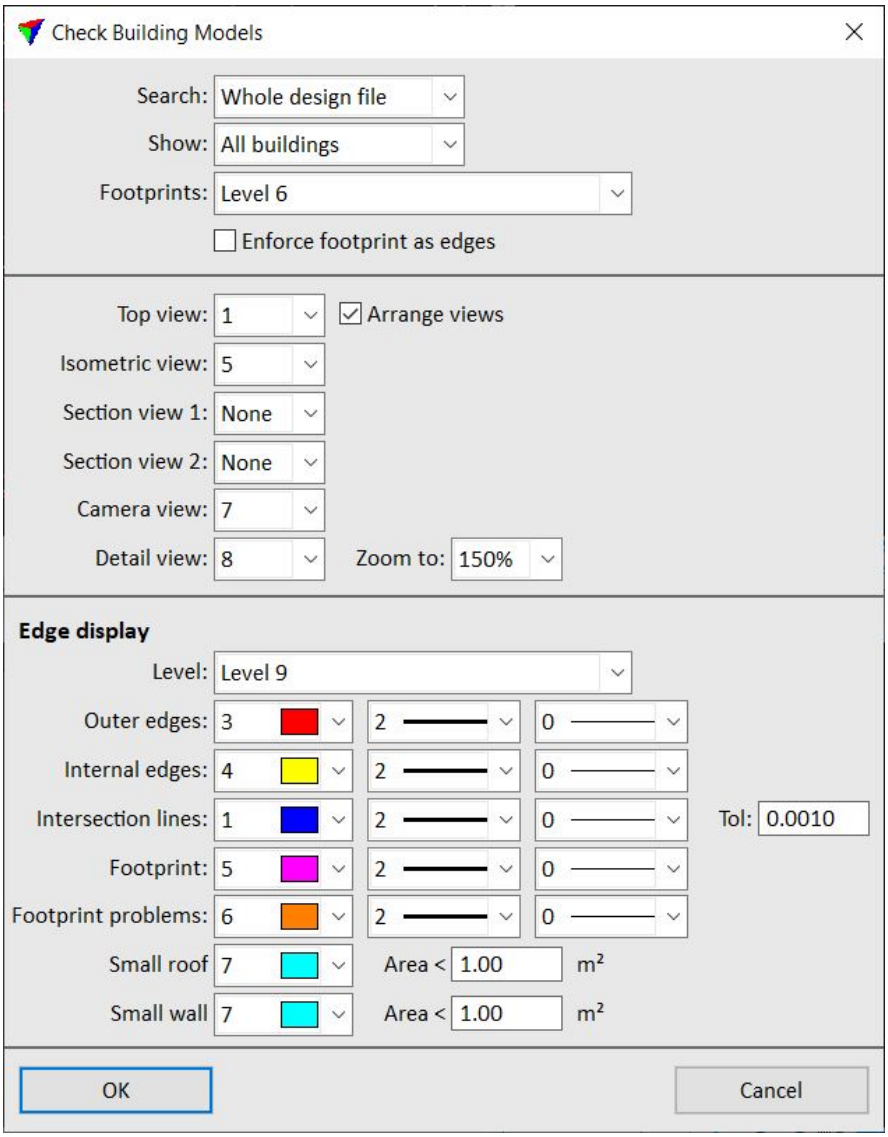
The **Check Building Models** dialog shows a list that contains building models in the CAD file that are drawn on the levels defined in Building vectorization / Levels category of TerraScan **Settings**. If a line in the list is selected, the software updates the display in a number of CAD file views as defined in the tool settings. The tool can update different view types, such as top, isometric, section, and camera views which may show the building models within a point cloud and/or on top of images.

With the help of the list, you can start to work with tools of the Building Patches toolbox and the Building Edges toolbox in order to improve the accuracy of the building models.

To check building models:

1. Select **Check Building Models** tool.

   This opens the **Check Building Models** dialog:



2. Define settings and click OK.

   This opens the Check Building Models dialog.

SETTING	EFFECT
Search	Area where the software searches for building models: • **Whole design file** - all models in the CAD file. • **Active block** - all models inside the active project block. This is defined by the TerraScan project block that is loaded or was last loaded into TerraScan.
Show	Building models shown in the list of the **Check Building Models** dialog: • **All buildings** - all building models on **Model to check** levels defined in Building vectorization / Levels.

SETTING	EFFECT
	• **Unchecked buildings** - building models on **Model to check** levels defined in Building vectorization / Levels with the status *Need to check* or any "problem" status. • **Problem buildings** - building models on **Model to check** levels defined in Building vectorization / Levels with any "problem" status. See Validate models for more information.
Footprints	CAD file level on which the footprint polygons are drawn. This is required for detecting and highlighting **Footprint problems**.
Enforce footprint as edges	If on, the tool clips automatically all building model edges to the footprint polygons. You should switch this option on if model edges must match to footprint polygons.
Top view	A top view showing the active building model is displayed in the given view.
Isometric view	An isometric view showing the active building model. It is recommended to set the display style for this view to **Smooth rendering** in CAD file view controls.
Section view 1	A section view showing the active building model. The section is drawn along the major direction of the building.
Section view 2	A section view showing the active building model. The section is drawn across the major direction of the building.
Camera view	A camera view showing the active building model is displayed in the given view. This view works only if a mission, camera, and image list are loaded into TerraPhoto.
Detail view	A camera view showing the location of an active building model in detail is displayed in the given view. The zoom level is determined by the given **Zoom to** value. This view works only if a mission, camera, and image list are loaded into TerraPhoto and if you select a building edge or corner for modification.
Arrange views	If on, the CAD file views are arranged on the screen according to the given view settings. The software opens the views and places them within the CAD platform interface without overlap.

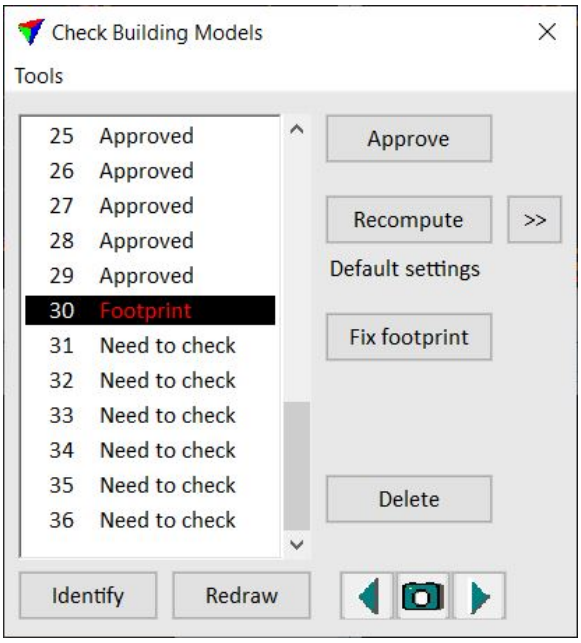
SETTING	EFFECT
Level	CAD file level on which the colored edges of an active model are drawn. The level should be switched on in the views that display the active model.
Outer edges	Display color, line weight, and line style of outer edges of the active model. Uses the active color table and standard line weights and styles of the CAD file.
Internal edges	Display color, line weight, and line style of internal edges of the active model. Uses the active color table and standard line weights and styles of the CAD file.
Intersection lines	Display color, line weight, and line style of intersection lines of the active model. Uses the active color table and standard line weights and styles of the CAD file. The **Tol**erance value determines up to which maximum positional difference vertices of intersecting planes are considered to be at exactly the same location.
Footprint	Display color, line weight, and line style of lines of the footprint polygon. Uses the active color table and standard line weights and styles of the CAD file.
Footprint problems	Display color, line weight, and line style of lines of the active model that do not match to footprint polygons. This highlights problems if a vector model edge is inside the footprint polygon drawn on the **Footprints** level. Uses the active color table and standard line weights and styles of the CAD file.
Small roof	Display color of roof planes smaller than the given **Area** value. The color is visible if the **Fill** attribute is switched on in the **View Attributes** dialog of a CAD file view.
Small wall	Display color of wall planes smaller than the given **Area** value. The color is visible if the **Fill** attribute is switched on in the **View Attributes** dialog of a CAD file view.

Check Building Models

The **Check Building Models** dialog shows a list that contains all building models drawn on the CAD file levels that are defined in Building vectorization / Levels category of TerraScan **Settings**. The status of each model after automatic creation is *Need to check* by default. In addition, the status of a model can indicate a problem. Then, the status is shown in red color. A "problem" status can be

related to footprints, such as mismatch between model edges and footprint boundaries, or to non-planar roof patches.

Further, the dialog contains buttons that can be used to change the status of a model to *Approved*, to recompute or delete a model, to fix footprint or planarity issues, to display a certain model, and to select another image that is displayed in the background of the active model. Menu commands can be used to display the Vectorize Buildings dialog, to validate models, and to change the status of all models to *Approved*.



To show the location of a building model, select a line in the **Check Building Models** dialog. This updates the display in all views that are set up for checking building models.

COMMAND/BUTTON	EFFECT
Approve	After checking a building model and possibly modifying it, click on the **Approve** button. This changes the status of the selected model to *Approved*. It moves the model to the **Approved models** levels defined in Building vectorization / Levels category of TerraScan **Settings**.
Recompute	Recomputes a building model based on loaded points and tailored settings. This might be necessary, if the settings of the automatic vectorization process did not provide a reasonable model for this building.
Fix footprint	Fix small mismatch between roof edges and a footprint polygon. This is only available if models have been validated using the **Footprint mismatch** option.

COMMAND/BUTTON	EFFECT
Fix planarity	Fixes planarity issues of roof patches. Exactly planar roof patches are enforced. This also requires that **Average elevations** is set to 0.0 in Building vectorization / Models category of TerraScan **Settings**. Exact planarity may lead to small differences between vertices of intersection lines. This is only available if models have been validated using the **Non-planar roof** option.
Delete	Deletes a model from the CAD file and from the list. If you undo the delete action by using the **Undo** command of the CAD platform, the model is returned into the CAD file but not into the list of the dialog. You need to re-open the dialog in order to see the model again in the list.
Identify	To identify a building model, click on the **Identify** button and place a data click close to a model in a view. This selects the corresponding model in the **Check Building Models** dialog.
	You can use the buttons in the lower right corner in order to select images from the TerraPhoto image list. The image is displayed in the camera views used for checking building models. By default, the software selects the image for display that sees the building (detail) location best. Click on the camera button in the middle of the button group in order to identify an image for display. Move the mouse pointer into a view. The image footprint closest to the mouse pointer is dynamically displayed. Select an image for display with a data click. Click on the arrow buttons left and right in the button group in order to select the previous or next image from the currently displayed image in the images list.
Tools / Computation settings	Opens the Vectorize Buildings dialog with the settings used in the automatic vectorization process. This might be useful to check before recomputing a model for which the settings do not apply sufficiently.
Tools / Validate models	Checks building models for footprint or planarity problems.
Tools / Approve all	Changes the status of all models in the list to *Approved*. It moves the models to the **Approved models** levels defined in Building

COMMAND/BUTTON	EFFECT
	vectorization / Levels category of TerraScan **Settings**.

To recompute a building model:

1. Load laser data into TerraScan for the location of the building model.
2. Click on the >> button in order to open the **Vectorize Buildings** dialog.

   The settings of the dialog are described for the Vectorize Buildings tool.
3. Change settings in the dialog and click OK.
4. Click on the **Recompute** button.

   This recomputes the selected building model by using the tailored settings.

Validate models

Validate models command checks the vector models in the list for footprint and/or planarity issues. The check is useful to find models that do not match to footprint polygons or that contain non-planar building patches.

The validation process may change the status of vector models to a "problem" status:

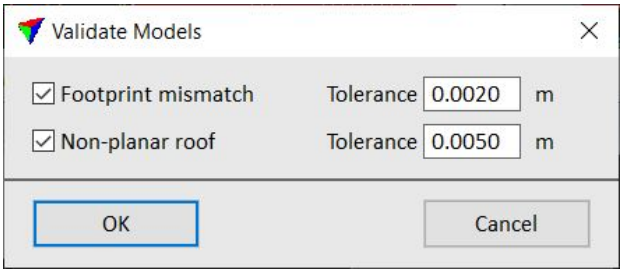
- *Footprint* - the model edges do not match to the footprint boundaries.
- *No footprint* - no footprint is available for validation.
- *Non-planar* - one or more patches of the model are not planar. Exactly planar roof patches can only be created if **Average elevations** is set to 0.0 in Building vectorization / Models category of TerraScan **Settings**. Otherwise, patches are non-planar which is fine for most of the building model applications.

A "problem" status is displayed in red in the list of building models.

To validate models:

1. Select **Validate models** command from the **Tools** pulldown menu.

   This opens the **Validate Models** dialog:



2. Select settings and click OK.

This starts the validation process. An information dialog shows the amount of detected problems. The status for models is changed in the list if a problem is detected.

SETTING	EFFECT
Footprint mismatch	If on, the process checks mismatch between footprint polygons and vector model edges. A problem is detected if the model vertex differs more than the given **Tolerance** value from the footprint vertex.
Non-planar roof	If on, the process checks the planarity of roof patches. A problem is detected if the patch planarity differs more than the given **Tolerance** value from a perfect plane.

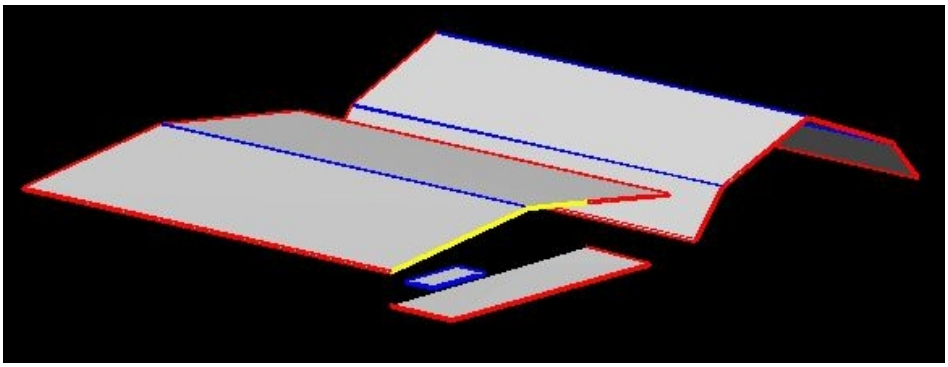
It is virtually impossible to create roof patches with exact planarity, perfect geometry and even shapes. Therefore, enforce exact planarity for roof patches only if it is explicitly required. Otherwise, it may be more convenient to create building models with correct outer boundaries and intersection lines for roof patches and at the same time, accepting non-planar patches.

Construct Roof Polygons

Construct Roof Polygons tool creates 3D roof polygons from selected 3D roof lines. It tries to determine closed polygons for each roof plane from the line work. The resulting polygons can then be used by Create Buildings from Polygons tool in order to create 3D building models.

The line elements must represent different types of roof edges, such as outer edges, internal edges along elevation jumps, and intersection lines, and they must form a closed line work for each building. The tool does not rely on lines being drawn on different levels or using different symbology. It tries to determine which elevations to keep and which to ignore in the polygon-building process only from the geometrical configuration of the line work.

The following figure illustrates the result of the roof polygon construction. Lines are roughly colored according to their roof edge type: outer edges = red, elevation jumps = yellow, intersection lines = blue. The resulting roof polygons are displayed in gray.



To construct roof polygons from lines:

1. Select the lines that represent roof edges.

2. Select **Construct Roof Polygons** tool.

   This creates the polygons. The polygons are drawn on the active level using the active symbology settings of the CAD file. An information dialog shows the number of created polygons.

   It is recommended to check the polygons, for example, by using Smooth rendering display of the CAD platform. This shows gaps or other issues in the roof polygons which may be caused by flaws in the line work. In this case, correct the line work and run the roof polygon construction again.

You can undo the creation of roof polygons by using the **Undo** command of the CAD platform.

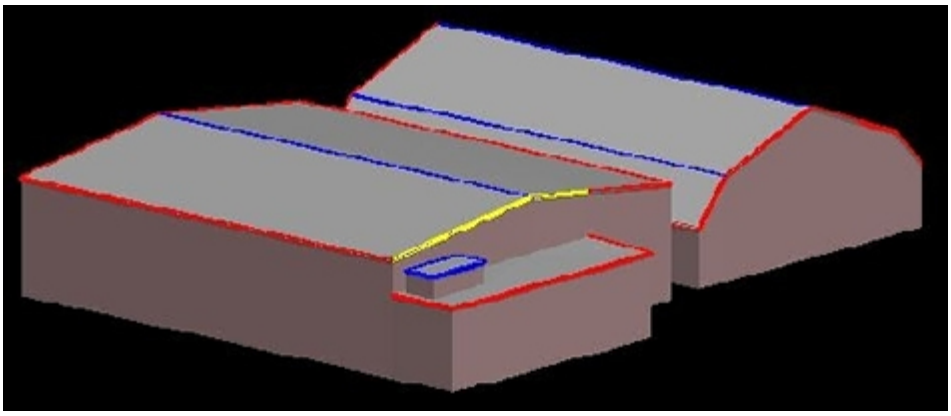
Create Buildings from Polygons

Not Spatix

Create Buildings from Polygons tool creates 3D building models from selected 3D roof polygons. The polygons are usually a result of the Construct Roof Polygons tool.

The tools uses laser points to get the base elevation for walls. The building models are created as MicroStation cell elements that contain shapes for each roof plane, possibly roof sides which determine the roof's thickness according to settings in Building vectorization / Model, and wall shapes for each outer roof edge.

The following figure illustrates the result of the building model creation using the same example as shown for the Construct Roof Polygons tool.

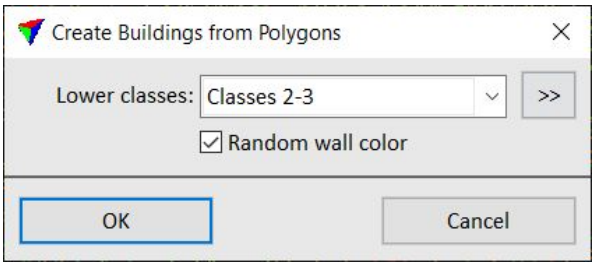


To construct building models from polygons:

1. Select the polygons that represent roof planes.

2. Select **Create Buildings from Polygons** tool.

   This opens the **Create Building from Polygons** dialog:



3. Define settings and click OK.

   This creates the building models. The building models are drawn into the CAD file according to the settings in Building vectorization / Levels category of TerraScan **Settings**.

SETTING	EFFECT
Lower classes	Point class(es) consisting points next to buildings, for example, ground or low vegetation. The points are used to get the base elevation of the building walls.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Lower classes** field.
Random wall color	If on, wall shapes are colored randomly by using a selection of colors from the active color table of the CAD file. If off, the color defined in Building vectorization / Model category of TerraScan **Settings** is used for all wall shapes.

You can undo the creation of building models by using the **Undo** command of the CAD platform.

Delete Database Buildings

Not Spatix

Delete Database Buildings tool deletes building models from a 3D City Database (3DCityDB). This requires the connection of TerraScan with the 3DCityDB. The connection parameters are defined in the City databases category of TerraScan **Settings**.

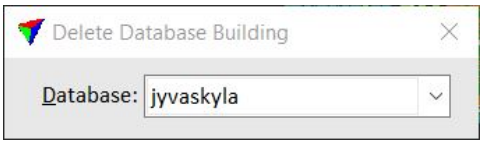
You may use the tool for deleting demolished buildings from a 3DCityDB. The building models must be read from the database using the Read Buildings from Database tool. This ensures that the software identifies a building model by its unique ID stored in the database.

To delete buildings from a 3DCityDB:

1. Select the building models you want to delete.

2. Select **Delete Database Buildings** tool.

   This opens the **Delete Database Buildings** dialog:



3. Select the **Database** from the selection list.

4. Confirm the process with another data click inside a CAD view.

   This deletes the selected building models from the database based on the model ID. It also removes the models from the CAD file.

Draw Roof Lines

Not Spatix

Draw Roof Lines tool draws relevant roof lines of selected building models as permanent elements in the CAD file. This might be useful for checking the quality of building vector models without modifying the models at the same time.

To draw roof lines in the CAD file:

1. Select the building models for which to draw the roof lines.

2. Select **Draw Roof Lines** tool.

   This opens the **Draw Roof Lines** dialog:

3. Define settings and click OK.

   This draws the line elements on the active level of the CAD file.

SETTING	EFFECT
Footprints	CAD file level on which the footprint polygons are drawn. This is required for detecting and

SETTING	EFFECT
	drawing **Footprint problems**.
Outer edges	Display color, line weight, and line style of outer edges of the model. Uses the active color table and standard line weights and styles of the CAD file.
Internal edges	Display color, line weight, and line style of internal edges of the model. Uses the active color table and standard line weights and styles of the CAD file.
Intersection lines	Display color, line weight, and line style of intersection lines of the model. Uses the active color table and standard line weights and styles of the CAD file. The **Tol**erance value determines up to which maximum positional difference vertices of intersecting planes are considered to be at exactly the same location.
Footprint problems	Display color, line weight, and line style of roof lines of the model that do not match to footprint polygons. This highlights problems if a vector model edge is inside the footprint polygon drawn on the **Footprints** level. Uses the active color table and standard line weights and styles of the CAD file.

You can undo the drawing of roof lines by using the **Undo** command of the CAD platform.

Merge Footprint Polygons

Merge Footprint Polygons tool can be used to merge several polygons that form a single building footprint into one polygon. A typical example is that the main building and balconies or other small building extensions are outlined by separate polygons. In order to create a 3D vector model of the building including these extensions, there must be one single footprint polygon.

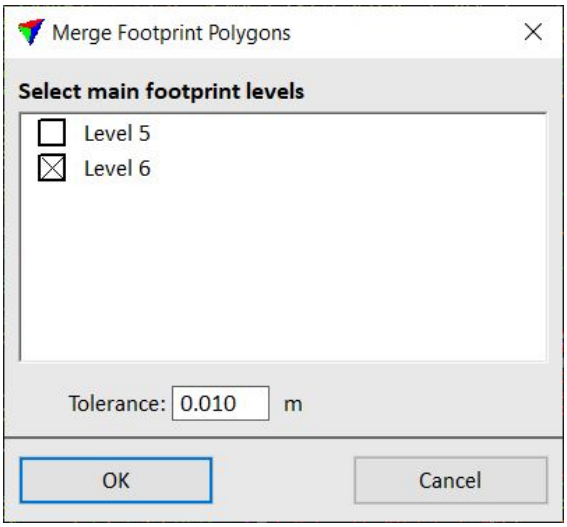
The polygons to be merged must share at least one edge. The larger polygon is considered as the 'main footprint'. Smaller polygons are merged to the main footprint. The polygons may be drawn on different CAD file levels but the main footprints for all buildings should be on one level.

To merge footprint polygons:

1. Select the footprint polygons. Polygons of several buildings may be selected.

2. Select **Merge Footprint Polygons** tool.

   This opens the **Merge Footprint Polygons** dialog:



3. Define settings and click OK.

   This creates a new polygon for each building footprint by merging smaller polygons to the main footprint. The polygon is drawn on the active level using the active symbology settings of the CAD file.

SETTING	EFFECT
Select main footprint levels	Determines the CAD file level on which the main footprint polygons are drawn.
Tolerance	Maximum distance limit by which the shared edge(s) of the main footprint polygon and the extension polygons may differ in location. If the distance is smaller, the edge is still considered as shared and polygons are merged.

You can undo the merging of footprint polygons by using the **Undo** command of the CAD platform.

Read Buildings from Database

Not Spatix

Read Buildings from Database tool reads building models from a 3D City Database (3DCityDB). This requires the connection of TerraScan with the 3DCityDB. The connection parameters are defined in the City databases category of TerraScan **Settings**.

The 3D City Database is a free geo database to store, represent, and manage virtual 3D city models on top of a standard spatial relational database. The database schema implements the CityGML standard for storing geometry and semantic information of urban objects. More information, instructions for 3DCityDB setup and examples are available on the official internet pages of 3D City Database.

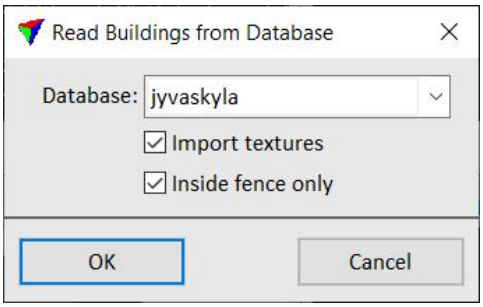
You may update building models stored in a 3DCityDB by reading the models, modify them based on a newer point cloud/image data set, and write them back to the database. In this scenario, the CAD file acts only as temporary working environment. The building models are only stored in the database.

To read buildings from a 3DCityDB:

1. (Optional) Define the area for reading building models by drawing a fence.

2. Select **Read Buildings from Database** tool.

   This opens the **Read Buildings from Database** dialog:



3. Define settings and click OK.

   This reads the building models from the database and draws them into the CAD file. The building models are drawn according to the settings for **Models to check** in Building vectorization / Levels category of TerraScan **Settings**.

SETTING	EFFECT
Database	Name of the 3DCityDB as defined in the City databases category of TerraScan **Settings**.
Import textures	If on, texture images are imported as well. If off, only the geometry of the models is imported.
Inside fence only	If on, only building models located inside a fence are imported. This is only active if a fence has been defined before the tool is started.

Vectorize Buildings

Not Lite, Not Spatix

Vectorize Buildings tool creates 3D building models based on loaded point cloud data. The point cloud has to be classified into:

- ground points using the Ground classification routine.
- above-ground points which may be hits on building roofs using Compute distance followed by the By distance classification routine. This classification also includes points from high vegetation and other high objects.
- points on building roofs using the Buildings classification routine.

The tool creates MicroStation cell elements that contain shapes for each roof plane, possibly roof sides which determine the roof's thickness according to settings in Building vectorization / Model, and wall shapes for each outer roof edge.

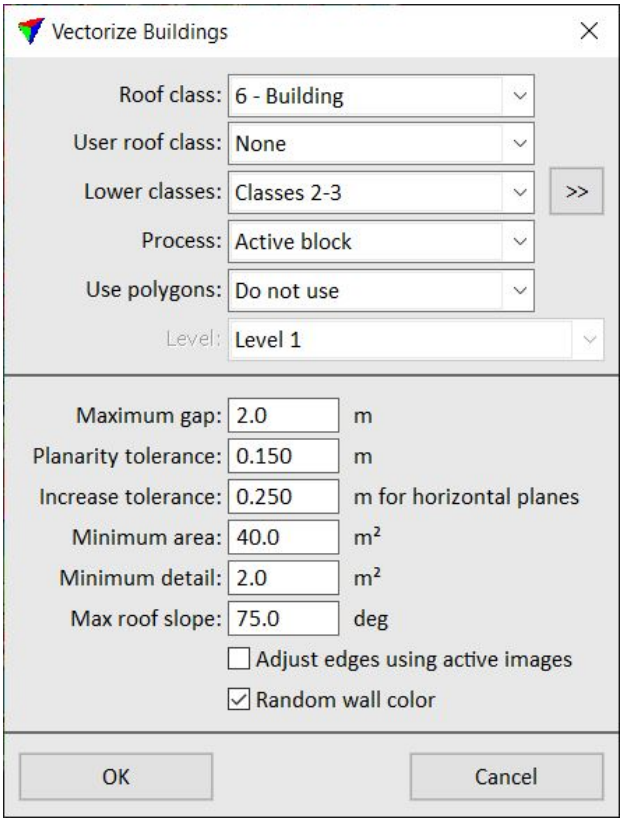
Building vectorization can be also performed on project level by using the Vectorize buildings macro action.

To vectorize buildings based on loaded points:

1. Load point cloud data into TerraScan.

2. Select the **Vectorize Buildings** tool.

   This opens the **Vectorize Buildings** dialog:



3. Define settings and click OK.

   This starts the vectorization process. It may take a while until the first models are created because the routine creates models for large buildings first. The building models are drawn into the CAD file according to the settings in Building vectorization / Levels category of TerraScan **Settings**.

SETTING	EFFECT
Roof class	Point class consisting points on building roofs.
User roof class	Point class consisting points on building roof details. The details are vectorized even if the area is smaller than the **Minimum detail** value.

SETTING	EFFECT
Lower classes	Point class(es) consisting points next to building roof edges, for example, ground or low vegetation. The points are used to determine the base elevation of building walls and help to place outer roof edges more accurately.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Lower classes** field.
Process	Area to be processed: • **All points** - all loaded points are processed. This may include points from neighbour blocks. • **Active block** - points of the active block are processed. • **Inside fence** - points inside a fence or selected polygon are processed. This is only active if a MicroStation fence is drawn or a polygon is selected.
Use polygons	Defines how polygons are used in addition to point cloud data: • **Do not use** - no polygons are used. • **As bounding polygons** - polygons define boundaries that divide large building blocks into separate models. Example: land property polygons. • **As roof edges** - polygons define the XY shape of outer edges of buildings. Example: footprint polygons.
Level	CAD file level on which the polygons are located that are used in the vectorization process. This is only active if **Use polygons** is set to **As bounding polygons** or **As roof edges**.
Maximum gap	Maximum distance between building parts belonging to the same model. If the distance is larger, separate building models are created.
Planarity tolerance	Defines how closely a point must match a plane equation to belong to that roof plane.
Increase tolerance	Additional tolerance for merging close to horizontal planes together.
Minimum area	Minimum size of a building footprint.
Minimum detail	Minimum size of a building part footprint.

SETTING	EFFECT
Max roof slope	Maximum gradient of a roof plane.
Adjust edges using active images	If on, building edges are adjusted based on images. The images must be referenced by an image list loaded into TerraPhoto.
Random wall color	If on, wall shapes are colored randomly by using a selection of colors from the active color table of the CAD file. If off, the color defined in Building vectorization / Model category of TerraScan **Settings** is used for all wall shapes.

Write Buildings to Database

Not Spatix

Write Buildings to Database tool writes building models to a 3D City Database (3DCityDB). This requires the connection of TerraScan with the 3DCityDB. The connection parameters are defined in the City databases category of TerraScan **Settings**.

The 3D City Database is a free geo database to store, represent, and manage virtual 3D city models on top of a standard spatial relational database. The database schema implements the CityGML standard for storing geometry and semantic information of urban objects. More information, instructions for 3DCityDB setup and examples are available on the official internet pages of 3D City Database.

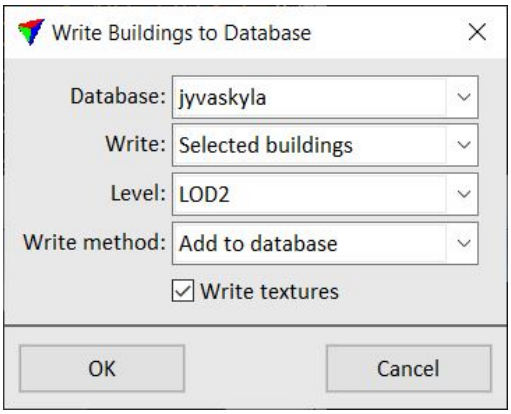
You may update building models stored in a 3DCityDB by reading the models, modify them based on a newer point cloud/image data set, and write them back to the database. In this scenario, the CAD file acts only as temporary working environment. The building models are only stored in the database.

To write buildings to a 3DCityDB:

1. (Optional) Define building models you want to write to the database either by selecting the models or by drawing a fence around the models.

2. Select **Write Buildings to Database** tool.

   This opens the **Write Buildings to Database** dialog:



3. Define settings and click OK.

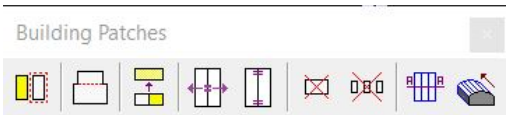
   This writes the building models to the database.

SETTING	EFFECT
Database	Name of the 3DCityDB as defined in the City databases category of TerraScan **Settings**.
Write	Determines what building models are written to the database: • **All buildings** - all building models in the CAD file. • **Selected buildings** - building models that have been selected before starting the tool. • **Inside fence** - buildings models that are inside a fence drawn before starting the tool.
Level	Level-of-detail of the building models. Refers to the LOD definition of the CityGML standard: • **LOD1** - building models without roof structure and plane walls. • **LOD2** - building models with roof structure and plane walls. This is the LOD of building models produced by TerraScan vectorization tools from aerial point cloud data.
Write method	Method of writing building models to the database: • **Add to database** - new models are written to the database. Use this if models do not yet exist in the database. • **Replace by Id** - a model replaces an existing model with identical ID. This can be used if existing models in the database are updated. The ID identifies each model in a unique way. • **Replace by centroid** - a model replaces an existing model with the same centroid coordinate. This may be used if the model ID is not usable for an update case.

SETTING	EFFECT
Write textures	If on, texture images are stored in the database as well. If off, only the geometry of the models is stored.

Building Patches toolbox

The tools in the **Building Patches** toolbox are used to modify roof patches of 3D building models. The term “patch” is used for the single roof planes that form a roof.



TO	USE TOOL	
Split building into two separate models		Split Building
Split patch into two separate patches		Split Patch
Merge two patches into one		Merge Patches
Enforce symmetry for planar roof patches		Apply Plane Symmetry
Enforce the same elevation for vertices of adjacent patches		Match Roof Elevations
Remove a patch by mouse click		Remove Patch
Remove small patches		Remove Details
Display a building cross section		Draw Building Section
Extrude a building model from a cross section		Extrude Building

Building Patches tools work only when the Check Building Models dialog is open. You can undo the actions of the tools by using the **Undo** command of the CAD platform.

At the moment, cells and thus, the tools of the **Building Patches** toolbox do only work in MicroStation. There is not yet any corresponding element type in Spatix.

Apply Plane Symmetry

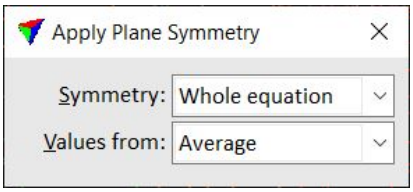
Not Spatix

Apply Plane Symmetry tool forces plane equations, normal directions and/or slope gradients of planar roof patches to match. The symmetry adjustment requires that at least two patches are selected. However, the tool is also applicable to more than two patches in one step.

To apply plane symmetry:

1. Select **Apply Plane Symmetry** tool.

   The **Apply Plane Symmetry** dialog opens:



2. Select settings.

3. Move the mouse pointer inside a view.

   A patch is highlighted if the mouse pointer is inside.

4. Define the first roof patch with a data click.

   If **Values from** is set to **First patch**, this patch determines the symmetry change for the other patches.

5. Define the next roof patch with a data click.

   Repeat step 5 for all patches that you want to include in the symmetry adjustment.

6. Confirm the patch selection with another data click inside one of the selected patches.

   This applies the symmetry to all selected patches.

SETTING	EFFECT
Symmetry	Determines the symmetry value that is enforced for all selected roof patches: • **Whole equation** - the plane equation is recomputed and selected patches are adjusted accordingly. Symmetric planes have the same plane equation. • **Direction & Slope** - normal direction and slope gradient are adjusted.

SETTING	EFFECT
	• **Direction** - the normal direction of a plane is adjusted. Symmetric planes have normal directions that differ by 90 degree. • **Slope** - the slope of a plane is adjusted. Symmetric planes have mirrored slope gradients.
Values from	Defines how the new values for roof patches are derived: • **Average** - the values of all selected patches are averaged. • **First patch** - the value of the first selected patch is used for all selected patches.

Draw Building Section

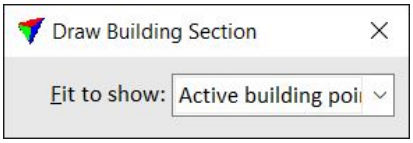
Not Spatix

Draw Building Section tool displays a cross section of a building. The tool is intended to be used, for example, before Extrude Building tool. It provides a cross section view of the laser data that is suited for drawing a building roof profile line. In addition, the tool can be helpful in cases there are no image available for modifying vector models manually. Then, it speeds up the drawing of sections for single building planes.

To draw a building section:

1. Load point cloud data into TerraScan.
2. (Optional) Open one or two additional CAD file view(s) that can be used for the section display.
3. (Optional) Draw a fence around the area from which you want to create a cross section.
4. Select **Draw Building Section** tool.

   The **Draw Building Section** dialog opens:



5. Select an option for the tool setting.
6. If **Fit to show** is set to **Single patch**, move the mouse pointer in the top view. If the mouse point is inside a patch, it is highlighted. Define the patch with a data click in the top view.
7. Define a view with a data click.

   This displays the building or patch section in the selected view.

8. If **Fit to show** is set to **Single patch**, define the second view with a data click.

   This displays the second patch section in the selected view.

SETTING	EFFECT
Fit to show	Defines the area that is used to select a location for the cross section: • **Active building points** - area covered by all points that are inside the active building. • **Inside fence** - area covered by a MicroStation fence. This requires a fence element drawn into the CAD file. • **Single patch** - area covered by a selected patch. The patch is selected with a separate data click. The tool draws one along-patch section and one across-patch section.

Extrude Building

Not Spatix

Extrude Building tool creates a building model by extruding a cross section line. The tool is especially useful for modeling buildings with round roofs. For such roof shapes, the automatic vectorization process usually does not provide a good result. The tool can also be used for adding or replacing planar roof patches of an existing model. This may be useful if a part of a roof can not be modeled from the point cloud due to the lack of points.

The cross section line of the building roof needs to be digitized manually based on a vertical section view of the laser data. The line string element should be placed on a CAD file level that is not used for building models. It can also be deleted after the building model has been created.

To create a building model from a cross section line:

1. Remove all patches that you want to replace from the existing model by using Remove Patch or Remove Details tools.

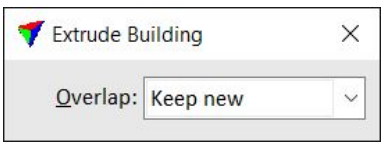
2. Create a cross section view of the building by using the Draw Building Section tool.

   It is recommended to display the section in an additional CAD file view.

3. Digitize the shape of the cross section based on the laser data that is displayed in the section view. You can use any CAD tool for line string placement.

4. Select **Extrude Building** tool.

   This opens the **Extrude Building** dialog:



5. Select an option for handling overlap areas.

6. Select the cross section line with a data click.

7. Define the first edge of the building with a data click, preferable in a top view.

8. Define the second edge of the building with a data click, preferable in a top view.

   This creates a building model between the two edges defined by the data clicks. For each intermediate vertex of the cross section line, an intersection line is created in the building model. The outer boundaries of the new model are defined by the first and last vertex of the cross section line.

SETTING	EFFECT
Overlap	Defines which roof patch is used in overlapping roof patch areas: • **Keep new** - replaces old patches with new ones. • **Keep old** - keeps the old patches and adds new patches only in areas outside the old ones.

Match Roof Elevations

Not Spatix

Match Roof Elevations tool modifies the plane equation in order to set the elevation of roof plane vertices to the elevation of a neighbouring patch.

To match roof elevations:

1. Select **Match Roof Elevations** tool.

2. Move the mouse pointer inside a view.

   A patch is highlighted if the mouse pointer is inside.

3. Define the roof patch to modify with a data click.

   The vertex closest to the mouse pointer is highlighted if the tool is applicable.

4. Define the first vertex to adjust with a data click.

   Repeat step 4 for all vertices that you want to adjust in elevation.

5. Confirm the vertex selection with another data click on the last selected vertex.

   This applies the elevation of all selected vertices to the elevation of the neighbouring patch.

Merge Patches

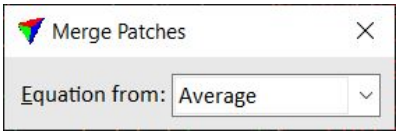
Not Spatix

Merge Patches tool combines two neighbouring patches into one patch. The process recomputes the plane equation for the new patch as average of the two original planes.

To merge two patches:

1. Select **Merge Patches** tool.

   This opens the Merge Patches dialog:



2. Define settings.

3. Move the mouse pointer inside a view.

   Potential patches for merging are dynamically highlighted if the mouse pointer is inside a patch.

4. Select the first patch with a data click.

   If you move the mouse pointer, the possible patches for merging are dynamically displayed.

5. Select the second patch with a data click.

   This merges the two patches into one patch and recomputes the plane equation of the new patch. You can continue with step 3.

SETTING	EFFECT
Equation from	Method of how the plane equation for the merged patch is defined: • **First patch** - the equation of the first selected patch is used. • **Average** - the average of the two patches is computed.

Remove Details

Not Spatix

Remove Details tool removes all patches of a building roof which are of the same size or smaller than a patch identified by a data click or a specified value. This can be used, for example, for removing patches of unnecessary structures on a roof, such as roof windows.

To remove a building patch:

1. Select **Remove Details** tool.

   The **Remove Details** dialog opens:

2. Define settings.

3. Move the mouse pointer inside a view.

   The patches effected by the removal action are dynamically highlighted if the mouse pointer is inside a patch (**Limit by** = **Identify patch**) or inside the view (**Limit by** = **Keyin value**).

4. Confirm the selection of patches with a data click.

   This removes all patches from the building model that are defined by the tool's setting. You can continue with step 4.

SETTING	EFFECT
Remove	Defines what patches are effected by the removal action: • **All small patches** - all patches that are of the same size or smaller than the patch selected by the data click. • **Internal patches** - only patches that are completely inside a building roof are effected. The same size rules as for All small patches apply. • **Outer patches** - only patches that share an outer boundary of the building roof are effected. The same size rules as for All small patches apply.
Limit by	Method of how the maximum size of a patch to be deleted is determined:

SETTING	EFFECT
	• **Identify patch** - identified with a data click inside the patch. • **Keyin value** - the size is determined by the given **Area** value.

You can remove all patches of an active building model. Nevertheless, the model still exists in the list and stays active. You can apply additional processing steps, such as recomputing the model using the **Recompute** button of the Check Building Models dialog or creating a new model with the help of the Extrude Building tool. If you want to delete a model completely, use the **Delete** button of the Check Building Models dialog.

Remove Patch

Not Spatix

Remove Patch tool removes a single building patch.

To remove a building patch:

1. Select **Remove Patch** tool.
2. Move the mouse pointer inside a view.

   A patch is dynamically highlighted if the mouse pointer is inside a patch.
3. Select a patch with a data click.

   This removes the patch from the building model. You can continue with step 3.

You can remove all patches of an active building model. Nevertheless, the model still exists in the list and stays active. You can apply additional processing steps, such as recomputing the model using the **Recompute** button of the Check Building Models dialog or creating a new model with the help of the Extrude Building tool. If you want to delete a model completely, use the **Delete** button of the Check Building Models dialog.

Split Building

Not Spatix

Split Building tool cuts out a part of a building complex. This part is then treated as own separate building model.

To split a building into two building models:

1. Draw a fence around the building part that you want to cut out.
2. Select **Split Building** tool.

OR

1. Select **Split Building** tool.

2. Draw a line by placing the two end points with data clicks. The line represents the splitting line for the building model.

3. Move the mouse pointer inside a CAD file view.

   The two building parts are highlighted with blue and red coloring.

4. Accept the two building parts with a data click inside a view.

   This splits the building. The area outside the fence or on one side of the line stays as active building. The area completely inside the fence or on the other side of the line becomes a new building model that is put at the end of the list in the Check Building Models dialog.

Split Patch

Not Spatix

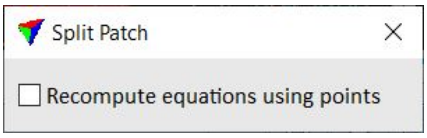
Split Patch tool splits a patch at edge vertices into two separate patches. The process can recompute the plane equations for the two patches if laser points of the roof class are loaded in TerraScan.

To split a patch:

1. (Optional) Load laser data into TerraScan. Only points in the building roof class are required.

2. Select **Split Patch** tool.

   The **Split Patch** dialog opens:



3. Define, whether the process should **Recompute** plane **equations using points** or not. The setting is only available if points are loaded in TerraScan.

4. Move the mouse pointer inside a view.

   A potential vertex for splitting is dynamically highlighted if the mouse pointer comes close to it.

5. Select the first edge vertex with a data click.

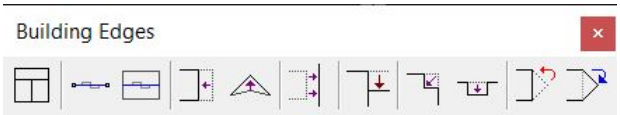
   If you move the mouse pointer, the possible split lines are dynamically displayed.

6. Select the second edge vertex with a data click.

   This splits the patch into two patches and recomputes the plane equation of the patches, if applicable. You can continue with step 5.

Building Edges toolbox

The tools in the **Building Edges** toolbox are used to modify roof edges and corners of 3D building models.



TO	USE TOOL	
Set all edges of a building model to rectangular		Set All Edges
Apply a straight line between two vertices		Apply Straight Line
Apply an intersection line of two patches		Apply Intersection Line
Move an edge of a building		Modify Edge
Move an edge vertex		Move Edge Vertex
Align an edge segment perpendicular or parallel to another edge		Align Edge Segment
Create a step corner		Build Step Corner
Cut an edge corner		Cut Edge Corner
Cut an edge segment		Cut Edge Segment
Delete a vertex from an edge		Delete Edge Vertex
Add a new vertex to an edge		Insert Edge Vertex

Building Edges tools work only when the Check Building Models dialog is open. You can undo the actions of the tools by using the **Undo** command of the CAD platform.

At the moment, cells and thus, the tools of the **Building Edges** toolbox do only work in MicroStation. There is not yet any corresponding element type in Spatix.

Align Edge Segment

Not Spatix

Align Edge Segment tool moves an edge segment. At the same time, it aligns the edge segment according to a base direction defined by a reference edge segment. The alignment is either parallel or perpendicular to the reference edge segment.

To align an edge segment:

1. Select **Align Edge Segment** tool.
2. Move the mouse pointer inside a view.

   The edge segment closest to the mouse pointer is dynamically highlighted.
3. Define the reference edge segment with a data click. This defines the base direction.
4. Define the edge segment to align with a data click.

   This updates the **Detail view** and displays the image that sees the selected edge segment location best. If you move the mouse pointer, the new edge location is dynamically displayed.
5. Define the new location of the edge segment with a data click.

   This aligns and places the edge segment at the new location. You can continue with step 4. After placing a reset click, you continue with step 3.

Apply Intersection Line

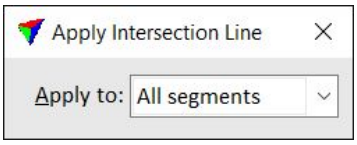
Not Spatix

Apply Intersection Line tool replaces edge segment(s) between two planar patches with an intersection line of two planes. This may move the original edge segment(s) and vertices to another location in order to match the exact intersection of the planes. Unnecessary intermediate vertices are removed from the resulting intersection line.

To apply an intersection line:

1. Select **Apply Intersection Line** tool.

   The **Apply Intersection Line** dialog opens:



2. Define whether all segments or only one segment is effected by the tool.
3. Move the mouse pointer inside a view.

The location of the intersection line between two patches closest to the mouse pointer is dynamically highlighted.

4. Accept the intersection line with a data click.

   This sets the intersection line, adjusts vertices if necessary, and deletes unnecessary intermediate vertices along the intersection line. You can continue with step 2 or 4.

SETTING	EFFECT
Apply to	Defines the edge segments effected by the tool: • **All segments** - all edge segments are replaced by the intersection line. • **One segment** - only one edge segment is replaced by the intersection line.

The tool adjusts vertices in order to apply an intersection line between two patches. If several intersection lines are connected in one vertex, it might be necessary to apply the tool several times to the edges. Then, the location of the vertices is more and more refined until intersection lines can be applied to all edges.

Apply Straight Line

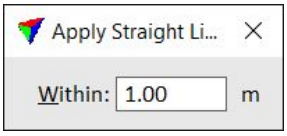
Not Spatix

Apply Straight Line tool moves all close by vertices to match a straight line between two selected vertices. Unnecessary vertices are removed from the resulting edge.

To apply a straight line:

1. Select **Apply Straight Line** tool.

   The **Apply Straight Line** dialog opens:



2. Define an offset within which the vertices are moved to match the straight line.

3. Move the mouse pointer inside a view.

   The vertex closest to the mouse pointer is dynamically highlighted.

4. Define the first vertex of the straight connection line with a data click.

   If you move the mouse pointer, the area within which vertices are effected is dynamically displayed.

5. Define the second vertex of the straight connection line with a data click.

   This moves all vertices within the given offset to the straight line and deletes unnecessary intermediate vertices along the edge. You can continue with steps 2 or 4.

SETTING	EFFECT
Within	Offset within which vertices are effected. Half of the given offset value applies to the left side and half to the right side of the straight line.

Build Step Corner

Not Spatix

Build Step Corner tool detaches a vertex and moves it along an incoming/outgoing edge segment. Only the vertex is moved, the effected segment should be aligned in a separate step using the Align Edge Segment tool.

To build a step corner:

1. Select Build Step Corner tool.
2. Move the mouse pointer inside a view.

   The edge segment and vertex to be detached closest to the mouse pointer is dynamically highlighted.
3. Define the vertex to detach and move with a data click.

   This updates the **Detail view** and displays the image that sees the selected vertex location best. If you move the mouse pointer, the new vertex location is dynamically displayed.
4. Define the new location of the vertex with a data click.

   This places the vertex at the new location. You can continue with step 3.

Cut Edge Corner

Not Spatix

Cut Edge Corner tool can modify a patch corner in two ways. It cuts off a piece from a corner or it adds a piece to a corner. In any case, the new edges are aligned perpendicular to the edges the form the original corner.

To add/cut off a piece to/from an edge corner:

1. Select **Cut Edge Corner** tool.
2. Move the mouse pointer inside a view.

   The edge corner closest to the mouse pointer is dynamically highlighted.
3. Define the edge corner to modify with a data click.

   This updates the **Detail view** and displays the image that sees the selected corner location best. If you move the mouse pointer, the new edge of the corner is dynamically displayed.

4. Define the location of one edge segment with a data click.

5. Define the location of the other edge segment with a data click.

   This places the new corner at the defined location. You can continue with step 3. You can go back from steps 5 to 4 and 4 to 3 by placing a reset click.

Cut Edge Segment

Not Spatix

Cut Edge Segment tool can modify an edge segment in two ways. It cuts off a piece from a segment or it adds a piece to a segment. The cut off or added part is formed by three new edge segments of which two are perpendicular and one is parallel to the original edge segment.

To add/cut off a piece to/from an edge segment:

1. Select **Cut Edge Segment** tool.

2. Move the mouse pointer inside a view.

   The edge closest to the mouse pointer is dynamically highlighted.

3. Define the edge to modify with a data click.

   This updates the **Detail view** and displays the image that sees the selected edge location best. If you move the mouse pointer, the new edge is dynamically displayed.

4. Define the location of one perpendicular edge segment with a data click.

5. Define the location of the other perpendicular edge segment with a data click.

6. Define the location of the parallel edge segment with a data click.

   This places the new edge segments at the defined locations. You can continue with step 3. You can go back from steps 6 to 5, 5 to 4, and 4 to 3 by placing a reset click.

Delete Edge Vertex

Not Spatix

Delete Edge Vertex tool removes a vertex from an edge. Only vertices that connect two edge segments can be removed.

To delete an edge vertex:

1. Select **Delete Edge Vertex** tool.

2. Move the mouse pointer inside a view.

   The vertex closest to the mouse pointer is dynamically highlighted.

3. Define the vertex to delete with a data click.

This removes the vertex. You can continue with steps 3.

Insert Edge Vertex

Not Spatix

Insert Edge Vertex tool adds a new vertex to an edge segment. It also defines the location of the new vertex.

To add an edge vertex:

1. Select **Insert Edge Vertex** tool.

   The **Insert Edge Vertex** dialog opens:

2. Define setting.

3. Move the mouse pointer inside a view.

   The edge segment closest to the mouse pointer is dynamically highlighted.

4. Define the edge segment to which to add a vertex with a data click.

   This updates the **Detail view** and displays the image that sees the selected edge location best. If you move the mouse pointer, the new vertex location is dynamically displayed.

5. Define the location of the new vertex with a data click.

   This adds the vertex and places the edge segments according to the location of the new vertex. You can continue with step 3.

SETTING	EFFECT
Insert at	Determines a rule for inserting an edge vertex: • **Free position** - the vertex can be inserted freely without limitations. • **90 degree angle** - the vertex can be inserted only at a place where it creates a 90 degree corner. • **To base 90 degree angle** - the vertex can be inserted only at a place where it creates a 90 degree corner of the base polygon. • **To intersection** - the vertex can be inserted only at a place where it creates an edge segment following the direction of an intersection line. This applies only if a vertex is inserted to an edge that shares an end point with an intersection line.

Modify Edge

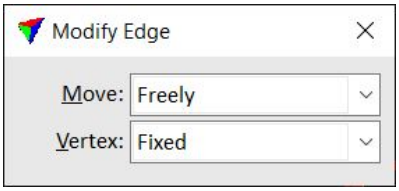
Not Spatix

Modify Edge tool moves an edge vertex or segment. The modification effects all parallel segments of the same edge.

To modify an edge vertex or segment:

1. Select **Modify Edge** tool.

   The **Modify Edge** dialog opens:



2. Define settings.

3. Move the mouse pointer inside a view.

   The edge segment or vertex closest to the mouse pointer is dynamically highlighted.

4. Define the edge segment or vertex to move with a data click.

   This updates the **Detail view** and displays the image that sees the selected edge/vertex location best. If you move the mouse pointer, the new edge segment or vertex location is dynamically displayed.

5. Define the new location of the edge segment or vertex with a data click.

   This places the edge segment or vertex at the new location and adjusts all other parallel segments along the same edge accordingly. You can continue with steps 2 or 4.

SETTING	EFFECT
Move	Determines a rule for moving an edge vertex: • **Freely** - the vertex can be moved freely without limitations. • **Along closest line** - the vertex can be moved only in the direction of the closest line. This is basically the incoming and outgoing edge segment. • **Along incoming line** - the vertex can be moved only in the direction of the incoming edge segment. • **Along outgoing line** - the vertex can be moved only in the direction of the outgoing edge segment.

SETTING	EFFECT
	• **To 90 degree angle** - the vertex can be moved only to a place where it creates a 90 degree corner. • **To base 90 degree angle** - the vertex can be moved only to a place where it creates a 90 degree corner of the base polygon. • **To intersection** - the vertex can be moved only in the direction of an intersection line. This applies only to vertices that are end points of intersection lines.
Vertex	Status of the vertex for tools that automatically adjust edges and vertices, such as the Set All Edges tool: • **Free** - sets a vertex to be freely movable for automatic edge and vertex adjustment. • **Fixed** - sets a vertex to be fixed in automatic edge and vertex adjustment.

Move Edge Vertex

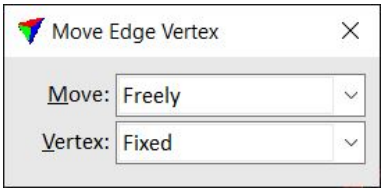
Not Spatix

Move Edge Vertex tool moves a vertex. The modification only effects the edge segments that are connected at the vertex but does not move other parallel edge segments.

To modify an edge vertex or segment:

1. Select **Move Edge Vertex** tool.

   The **Move Edge Vertex** dialog opens:



2. Define settings.

3. Move the mouse pointer inside a view.

   The vertex closest to the mouse pointer is dynamically highlighted.

4. Define the vertex to move with a data click.

   This updates the **Detail view** and displays the image that sees the selected vertex location best. If you move the mouse pointer, the new vertex location is dynamically displayed.

5. Define the new location of the vertex with a data click.

This places the vertex at the new location. You can continue with steps 2 or 4.

SETTING	EFFECT
Move	Determines a rule for moving an edge vertex: • **Freely** - the vertex can be moved freely without limitations. • **Along closest line** - the vertex can be moved only in the direction of the closest line. This is basically the incoming and outgoing edge segment. • **Along incoming line** - the vertex can be moved only in the direction of the incoming edge segment. • **Along outgoing line** - the vertex can be moved only in the direction of the outgoing edge segment. • **To 90 degree angle** - the vertex can be moved only to a place where it creates a 90 degree corner. • **To base 90 degree angle** - the vertex can be moved only to a place where it creates a 90 degree corner of the base polygon. • **To intersection** - the vertex can be moved only in the direction of an intersection line. This applies only to vertices that are end points of intersection lines.
Vertex	Status of the vertex for tools that automatically adjust edges and vertices, such as the Set All Edges tool: • **Free** - sets a vertex to be freely movable for automatic edge and vertex adjustment. • **Fixed** - sets a vertex to be fixed in automatic edge and vertex adjustment.

Set All Edges

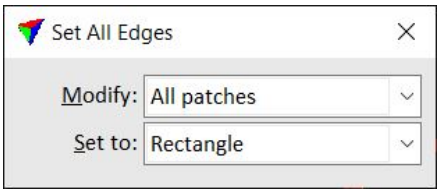
Not Spatix

Set all edges tool adjusts the edges of roof patches. As a result, all patches are set to a rectangle or rectangular shape. The tool can adjust all patches of a roof or only a single patch. Vertices that have been set by the Modify Edge or Move Edge Vertex tools to status *Fixed* are not changed. The other edges/vertices are adjusted to the fixed vertices.

To set all edges:

1. Select **Set All Edges** tool.

   The **Set All Edges** dialog opens:



2. Select settings.

3. Move the mouse pointer into a view.

   This displays the adjusted shape of one or more patches as preview.

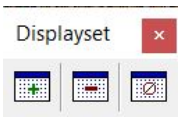
4. Apply the edge adjustment with a data click inside the patch or roof area.

   This sets the edges and thus, the shape of the patch(es).

SETTING	EFFECT
Modify	Determines the patches to modify: **All patches** or **One patch**.
Set to	Defines the shape of the patch(es): • **Rectangle** - all patches are set to rectangles. • **Rectangular** - all patches are set to rectangular shapes.

Displayset toolbox

Tools in the **Displayset** toolbox are used to add points to, remove points from, and clear displaysets. Displaysets can be used to display only a subset of points from a larger point cloud. This may be helpful to focus on a specific area by only displaying the points of this area in a view. Classification tools with setting **Any visible point** as source class classify only points that are included in the displayset.



TO	USE TOOL	
Add points to a displayset		Add Points To Displayset
Remove points from a displayset		Remove Points From Displayset
Clear a displayset		Clear Displayset

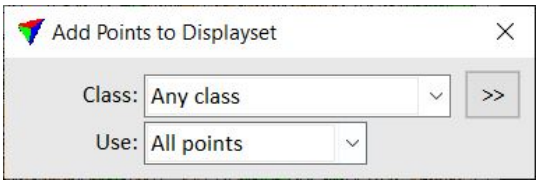
Add Points To Displayset

Add Points To Displayset tool adds points to a displayset. If there is not yet a displayset, the tool creates one. You can add all points of all or selected classes to a displayset, or a specific group, or points inside a fence.

To add points to a displayset:

1. Select **Add Points To Displayset** tool.

   This opens the **Add Points to Displayset** dialog:



3. Define settings.

4. If **Use** is set to **All points**, place a data click inside a view.

   If **Use** is set to **Group**, place a data click on the group to add to the displayset.

   If **Use** is set to **Fence**, draw a fence by placing data clicks inside a view. The last data click must close the fence. Confirm the fence content with an additional data click.

   This adds the points to the displayset. If **Points** is set to **Displayset only** for a view in the Display mode dialog, only points of the displayset are visible in this view.

SETTING	EFFECT
Class	Point class(es) added to the displayset.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Class** field.
Use	Determines which points are added to the displayset: • **All points** - all points in the given source class(es). • **Group** - points of one group in the given source class(es). This requires that groups are assigned to the points. • **Fence** - points inside a fence in the given source class(es).

Clear Displayset

Clear Displayset tool removes all points from a displayset.

To clear a displayset:

1. Select **Clear Displayset** tool.

   This removes all points from the displayset. The point display changes to all points, even if **Points** is set to **Displayset only** in the Display mode dialog.

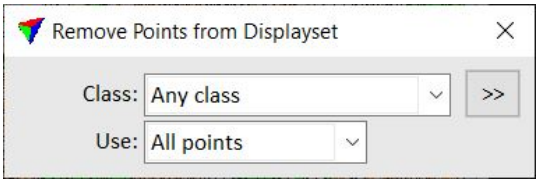
Remove Points From Displayset

Remove Points From Displayset tool removes points from a displayset. This requires that there is already a displayset created with the Add Points To Displayset tool. You can remove all points of all or selected classes from a displayset, or a specific group, or points inside a fence.

To remove points from a displayset:

1. Select **Remove Points From Displayset** tool.

   This opens the **Remove Points from Displayset** dialog:



3. Define settings.

4. If **Use** is set to **All points**, place a data click inside a view.

   If **Use** is set to **Group**, place a data click on the group to add to the displayset.

   If **Use** is set to **Fence**, draw a fence by placing data clicks inside a few. Confirm the fence content with an additional data click.

   This removes the points from the displayset. If all points are removed from the displayset, the displayset is cleared. The point display changes to all points, even if **Points** is set to **Displayset only** in the Display mode dialog.

SETTING	EFFECT
Class	Point class(es) removed from the displayset.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Class** field.

SETTING	EFFECT
Use	Determines which points are removed from the displayset: • **All points** - all points in the given source class(es). • **Group** - points of one group in the given source class(es). This requires that groups are assigned to the points. • **Fence** - points inside a fence in the given source class(es). This requires that a fence is drawn in the CAD file before starting the tool

Draw toolbox

The tools in the **Draw** toolbox are used to fit or adjust vector elements to point cloud data. In addition, there are tools for validating vector elements.

TO	USE TOOL	
Adjust mouse clicks to laser point coordinates		Mouse Point Adjustment
Fit linear element by laser points		Fit Linear Element
Drape linear element to laser surface		Drape Linear Element
Find breakline running parallel to an element		Find Breakline Along Element
Find curb stone running along a 2D element		Find Curb Along Element
Cut linear element with other features close-by		Cut Linear Element
Compare footprint polygons and classified roof hits		Check Footprint Polygons
Adjust shape element to laser elevation		Set Polygon Elevation
Place shape around a group of laser points		Place Collection Shape

TO	USE TOOL	
Find and classify pipes		Find Pipes
Draw tunnel sections into the CAD file		Vectorize Tunnel Sections
Check tunnel cross sections one at a time		Check Tunnel Sections
Inspect elements one at a time		Inspect Elements

Check Footprint Polygons

Check Footprint Polygons tool compares building footprint polygons with laser points, usually points classified into building class. There are two methods implemented in the tool:

- **Triangle network** - a TIN is created from the points, analyzed and compared with footprint polygons. The tool creates new polygons at locations where there is a footprint polygon but no points in the given source class(es), or where there are points but no footprint polygon.
- **Point radius** - the point-to-point distance is analyzed and the point coverage inside footprint polygons is computed. The tool moves polygons to another CAD file level if the point coverage is below a given percentage. Optionally, the coverage percentage can be drawn in the CAD file.

The tool is useful for finding flaws in the building classification, places with no or very sparse laser points on building roofs, and flaws in building footprint vector data.

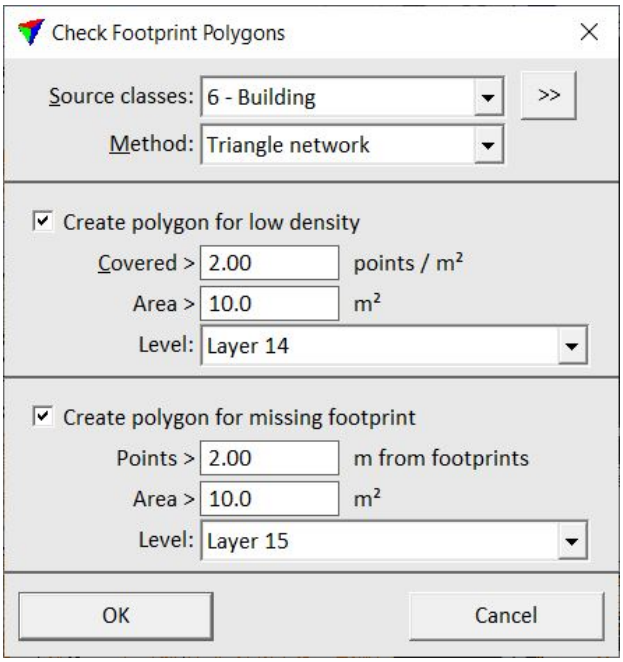
To compare footprint polygons and laser data:

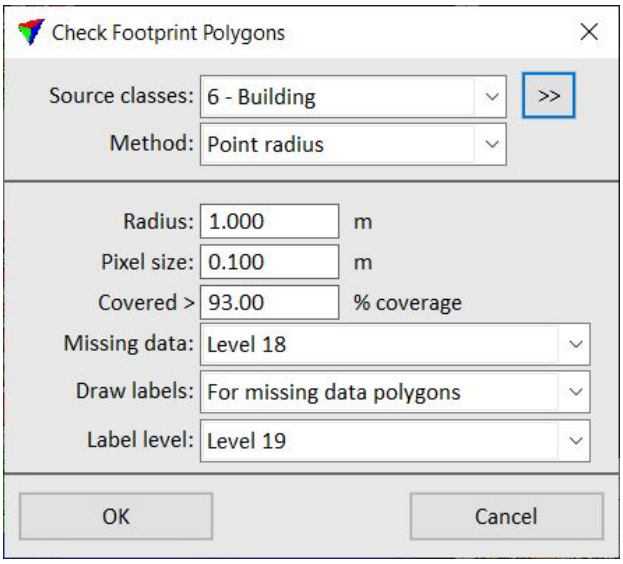
1. Load laser data into TerraScan. Only points in class(es) for the comparison with footprint polygons are required.

2. Select the footprint polygons that you want to include in the comparison.

3. Select the **Check Footprint Polygons** tool.

   This opens the **Check Footprint Polygons** dialog:





4. Define settings and click OK.

   The comparison starts and the software creates polygons for areas of low point density/coverage and/or missing footprints according to the given settings. An information dialog shows the number of created or modified polygons.

SETTING	EFFECT
Source classes	Point class(es) used for the comparison. The list contains the active classes in TerraScan.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Source classes** field.

SETTING	EFFECT
Method	Method for computing point coverage in a polygon area: • **Triangle network** - based on a TIN. This allows the detection of areas where there are not enough points inside a polygon as well as areas of missing footprint polygons. • **Point radius** - based on radius around each point. This allows the detection of areas where there are not enough points inside a polygon. This determines the availability of further settings in the dialog.
Create polygons for low density	If on, polygons are created on places where there is a selected polygon but no or only sparse laser points in the given **Source class(es)**.
Covered	Defines the minimum point density inside a footprint polygon. If the density is lower, a polygon is created.
Area	Defines the minimum area of a building. Only areas larger than the given value are considered in the comparison.
Level	Polygons marking low density places are drawn on the given level using the active symbology settings of the CAD file.
Create polygons for missing footprint	If on, polygons are created on places where there are laser points in the given **Source class(es)** but no selected polygon.
Points	Defines the minimum distance between a laser point and a selected polygon. If the distance is larger, a polygon is created.
Area	Defines the minimum area of a building. Only areas larger than the given value are considered in the comparison.
Level	Polygons marking missing footprint places are drawn on the given level using the active symbology settings of the CAD file.
Radius	Radius around a point within which the software checks for neighbor points.
Pixel size	Size of a pixel for analyzing point coverage inside a polygon area.
Covered >	Minimal expected percentage of polygon area covered with points. If the percentage is

SETTING	EFFECT
	lower, the polygon is identified as missing data area.
Missing data	Polygons identified as areas of missing data are moved to the given CAD file level.
Draw labels	Determines the polygons for which coverage labels are drawn: • **Do not draw** - no labels are drawn. • **For all polygons** - labels for all areas. • **For missing data polygons** - labels for areas that are missing data.
Label level	Labels are drawn on the given CAD file level. This is only active if **Draw labels** is set to **For all polygons** or **For missing data polygons**.

You can check the polygons created by this tool in a structured way with the help of the Inspect Elements tool.

Check Tunnel Sections

Check Tunnel sections tool supports the systematic check of tunnel sections created by the Vectorize Tunnel Sections tool. It provides a list of tunnel sections from which you can select one element after the other.

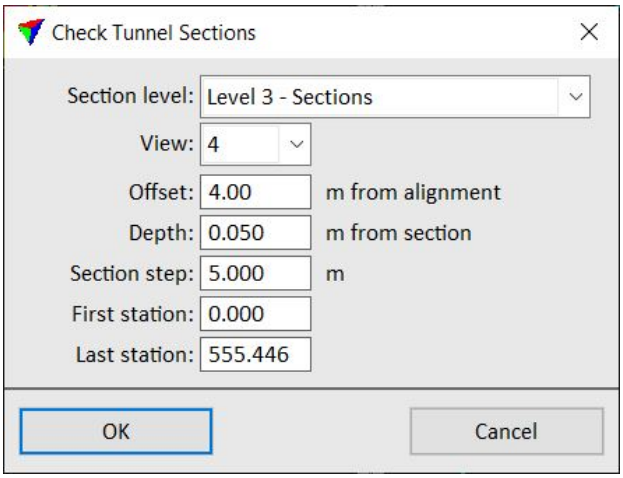
The tool includes settings that define a CAD file section view displaying a selected tunnel section. The selected section is automatically centered in this view.

To check tunnel sections:

1. Select the alignment element that has been used for vectorizing the tunnel sections.

2. Select the **Check Tunnel Sections** tool.

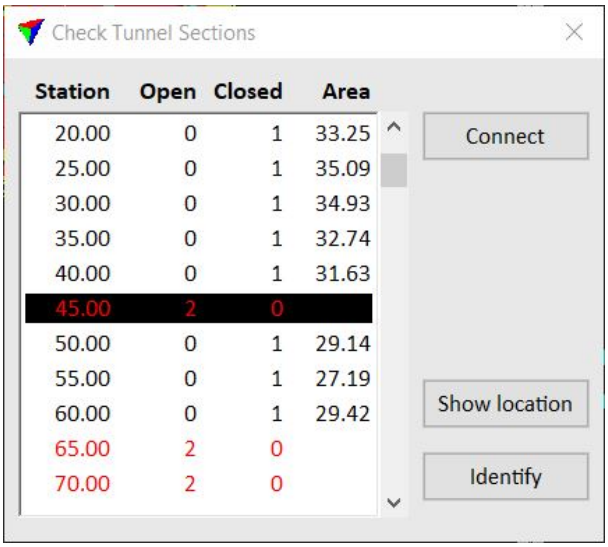
   This opens the **Check Tunnel Sections** dialog:



SETTING	EFFECT
Section level	CAD file level on which the tunnel sections have been drawn.
View	CAD file view used for displaying the selected tunnel section in a section view.
Offset	Distance to the left and right from the alignment element that is displayed in the section view.
Depth	Distance forward and backward from the section centerline. Defines the depth of the section displayed in the section view.
Section step	Distance between consecutive sections. This should be the same value as used for vectorizing the tunnel sections.
First station	Location of the first section for display along the alignment element. By default, this is the starting point of the alignment element.
Last station	Location of the last section for display along the alignment element. By default, this is the end point of the alignment element.

3. Define settings and click OK.

   This opens another **Check Tunnel Sections** dialog that contains the list of all sections that belong to the selected alignment element:



- **Station** - location of a tunnel section along the alignment element.
- **Open** - amount of gaps along the tunnel section line string. Entries with gaps are highlighted in red.
- **Closed** - amount of closed tunnel section line strings.
- **Area** - size of the area enclosed by the tunnel section line string. Given in CAD file units.

4. Select a row in the list of sections.

   This centers the selected section in the CAD file view defined in the tool's **View** setting.

SETTING	EFFECT
Connect	Creates a connection line between open end points of the tunnel section line string.
Show location	Select a row in the list, click on the button and move the mouse pointer inside a CAD file view. This highlights the selected section location in the view.
Identify	Click on the button and identify an element with a data click in a CAD file view. This selects the corresponding row in the list.

To close a gap in a tunnel section

1. Select the section in the **Check Tunnel Sections** dialog list.

2. Select the Connect button in the **Check Tunnel Sections** dialog.

3. Move the mouse pointer inside the CAD file section view close to the gap you want to close.

   This shows a preview of the connection line.

4. Place a data click in order to confirm the connection line. Continue placing connection lines until all gaps are closed.

This connects the end points of the existing section line string, moves the section drawing to the active level of the CAD file and applies the active symbology settings. The entry in the **Check Tunnel Sections** dialog list is updated.

You may use other CAD tools to modify tunnel sections as well. This is not updating the list in the **Check Tunnel Sections** dialog automatically.

Cut Linear Element

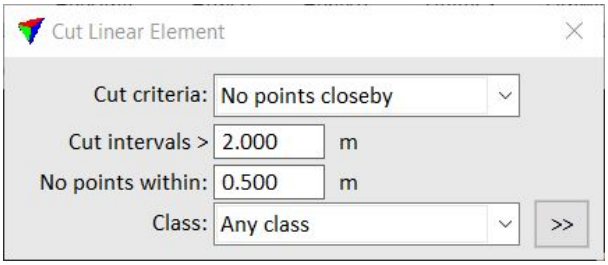
Cut Linear Element tool determines the distance between linear elements and laser points or other linear elements. It removes parts of linear elements for which there are no laser points, no other linear elements, or other linear elements within a certain 3D distance.

It may be used, for example, to mark places where there are elements close to rail tracks or wires.

To cut linear elements:

1. (Optional) Select the element(s) that you want to cut.
2. Select the **Cut Linear Element** tool.

   This opens the **Cut Linear Element** dialog:



3. Define settings.
4. If elements have been selected, start the process with a data click inside the CAD file view.

   This compares the selected elements to laser points or other elements and removes parts if applicable.

   OR

4. Identify the element to cut.

   This highlights the given element.

5. Accept the highlighted element with a data click.

   This compares the selected element to the laser points or other elements and removes parts if applicable. You can continue to steps 3 or 4.

SETTING	EFFECT
Cut criteria	Defines which element parts are removed:

SETTING	EFFECT
	• **No points closeby** - no laser points are close to the element. • **No elements closeby** - no other vector elements are close to the element. • **Another element closeby** - another element is close to the element.
Cut intervals	Minimum size of a linear element part to be removed by the tool. An element (part) is removed if the resulting 3D gap along the element longer than the given value.
No points within	An element (part) is removed if the 3D distance to laser points/other elements is larger than the given value.
Class	Point class(es) considered in the distance computation. This is only active if **Cut criteria** is set to **No points closeby**.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Class** field.
Level	Elements of the given CAD file level are considered for distance computation. This is only active if **Cut criteria** is set to **No elements closeby** or **Another element closeby**.

Drape Linear Element

Drape Linear Element tool fits linear elements to the elevation of laser points. The xy position of the elements is not effected.

The tool is typically used to drape linear elements which run on a smooth, planar ground surface or along edges of slopes. Valid CAD element types for this tool include lines, line strings, shapes, and complex shapes. You can fit several selected elements in a single process.

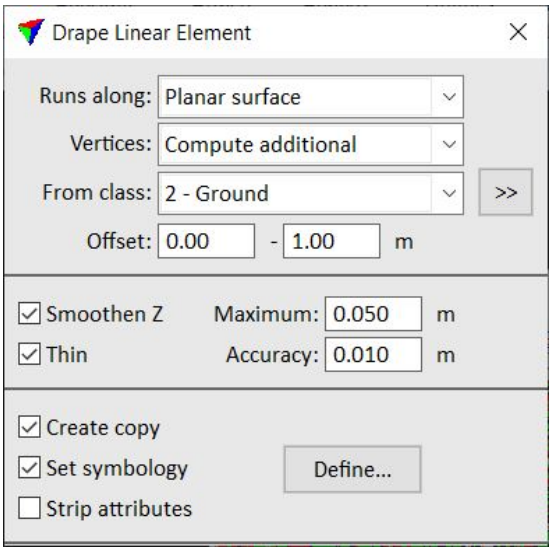
You can decide whether you want to adjust only existing vertices or add intermediate vertices so that the resulting element follows changes of the surface more closely. The density of automatically added vertices depends on the density of laser points. In addition, smoothing and thinning can be applied to the adjusted element.

To drape linear elements to laser points:

1. (Optional) Select the element(s) that you want to drape.

2. Select the **Drape Linear Element** tool.

   This opens the **Drape Linear Element** dialog:



3. Define settings.

4. If elements have been selected, start the draping process with a data click inside a view.

   This drapes the selected elements to the laser points.

   OR

4. Identify the element to drape.

   This highlights the given element.

5. Accept the highlighted element with a data click.

   This drapes the linear element to the laser points. You can continue to steps 3 or 4.

SETTING	EFFECT
Runs along	Type of surface structure the linear element runs along: • **Planar surface** - smooth planar surface. • **Juncture of surfaces** - intersection of one planar surface on the left side of the element and another planar surface on the right side. • **Edge of surface** - edge of a surface where only points on the left or on the right side of the edge are used for draping. • **Fixed height curb stone** - curb stone of constant height. Two linear elements are generated with a given elevation difference. • **Auto height curb stone** - curb stone of varying height. The average elevation difference is derived from the points on the left and the right side of the curb stone edge. Two linear elements are generated which have the derived elevation difference.

SETTING	EFFECT
Vertices	Determines the computation of additional vertices: • **Compute additional** - additional vertices are computed, the draped element follows the surface structure more closely. • **Drape original only** - no additional vertices are added.
From class	Point class(es) to drape to. Contains the list of active classes in TerraScan.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **From class** field.
Offset	Offset distance range from which to use laser points for computing elevation values for the linear element.
Smoothen Z	If on, smoothing is applied to the elevation of vertices of the draped element. The elevation of vertices can change up to the value given in the **Maximum** field.
Thin	If on, unnecessary vertices are removed from the draped element. The allowed change in position caused by thinning is defined by the value in the **Accuracy** field.
Create copy	If on, a new element is created and draped. If off, the original element is draped.
Set symbology	If on, you can define new symbology settings for the draped element. Click on the **Define** button in order to open the **Draped element symbology** dialog. You can define **Level**, **Color**, **Weight**, and **Style** settings.
Strip attributes	If on, any attribute linkages are removed from the draped element.
Shift	If on, the draped element is created at the given xy Distance from the surface edge location. This is only active if **Runs along** is set to **Edge of surface**.
Curb width	Width of a curb stone, xy offset between the upper and lower linear element of a curb stone. This is only active if **Runs along** is set to **Fixed height curb stone** or **Auto height curb stone**.

SETTING	EFFECT
Curb height	Height of a curb stone, elevation offset between the upper and lower linear element of a curb stone. This is only active if **Runs along** is set to **Fixed height curb stone**.

To drape shapes to a constant elevation derived from laser points inside the shapes, you may also check the Set Polygon Elevation tool.

Find Breakline Along Element

Find Breakline Along Element tool creates a linear element which runs along a breakline in the terrain. The search starts with an existing 2D linear element which runs close to the actual breakline location.

Valid CAD element types for this tool include lines, line strings, shapes, and complex shapes. You can fit several selected elements in a single process.

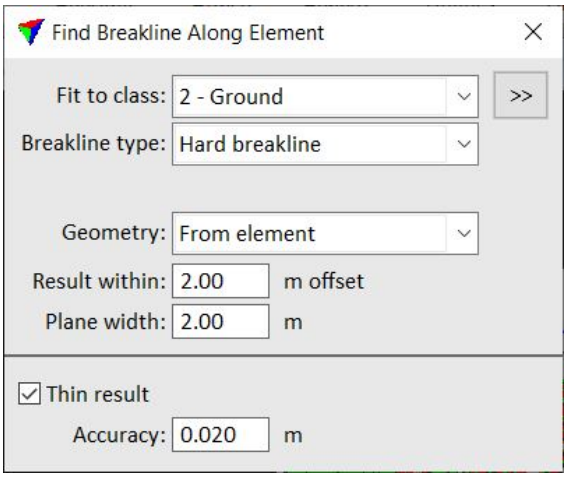
The tool finds the more accurate breakline position if there is a planar surface on both sides of the breakline. It is most useful to create hard breakline elements such as the top of man-made slopes.

To find a breakline feature:

1. (Optional) Select the element(s) from which you want to create breaklines.

2. Select the **Find Breakline Along Element** tool.

   This opens the **Find Breakline Along Element** dialog:



3. Define settings.

4. If elements have been selected, start the process with a data click inside a view.

   This creates new 3D elements at the most probably position of terrain breaklines close to the selected elements.

OR

4. Identify the 2D linear element running close to a terrain breakline.

   This highlights the given element.

5. Accept the highlighted element with a data click.

   The application determines the most probable position for a breakline using the given parameters and creates a 3D element running along the breakline. You can continue to steps 3 or 4.

SETTING	EFFECT
Fit to class	Point class from which to find the breakline in the terrain. The list contains the active point classes in TerraScan.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Fit to class** field.
Breakline type	Type of the breakline: • **Hard breakline** - breakline which forms a sharp corner when viewed in a cross section. • **Soft breakline** - breakline which forms a soft corner when viewed in a cross section. • **Slope change** - terrain slope changes at the breakline. • **Elevation jump top** - top of a breakline feature forming a drop in elevation. • **Elevation jump bottom** - bottom of a breakline feature forming a drop in elevation.
Soft breaks	Distance between single linear elements that form a soft breakline. This is only active if **Breakline type** is set to **Soft breakline**.
Geometry	Geometry type of the resulting element: • **Line** - feature may have sharp turns. • **Curve** - feature has smooth turns only. • **From element** - derived from the original element.
Result within	Determines how close the original 2D element is to the true xy position of the breakline.
Plane width	Width of the planar surfaces on both sides of the breakline.
Thin result	If on, unnecessary vertices are removed from the breakline element. The allowed change in position caused by thinning out vertices is defined by the value in the **Accuracy** field.

Find Curb Along Element

Find Curb Along Element tool creates longitudinal breakline elements along sharp edges in a point cloud. It requires a 2D line element running along the approximate location of a curb stone.

The tool creates two 3D line elements following the lower and upper edges of curb stones in a point cloud. It separates the lower and upper edge up to a given minimum vertical distance. If the height of the curb stone is smaller or if there are no sharp edges in the point cloud, only one line is drawn or the detection fails. The edge detection method is based on cross-sections every 25 cm along the 2D line element and the elevation jump of two planes (sidewalk and road surface). The planes must be represented by at least 40 points for a confident edge detection.

The suggested workflow for preparing the curb stone detection is as follows:

1. Classify ground and low vegetation points. The points should include the upper and lower ground level as well as the vertical side of a curb stone.

2. Classify model keypoints from ground points.

3. Create a TIN model and display the model as shaded surface in TerraModeler.

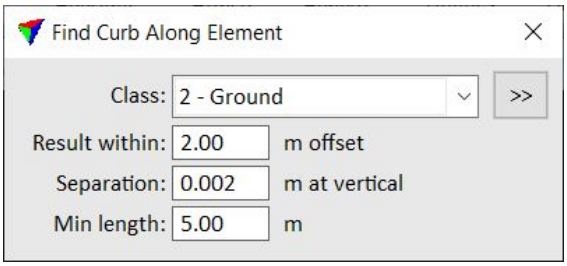
4. Digitize an approximate 2D line string element along the curb stone edge. You may use any TerraScan or CAD tool for line placement, such as Place railroad string tool, **Place Polyline** tool (Spatix) or **Place SmartLine** tool (MicroStation).

To vectorize curb stones:

1. (Optional) Select the 2D line string element(s) running along the approximate curb stone location.

2. Select **Find Curb Along Element** tool.

   This opens the **Find Curb Along Element** dialog:



3. Define settings.

4. If elements have been selected, place a data click inside a view in order to start the process.

   OR

4. Identify the 2D line element running along the approximate curb stone location with a data click.

   This highlights the given element.

5. Accept the highlighted element with another data click.

This creates new 3D line elements at the most probably position of curb stones close to the selected element(s). The line elements are drawn on the active level using the active symbology settings of the CAD file. You can continue to steps 3 or 4.

SETTING	EFFECT
Class	Point class(es) from which to find the curb stones.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Class** field.
Result within	Determines how close the original 2D element is to the true xy position of the curb stone.
Separation	Minimum distance between lower and upper edge of a curb stone for which two separate lines are drawn. If the height of a curb stone is smaller, only one line is drawn.
Min length	Minimum length of a curb stone line element.

You can undo the curb stone drawing by using the **Undo** command of the CAD platform.

Find Pipes

Find Pipes tool is used for the automatic detection of circular pipes based on dense point clouds. It may be used to detect pipes running horizontally or vertically for a longer distance without turns, such as pipes running inside a tunnel. The tool is not suited for pipes that run with a lot of turns such as indoor pipes of industrial sites.

Before running the tool, normal vectors must be computed for the point cloud. This can be done using the Compute normal vectors command for loaded points or the Compute normal vectors macro action. The software assumes that the normal direction points towards the outside of a pipe. In addition, some thinning may be applied to a very dense point cloud in order to speed up the detection process.

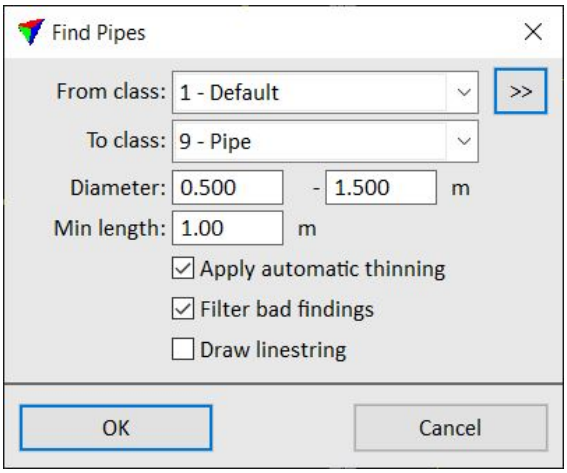
The **Find Pipes** tool runs on points loaded in TerraScan. It classifies laser points on pipes into a separate class and optionally creates line string elements that represent the centerline of a pipe.

To detect pipes automatically:

1. Load laser points into TerraScan.

2. Select **Find Pipes** tool.

   This opens the **Find Pipes** dialog:



3. Define settings and click OK.

   This starts the detection process. The software classifies the points and draws a line string in the center of a detected pipe, if the setting is selected. The level, color, line weight, and line style of the line string are determined by the active level and symbology settings of the CAD file.

SETTING	EFFECT
From class	Point class used for detecting pipes. The list contains the active classes in TerraScan.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **From class** field.
To class	Target class into which points on detected pipes are classified.
Diameter	Minimum and maximum outer diameter of the pipes to detect. Choose values a bit smaller (minimum) and bigger (maximum) than the true pipe diameter.
Min length	Minimum length of pipe structure to be detected. This avoids false findings of other circular objects in the point cloud.
Apply automatic thinning	If on, thinning is applied to the point cloud for detecting the pipes. The thinning distance is automatically derived from the pipe diameter settings. The thinning is only done internally for the pipe detection process and has no permanent effect on the point cloud or pipe classification result. Thinning speeds up the process significantly and should be switched on.

SETTING	EFFECT
Filter bad findings	If on, the software refines the pipe detection in order to avoid false findings of other circular objects in the point cloud.
Draw linestring	If on, the software draws a line string element at the center location of a pipe. The element is drawn using the active level and symbology settings of the CAD file.

If you only want to draw a centerline for already classified pipes, select the same class for **From class** and **To class** settings of the tool.

Fit Linear Element

Fit Linear Element tool improves the horizontal accuracy of manually placed linear elements by fitting the xy location of vertices to laser points. The resulting linear element follows laser points more closely.

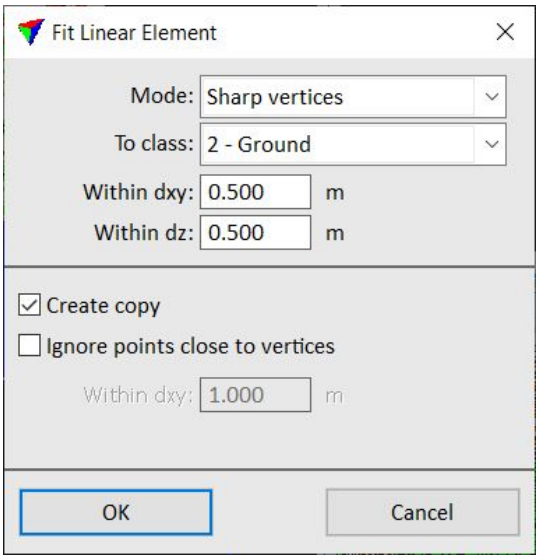
Valid CAD element types for this tool include lines, line strings, shapes, and complex shapes. You can fit several selected elements in a single process.

To fit linear elements:

1. (Optional) Select the element(s) that you want to fit.

2. Select the **Fit Linear Element** tool.

   This opens the **Fit Linear Element** dialog:



3. Define settings and click OK.

   If elements have been selected, they are fitted to the laser points.

4. Identify the element to fit with a data click.

   This highlights the given element.

5. Accept the highlighted element with another data click.

   This fits the selected element to follow laser points more accurately. You can continue to step 4.

SETTING	EFFECT
Mode	Defines whether vertices are added to the fitted element or not: • **Sharp vertices** - no additional vertices are added. Suitable for fitting elements with straight line segments between sharp turns (for example overhead wires). • **Smooth curvature** - additional vertices may be added. Suitable for fitting an element which has smooth curvature only (for example paint line on a road).
To class	Point class to fit to. Contains the list of active classes in TerraScan.
Within dxy	Maximum horizontal offset of laser points to be used in the fitting process.
Within dz	Maximum vertical offset of laser points to be used in the fitting process. Enter a large value such as 999.000 if you want to use all laser points regardless of their elevation.
Create copy	If on, a copy of the original element is created and fitted. The new element is placed on the active level using active symbology settings of the CAD file. If off, the original element is fitted.
Ignore points close to vertices	If on, the fitting process ignores points within the distance to element vertices given in the **Within dxy** field. This is only active if **Mode** is set to **Sharp vertices**.
Add vertices to long segments	If on, the fitting process adds intermediate vertices along long segments. The distance between consecutive vertices is given in the **Step** field. This is only active if **Mode** is set to **Smooth curvature**.
Smoothen curvature	If on, the curvature of the fitted element is smoothed by balancing angular direction changes between consecutive vertices. This is only active if **Mode** is set to **Smooth curvature**.

Inspect Elements

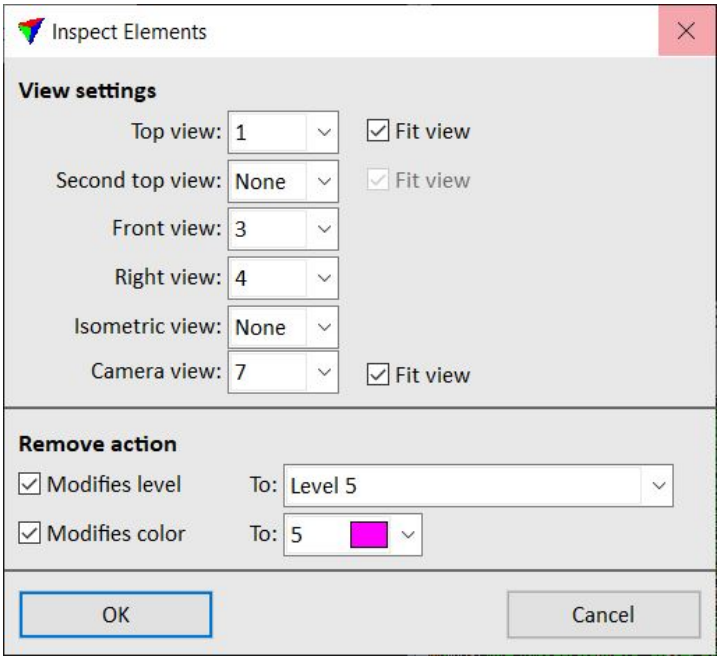
Inspect Elements tool supports the systematic check of vector elements in a CAD file. It provides a list of elements from which you can select one element after the other.

The tool includes view settings that define CAD file views displaying the selected element in different view orientations. The selected element is automatically centered in these views.

To inspect vector elements:

1. Select the elements you want to check.
2. Select the **Inspect Elements** tool.

   This opens the **Inspect Elements** dialog:

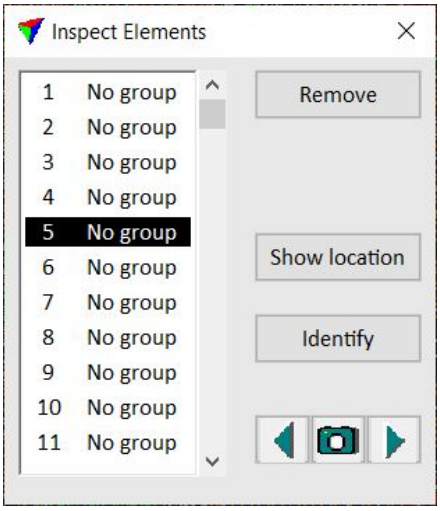


SETTING	EFFECT
Top view	CAD file view that displays the selected element in a top view.
Second top view	CAD file view that displays the selected element in a top view.
Front view	CAD file view that displays the selected element in a front section view. The cross section is 90 degree rotated compared with the **Right view**.
Right view	CAD file view that displays the selected element in a right section view. The cross section is 90 degree rotated compared with the **Front view**.

SETTING	EFFECT
Isometric view	CAD file view that displays the selected element in an isometric view.
Camera view	CAD file view that displays the selected element in a camera view. The view can display images that are referenced by an active image list in TerraPhoto.
Fit view	If on, the selected element is automatically fitted in the view.
Modifies level	If on, an element removed by the **Remove** button of the **Inspect Elements** dialog is moved to the given CAD file level.
Modifies color	If on, the selected color is applied to an element removed by the **Remove** button of the **Inspect Elements** dialog. The list contains the active color table of the CAD file.

3. Define settings and click OK.

   This opens another **Inspect Elements** dialog that contains the list of all selected elements:



4. Select a line in the list of elements.

   This centers the selected element in all CAD file views defined in the tool's **View settings**. You can use the remove button of the dialog to remove an element or take any other appropriate action.

SETTING	EFFECT
Remove	Removes the selected element from the list. The element itself is not deleted but it can be moved to another level and/or get another color according to the tool's Remove action settings.

SETTING	EFFECT
Show location	Select a line in the list, click on the button and move the mouse pointer inside a CAD file view. This highlights the selected element in the view.
Identify	Click on the button and identify an element with a data click in a CAD file view. This selects the corresponding line in the list.
	Moves one image backward in the active image list and displays the new image in the camera view.
	Click on the button and move the mouse pointer inside a CAD file view. The image closest to the mouse pointer is highlighted. Select an image for the camera view display with a data click.
	Moves one image forward in the active image list and displays the new image in the camera view.

Mouse Point Adjustment

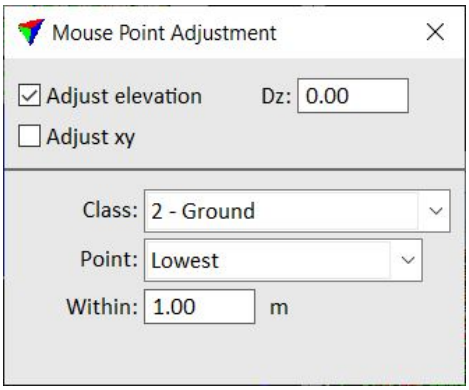
Mouse Point Adjustment tool adjusts vector data to the location and/or the elevation of laser points. You can use this tool with any CAD or TerraScan element placement tool. **Mouse Point Adjustment** simply fixes the coordinates of data clicks, which means the vertices of vector elements to laser data coordinates. It might be considered as a “snap to laser points” tool.

You can choose whether the elevation and/or the xy location of the vertices are adjusted. If you want to digitize a linear object, such as a wire, you probably want to adjust both, the xy and the elevation of the vertices to the laser points. On the other hand, if you want to place an object on the ground, you probably want to adjust only the elevation of the vertices.

To place elements adjusted to laser points:

1. Select the **Mouse Point Adjustment** tool.

   This opens the **Mouse Point Adjustment** dialog:



2. Define setting.

3. Start the drawing tool that you want to use and digitize elements.

   As long as the **Mouse Point Adjustment** dialog is open and any adjustment option is switched on, all vertices of elements are adjusted according to the settings.

SETTING	EFFECT
Adjust elevation	If on, the elevation of data clicks (vertices) is adjusted.
Dz	Constant offset from the laser data elevation value that is added to the elevation coordinate of element vertices.
Adjust xy	If on, the xy location of data clicks (vertices) is adjusted.
Class	Point class to adjust to. Contains the list of active classes in TerraScan.
Point	Points or surface model from which element vertex coordinates are derived: • **Closest** - point closest to the data click. • **Highest** - highest point within a search area. • **Average** - average xy and/or z of all points within a search area. • **Percentile** - average xy and/or z of a given percentile of points within a search area. • **Lowest** - lowest point within search area. • **TIN model** - elevation of a triangulated surface model and xy from the closest point.
Within	Radius of the search area around the mouse pointer location.

Be sure to always close the **Mouse Point Adjustment** dialog if you do not want to adjust data clicks to laser points. As it effects all data clicks, it may interfere with your normal work if it is active.

Place Collection Shape

Place Collection Shape tool lets you create shape elements that are associated with certain thematic types. Collection shapes are used to group laser points together that belong, for example, to a topographic object, such as a building or a road.

Collection shapes can be used later to output groups of laser points.

You need to define collection shape types before you can use the tool. See Collection shapes category of TerraScan **Settings** for more information.

Collection shapes can be created manually by using the digitization function of the **Place Collection Shape** tool. Alternatively, they can be produced automatically from already existing shape elements that have been created using any CAD tool for shape drawing.

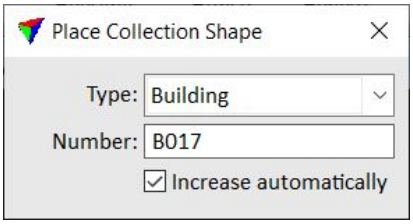
Collection shapes do not store any attribute information. The application only uses element level, color, line weight, and line style to recognize a shape as a collection shape of a specific type.

To place a collection shape:

1. (Optional) Select shape elements that you want to turn into collection shapes.

2. Select the **Place Collection Shape** tool.

   The **Place Collection Shape** dialog opens:



3. Select a collection shape type in the **Type** list.

4. If shape elements have been selected, apply the collection shape symbology with a data click inside a view.

   This turns all selected shapes into collection shapes. An information dialog shows the number of effected shape elements.

   OR

4. (Optional) Define additional settings in the **Place Collection Shape** dialog.

5. Digitize the shape boundary by placing vertices with data clicks. To close the shape, place a data click close to the first vertex. You can undo a vertex placement with a reset click.

   This creates a shape element on the level and using the symbology specified for the collection shape type. If a **Number** is defined, a text element is drawn inside the shape element using active text settings of the CAD file.

SETTING	EFFECT
Type	Collection shape type. The list contains all shape types that are defined in Collection shapes category of TerraScan **Settings**.
Number	Text string that is drawn inside the shape element. This works only if collection shapes are digitized manually.
Increase automatically	If on, and if the text string in the **Number** field ends with a number, the number increases by 1 after a collection shape has been created manually.

Set Polygon Elevation

Set Polygon Elevation tool sets the elevation of a closed element based on laser points inside it.

This tool is typically used to drape a digitized 2D shape to the elevation of laser points on a building roof or a bridge. All vertices of the shape are set to the same elevation value derived from laser points.

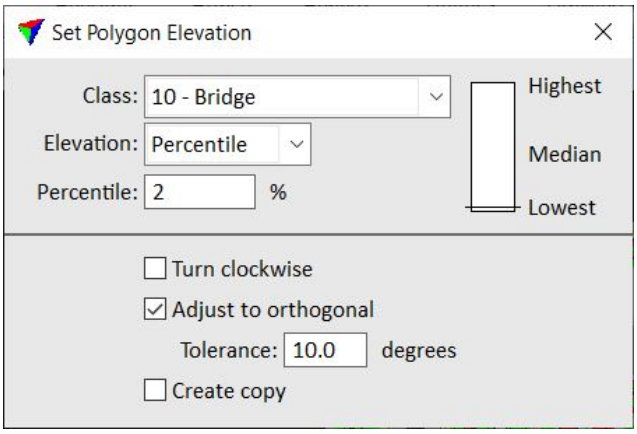
Valid CAD element types include shapes, complex shapes, and ellipses. You can fit several selected elements in a single process.

To set the elevation of a closed element:

1. (Optional) Select elements for which you want to set the elevation.

2. Select the **Set Polygon Elevation** tool.

   This opens the **Set Polygon Elevation** dialog:



3. Define settings.

4. If elements have been selected, start the process with a data click inside the CAD file view.

This computes an elevation value from laser points for each selected element and adjusts all selected elements.

OR

4. Identify the element to adjust with a data click.

   This highlights the element.

5. Accept the highlighted element with a data click inside the CAD file view.

   This computes an elevation value from laser points and adjusts the element. You can continue with steps 3 or 4.

SETTING	EFFECT
Class	Point class from which to derive the elevation. Only points inside the shape element are used.
Elevation	Method of elevation computation: • **Percentile** - elevations from the given Percentile value of points. The scale shows which percentile of points is used: Lowest, Median, or Highest points, and can be used to set the percentile value with a data click. • **Average** - average of all laser point elevation values inside the shape.
Turn clockwise	If on, the drawing direction of the adjusted shape is forced to run clockwise.
Adjust to orthogonal	If on, the corner angles of the adjusted shape are fixed to 90 degree turns if the angles of the original element are within the **Tolerance** value off from 90 degrees.
Create copy	If on, a new element is created and adjusted using the active level and symbology settings of the CAD file. If off, the original element is modified.

Vectorize Tunnel Sections

Vectorize Tunnel Sections tool draws tunnel cross sections into the CAD file. The sections are drawn as line string elements which are fitted to the cross section provided by the point cloud. The tool requires a 3D linear element that runs along the approximate center of the tunnel. The tool does not try to create closed line strings for the sections.

Optionally, the tool draws additional line elements as markers for the section location. These horizontal marker lines are drawn at the alignment element elevation and extended up to the length of the given offset from alignment setting. The line may be useful for creating a vertical section at a tunnel cross section location.

Before vectorizing tunnel sections, it may be useful the classify the points on the tunnel floor, roof and walls by using the Tunnel surfaces classification routine.

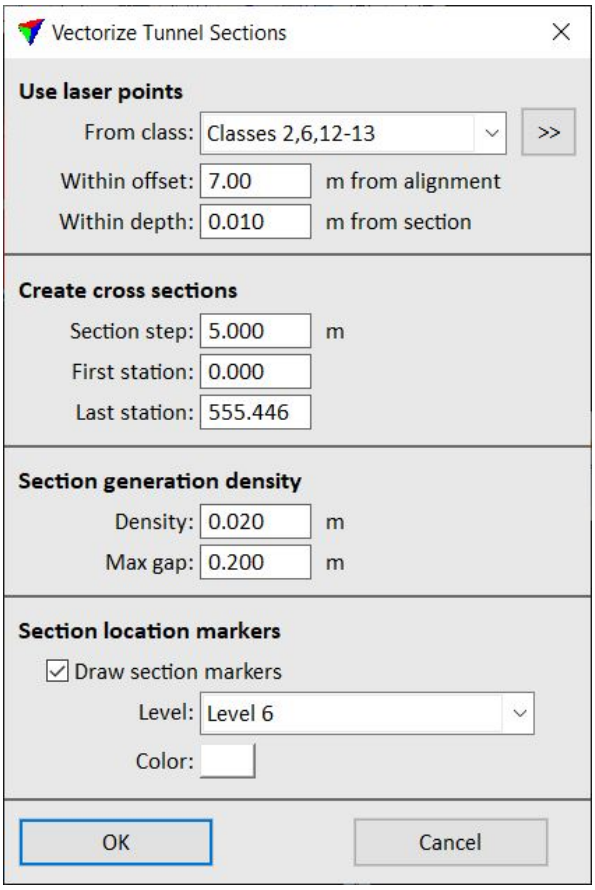
The Check Tunnel Sections tool may be used to check and modify the tunnel sections in an organized and automated way.

To vectorize tunnel sections:

1. Select the 3D alignment element.

2. Select the **Vectorize Tunnel Sections** tool.

   This opens the **Vectorize Tunnel Sections** dialog:



3. Define settings and click OK.

   This draws the sections and possibly the location markers into the CAD file. The sections are created as line string elements and drawn on the active level with the active symbology settings of the CAD file.

SETTING	EFFECT
From class	Point class(es) that are considered for the section line fitting.
>>	Opens the **Select classes** dialog which contains the list of active classes in

SETTING	EFFECT
	TerraScan. You can select multiple source classes from the list that are then used in the **From class** field.
Within offset	Maximum distance of points from the alignment element that are considered for the section line fitting. Determines also the length of the section marker lines left and right from the alignment.
Within depth	Depth of a tunnel cross section within which points are used for the section line fitting.
Section step	Distance between consecutive sections.
First station	Location of the first cross section along the alignment element.
Last station	Location of the last cross section along the alignment element. By default, the software suggests the last possible location at the end of the alignment element.
Density	Minimum length of a line segment along the cross section line string. Determines the complexity of the cross section line string element.
Max gap	Maximum length of a gap in the point cloud cross section that is ignored for the section drawing. A larger gap breaks the section line string element.
Draw section markers	If on, an additional line element is drawn in the CAD file at the location of the cross section.
Level	CAD file level on which the section marker lines are drawn. This is only active if **Draw section markers** is switched on.
Color	Color used for drawing the section marker lines. Uses the active color table of the CAD file. This is only active if **Draw section markers** is switched on.

General toolbox

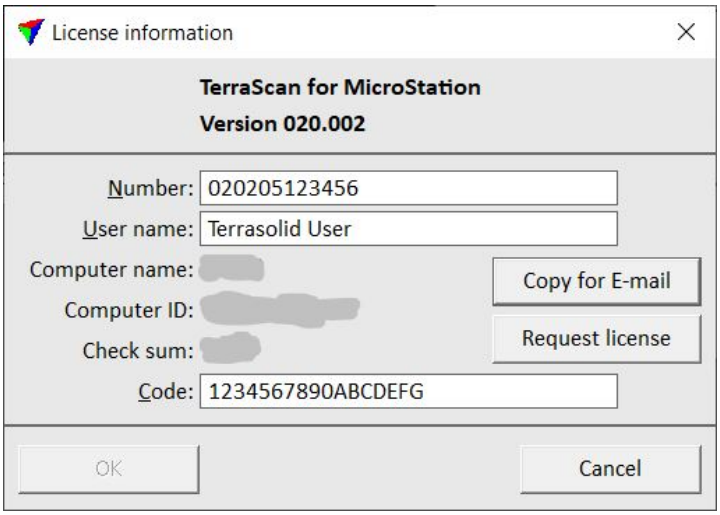
The tools in the **General** toolbox are used to define user settings, to define point classes, to define project blocks, to manage trajectories, to load points and to access license information and the users' guide document.

TO	USE TOOL	
Change user settings		Settings
Define coordinate range and resolution		Define Coordinate Setup
Define point classes and drawing symbology		Define Classes
Design project block boundaries		Design Block Boundaries
Define project and data blocks		Define Project
Manage trajectory information		Manage Trajectories
Load points from airborne / mobile scanning		Load Airborne Points
Load points from static terrestrial scanning		Load Ground Points
Show about TerraScan and license information		About TerraScan
Open online help		Help on TerraScan

About TerraScan

About TerraScan tool opens a dialog which shows information about TerraScan and about the license.

From this dialog, you can open the **License information** dialog which looks the same for all Terra Applications:



Use the **Request license** button to start the online registration for node-locked licenses.

More information about license registration is available on the Terrasolid web pages.

Define Classes

Define Classes tool opens a dialog for managing point classes and related drawing rules.

A point class definition includes descriptive information for the class, such as a unique number, code, and description, and rules for displaying the point on the screen or drawing it into the CAD file.

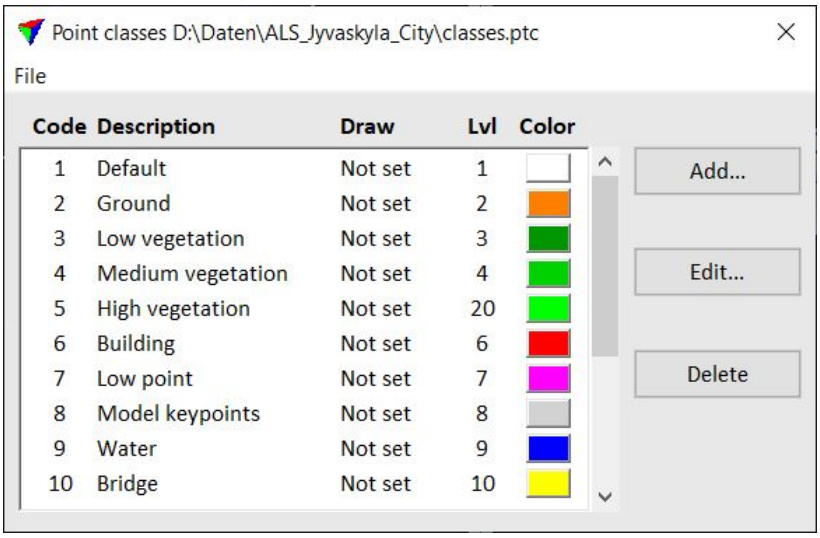
TerraScan provides a default class definition file TSCAN.PTC which is stored in the TerraScan installation folder. You can **Add**, **Edit**, and **Delete** point classes by using the corresponding buttons in the **Point classes** dialog.

You can create a **New** empty class definition file, **Open** an existing file, **Save** changes to an existing file, and **Save** class definitions **as** a new file by using the corresponding commands from the **File** pulldown menu of the **Point classes** dialog. The class definitions are saved into files with the default extension .PTC. Usually, there are different class definition files for different project types. If TerraScan is loaded, the last-used class definition file is still active.

To add a new class to the active class list:

1. Select the **Define Classes** tool.

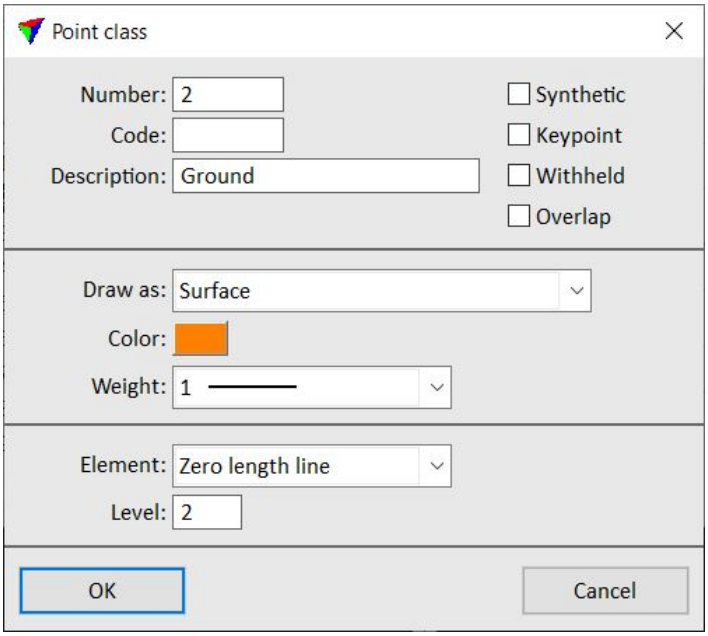
   The **Point classes** dialog opens:



The dialog contains a list of all point classes in the active class list.

2. Click the **Add** button.

   The **Point class** dialog box opens:



3. Define settings and click OK.

   The class is added to the active class list.

4. Select **Save** or **Save as** from the **File** pulldown menu in order to save the class definitions into a file.

SETTING	EFFECT
Number	Unique number of the point class.
Code	Code of the point class. The code is a text string which can include any kind of characters.

SETTING	EFFECT
Description	Descriptive name of the class.
Synthetic	If on, the classification bit for synthetic points is set in LAS 1.4 files.
Keypoint	If on, the classification bit for model keypoints is set in LAS 1.4 files.
Withheld	If on, the classification bit for withheld (deleted) points is set in LAS 1.4 files.
Overlap	If on, the classification bit for overlap points is set in LAS 1.4 files.
Draw as	Determines how points of a class are displayed in rendered views: • **Not set** - points are drawn normally. • **Thin object** - points are drawn in a thin-object way in rendered views. • **Surface** - points are drawn in a surface-like way in rendered views.
Color	Color for displaying points of the class. Uses the active color table of the CAD file.
Weight	Size for displaying points of the class. Uses the CAD file line weights.
Element	Element type used for drawing points of the class permanently into the CAD file.
Level	Level on which point of the class are drawn permanently into the CAD file.
Character	Character used for drawing points permanently into the CAD file. This is only active if **Element** is set to **Character**.
Font	Font type used for drawing points permanently into the CAD file. Uses the CAD file font types. This is only active if **Element** is set to **Character** or **Elevation text**.
Size	Size of the text used for drawing points permanently into the CAD file. This is only active if **Element** is set to **Character** or **Elevation text**.
Justify	Justification of the text relative to the original point. Uses the CAD file justification options for text elements. This is only active if **Element** is set to **Elevation text**.
Dx	Offset in X direction between the original point and the origin point of the text element. This is only active if **Element** is set to **Elevation text**.

SETTING	EFFECT
Dy	Offset in Y direction between the original point and the origin point of the text element. This is only active if **Element** is set to **Elevation text**.
Diameter	Diameter of a circle used for drawing points permanently into the CAD file. This is only active if **Element** is set to **Circle**.
Style	Style of the outline of a circle for drawing points permanently into the CAD file. Uses the CAD file line styles. This is only active if **Element** is set to **Circle**.

Since point class definitions are stored in a text file format, you can edit the file in a text editor as well. This may be useful if you need to copy classes and drawing rules from one class list to another.

Define Coordinate Setup

Define Coordinate Setup tool sets up coordinate system values that a Terra Application uses for laser points and images. It determines the coordinate range inside which all data must be located and the resolution to which coordinate values are rounded. The coordinate setup is stored into the active CAD file and is used by all Terra Applications.

Terra Applications use signed 32 bit integer values for storing coordinates of laser points and images. This has the advantage of using only 12 bytes of memory for the coordinate information of each point. You can control how accurately coordinate values are stored by defining how big each integer step is.

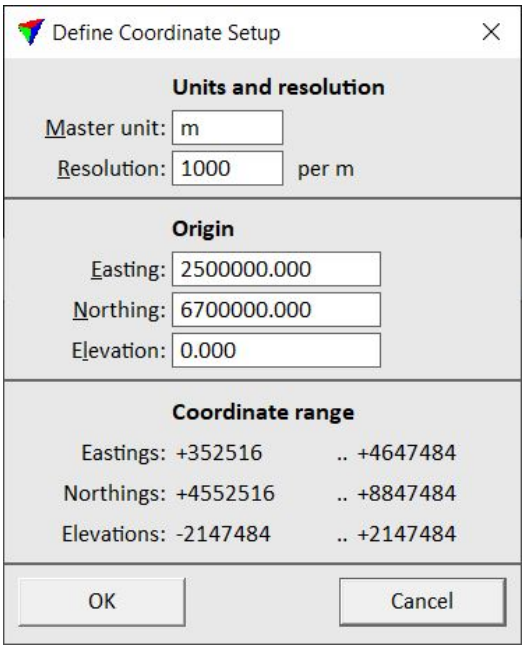
If, for example, one integer step is equal to one millimeter, all coordinate values are rounded to the closest millimeter. At the same time it would impose a limitation on how far apart points can be or how big the coordinate ranges are. Millimeter steps produce a coordinate cube which has a size of 2^{32} millimeters or 4294967.296 meters. If the origin of the coordinate system is at [0.0, 0.0, 0.0], the coordinate ranges are limited to values between -2147483 and +2147483. If necessary, you can fit the coordinate ranges to your data by modifying the Easting and Northing coordinates of the coordinate system origin.

If one integer step is equal to one centimeter, the coordinate values can range from -21 million to +21 million which is large enough for most coordinate systems.

To define the coordinate setup:

1. Select the **Define Coordinate Setup** tool.

   This opens the **Define Coordinate Setup** dialog:



2. Define settings and click OK.

   This modifies the coordinate system values used by all Terra Applications in the active CAD file.

Define Project

Not UAV

Define Project tool opens the TerraScan **Project** window. The window displays the active project and contains menu commands for handling TerraScan projects.

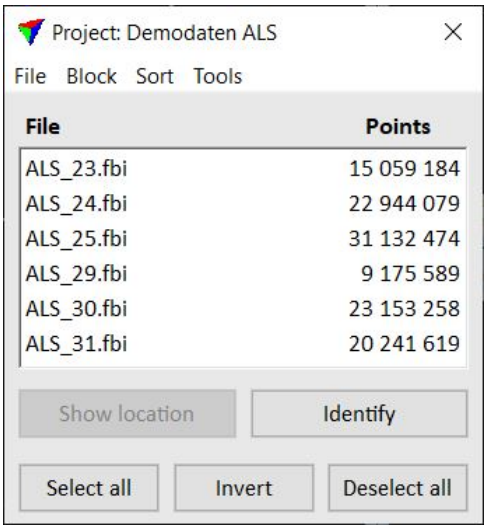
The main benefits of organizing point cloud data in a project are:

- The project definition divides a large data set into smaller parts which are easy to manage. Each part should contain an amount of points that can be loaded into memory and still allow processing of the points.
- When points are imported into a project, the application automatically divides the large point cloud into geographical regions (called 'blocks' in TerraScan's terminology). This is required because raw laser data is often provided in line order while some classification routines and other processing steps rely on geographical regions.
- You can run macros that process the data of all or selected blocks of a project. You can also start other processing routines from the Project window. This is essential for the automated processing of large point cloud data sets.
- TerraScan projects can be directly used in TerraStereo, Terrasolid's software for advanced visualization of huge point clouds in mono and stereo mode.

To view the active project:

1. Select the **Define Project** tool.

   The **Project** window opens:



If there is an active project in TerraScan, the title bar of the window displays the name of the project. Further, the window shows the list of blocks that belong to the project. For each block, the name and the amount of points in the file are displayed.

The menu commands of the **Project** window are described in detail in Chapter Working with Projects.

Design Block Boundaries

Not UAV

Design Block Boundaries tool creates shape elements that can be used as block boundaries for a TerraScan project. The block boundary creation can start from line or shape elements. If points are loaded in TerraScan, the tool can also compute the amount of points inside each block boundary.

The line elements used as starting elements for the tool should cross each other in order to create a closed line work for the shape creation. If shape elements are used as starting elements, the tool does not create new shapes. It only computes the amount of points inside the existing shapes.

The amount of points inside each block area is shown by text elements, which are drawn into the CAD file. The color of the label indicates whether the amount of points inside a block is within a given range. The point count is given in values rounded to million points. If no points are loaded in TerraScan, the tool ignores settings related to labels and point counts.

The tool supports significantly the creation of block boundaries for TerraScan projects, especially if the point density is varying in the project area. This is often the case in mobile mapping projects if the driving speed is not constant.

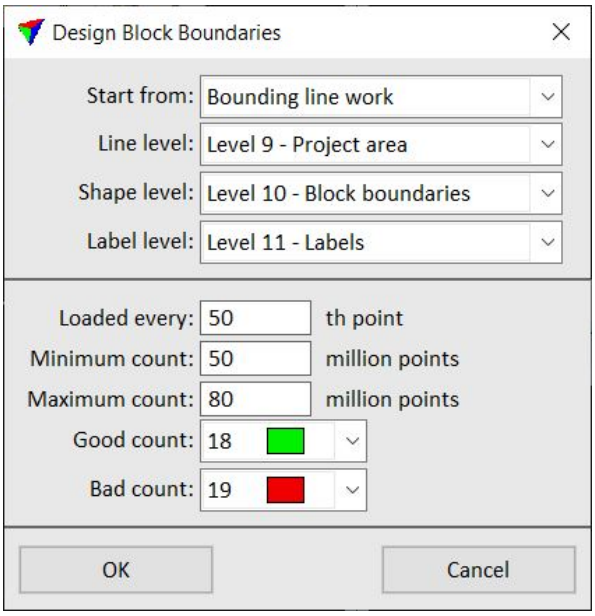
To design block boundaries:

1. (Optional) Load points from the whole project area into TerraScan using Read points command or Load Airborne Points tool. Load only a subset of points if the project area is too big to load all points.

2. Use CAD tools or TerraScan tools to digitize line elements around your project area and to separate the project area into smaller parts.

3. Select the **Design Block Boundaries** tool.

   This opens the **Design Block Boundaries** dialog:



4. Define settings and click OK.

   This creates shapes on the given **Shape level**. If points are loaded in TerraScan, the tool creates text elements on the given **Label level**.

5. If the amount of points per block is not within the given limits, modify the line work. Run the **Design Block Boundaries** tool again in order to update the shapes and labels. Continue until the point counts are within the limits.

6. Continue with the **Define Project** tool in order to add the shapes as block boundaries to a project.

SETTING	EFFECT
Start from	Elements that are used to create block boundaries: • **Bounding line work** - line elements that form closed areas. • **Shapes already drawn** - already existing shape elements.
Line level	CAD file level on which the line elements are drawn. This is only active if **Start** from is set to **Bounding line work**.
Shape level	CAD file level on which the shape elements are drawn. The shapes are created if **Start** from is set to **Bounding line work**.
Label level	CAD file level on which text elements are drawn. The texts show the amount of points inside a shape area if points are loaded in TerraScan.

SETTING	EFFECT
Load every	Indicates the subset of points that is loaded into TerraScan. This is ignored if no points are loaded.
Minimum count	Minimum amount of points accepted in one project block. Rounded to million points.
Maximum count	Maximum amount of points accepted in one project block. Rounded to million points.
Good count	Display color of a label for shape areas, where the amount of points is within the given **Minimum** and **Maximum count** values. Uses the active color table of the CAD file.
Bad count	Display color of a label for shape areas, where the amount of points is outside the given **Minimum** and **Maximum count** values. Uses the active color table of the CAD file.

Help on TerraScan

Help on TerraScan tool opens the online help in the standard web browser.

Load Airborne Points

Load Airborne Points tool performs exactly the same action as the Read points command in the **File** pulldown menu of the **TerraScan** window.

Load Ground Points

Load Ground Points tool is used to load laser points from a static ground-based scanner into TerraScan. The tool works in the same way as the Load Airborne Points tool and the Read points command.

The important difference is that the **Measurement** pulldown menu replaces the **Line** pulldown menu in the **TerraScan** window. The pulldown menu contains commands tailored for processing data of static ground-based laser scanners.

Manage Trajectories

Manage Trajectories tool opens the TerraScan **Trajectories** window. The window displays the active trajectories and contains menu commands for handling trajectory information in TerraScan.

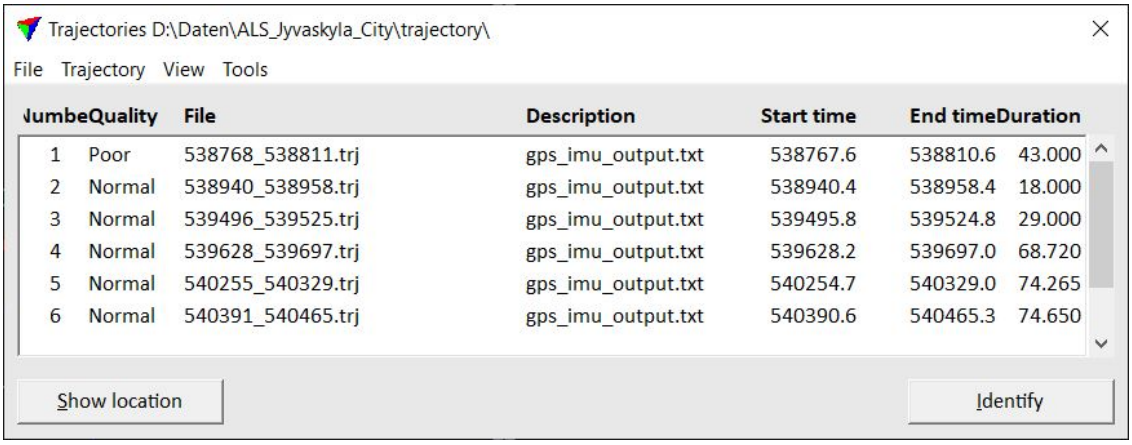
Trajectory information is required by the following processing steps:

- Cut overlap command for identifying points from overlapping lines.
- Adjust laser angles command for applying heading, roll, and pitch corrections to laser data.
- TerraMatch tools for fixing mismatch in laser data.

To view information about active trajectories:

1. Select the **Manage Trajectories** tool.

   This opens the **Trajectories** window:



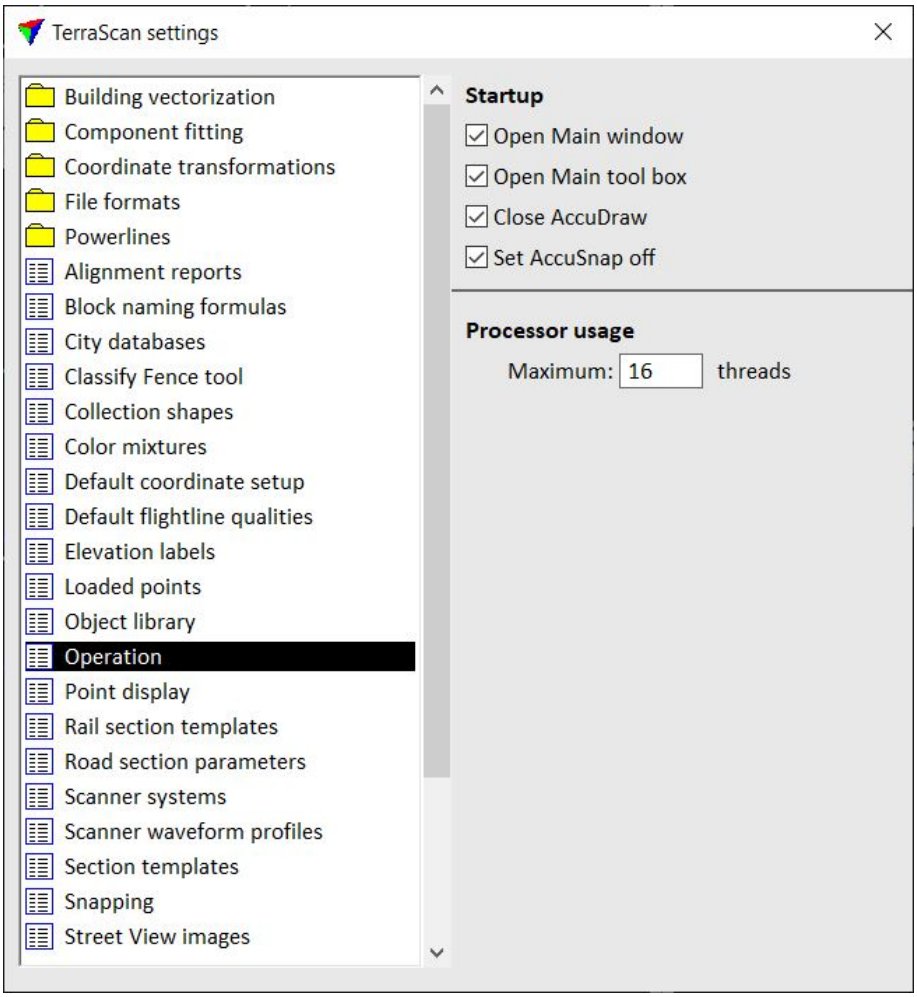
Number	Quality	File	Description	Start time	End time	Duration
1	Poor	538768_538811.trj	gps_imu_output.txt	538767.6	538810.6	43.000
2	Normal	538940_538958.trj	gps_imu_output.txt	538940.4	538958.4	18.000
3	Normal	539496_539525.trj	gps_imu_output.txt	539495.8	539524.8	29.000
4	Normal	539628_539697.trj	gps_imu_output.txt	539628.2	539697.0	68.720
5	Normal	540255_540329.trj	gps_imu_output.txt	540254.7	540329.0	74.265
6	Normal	540391_540465.trj	gps_imu_output.txt	540390.6	540465.3	74.650

If there are active trajectories in TerraScan, the title bar of the window displays the active trajectory folder. Further, the window shows the list of trajectory files that are stored in the active trajectory folder.

The menu commands of the **Trajectory** window are described in detail in Chapter Manage Trajectories.

Scan Settings

Scan Settings tool lets you change a number of settings that control the way how TerraScan works. Selecting this tool opens the **TerraScan settings** dialog:



The settings are grouped into logical categories. Selecting a category in the list displays the appropriate controls next to the category list.

The different categories and related settings are described in detail in Section TerraScan Settings.

Groups toolbox

Tools in the **Groups** toolbox are used to manipulate point groups manually. The tools require that points have been assigned to groups using the Assign groups command for loaded points or the Assign groups macro action.

TO	USE TOOL	
Create a new point group		Create Point Group
Merge two point groups into one group		Merge Point Groups
Add points to an existing group		Add Points To Group
Remove a group assignment from points inside a fence		Clear Group Fence

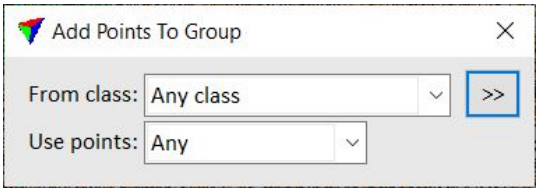
Add Points To Group

Add Points To Group tool adds points to an existing group. Points to be added are determined by a fence and optionally also specified by class.

To add points to a group:

1. Select **Add Points To Group** tool.

   This opens the **Add Points To Group** dialog:



2. Define settings.

3. Select the target group with a data click inside or close to the group.

4. Draw a fence around the points to add to the group by placing data clicks inside a few. The last data click must close the fence.

5. Confirm the fence contents with another data click inside the few.

   This adds the points inside the fence to the target group.

SETTING	EFFECT
From class	Point class(es) added to the target group.

SETTING	EFFECT
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **From class** field.
Use points	Determines which points are added to the target group: • **Any** - all points inside the fence. • **One group** - points of the one group from which the highest amount of the points is inside the fence. • **Non-grouped** - points without a group assignment.

In MicroStation, a fence can be drawn before the tool is started using the **Place fence** tool. Then, the fence element defines the fence area for the tool.

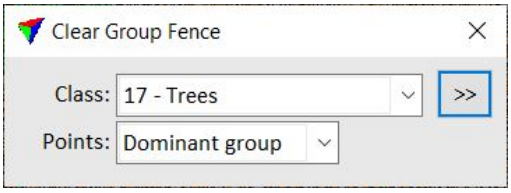
Clear Group Fence

Clear Group Fence tool clears the group assignment from points inside a fence.

To clear groups:

1. Select **Clear Group Fence** tool.

   This opens the **Clear Group Fence** dialog:



2. Define settings.

3. Draw a fence around the points to add to the group by placing data clicks inside a few. The last data click must close the fence.

4. Confirm the fence contents with another data click inside the few.

   This removes the group assignment from points inside the fence.

SETTING	EFFECT
Class	Point class(es) added to the target group.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can

SETTING	EFFECT
	select multiple source classes from the list that are then used in the **Class** field.
Points	Determines from which points the group is cleared: • **All** - all points inside the fence. • **Dominant group** - points of the one group from which the highest amount of the points is inside the fence.

In MicroStation, a fence can be drawn before the tool is started using the **Place fence** tool. Then, the fence element defines the fence area for the tool.

Create Point Group

Create Point Group tool assigns points to a group. The assignment can be done using two alternative methods:

- **Inside fence** - points inside a fence are assigned to one group.
- **Tree tip point** - the process starts from a user-defined tree tip point and assigns all points belonging to the same tree to one group.

The fence is drawn manually after starting the tool and defining the settings. The fence can be drawn in a top view or a section view.

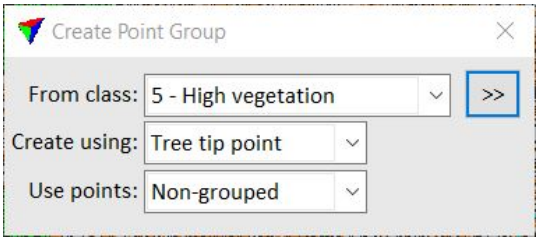
The tool expects that a distance from ground value is computed for the points in the source class for grouping. The value can be computed by using the Compute distance command for loaded points or the Compute distance macro action.

It is recommended to use the tool only for cases where the automatic group assignment failed. Points can be automatically assigned to groups by using the Assign groups command or Assign groups macro action.

To create a new point group:

1. Select the **Create Point Group** tool.

   This opens the **Create Point Group** dialog:



2. Define settings.

3. If **Create using** is set to **Tree tip point**, identify the top of a tree with a data click.

   This assigns all points found for the tree to a group

   OR

3. If **Create using** is set to **Inside fence**, draw the fence around points that you want to be assigned to one group. Close the fence by placing the last data click close to the first vertex of the fence. Confirm the fence with another data click.

   This assigns all points inside the fence to a group.

SETTING	EFFECT
From class	Point class(es) from which points are assigned to a group.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **From class** field.
Create using	Method used for group assignment: • **Inside fence** - points inside a fence are assigned to a group. • **Tree tip point** - a watershed algorithm is applied in order to assign points belonging to one tree to a group.
Use points	Determines which points create the new group: • **Any** - any point in the given source class(es). • **Current group** - points that are already assigned to a group. • **Non-grouped** - points that are not assigned to any group.

In MicroStation, a fence can be drawn before the tool is started using the **Place fence** tool. Then, the fence element defines the points that are add to a group and you only need to confirm the fence with a data click.

Merge Point Groups

Merge Point Groups tool can be used to manually merge two point groups into one group. Before using the tool, points must be assigned to groups by using the Assign groups command or Assign groups macro action.

To merge two point groups:

1. Select the **Merge Point Groups** tool.

2. Identify the first group with a data click. This group determines the number of the merged group.

3. Identify the second group with another data click.

   This merges the two groups.

Model toolbox

The tools in the **Model** toolbox are used to create an editable surface model, to classify laser points manually, to fix the elevation of points, and to update the surface model.

The tools for manual point classification effect points loaded in TerraScan memory. You have to save the points in order to store changes permanently into point files.

TO	USE TOOL	
To create an editable triangulated model		Create Editable Model
Assign class to a laser point		Assign Point Class
Classify points using a brush		Classify Using Brush
Classify points inside fence		Classify Fence
Classify points above line in section view		Classify Above Line
Classify points below line in section view		Classify Below Line
Classify points close to line in section view		Classify Close To Line
Add a synthetic point using mouse click		Add Synthetic Point
Classify vegetation points out of ground		Remove Vegetation
Set elevation of points inside polygon(s)		Fix elevation
Rebuild model after classification		Rebuild Model

Create Editable Model and **Rebuild Model** tools require TerraModeler to run.

Add Synthetic Point

Add Synthetic Point tool adds synthetic points to a point data set loaded in TerraScan. It may be used to create single points, points along selected vector elements or inside selected polygons.

If a single point is added in a top view, its xy location is determined by the data click position and its elevation is fixed to the active Z setting in the CAD file. If a point is added in a section view, its xy location is fixed to the center of the section and its elevation is set by the data click position.

Points along a vector element are created at the exact 3D position of the element and with a given spacing between the points.

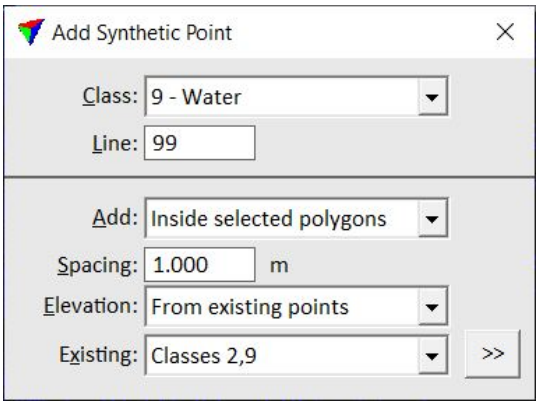
Points inside a polygon are created as grid with a given horizontal point distance. The elevation can be determined in different ways, such as a key-in value, the mouse pointer position, derived from other point classes or from the polygon elevation.

To add synthetic points:

1. Select vector elements or polygons if you want to add points along or inside them.

2. Select the **Add Synthetic Point** tool.

   The **Add Synthetic Point** dialog opens:



3. Define settings.

4. Place a single synthetic point with a data click in a view.

   This adds a new point to the loaded point data set. You can continue with step 2 if you want to change settings, or with step 3.

   OR

4. Confirm selected vector elements or polygons with a data click inside the CAD view. If **Elevation** is set to **From mouse click**, place a data click inside a section view in order to determine the elevation value for the new points.

   This adds the new points to the loaded point data set.

5. Use Save points or Save points As commands in order to save added points permanently into a laser point file.

SETTING	EFFECT
Class	Target class into which synthetic points are added. The list contains the active classes in TerraScan.
Line	Line number assigned to synthetic points.
Add	Method of adding new points: • **Single point** - a single point is added by a data click. • **Along selected vectors** - points are added along selected vector elements. • **Inside selected polygons** - points are added inside selected polygons.
Spacing	Distance between new points along an element or inside a polygon. This is only active if **Add** is set to **Along selected vectors** or **Inside selected polygons**.
Elevation	Method of how the elevation of new points inside a polygon is computed: • **From existing points** - points in given **Existing** class(es) are used. The **>>** button opens the **Select Classes** dialog which lets you select several classes. • **From polygons** - the average elevation of a polygon is used for the points falling inside the polygon. • **From mouse click** - a data click in a section view determines the elevation of the new points. • **As keyin value** - the given **Keyin Z** value is used. This is only active if **Add** is set to **Inside selected polygons**.

Assign Point Class

Assign Point Class tool classifies a single laser point or points that belong to a group of points. It classifies either the closest point to the data click, or the highest or lowest point within a circular search area. For group classification, the point selection method determines which group is classified.

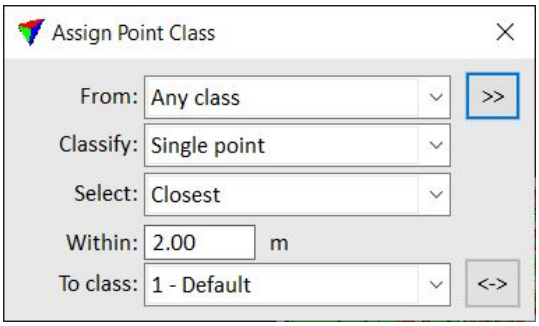
Classifying points of a group requires the assignment of group numbers to laser points. This can be done by using the Assign groups command for loaded points or the corresponding macro action for macro processing of project blocks.

The tool works in top views as well as in section views or any rotated views.

To classify a single point or a group of points:

1. Select the **Assign Point Class** tool.

   The **Assign Point Class** dialog opens:



2. Define settings.

3. Move the mouse pointer inside a view.

   In a top view, the search area is shown at the mouse pointer position.

4. Identify the single point or the group of points with a data click.

   This classifies the identified point or point group. You can continue with step 2 if you want to change settings, or with step 3.

5. Use Save points or Save points As commands in order to save changes to point classes permanently into a laser point file.

SETTING	EFFECT
From	Source class; only points from this class are effected. The list contains the active classes in TerraScan. Alternatively, **Any visible point** can be classified.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **From class** field.
Classify	Defines which points are classified: **Single point** or **Whole group**. **Whole group** is only active if group numbers are assigned to laser points.
Select	Method how the software selects a point or group for classification: • **Closest** - the point closest to the data click is classified. • **Highest** - the highest point within the search area is classified. • **Lowest** - the lowest point within the search area is classified.

SETTING	EFFECT
Within	Radius of the search area. Given in master units of the CAD file.
To class	Target class into which points are classified. The list contains the active classes in TerraScan.
<->	Switches **From** and **To class** classes. If **From class** is set to multiple classes or **Any visible point**, **To class** is switched to the source class with the lowest class number.

Classify Above Line

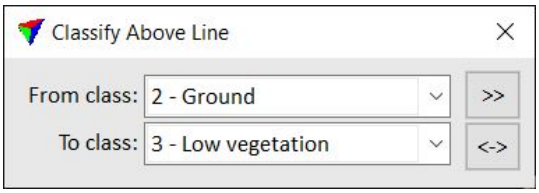
Classify Above Line tool classifies points above a line drawn in a cross section view. The classification effects only points that are inside the extend and display depth of the cross section view.

The tool works in section views.

To classify points above a line:

1. Select the **Classify Above Line** tool.

   The **Classify Above Line** dialog opens:



2. Define settings.

3. Draw a line by placing data clicks in a section view. The line and the area effected by the classification are temporarily displayed after placing the start point.

   This classifies points which are above the line, between the line's start and end point, and within the section view extend and depth. You can continue with step 2 if you want to change settings, or with step 3.

4. Use Save points or Save points As commands in order to save changes to point classes permanently into a laser point file.

SETTING	EFFECT
From class	Source class(es); only points from selected class(es) are effected. The list contains the active classes in TerraScan. Alternatively,

SETTING	EFFECT
	points from multiple classes or **Any visible point** can be classified.
To class	Target class into which points are classified. The list contains the active classes in TerraScan.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **From class** field.
<->	Switches **From** and **To class** classes. If **From class** is set to multiple classes or **Any visible point**, **To class** is switched to the source class with the lowest class number.

Classify Below Line

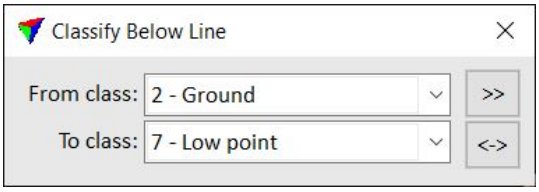
Classify Below Line tool classifies points below a line drawn in a cross section view. The classification effects only points that are inside the extend and display depth of the cross section view.

The tool works in section views.

To classify points below a line:

1. Select the **Classify Below Line** tool.

   The **Classify Below Line** dialog opens:



2. Define settings.

3. Draw a line by placing data clicks in a section view. The line and the area effected by the classification are temporarily displayed after placing the start point.

   This classifies points which are below the line, between the line's start and end point, and within the section view extend and depth. You can continue with step 2 if you want to change settings, or with step 3.

4. Use Save points or Save points As commands in order to save changes to point classes permanently into a laser point file.

SETTING	EFFECT
From class	Source class(es); only points from selected class(es) are effected. The list contains the active classes in TerraScan. Alternatively, points from multiple classes or **Any visible point** can be classified.
To class	Target class into which points are classified. The list contains the active classes in TerraScan.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **From class** field.
<->	Switches **From** and **To class** classes. If **From class** is set to multiple classes or **Any visible point**, **To class** is switched to the source class with the lowest class number.

Classify Close To Line

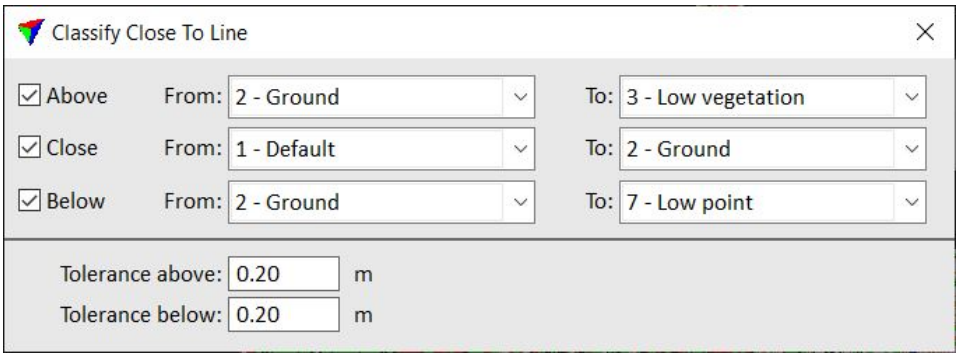
Classify Close To line tool classifies points that are close to a given line in a cross section view. It can combine up to three classification steps, above, close, and below a line. The classification effects only points that are inside the extend and display depth of the cross section view.

The tool works in section views.

To classify points close to lines:

1. Select the **Classify Close To Line** tool.

   The **Classify Close To Line** dialog opens:



2. Define settings.

3. Draw a line by placing data clicks in a section view. The line and the area effected by the classification are temporarily displayed after placing the start point.

This classifies points which are above the line, below the line, and/or close to the line, between the lines' start and end point, and within the section view extend and depth. You can continue with step 2 if you want to change settings, or with step 3.

4. Use Save points or Save points As commands in order to save changes to point classes permanently into a laser point file.

SETTING	EFFECT
Above	If on, above line classification is applied. Source and target classes are selected in the **From** and **To** lists of active classes. Alternatively, **Any visible point** can be selected in the **From** list.
Close	If on, close to line classification is applied. Source and target classes are selected in the **From** and **To** lists of active classes. Alternatively, **Any visible point** can be selected in the **From** list.
Below	If on, below line classification is applied. Source and target classes are selected in the **From** and **To** lists of active classes. Alternatively, **Any visible point** can be selected in the **From** list.
Tolerance above	Distance from the drawn line to the line that defines the above line classification limit. Together with the **Tolerance below** value, this defines the area of close to line classification.
Tolerance below	Distance from the drawn line to the line that defines the below line classification limit. Together with the **Tolerance above** value, this defines the area of close to line classification.

Classify Fence

Classify Fence tool classifies points inside a fence area.

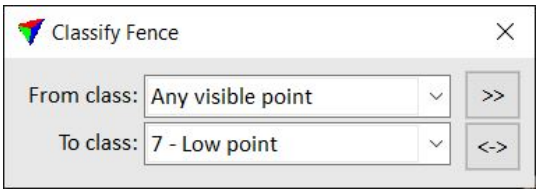
In MicroStation, the fence area can be defined by drawing a fence element before starting the tool.

The tool works in top views as well as in section views.

To classify points inside a fence:

1. Select the **Classify Fence** tool.

   The **Classify Fence** dialog opens:



SETTING	EFFECT
From class	Source class(es); only points from selected class(es) are effected. The list contains the active classes in TerraScan. Alternatively, points from multiple classes or **Any visible point** can be classified.
To class	Target class into which points are classified. The list contains the active classes in TerraScan.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **From class** field.
<->	Switches **From** and **To class** classes. If **From** is set to multiple classes or **Any visible point**, **To class** is switched to the source class with the lowest class number.

Classify Using Brush

Classify Using Brush tool classifies points inside a circular or rectangular brush moved in a CAD file view.

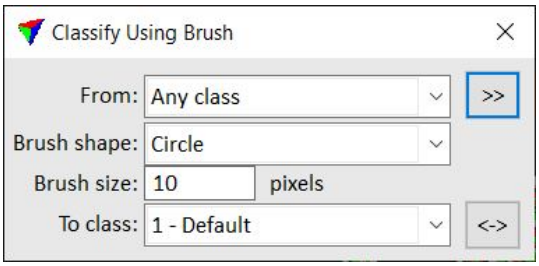
The tool can be utilized with two different kinds of mouse action. You can use it with two separate data clicks and mouse pointer movement in between, or you can keep the data button pressed down while moving the mouse pointer. A circle or rectangle around the mouse pointer indicates the brush of the given size. The brush shape is displayed brighter when the tool is active.

The tool works in top views as well as in section views.

To classify points inside a brush using two mouse clicks:

1. Select the **Classify Using Brush** tool.

   The **Classify Using Brush** dialog opens:



2. Define settings.

3. Place a data click to start the classification.

   OR

3. Press the data button down to start the classification.

   This classifies points inside the brush area.

4. Move the mouse pointer to classify additional points.

5. Place another data click to stop the classification.

   OR

5. Release the data button to stop the classification.

   This classifies all points touched by the mouse pointer. You can continue with step 2 if you want to change settings, or with step 3.

6. Use Save points or Save points As commands in order to save changes to point classes permanently into a laser point file.

SETTING	EFFECT
From	Source class; only points from this class are effected. The list contains the active classes in TerraScan. Alternatively, **Any visible point** can be classified.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **From** field.
Brush shape	Shape of the brush: **Circle** or **Rectangle**.
Brush size	Size of the brush. Given in pixels on the screen.
To class	Target class into which points are classified. The list contains the active classes in TerraScan.
<->	Switches **From** and **To class** classes. If **From** is set to **Any visible point**, **To class** is switched to the source class with the lowest class number.

Create Editable Model

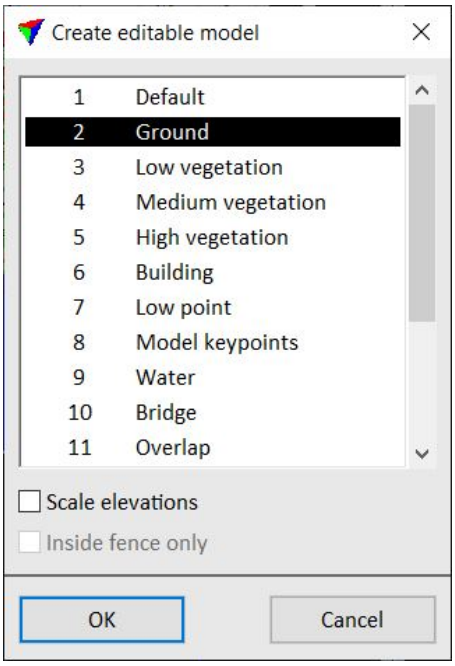
Create Editable Model tool creates a surface model from loaded laser points which can be visualized in TerraModeler. The tool starts TerraModeler automatically if the application is not yet running.

The surface model is actively linked to the loaded laser points, which means that all surface model displays are updated immediately to reflect any change in point classification. The active surface model display is particularly useful for validating ground classification. The best display method for this purpose is a shaded surface drawn by the Display Shaded Surface tool in TerraModeler's Display Surface toolbox.

To create an editable model and display a shaded surface:

1. Load laser data into TerraScan.
2. Select the **Create Editable Model** tool.

   This opens the **Create editable model** dialog:



3. Select class(es) which you want to include in the surface model.

4. (Optional) Switch on **Scale elevations** and type a factor by which you want to scale the elevations. A factor > 1.0 results in a model with exaggerated elevation values.

5. Click OK.

   This opens the **Surface settings** dialog in TerraModeler.

6. Enter a descriptive name for the new surface, define other settings if required, and click OK.

   TerraModeler creates the surface model.

7. Select the Display Shaded Surface tool in TerraModeler's Display Surface toolbox.

8. Define display settings and click OK.

   This displays the shaded surface using the elevation values of the laser points and given lightning conditions.

If TerraModeler is not available, you can use the Color by Shading display method of TerraScan to display a shaded surface visualization of the laser points based on class coloring.

Fix elevation

Fix Elevation tool fixes elevation values of points to a constant value. The tool processes data inside limited areas that can be defined by selected polygons, polygons on a given CAD file level, or a fence (MicroStation).

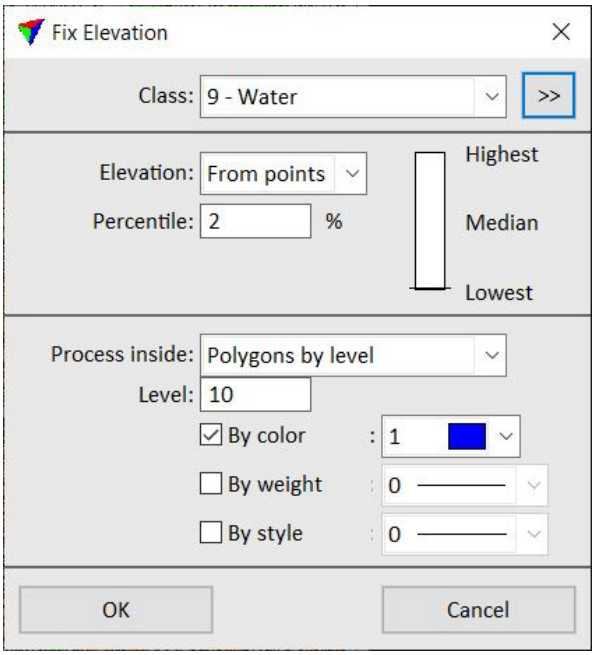
This tool may be useful, for example, to get a smooth surface from points on water.

To fix elevations of points:

1. Use CAD platform tools to draw polygons or a fence *(MicroStation only)* around the area(s) for processing. Select polygons, if required.

2. Select the **Fix Elevation** tool.

   The **Fix Elevation** dialog opens:



3. Define settings and click OK.

   This sets the elevation values of the laser points in the selected class(es) and inside the processing area to a constant elevation value.

4. Use Save points or Save points As commands in order to save changes to point classes permanently into a laser point file.

SETTING	EFFECT
Class	Source class(es); only points from selected class(es) are effected. The list contains the active classes in TerraScan. Alternatively, points from multiple classes or **Any class** can be selected.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Class** field.
Elevation	Method of calculating the constant elevation value: • **From points** - elevation derived from the given **Percentile** value of points. The scale

SETTING	EFFECT
	shows which percentile of points is used: **Lowest**, **Median**, or **Highest** points, and can be used to set the percentile value with a data click. • **Keyin value** - absolute elevation value given in the Value field.
Process inside	Determines the processing area: • **Fence** - points inside a fence are effected. *(MicroStation only)* • **Selected polygons** - points inside selected polygons are effected. • **Polygons by level** - points inside polygons drawn on the given CAD file **Level** are effected. The polygons can be further specified by selecting **By color**, **By weight**, and/or **By style** options. The selection lists use the active color table, line weights and line styles of the CAD file.

Rebuild Model

Rebuild Model tool runs a complete update of an editable surface model. It effects the surface model that has been created by the Create Editable Model tool. The process re-creates the TIN structure from all points in the model class(es) and updates all active displays of the surface model.

You should use this tool if you classify points to/from the surface model class(es) using tools other than those in the **Model** toolbox or if the automatic update of the editable surface model does not work correctly.

To rebuild a model:

1. Select the **Rebuild Model** tool.

   If an editable model is available, the re-building process starts. A progress bar shows the progress of the process.

   If no editable model is available, an information dialog is shown.

Remove Vegetation

Remove Vegetation tool removes points in vegetation from points classified as ground. It works within a limited area defined by a selected polygon or MicroStation fence.

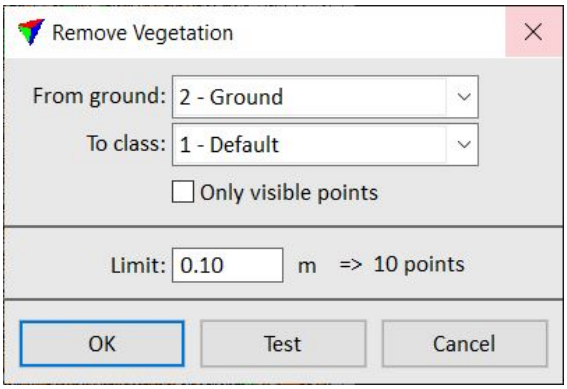
The can be used, for example, to smooth the ground surface after automatic ground classification.

To remove vegetation from ground points:

1. Use CAD tools to draw a polygon or fence *(MicroStation only)* around the area for processing. Select the polygon.

2. Select the **Remove Vegetation** tool.

   The **Remove Vegetation** dialog opens:



3. Define settings.

4. Click on the **Test** button to see the result of the settings in a preview.

5. Click OK to apply the classification.

   This classifies the points from the ground class to the target class.

6. Use Save points or Save points As commands in order to save changes to point classes permanently into a laser point file.

SETTING	EFFECT
From ground	Source class, usually a ground class; only points from the selected class is effected. The list contains the active classes in TerraScan.
To class	Target class into which points are classified. The list contains the active classes in TerraScan.
Only visible points	If on, only points that are visible in the marked processing area are effected. If off, all points in the marked processing area are effected, even if their display is switched off.
Limit	Distance up to which points above a surface defined by the lowest ground points are classified. Given in the master unit of the CAD file. The amount of effected points is shown next to the input field.

Powerlines

TerraScan has a number of tools which are dedicated to powerline processing. These include classification, vectorization, and reporting tools.

Tools for powerline processing are divided into three parts:

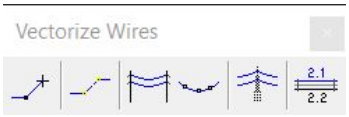
- Vectorize Wires toolbox - tools for automatic detection and manual placement of wires, check and correction of wire attachments as well as assigning wire attributes.
- Vectorize Towers toolbox - tools for manual placement of tower models, manipulation of towers, cross arms, and attachments.
- View Powerline toolbox - tools for labeling towers, finding danger objects, and creating reports.

A general strategy for processing powerline data can be outlined as follows. The final workflow depends on the project requirements.

1. Classify the point cloud in a way that points on powerline structures (towers, wires) stand out. This may involve classification of points
   - on the ground using the Ground routine
   - above the ground using the By distance routine
   - relative to the centerline of the powerline using the By centerline routine
   - with echo type “first of many” using the By echo routine
2. Create a tower string using the Place Tower String tool or the Draw as line strings command.
3. Detect wires automatically using the Detect Wires tool.
4. Manually place wires using the Place Catenary String tool in places where the automatic detection does not work.
5. Validate and adjust catenary attachment points using the Check Catenary Attachments tool.
6. Assign wire attributes using the Assign Wire Attributes tool.
7. Define tower models in the Powerlines / Tower types category of TerraScan **Settings**.
8. Manually place towers using the Place Tower tool.
9. Create labels with the help of labeling tools in the View Powerline toolbox.
10. Detect and analyze danger objects close to the powerline using the Find Danger Objects tool or the Wire danger points macro action.
11. Create output reports with tools in the View Powerline toolbox.

Vectorize Wires toolbox

The tools in the **Vectorize Wires** toolbox are used to place a powerline centerline, to detect wires automatically, to manually place wire strings, to validate wire attachment points and to assign attributes to wires.



TO	USE TOOL	
Place a line string from tower to tower		Place Tower String
Activate a powerline for viewing and modification		Activate Powerline
Detect wires along active powerline		Detect Wires
Digitize a wire line string		Place Wire String
Check wire attachment points at tower locations		Check Wire Attachments
Assign number and description to wire		Assign Wire Attributes

Activate Powerline

Activate Powerline tool activates a tower string element for further processing steps.

Most of the powerline processing tools, such as detection of wires, check of catenary attachments, placement of towers as well as labeling and report tools, are applied based on the activated tower string.

To activate a powerline:

1. Select the **Activate Powerline** tool.
2. Click on the tower string element that you want to activate.

   OR

1. Select the tower string element you want to activate.
2. Select the **Activate Powerline** tool.

This activates the selected tower string. The tower string is displayed as defined in the Powerlines / Active line category of TerraScan **Settings**. An information dialog shows the number of the activated powerline.

To deactivate a powerline:

1. Select the **Activate Powerline** tool.

   This deactivates any activated tower string element. You can continue with activating another tower string element or with selecting any other tool.

Assign Wire Attributes

Not Lite

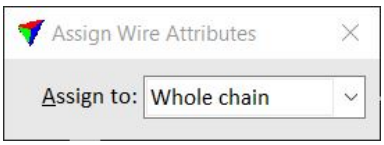
Assign Wire Attributes tool lets you define attributes for wire strings. You can also change the symbology of the line elements. Changes can be applied for the wire along the whole tower string, within a certain span range, or for a single wire string from one tower to another. The attributes can be included in reports created with the Output Catenary tool.

To assign attributes to catenary strings:

1. Activate a tower string element using Activate Powerline tool.

2. Select the **Assign Wire Attributes** tool.

   This opens the **Assign Wire Attributes** dialog:

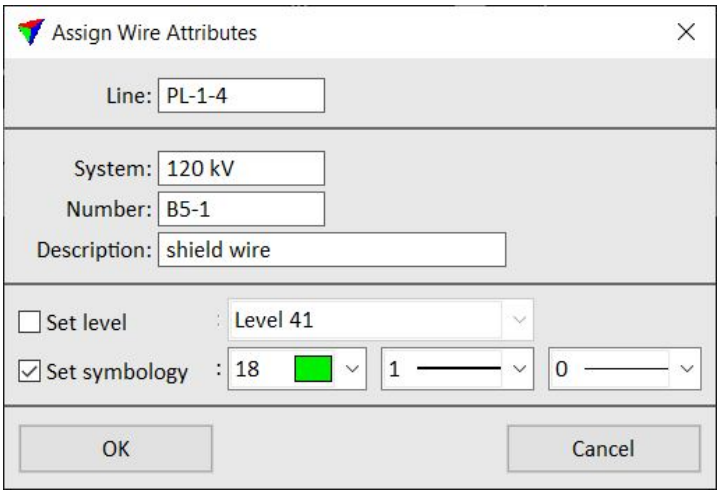


3. Select, for what to assign attributes: **Whole chain**, **Span range**, or **Single span**.

4. If **Assign** to is set to **Span range**, select the start tower and the end tower of the span range with data clicks.

5. Select a wire string for which to assign attributes.

   This highlights the selected wire string and opens another **Assign Wire Attributes** dialog:



6. Define settings and click OK.

   This assigns the given attributes to the wire. You can continue with steps 3 or 5.

SETTING	EFFECT
Line	Number of the powerline. This is filled automatically from the activated tower string.
System	Text field for entering a free system identifier.
Number	Text field for entering a free wire number.
Description	Text field for entering a free description for the wire.
Set level	If on, the selected wire is moved to the given CAD file level.
Set symbology	If on, the given **Color**, **Weight** and line **Style** is applied for the selected wire. The selection lists include the standard colors, line weights and styles of the CAD file.

Check Wire Attachments

Not Lite

Check Wire Attachments tool validates and adjusts end points of wire strings. At tower locations, the incoming and the outgoing wire strings do not meet exactly because each wire string has been computed using laser points from one tower-to-tower span only. The magnitude of the gap between wire strings gives some indication of how accurately the wires have been detected or placed.

Check Wire Attachments tool produces a list of all tower locations as they are defined by the tower string. For each tower location, the size of the gaps at the wire attachment points are computed in XY and Z. In addition, a shift of the tower positions is computed. A shift unequal to 0 indicates a horizontal shift off from the tower center in direction of the tower string (longitudinal shift). In this case, all incoming wire strings are lower or higher than all outgoing wire strings.

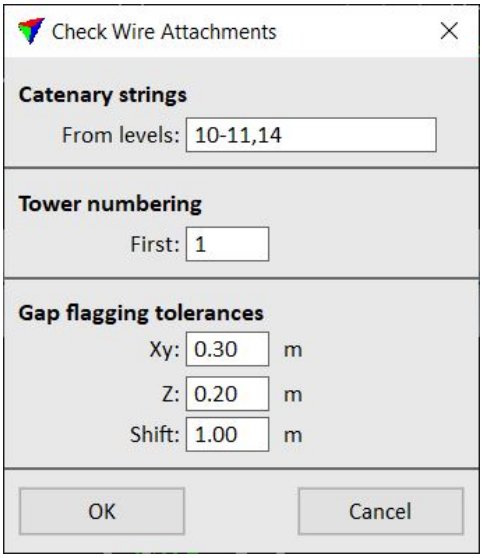
The window displaying the list contains pulldown menus with commands for fixing the gaps automatically or manually.

To validate wire attachment points:

1. Activate a tower string element using the Activate Powerline tool.

2. Select the **Check Wire Attachments** tool.

   This opens the **Check Wire Attachments** dialog:



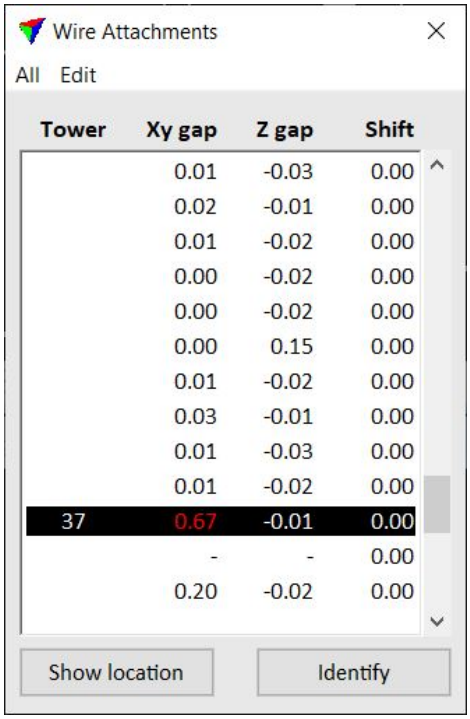
3. Define settings and click OK.

   The application searches for wire strings on the given levels and computes gaps between the incoming and the outgoing wires. After the process finished, it opens the Wire Attachments window.

SETTING	EFFECT
From levels	CAD file level(s) from which to search catenaries. Define a range of level numbers by using a minus sign, separate different level numbers by comma.
First	Number of the tower at the first vertex of the activated tower string. The numbering is only relevant for the list in the **Wire Attachments** window.
Xy	Xy gap flagging limit. Gaps exceeding this value are displayed with red color in the list.
Z	Z gap flagging limit. Gaps exceeding this value are displayed with red color in the list.
Shift	Tower shift flagging limit. Shifts exceeding this value are displayed with red color in the list.

Wire Attachments window

The **Wire Attachment** window displays the list of wire attachment points for each tower location.



The columns in the window are: tower location number, size of XY gap, size of Z gap, horizontal shift of the tower position. The values are given in 1/100 of the master unit of the CAD file. A - sign for the gaps indicates an open end of a wire string. This is acceptable if a wire starts or ends at a tower. Otherwise, it may indicate very big gaps between incoming and outgoing wire strings.

To show the location of an attachment point, select a line in the window. Click on the **Show location** button and move the mouse pointer into a view. This highlights the selected location with a cross. Place a data click inside a view in order to center the selected location in the view.

To identify a point, click on the **Identify** button and place a data click close to a point in a view. This selects the corresponding line in the list of attachment points.

It is recommended to check at least bigger gaps and fix them manually. The commands from the **Edit** pulldown menu provide tools for moving and connecting wire end points in XY and/or Z. Small gaps can be fixed automatically with commands from the **All** pulldown menu. In a similar way, shifts of tower positions can be fixed. The manual work of fixing wire attachment points can be supported by laser data loaded in TerraScan. This helps to move wire end points to correct locations.

You can undo actions of the tools by using the **Undo** command of the CAD platform.

TO	USE COMMAND
Shift all tower positions	All / Shift all
Adjust all attachment points to the average	All / Adjust all

TO	USE COMMAND
Recompute gaps and shifts	All / Update list
View statistics about gaps	All / Statistics
Shift a single tower position	Edit / Shift tower
Set two connecting wire end points to a given xy location	Edit / Set attachment xy
Adjust a single tower position to the average	Edit / Adjust attachment
Manually enter the location of a wire end point	Edit / Move wire end

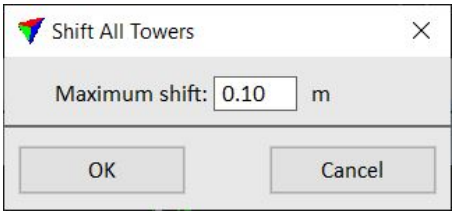
All / Shift all

Shift all command preforms the same action as the Edit / Shift tower command but for all tower positions of the activated tower string. The command should be used if the shift values are small and tower position are only moved within an acceptable distance.

To shift all towers:

1. Select **Shift all** command from the **All** pulldown menu.

   This opens the **Shift All Towers** dialog:



2. Define a **Maximum shift** distance and click OK.

   This moves the vertices of the tower string representing the tower position and the wire attachment points for all positions for which a shift correction applies. If the computed shift of a tower position exceeds the **Maximum shift** value, the additional shift remains. The gap values in the **Wire Attachments** window are updated automatically.

   An information dialog shows the number of shifted tower positions.

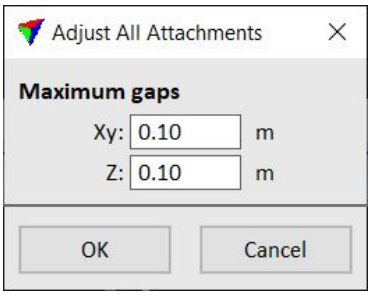
All / Adjust all

Adjust all command performs the same action as the Edit / Adjust attachment command but for all attachment points. The command should be used if the gap values are small and the attachment points are only moved within an acceptable distance.

To adjust all attachment points:

1. Select **Adjust all** command from the **All** pulldown menu.

   This opens the **Adjust All Attachments** dialog:



2. Define maximum values for **Xy** and **Z** gaps to be corrected and click OK.

   This adjusts all attachment points for which both, the horizontal gap and the vertical gap are within the defined limits. The end point of the incoming wire string and the start point of the outgoing wire string are adjusted to the average XYZ position between the two. The gap values in the **Wire Attachments** window are updated automatically.

All / Update list

Update list command recomputes the gap and shift values for wire strings and updates the list. The command should be used if

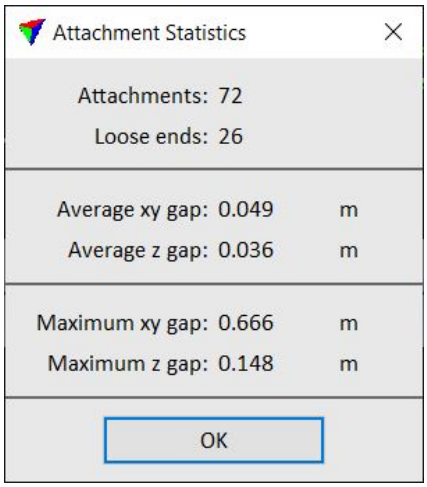
- a wire string has been deleted or added.
- an action has been undone using the **Undo** command of the CAD platform.

All / Statistics

Statistics command displays statistical information about wire attachment points. This includes:

- **Attachments** - number of wire attachment points.
- **Loose ends** - number of wire end points that do not have an attachment to another wire string. This is the case at the start and end of a tower string because there are no incoming/outgoing wire. If there are more loose ends, it indicates missing wire strings due to problems in wire detection, or too big gaps between incoming and outgoing wire strings at an attachment point.
- **Average xy gap** - average size of horizontal gaps at attachment points.
- **Average z gap** - average size of elevation gaps at attachment points.
- **Maximum xy gap** - size of the largest horizontal gap.
- **Maximum z gap** - size of the largest elevation gap.

The values may be helpful to decide whether manual work for fixing gaps is required or not. In the end, when all gaps are closed, all values should be 0.



Edit / Shift tower

Shift tower command shifts a tower position along tower string direction (longitudinal shift). In can be used for places where all incoming wires are higher/lower than corresponding outgoing wires. Shifting the tower position moves all attachment points towards the previous/next tower and thus, lowers/raises the attachment points of incoming /outgoing wires.

To shift a single tower position:

1. Select **Shift tower** command from the **Edit** pulldown menu.

2. Move the mouse pointer inside a view.

   The tower closest to the mouse pointer is highlighted by a red or green square. The square shows the target location of the tower position shift. The color indicates whether the shift can be performed (green) or not (red).

3. Confirm the new position with a data click.

   This moves the vertex of the tower string representing the tower position and the wire attachment points. The gap values in the **Wire Attachments** window are updated automatically.

Edit / Set attachment xy

Set attachment xy command moves a single attachment point to a given XY location. At the same time, it moves the end/start points of the incoming and outgoing wire strings to the same XY location. The elevation of the wire end/start points stays unaffected.

To set xy location of an attachment point:

1. Select **Set attachment xy** command from the **Edit** pulldown menu.

2. Move the mouse pointer inside a top view.

   The attachment point closest to the mouse pointer is highlighted by a green square. The square shows the target location of the attachment point.

3. Select the attachment point with a data click.

4. Define the XY location with a data click.

   This moves the wire end points to the XY position of the data click. It does not modify the wire constants or the elevation of the wire end points. The gap values in the **Wire Attachments** window are updated automatically.

   You can continue with step 3.

Edit / Adjust attachment

Adjust attachment command adjust a single attachment point to the average of the incoming and the outgoing wires' end/start points. It effects both, the XY and Z position of the attachment point.

To adjust a single attachment:

1. Select **Adjust attachment** command from the **Edit** pulldown menu.

2. Move the mouse pointer inside a top or section view.

   The attachment point closest to the mouse pointer is highlighted by a green square. The square shows the target location of the attachment point. The color indicates whether the adjustment can be performed (green) or not (red).

3. Select the attachment point with a data click.

   This moves the end point of the incoming wire string and the start point of the outgoing wire string to the new location. The gap values in the **Wire Attachments** window are updated automatically.

   You can continue with step 3.

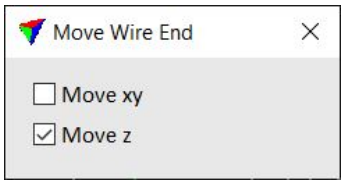
Edit / Move wire end

Move wire end command lets you move a wire start/end point to a new location. The tool should be used if one of the wires matches laser points better than the other one. You can either move the XY position, the Z position, or both at the same time. CAD platform snapping functionality can be used to snap start/end points to each other.

To move a wire end or start point:

1. Choose **Move wire end** command from the **Edit** pulldown menu.

   This opens the **Move Wire End** dialog:



2. Select the position(s) you want to modify.

3. Move the mouse pointer inside a top or section view.

   The wire start/end point closest to the mouse pointer is highlighted by a green square.

4. Select the start or end point with a data click.

5. Define the new location of the start/end point with another data click.

   This recomputes and redraws the wire string. The gap values in the **Wire Attachments** window are updated automatically.

   You can continue with step 4.

Detect Wires

Not Lite

Detect Wires tool is used to vectorize wires and classify points on wires of a powerline. It draws line string elements in the CAD file for all detected wires. The tool runs on points loaded in TerraScan.

The tool searches points along a catenary curve. Catenary curves are mathematical descriptions of wires that are connected at their end points but hanging free between these end points. The process involves least squares fitting for both, the xy line equation and the elevation curve equation of the catenary.

The most important parameter controlling wire detection is **Max gap** which defines the maximum gap between consecutive laser points on a wire. It is not advisable to run the detection on the whole data set with a large maximum gap value because the chance of false detections increases. It is recommended to run the detection first with a relatively small maximum gap value which does not necessarily detect all wires. For locations with fewer points on the wires, the detection should be done for single segments with a smaller value.

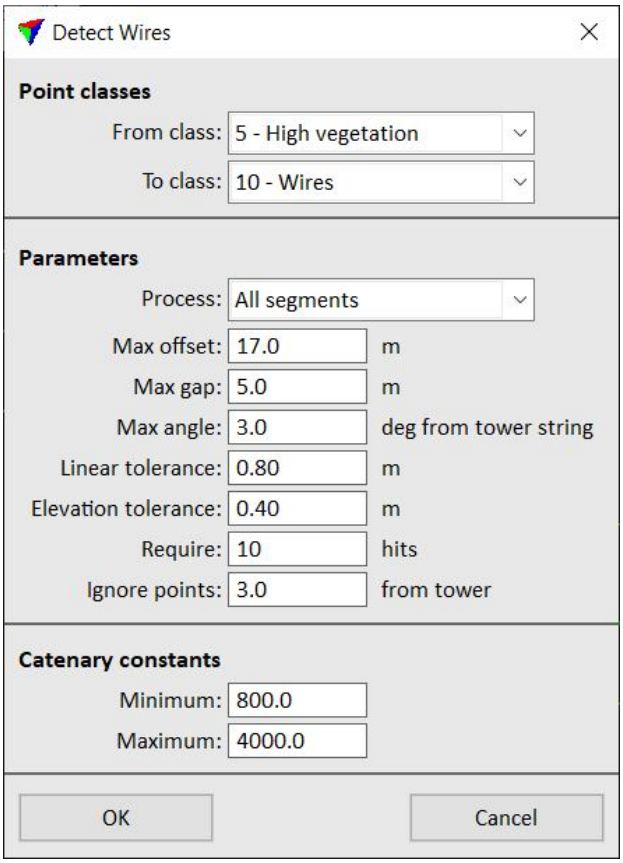
The **Detect Wires tool** requires a tower string element that is has been created, for example, with the Place Tower String tool. The tower string must be activated. The line string elements for detected wires are drawn on the active level using the active symbology settings of the CAD file.

To detect wires:

1. Activate a tower string element using the Activate Powerline tool.

2. Select the **Detect Wires** tool.

   This opens the **Detect Wires** dialog:



3. Define settings and click OK.

   If **Process** is set to **All segments**, the application starts to detect wires.

4. If **Process** is set to **Single segment**, move the mouse pointer inside a top view. The segment closest to the mouse pointer is highlighted.

5. Select the segment to process with a data click.

   This starts the detection of wires for the selected segment.

   A progress window shows the progress of the process. Depending on the data set, the process may take some time, especially if there is vegetation close to the wires.

SETTING	EFFECT
From class	Point class from which wires are detected.
To class	Target class into which points on wires are classified.
Process	Determines where wires are detected: • **All segments** - for all segments for which data is loaded in TerraScan. • **Single segment** - only for the selected tower-to-tower segment.
Max offset	Maximum distance from the tower string element to the left/right side to detect wires.

SETTING	EFFECT
	Defines the corridor in which the software searches for wires.
Max gap	Maximum allowed gap between consecutive points on a wire.
Max angle	Maximum allowed angle between a wire and the tower string element.
Linear tolerance	Tolerance for XY line fitting and classification of points on a wire.
Elevation tolerance	Tolerance for elevation curve fitting and classification of points on a wire.
Require	Minimum amount of laser points on a single wire required for detection. Values can range from 3 to 999.
Ignore points	Distance from tower within which points are ignored for wire detection. Points close to the tower can be from tower structures and should be ignored when determining the mathematical shape of the wire.
Minimum	Minimum catenary constant to accept a wire.
Maximum	Maximum catenary constant to accept a wire.

The vectorization of the wires can be undone by using the **Undo** command of the CAD platform. The classification of the wires can be undone by using the Undo command of TerraScan.

Place Tower String

Not Lite

Place Tower String tool is used to manually digitize a centerline of a powerline. The tool produces a line string element. The tower string must have vertices only at tower locations. The vertices should be located at the center point of a tower as accurate as possible. The tower string can be digitized based on laser data and/or rectified images that show the tower locations.

This tower string is used in later processing steps when detecting wires, validating catenary attachment points, placing towers, and producing reports.

Place Tower String tool integrates line string placement and view panning in one tool which speeds up the digitization process. The tower string is treated as 2D line element by other powerline processing tools. Therefore, the elevation of tower string vertices does not play a role.

In general, any line string or complex line string element can be used as a tower string. If tower positions are provided in a text file, the tower string can be generated automatically by using the

Draw as line strings command. In this case, the **Place tower string** tool is only used to assign a line number to the tower string element.

To prepare for tower string placement:

1. Classify the point cloud in a way that tower locations and wires stand out. This may include steps like
   - classify by echo type, classify first of many echoes into a separate class
   - classify by centerline using an approximate centerline element of the powerline
   - classify ground and above-ground points

   It depends on your workflow and project requirements whether the classification is applied to project blocks or only temporarily to loaded points. For placing a tower string, you should load a point cloud that is dense enough for visual tower detection and cover as much as possible from the powerline.

2. Choose a view used as **Top view** for tower string digitization. Optionally, choose a second view used as **Profile view** where a profile of the powerline is displayed while digitizing the tower string.

3. Set up the display of laser points in the Display mode dialog. The class(es) with points on towers/wires should be switched on, all other classes off. Set **Color by** to **Elevation**.

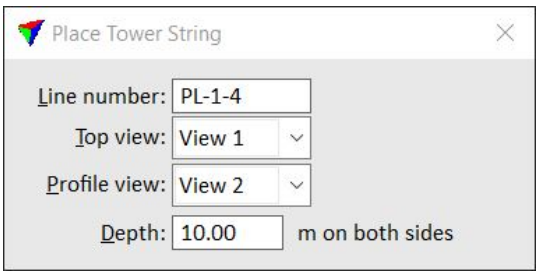
4. To emphasize the depth perception in the top view, set the **Borders** setting in the Display mode dialog to a value > 0%. This improves the visibility of towers and wires within their environment. You may also force the software to draw all points on the screen by setting **Speed** to **Normal** or **Slow**.

5. Select CAD file symbology settings to draw the tower string. Set an empty level as active level.

To place a tower string:

1. Select the **Place Tower String** tool.

   This opens the **Place Tower String** dialog:



2. Define settings.

SETTING	EFFECT
Line number	Defines a number or name for the tower string. Each tower string should have a unique number in order to clearly identify a powerline.
Top view	Number of the view used as top view. This view is primarily used for digitizing the tower string.

SETTING	EFFECT
Profile view	An optional view which displays a profile along the powerline. This is useful to see tower locations more clearly. The profile view can also be used to place vertices for the tower string.
Depth	Depth of the profile view. This should include the whole width of a powerline.

3. Enter a vertex at the center of the first tower.

   This defines the location of the first tower. The dynamic rectangle is displayed whenever the mouse pointer is inside a view. If you place a data click outside the rectangle, the application pans the view in the direction of the click. Placing a data click inside the rectangle adds a new vertex to the tower string.

4. Pan the view to the next tower location.

5. Place a vertex at the center of the tower.

6. Continue with steps 4 and 5 until the last tower of the powerline.

7. Press the reset button to finish the tower string.

   It is recommended to check the tower string for missing or unnecessary vertices. It is essential for automatic wire detection and tower model placement, that the tower string has vertices only at tower locations and that no tower is missing a vertex.

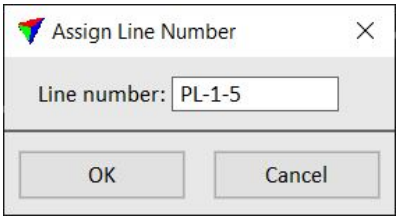
   It may happen that a tower string can not be placed along the complete powerline in one operation. In this case, continue placing another tower string that runs in the same direction and starts at the exact end point of the previous tower string. After placing several tower strings along one powerline, join them with the **Create Complex Chain** tool (MicroStation) or Construct Big Element tool (Spatix).

To assign a line number to an existing tower string element:

1. Select the line element that you want to use as tower string.

2. Select the **Place Tower String** tool.

   This opens the **Assign Line Number** dialog:



3. Type a number or text string in the **Line number** field.

4. Click OK.

This assigns the number to the tower string element.

Place Wire String

Not Lite

Place Wire String tool lets you manually place a line string for a single wire (catenary string) between two towers. You can use this tool in places where the automatic detection of wires fails.

You define the shape of the wire curve with three mouse clicks. Optionally, the start and end point of the line string are defined by mouse clicks as well. Start and end points only effect the length of the wire curve but not its shape.

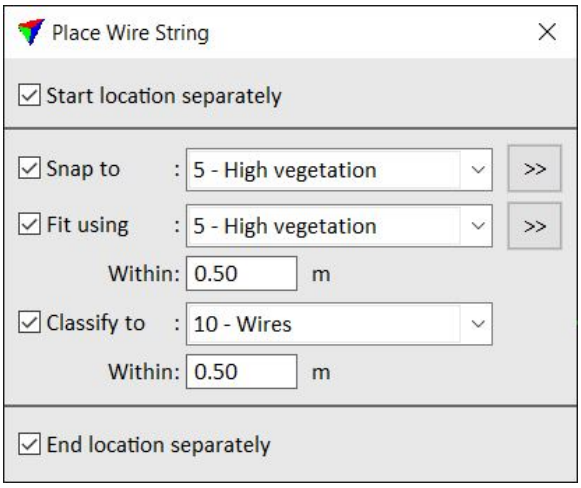
The line string element for the wire is drawn on the active level using the active symbology settings of the CAD file.

To place a wire string manually:

1. Use Draw Vertical Section tool to create a longitudinal section along the wire from tower to tower.

2. Select the **Place Wire String** tool.

   This opens the **Place Wire String** dialog:



3. Define settings.

SETTING	EFFECT
Start location separately	If on, the first data click defines the start point of the catenary string.
Snap to	If on, the three curvature points are snapped to the closest laser points in a given class. This lock is normally on.
>>	Opens the **Select classes** dialog which contains the list of active classes in

SETTING	EFFECT
	TerraScan. You can select multiple source classes from the list that are then used in the **Snap to** field.
Fit using	If on, the wire string is fitted to laser points in the given class within the given tolerance.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Fit using** field.
Classify to	If on, laser points within the given radius around the wire string are classified into the given class.
End location separately	If on, the last data click defines the end location of the wire string.

4. (Optional) Define the start point of the wire string with a data click.

5. Define the first, second, and third curvature point with data clicks.

6. (Optional) Define the end point of the wire string with a data click.

   The application draws the wire string defined by the given points temporarily into the CAD file.

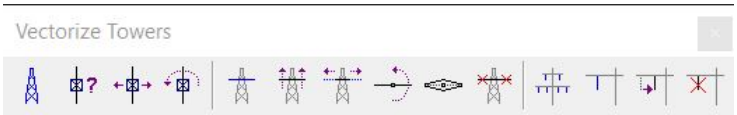
7. Accept the wire string with another data click.

   The wire string is drawn permanently into the CAD file and laser points are classified into the given target class, if the corresponding setting is switched on.

The placement of a wire string can be undone by using the **Undo** command of the CAD platform. The classification of the points along the wire string can be undone by using the Undo command of TerraScan.

Vectorize Towers toolbox

The tools in the **Vectorize Towers** toolbox are used to place towers and to manipulate towers, cross arms, and attachments.



TO	USE TOOL	
Place a tower		Place Tower
Edit tower attributes		Edit Tower Information
Move a tower to another location		Move Tower
Rotate a tower around its base point		Rotate Tower
Add a cross arm to a tower		Add Cross Arm
Set the height of a cross arm		Set Cross Arm Elevation
Set the length of a cross arm		Extend Cross Arm
Rotate a cross arm around the tower		Rotate Cross Arm
Modify a cross arm		Modify Cross Arm
Delete a cross arm		Delete Cross Arm
Create attachments automatically		Create Attachments
Add an attachment manually to a cross arm		Add Attachment
Move an attachment along the cross arm		Move Attachment
Delete an attachment		Delete Attachment

At the moment, cells and thus, the tools of the **Vectorize Towers** toolbox do only work in MicroStation. There is not yet any corresponding element type in Spatix.

Add Attachment

Not Lite, Not Spatix

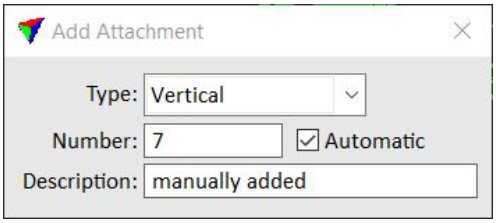
Add Attachment tool lets you add an attachment manually to a cross arm. Besides the options offered for automatic creation of attachments there are two more possibilities:

- **Side slope** - creates a sloped line in left or right direction relative to the tower string element.
- **Dual point** - connects two wire strings that are not joined.

To add an attachment to a cross arm:

1. Select the **Add Attachment** tool.

This opens the **Add Attachment** dialog:



2. Define settings.

SETTING	EFFECT
Type	Direction of the attachment: **Vertical**, **Side slope**, **Slope 3D**, or **Dual point**.
Number	Number of the attachment.
Automatic	If on, the number is set automatically based on the already existing number of attachments for the tower.
Description	Text field for defining a description for the attachment.

3. Select a cross arm for which to add an attachment.

4. Identify the wire end point, connection point, or the end point of the first catenary element with a data click.

   This finishes the creation of a **Vertical** attachment.

5. Define the location on the cross arm where the attachment is placed with a data click.

   This finishes the creation of a **Side slope** or a **Slope 3D** attachment.

6. Identify the end point of the second catenary element to connect it with the attachment.

   This finishes the creation of a **Dual point** attachment.

Add Cross Arm

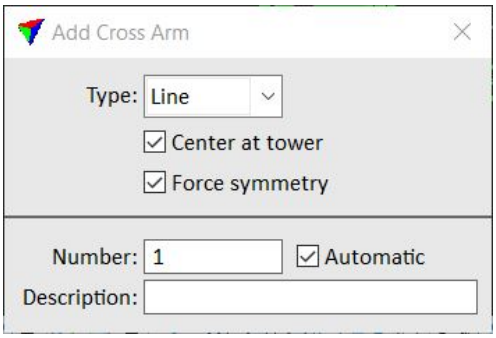
Not Lite, Not Spatix

Add Cross Arm tool lets you add a cross arm to a tower. The cross arm can be added as simple line or as shape. This is a way to create more complex cross arms than can be defined in Powerlines / Tower types category of TerraScan **Settings**.

To add a cross arm to a tower:

1. Select the **Add Cross Arm** tool.

   This opens the **Add Cross Arm** dialog:



2. Define settings.

SETTING	EFFECT
Type	Element type of the cross arm: **Line** or **Shape**.
Center at tower	If on, a cross arm is centered at the tower center point.
Force symmetry	If on, a symmetric cross arm is forced. The shape of a cross arm on the one side of the tower is the same as the shape on the other side of the tower.
Number	Number of the cross arm.
Automatic	If on, the number is set automatically based on the already existing number of cross arms for the tower.
Description	Text field for defining a description for the cross arm.

3. Select a tower for which to add a cross arm with a data click.

4. Define the height of the cross arm in a section view.

   A line is shown dynamically at the mouse location to indicate the location of the cross arm.

5. Define the first vertex of the cross arm with a data click.

6. Define the second vertex of the cross arm with a data click.

   This finishes the definition of a cross arm of the type **Line**.

7. Define additional vertices for a cross arm of the type **Shape**.

   When the mouse comes close to the first vertex, it snaps to the vertex in order to close the shape. If **Force symmetry** is switched on, an even amount of shape vertices is required.

8. Define the last vertex and close the shape.

   This finishes the creation of a cross arm of the type **Shape**.

When digitizing a cross arm, vertices can be added in a top view as well as in a section view. If placed in a top view, the elevation of the vertex is set by the cross arm height. If entered in a section view, the XY location of the vertex is defined by the centerline of the current section.

Create Attachments

Not Lite, Not Spatix

Create Attachments tool creates attachments for a tower. Attachments are line elements that connect wires at their end or connection points with the cross arms of a tower.

Before the creation of attachments, the vectorization of wires, the tower and its cross arms must be finished. Also, the location and rotation of towers and cross arms should be correct.

The attachment lines are created automatically if a connection from a wire end or connection point to a cross arm can be created. The connection can be:

- **Vertical** - vertical line. The location of the wire end or connection point is moved to the attachment's end point to enforce a vertical connection line to the tower arm.
- **Slope 3D** - sloped line in forward or backward direction along the tower string. Connects the wire end or connection points on their original location with the cross arm.

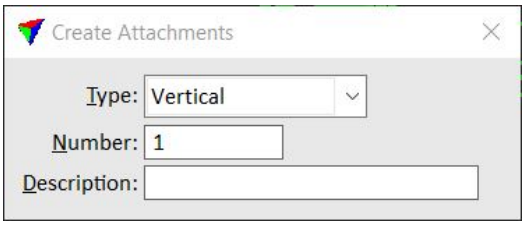
The automatic creation of attachments fails if the software can not create a linear connection in vertical or forward/backward direction between a wire end or connection point and a cross arm.

To create attachments for a tower:

1. Activate a tower string element using Activate Powerline tool.

2. Select the **Create Attachments** tool.

   This opens the **Create Attachments** dialog:



3. Define settings.

SETTING	EFFECT
Type	Direction of the attachment: **Vertical** or **Slope 3D**.
Number	Number of the first attachment for the tower. Further attachments of the same tower are numbered increasingly.

SETTING	EFFECT
Description	Text field for defining a description for all attachments of the tower.

4. Select a tower for which to create attachments with a data click.

5. Confirm the attachments with another data click.

   This creates the attachment lines as part of the tower model. The creation can be rejected by placing a reset click after step 4.

Delete Attachment

Not Lite, Not Spatix

Delete Attachment tool deletes an attachment.

The removes the connection between a wire end or connection point and a tower cross arm.

To delete an attachment:

1. Select the **Delete Attachment** tool.

2. Select the attachment to be deleted with a data click close to it.

3. Accept the removal with another data click.

Delete Cross Arm

Not Lite, Not Spatix

Delete Cross Arm tool deletes a cross arm.

If attachments have been created for a cross arm, they are deleted as well.

To delete a cross arm:

1. Select the **Delete Cross Arm** tool.

2. Select the cross arm to be deleted with a data click.

3. Accept the deleted cross arm with another data click.

Edit Tower Information

Not Lite, Not Spatix

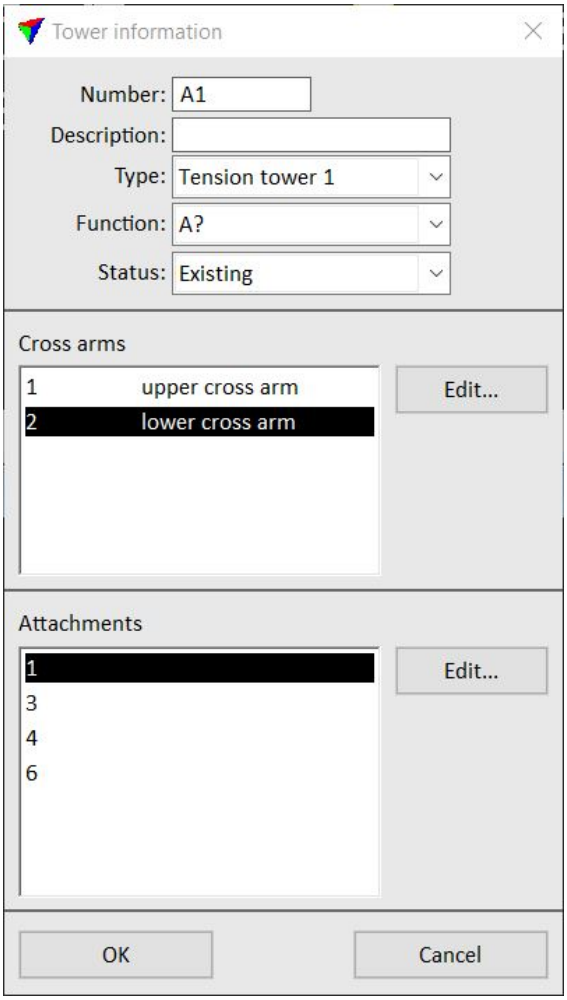
Edit Tower Information tool lets you edit the attributes of a tower. This includes information about tower number, description, type, status, and function as well as cross arm and attachment attributes.

To edit tower information:

1. Select the **Edit Tower Information** tool.

2. Select a tower for which to edit the attributes with a data click.

   This opens the **Tower information** dialog:



3. Modify tower attributes.

4. To edit the attributes of cross arms, select the line of the cross arm and click the **Edit** button.

5. To edit the attributes for attachments, select the lines of a cross arm and of the attachment and click the **Edit** button.

6. Click OK to apply the new attributes.

SETTING	EFFECT
Number	Tower number.
Description	Text field for entering a description of the tower.
Type	Type of the tower as defined in Powerlines / Tower types category of TerraScan **Settings** in the **Description** field.
Function	Function of the tower defined in Powerlines / Tower functions category of TerraScan **Settings**.
Status	Status of the tower defined in Powerlines / Tower statuses category of TerraScan **Settings**.
Cross arms	Cross arms defined for the tower. Click **Edit** to change cross arm **Number** and **Description**.
Attachments	Attachments defined for the selected cross arm. Click **Edit** to change attachment **Number** and **Description**.

Extend Cross Arm

Not Lite, Not Spatix

Extend Cross Arm tool lets you change the length of a cross arm either on both sides simultaneously or only on one side.

To extend a cross arm:

1. Select the **Extend Cross Arm** tool.

   This opens the **Extend Cross Arm** dialog:

2. Select whether to extend **Both sides** or **One side** in the **Extend** field.

3. Select the cross arm to be extended with a data click near the end point.

   The extend of the cross arm is dynamically displayed as the mouse pointer is moved.

4. Define a new end point for the cross arm with another data click.

Modify Cross Arm

Not Lite, Not Spatix

Modify Cross Arm tool modifies a cross arm by moving single vertices of the line or shape. Modification can be made to either the elevation, the XY location, or both.

To modify a cross arm:

1. Select the **Modify Cross Arm** tool.

   This opens the **Modify Cross Arm** dialog:

2. Select whether to change the **Elevation**, the **Xy position**, or the **Xyz position** of a vertex.

3. Select a vertex with a data click close to it.

   The vertex position is dynamically displayed as the mouse pointer is moved.

4. Define the new position for the vertex with a data click.

Move Attachment

Not Lite, Not Spatix

Move Attachment tool moves an attachment along the cross arm.

The tool is not yet implemented in the software.

Move Tower

Not Lite, Not Spatix

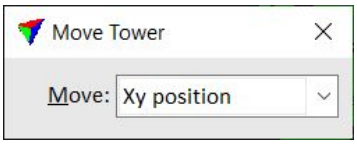
Move Tower tool lets you change the location or the height of a tower.

The tool works with and without an active tower string element. If the tower string element is activated, the tool moves the vertex of the tower string and the tower model if one is placed at the location of the vertex. It also moves attached wire elements if attachments have been placed and the tower string element has been activated. If the tower string element is not activated, only the tower model is moved.

To move a tower:

1. (Optional) Activate a tower string element using Activate Powerline tool.

2. Select the **Move Tower** tool.



3. Define what to move for the tower.

SETTING	EFFECT
Move	Determines what is moved for a tower: • **Xy position** - horizontal position. Can be moved in a top or section view of the tower. • **Xyz position** - horizontal and vertical position. If moved in a top view, the elevation of the tower is set to the active depth of the view. If moved in a section view, the XY position is determined by the centerline of the section. • **Base elevation** - base point elevation. Move this in a section view of the tower. Only the height of the tower is modified, not its location. • **Top elevation** - top point elevation. Move this in a section view of the tower. Only the height of the tower is modified, not its location.

4. Identify the tower with a data click.

   The tower model or the tower base/top point is dynamically displayed at the mouse pointer location.

5. Define the new location of the tower or the tower base/top point with a data click.

Place Tower

Not Lite, Not Spatix

Place Tower tool lets you place a tower model. The shape of the tower has to be defined in Powerlines / Tower types category of TerraScan **Settings**. A tower model is drawn as MicroStation cell element into the CAD file.

Towers are placed manually based on the tower type definition, an activated tower string element, and laser points. Tower placement can be supported by view arrangement and display options, for example by using the View Tower Spans tool. A typical setup may include views for displaying the span and the tower in (rotated) top views, the tower and the tower tip in cross section views.

The placement of towers with the **Place Tower** tool is straightforward and includes placing the base point, the tip point, and the end point(s) of the cross arm(s). The XY location of a tower is defined by the vertex of the tower string element. Classified laser points loaded in TerraScan can be used to fix the tower's base point elevation to the ground.

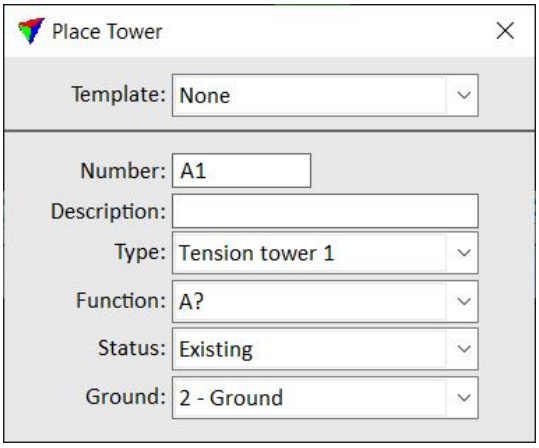
In addition to the simple tower type models that are defined in the TerraScan **Settings**, a more complex tower template can be created. Template creation may start from an already vectorized, simple tower model. Then, tools from the **Vectorize Towers** toolbox can be used to add, for example, more complex cross arms to the tower cell element. See Creating a tower template for more information about how to define a tower template.

To place a tower model:

1. Activate a tower string element using Activate Powerline tool.

2. Select the **Place Tower** tool.

   This opens the **Place Tower** dialog:



3. Define settings.

SETTING	EFFECT
Template	Use of a template for tower placement: • **None** - no template is used. • **Identify** - the tower model that is selected with the next data click is set as active template. • **Active** - tower models are placed using the active template.
Number	Tower number. The number may be preceded with alphanumerical characters. This can be used in reports created with the Export Powerline tool.
Auto increase	If on, tower numbers increase automatically while placing towers.
Description	Text field for typing a description for the tower. This can be used in reports created with the Export Powerline tool.
Type	Type of the tower. Defined in Powerlines / Tower types category of TerraScan **Settings** in

SETTING	EFFECT
	the **Description** field.
Function	Function of the tower. Defined in Powerlines / Tower functions category of TerraScan **Settings** in the **Description** field.
Status	Status of the tower. Defined in Powerlines / Tower statuses category of TerraScan **Settings** in the **Description** field.
Ground	Point class used to derive the tower's base point elevation. This is only used if the base point is placed in a top view.

4. Move the mouse pointer into a view.

   A line is displayed at the location of the tower defined by a vertex of the tower string. In a section view, the base point can be moved in elevation.

5. Define the base point of the tower with a data click.

   If the base point is defined in the top view, the elevation is derived from the laser points according to the Ground setting in the tool's dialog. In a section view, the elevation is defined by the data click.

6. Move the mouse pointer in a section view, preferable a view showing the tip of the tower.

   A dynamic preview of the tower center is displayed.

7. Define the tip of the tower with a data click in the section view.

   A dynamic preview of the first cross arm is displayed.

8. Place the end point of the first cross arm with a data click in the section view.

9. If the tower model defines additional cross arms, repeat step 8 for all cross arms.

   After placing the end point of the last cross arm, the placement of the tower is finished. If **Auto increase** is switched on in the **Place Tower** dialog, the number is increased.

   Use the list of the View Tower Spans tool to display the next tower. You may define new settings for the next tower in the tool's dialog and continue with step 4.

While placing a tower model, you can undo step-by-step with reset clicks.

Creating a tower template

A tower template can be created to place towers with a more complex shape as it can be defined in TerraScan Settings. The template defines the position, length and shape of the cross arms of the tower.

To create and place a tower template:

1. Place one simple tower model using Place Tower tool.
2. Modify the model using Add Cross Arm tool or other **Vectorize Towers** tools.
3. Select the Place Tower tool.
4. Set **Template** to **Identify**.
5. Select the tower model with a data click.

   This sets the selected tower model as active template.

   You can continue with placing the first tower using the template. Follow the steps 4-7 described above for placing the tower's base point and tip.
6. The position and length of the cross arms is defined by the template. Confirm the placement of the tower model with another data click in the section view.

Rotate Cross Arm

Not Lite, Not Spatix

Rotate Cross Arm tool rotates a cross arm around the center point of the tower.

The tool also moves attached wires if attachments have been placed and the tower string element is activated.

To rotate a cross arm:

1. (Optional) Activate a tower string element using Activate Powerline tool.
2. Select the **Rotate Cross Arm** tool.
3. Select the cross arm to be rotated with a data click.

   The cross arm direction is dynamically displayed as the mouse pointer is moved.
4. Define the new direction for the cross arm with a data click.

Rotate Cross Arm tool should not be used after attachments have been placed to prevent an incorrect replacement of attachments and wires.

Rotate Tower

Not Lite, Not Spatix

Rotate Tower tool lets you change the horizontal direction of the tower.

The tool works with and without an active tower string element. It also moves attached wire elements if attachments have been placed and the tower string element has been activated.

To rotate a tower:

1. (Optional) Activate a tower string element using Activate Powerline tool.
2. Select the **Rotate Tower** tool.
3. Identify the tower to be rotated with a data click.

   The tower model is dynamically displayed at the mouse pointer location.
4. Define the new direction of the tower with a data click.

Set Cross Arm Elevation

Not Lite, Not Spatix

Set Cross Arm Elevation tool lets you change the height of a cross arm.

To set the elevation for a cross arm:

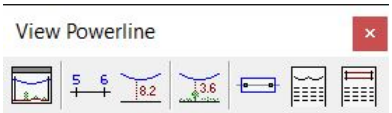
1. Select the **Set Cross Arm Elevation** tool.
2. Select a cross arm for which to set a new height with a data click.

   The cross arm location is dynamically displayed at the mouse pointer location.
3. Define the new height with a data click in a section view.

   This places the cross arm at the new height.

View Powerline toolbox

The tools in **View Powerline** toolbox are used to view tower spans, to label towers and heights of catenaries, to find danger objects and create reports of danger object locations, to create span tiles and to output catenaries and towers into text or files.



TO	USE TOOL	
View tower spans as profiles and cross sections		View Tower Spans
Label tower string with tower numbers		Label Towers
Label height from ground to catenary		Label Catenary Height
Find points close to the catenaries		Find Danger Objects
Create tiles rectangles for powerline spans		Create Span Tiles
Output catenary coordinates to text files		Output Catenary
Export powerline information to text file		Export Powerline

Create Span Tiles

Create Span Tiles tool creates rectangular shape elements for each span. These rectangles can be used, for example, as tiles for creating orthophotos with TerraPhoto where each resulting image covers one tower-to-tower distance.

There are two different types of tile elements that can be produced:

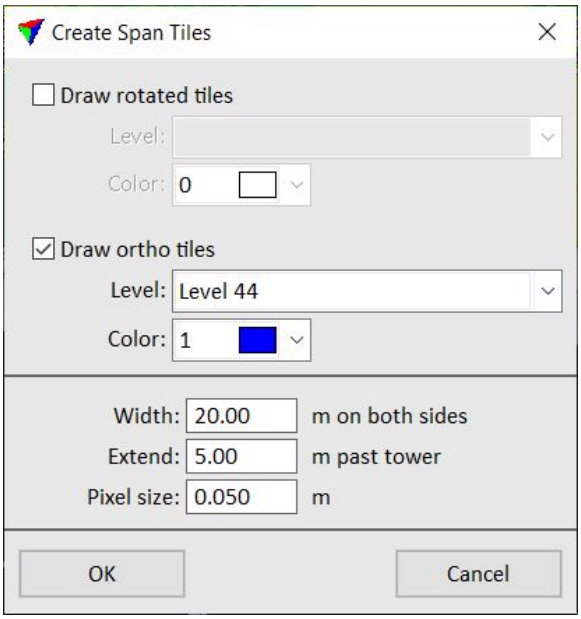
- **Rotated tiles** - the long rectangle sides are parallel to the tower string.
- **Ortho tiles** - tiles are drawn as orthogonal bounding box around a span.

The shape elements are drawn using the active line weight and style settings of the CAD file.

To create span tiles:

1. Activate the tower string element using the Activate Powerline tool.
2. Select the **Create Span Tiles** tool.

   This opens the **Create Span Tiles** dialog:



3. Define settings and click OK.

   This draws the tiles as shape elements into the CAD file.

SETTING	EFFECT
Draw rotated tiles	If on, rotated tiles parallel to the tower string are drawn on the defined **Level** using the selected **Color**.
Draw ortho tiles	If on, orthogonal tiles are drawn around each span on the defined **Level** with the selected **Color**.
Width	Width of the tile measured from a tower string vertex to the tile boundary. For orthogonal tiles, this is the minimum distance.
Extend	Distance by which a span tile is extended beyond the start and end tower of the span.
Pixel size	Intended pixel size to be used for orthophoto creation. See TerraPhoto User Guide for more information.

Export Powerline

Export Powerline tool creates a report for the towers that belong to an active tower string element. The output of the tool is a structured, delimited text file that lists all attributes of a tower selected in the tool's first dialog.

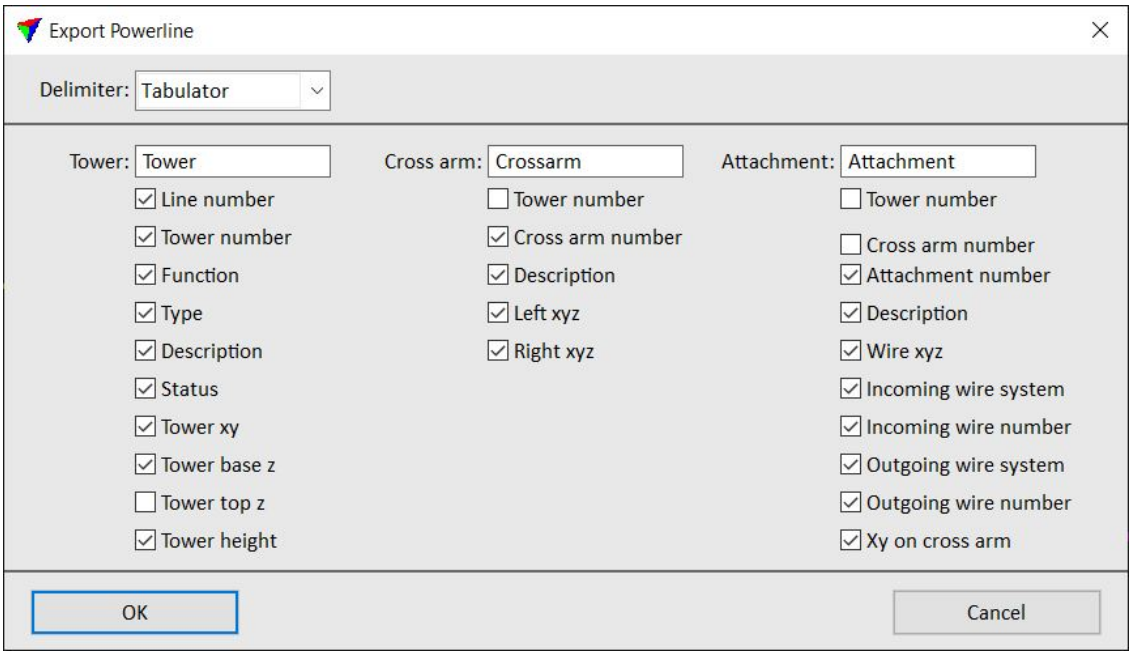
The tool requires that tower models have been placed using the Place Tower tool.

To output towers as text file:

1. Activate the tower string element using the Activate Powerline tool.

2. Select the **Export Powerline** tool.

   This opens the **Export Powerline** dialog:



3. Select a **Delimiter** used to separate columns in the text file.

4. Type texts in the **Tower**, **Cross arm** and **Attachment** fields. The texts are used in the report.

5. Select attributes to be included in the report.

6. Click OK.

   This opens the **Powerline export file** dialog, a standard Windows dialog for saving a file.

7. Define a name and location for storing the output file. Add a suitable extension to the file name, such as .TXT or .CSV for opening the file in a text or spreadsheet application.

8. Click **Save**.

   This creates the report. An information dialog informs about the success of the action.

The following figure illustrates the structure of the report:

Tower	**1**	**1 A**	**T1**			**Bestand**	**10321.12**	**735921.11**	**441.12**	**477.34**	**36.22**
Crossarm	***1***		***10321.04***	***735926.33***	***474.88***	***10321.2***	***735915.9***	***474.88***			
Attachment	2		10321.04	735926.29	472.41					10321.04	735926.29
Attachment	5		10321.2	735916.07	472.44					10321.2	735916.07
Crossarm	***2***		***10321.03***	***735927.48***	***469.48***	***10321.22***	***735914.75***	***469.48***			
Attachment	1		10321.03	735927.35	466.93					10321.03	735927.35
Attachment	6		10321.22	735915.01	466.97					10321.22	735915.01
Crossarm	***3***		***10321.04***	***735926.49***	***464***	***10321.21***	***735915.74***	***464***			
Attachment	3		10321.04	735926.27	461.46					10321.04	735926.27
Attachment	4		10321.2	735916.07	461.47					10321.2	735916.07
Tower	**1**	**3 A**	**T1**			**Bestand**	**10673**	**735926.52**	**436.97**	**473.06**	**36.09**
Crossarm			***10672.92***	***735931.99***	***470.66***	***10673.08***	***735921.04***	***470.66***			
Attachment	3		10672.93	735931.48	468.09					10672.93	735931.48

The first row shows the tower attributes, followed by the first cross arm attributes in the next row. Then, the attributes of the attachments belonging to the first cross arm are listed. This is done for all cross arms of the first tower. In the same way, all other towers and tower parts are listed in the report file.

Find Danger Objects

Find Danger Objects tool finds points which are within a given distance from vectorized wires. In addition, it lets you create reports in order to document danger point locations.

There are three different methods how the distance between wires and the danger objects can be defined:

- **Vertical distance to wire** - points within a vertical distance from the wire and within a given offset from the tower string.
- **3D distance to wire** - points within a 3D radius around a wire.
- **Falling tree logic** - each point in the source class is considered as the tip of a tree with its trunk at the xy location of this point and the elevation of the base point on the ground. If the point “falls like a tree” and falls into a 3D buffer area around a wire, it is listed as a danger point. The method requires a ground classification.

The tool produces a list of potential danger points. The list is traversable, you can scroll through the list and check each location.

A general workflow example for reporting danger object locations along a powerline is:

1. Detect wires by following the workflow for powerline processing as suggested, for example, at the beginning of the Powerlines chapter.

2. Start the **Find Danger Objects** tool. Use any class with points on potential danger objects as **Object class**. Report danger points **By minimum interval** to get at least one point from each potential danger object.

3. Check the list of danger points. Use the **Classify** button to classify points on danger objects into a separate class. Use the **Remove** button to remove points from the list if they are not on danger objects (such as noise close to the wires).

4. Save the point cloud after classifying danger points.

5. Close the **Danger Objects** dialog.

6. Start the **Find Danger Objects** tool again. Use the point class with points on danger objects as **Object class**. Set **Report** to **All** to get a list of all danger points.

7. Setup the view display in a way that suites for creating HTML reports. For creating text reports, no view setup is required.

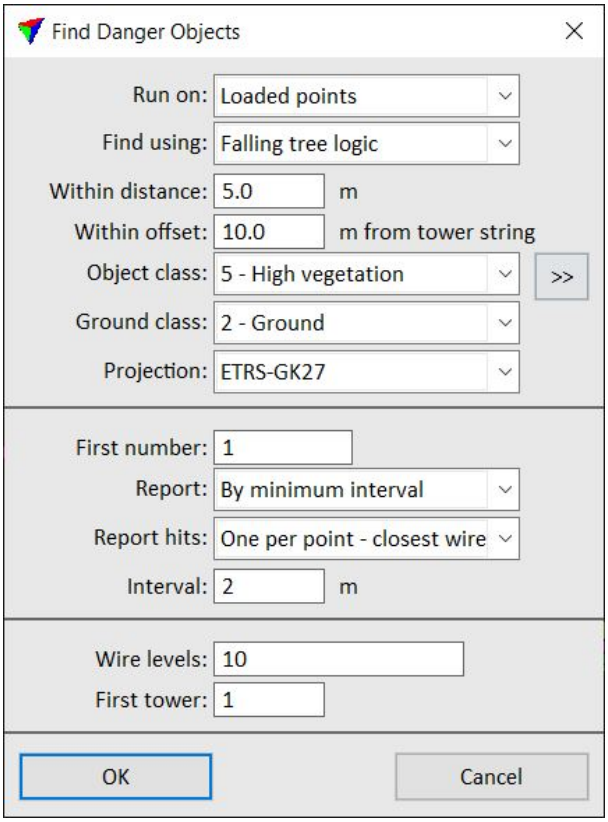
8. Create a text report by using the File / Output report command or HTML reports by using the File / Write HTML command.

To find danger points:

1. Activate the tower string element using the Activate Powerline tool.

2. Select the **Find Danger Objects** tool.

   This opens the **Find Danger Objects** dialog:



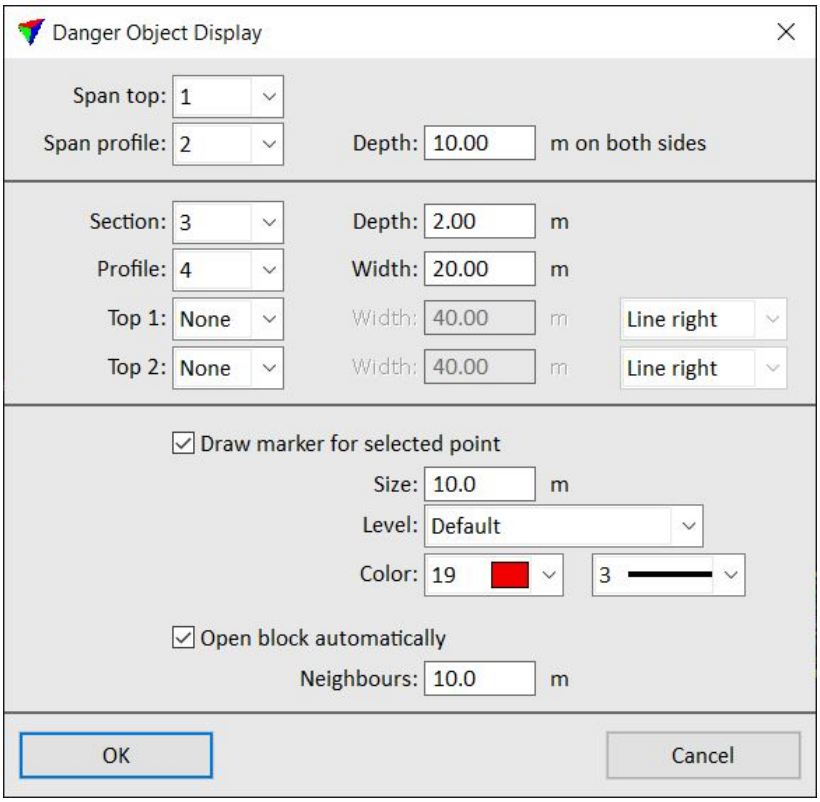
SETTING	EFFECT
Run on	Source data used for danger object detection: • **Loaded points** - points loaded in TerraScan. • **Active project** - points in blocks of the active project in TerraScan.
Find using	Method of danger object detection: **Vertical distance to wire**, **3D distance to wire** or **Falling tree logic**.

SETTING	EFFECT
Within distance	Radius that defines a 3D buffer around a wire. The buffer is added to a wire for computing distances to points.
Within offset	Horizontal distance from the tower string element that defines a corridor. Danger objects are searched within this corridor.
Object class	Point class(es) from which to search danger objects.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Object class** field.
Ground class	Class that contains points on the ground. This is only active if **Find using** is set to **Falling tree logic**.
Projection	Projection system that is used for the data set. This is required for writing latitude/longitude values into output reports.
First number	Number of the first potential danger point in the list.
Report	Reporting frequency: • **All** - each danger point is reported. This is useful, for example, if points on danger objects are already classified and you want to create reports for danger objects. • **One for every span** - only the danger point with the smallest distance within a span (distance between two towers) is reported. • **By minimum interval** - danger points are reported every given **Interval** distance. This is useful if you want to detect danger objects and classify points on them into another point class.
Report hits	Hits on danger objects to report: • **One per point - closest wire** - for each potential danger point in the point cloud, only the distance to the closest wire is reported.The distance to other wires is not reported even if it is also within the critical distance. • **Multiple - all wires** - for each potential danger point in the point cloud, distances to all wires are reported if they are within the critical distance.

SETTING	EFFECT
Wire levels	CAD file level(s) on which the vectorized wire elements are placed. Separate several level numbers by a comma or minus, for example 1-10,13. Use 0-63 for all levels.
First tower	Number of the tower at the first vertex of the selected tower string.

3. Define settings and click OK.

   The application searches for danger points. When the search is complete, the **Danger Object Display** dialog opens:

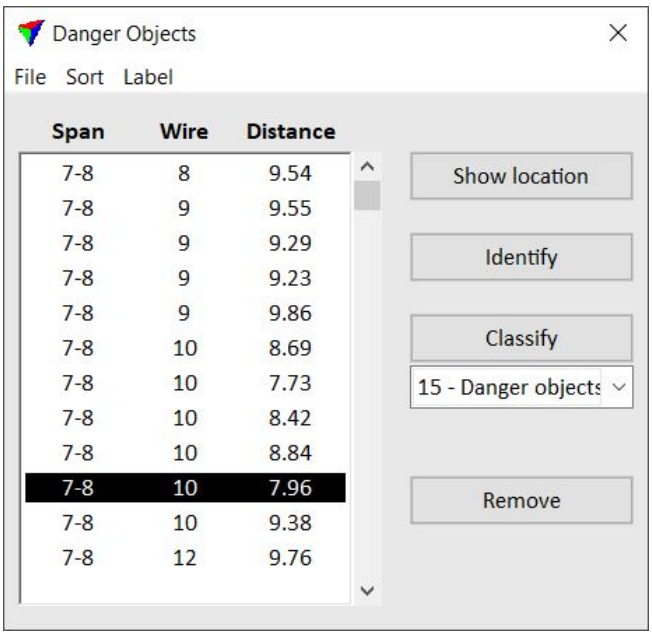


SETTING	EFFECT
Span top	Number of CAD file view used for displaying a span (distance between two towers) in a top view.
Span profile	Number of CAD file view used for displaying a span in a profile view. The centerline of the profile is defined by the tower string element, the depth by the **Depth** value.
Section	Number of CAD file view used for displaying a danger point location in a section view. The centerline of the section runs perpendicular to

SETTING	EFFECT
	the tower string, the depth is defined by the **Depth** value.
Profile	Number of CAD file view used for displaying a danger point location in a profile view. The centerline of the profile is defined by the tower string element, the width by the **Width** value.
Top 1, Top 2	Number of CAD file view used for displaying a danger point location in a top view. The width of the area shown in the view is defined by the **Width** value. The orientation of the view is defined by the selected option: • **North up** - north direction points to the top edge of the view. • **Line up** - the tower string element runs from the bottom to the top edge of the view. • **Line right** - the tower string element runs from the left to the right edge of the view.
Draw marker for selected point	If on a temporary marker is drawn for a danger point that is selected in the list. The marker is drawn with the given **Size**, on the selected **Level** and using the **Color** and line weight settings.
Open block automatically	If on, points from the block at the danger point location are loaded automatically. Points from neighbouring blocks are loaded in the given **Neighbours** distance.

4. Define settings and click OK.

   The **Danger Objects** dialog is now accessible:



The list contains all detected danger points. For each danger point, it shows the span (start and end tower numbers), wire number, and distance between the wire and the danger point. If you click on a row, the application automatically updates the previously defined CAD file views to display that danger point location.

The **Danger Objects** dialog contains several menu and button commands for viewing, classifying, labeling and reporting danger locations.

The list can be sorted using commands from the **Sort** pulldown menu.

The danger point locations can be labeled using commands from the **Label** pulldown menu. A label consists of a line and a text element. The line represents the shortest distance between the wire and the danger point. The text element shows the distance value in CAD file units.

TO	USE COMMAND
Output danger locations to a text file	File / Output report
Output danger locations to an HTML file	File / Write HTML
Classify all danger points into another class	File / Classify all
Change the view setup for danger point display	File / Display settings
Sort list by danger point distance	**Sort / By distance**
Sort list by wire number and secondarily by distance	**Sort / By wire and distance**
Sort list by span and secondarily by wire number	**Sort / By span and wire**
Sort list by span and secondarily by distance	**Sort / By span and distance**
Label the selected danger point in 3D position	**Label / In 3d**
Label the selected danger point in a profile drawing	**Label / In profile**
Label all danger points in 3D	**Label / All in 3d**
Label all danger points in a profile drawing	**Label / All in profile**
Show the location of the selected danger point in a view	**Show location**
Identify a danger point at a certain location	**Identify**
Classify the selected danger point to the given point class	**Classify**
Remove the selected row from the list	**Remove**

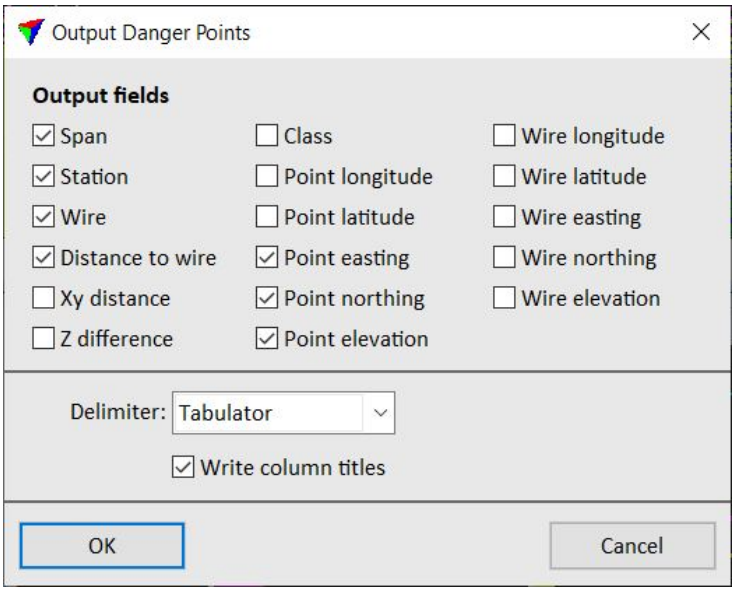
File / Output report

Output report command creates a report in a text file format. You can decide what information is included in the report. The result is a delimited text file that can be opened in text editors or spread sheet applications.

To create a report in text file format:

1. Select **Output report** command from the **File** pulldown menu.

   This opens the **Output Danger Points** dialog:



2. Select fields to be included in the report.

   Longitude and latitude values can only be computed if the projection system of the data is known. The correct projection system must be selected in the **Find Danger Objects** dialog. See Find Danger Objects tool for a description of the dialog.

3. Select a **Delimiter** character.

4. (Optional) Switch on **Write column titles** in order to add the field names as column titles to the report.

5. Click OK.

   This opens the **Danger point output file** dialog, a standard Windows dialog for saving files.

6. Define a storage location, name, and extension for the report and click **Save**.

   This creates the report file.

File / Write HTML

Write HTML command creates a report in HTML format. The report may contain images and textural information of each danger point location. You can decide what information is included in the report.

The command uses an HTML template that defines the layout of the report. The template defines variables for the textural information and placeholders for images. The following tables list all variables that can be included in report templates:

VARIABLE	TEXT CONTENT
#dgrnumber	Point number
#dgrspan	Span (start-end tower)
#dgrwire	Wire number
#dgrdist	Distance to wire
#dgrstation	Distance from start tower
#dgrclass	Point class
#dgrx	Easting coordinate
#dgry	Northing coordinate
#dgrz	Elevation
#dgrlon	Longitude
#dgrlat	Latitude

VARIABLE	CAPTURED VIEW
#spantop	**Span top** view as set in the Danger Object Display dialog.
#spanprofile	**Span profile** view as set in the Danger Object Display dialog.
#section	**Section** view as set in the Danger Object Display dialog.
#profile	**Profile** view as set in the Danger Object Display dialog.
#top1	**Top 1** view as set in the Danger Object Display dialog.
#top2	**Top 2** view as set in the Danger Object Display dialog.

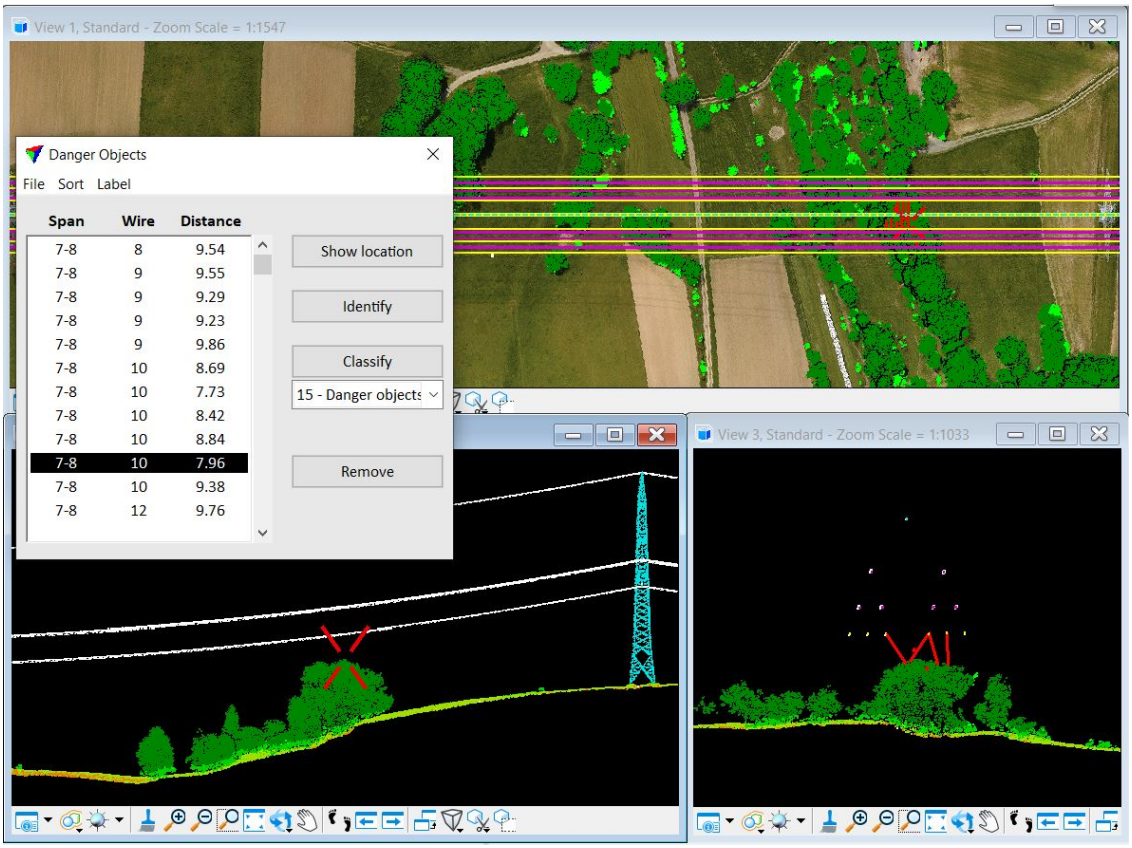
An example template is provided in the EXAMPLE folder of a TerraScan installation, for instance in C:\TERRA64\EXAMPLE\DANGERPOINT_REPORT.HTML.

Longitude and latitude values can only be computed if the projection system of the data is known. The correct projection system must be selected in the **Find Danger Objects** dialog. See Find Danger Objects tool for a description of the dialog.

The view setup determines the size of the images saved for the report. The content of the images is defined by the visibility of data in the views. Images may contain orthophotos attached as raster reference files in TerraPhoto or MicroStation, point cloud data displayed in TerraScan, and vector data in the CAD file.

If point cloud data from a TerraScan project is included in the images and reports are written for all danger points, it is required to switch the **Open block automatically** option on in the Danger Object Display dialog.

In the default template, there are placeholders for three images: a span top view, a section view, and a profile view. An example view setup for producing images from these three views is illustrated in the following figure. The views contain point cloud data colored by class, vector data for wires, and labels for danger points. In the span top view, the visibility of some point classes is switched off and orthophotos attached as TerraPhoto references are displayed.

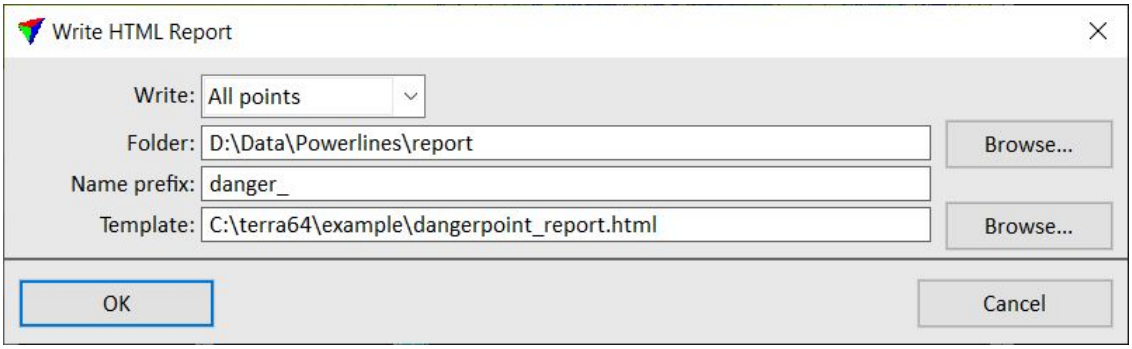


To create a report in HTML format:

1. Arrange the views in a way that fits to your report template. Set the visibility of data.

2. Select **Write HTML** command from the **File** pulldown menu.

   This opens the **Write HTML Report** dialog:



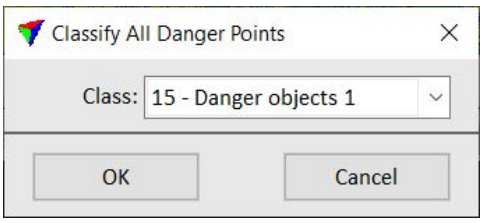
3. Define settings and click OK.

   This creates the report file(s) and the image(s). The names of the image files are a combination of the view type and an increasing danger object number, for example "*spantop1.jpg*", "*section1.jpg*", "*profile1.jpg*" are the images for the report file "*report1.html*". Images are stored as JPEG files.

SETTING	EFFECT
Write	A report is written: • **All points** - for all danger points in the list. • **Selected point** - for the danger point selected in the list.
Folder	Location where the reports are stored.
Name prefix	Defines the name prefix for the report files. The name is a combination of the prefix and the danger object number.
Template	Storage location and name of the HTML template file.

File / Classify all

Classify all command classifies all danger points in the list into another point class. The target class is defined in the **Classify all danger points** dialog:



Another automatic way to classify more points on danger objects is provided by the Wire danger points routine.

File / Display settings

Display setting command opens the Danger Object Display dialog described above.

Label Catenary Height

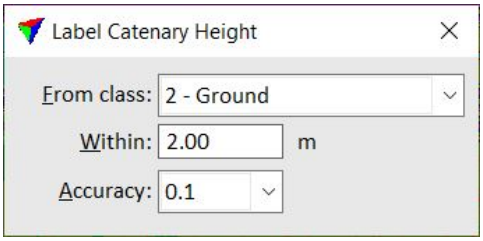
Label Catenary Height tool labels the location of the minimum elevation difference between a catenary element and laser points loaded in TerraScan.

It finds the lowest point of a catenary element, derives an elevation value from all points within a given radius around this XY location. Then, it computes the elevation difference between the catenary and the derived elevation value. The tool draws a vertical line that marks the lowest catenary point. In addition, a text element is drawn that shows the elevation difference value. The text is drawn using the active text, level, and symbology settings of the CAD file.

To label height from catenary to ground:

1. Select the **Label Catenary Height** tool.

   This opens the **Label Catenary Height** dialog:



2. Define settings.

3. Select a catenary string element with a data click.

   The catenary is highlighted.

4. Accept the selected catenary element with another data click inside the view.

   The location of minimum height difference is labeled. You can continue with step 3.

SETTING	EFFECT
From class	Point class used for the elevation difference computation.
Within	Radius within which points are used for computing the elevation value.
Accuracy	Number of decimals for drawing the label.

Label Towers

Label Towers tool draws a label for each tower represented by a vertex of the active tower string element. The label is drawn as cell element into the CAD file. The label is displayed in all views in a way that it is readable independently of the view type and rotation.

The location of the label is determined by the vertices of the tower string element and offset values. A positive offset value places the label elements on the right side of the tower string, a negative value on the left side. In addition, an elevation offset can be applied. Active text size, level, and symbology settings of the CAD file are used for drawing the labels.

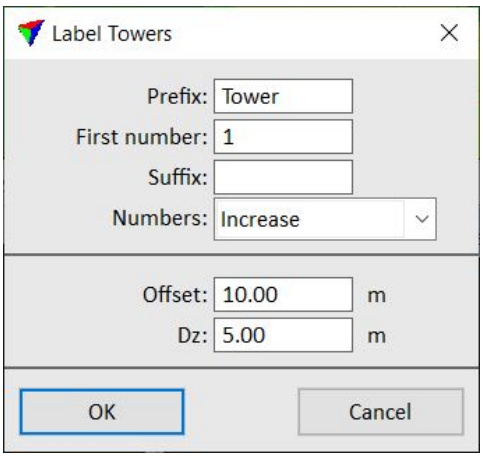
To place labels for tower locations:

1. Activate a tower string element using the Activate Powerline tool.

2. Define settings for texts using the CAD platform **Text** tools. Set the active level and symbology settings.

3. Select the **Label Towers** tool.

   This opens the **Label Towers** dialog:



4. Define settings and click OK.

   This draws the labels into the CAD file.

SETTING	EFFECT
Prefix	Text that is added before the tower number.
First number	Number of the first tower.
Suffix	Text that is added after the tower number.
Numbers	Method of numbering the towers along the tower string element: **Increase** or **Decrease**.
Offset	Horizontal offset of the label from the tower location. Measured between the tower string

SETTING	EFFECT
	vertex and the center point of the text element.
Dz	Vertical offset of the label from a tower location. Measured between the tower string vertex and the center point of the text element.

You can undo the placement of tower labels by using the **Undo** command of the CAD platform.

Output Catenary

Output Catenary tool creates a report for the catenaries that belong to an active tower string element. It creates a list of all wire strings, from which you can store a text file for one selected catenary string at a time.

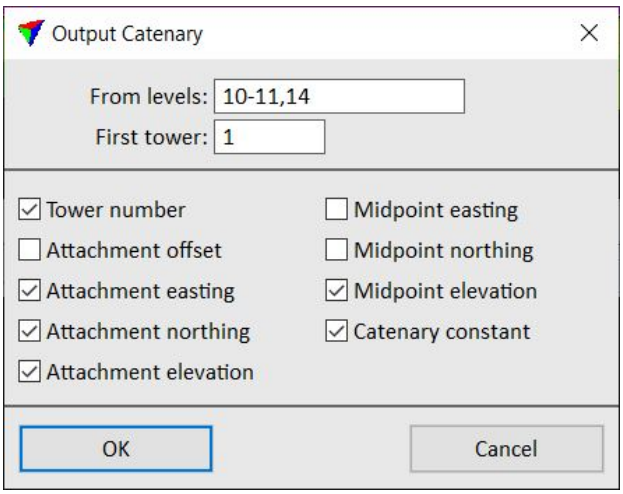
The output of the tool is a structured, space-delimited text file that lists all attributes of a wire string selected in the tool's first dialog.

To output a catenary into a text file:

1. Activate the tower string element using the Activate Powerline tool.

2. Select the **Output Catenary** tool.

   This opens the **Output Catenary** dialog:



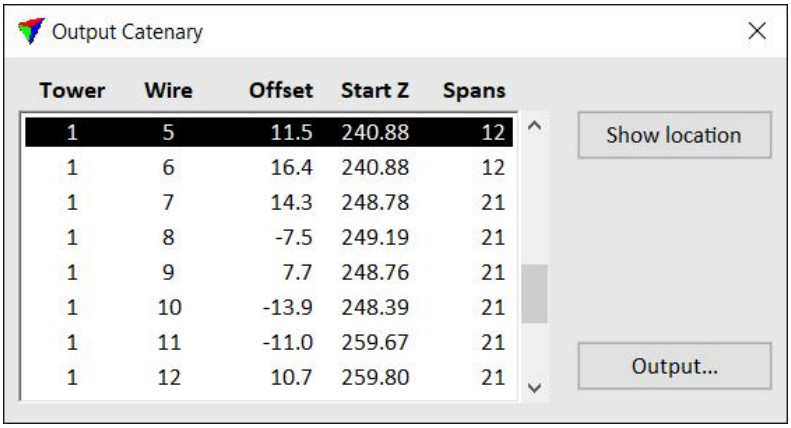
3. Type the number(s) of CAD file level(s) that contain catenary string elements into the **From levels** field.

4. Type a number for the **First tower**. The numbering is applied to the list of wires displayed by this tool and to the text reports.

5. Select attributes to be included in the report.

6. Click OK.

Another **Output Catenary** dialog opens:



Tower	Wire	Offset	Start Z	Spans
1	5	11.5	240.88	12
1	6	16.4	240.88	12
1	7	14.3	248.78	21
1	8	-7.5	249.19	21
1	9	7.7	248.76	21
1	10	-13.9	248.39	21
1	11	-11.0	259.67	21
1	12	10.7	259.80	21

The dialog shows a list of all wires found on the given levels, and for each wire string the number of the start tower, the number of the wire, the offset between tower string vertex and wire attachment, the elevation at the start tower, and the number of spans that include this wire string.

The location of a wire can be shown by selecting a wire in the list and using the **Show location** button. Move the mouse pointer into a view. The selected wire is highlighted. Place a data click in the view in order to move the view to the start of the highlighted wire.

7. Select a wire in the list.

8. Click **Output**.

   This opens the **Output selected catenary chain** dialog, a standard dialog for saving a file.

9. Define a name and location for storing the output file. Add a suitable extension to the file name, such as .TXT or .CSV for opening the file in a text or spreadsheet application.

10. Click **Save**.

    This creates a text file that contains a structured list with a column for each selected attribute. An information dialog informs about the success of the action.

View Tower Spans

View Tower Spans tool makes it easy to traverse through a powerline and display tower and span locations in different types of views, such as top views, sections views, and profile views.

Additionally, top and oblique camera views of towers can be displayed showing aerial images. This requires that TerraPhoto is running and the availability of images organized in a TerraPhoto image list. Camera views might be useful in addition to other display options for towers and spans to support classification tasks or to identify tower structures.

The tool handles the automatic update of views as you scroll through a list of towers. The following view types can be automatically updated:

- **Span top** - a top view showing one tower span. The display is rotated into longitudinal direction of the powerline.
- **Span profile** - a section view showing a longitudinal profile along a tower span.

- **Tower top** - a top view showing the tower location. The top view is rotated to north direction pointing up.
- **Tower rotated** - a top view from the tower location. The view is rotated into longitudinal direction of the powerline.
- **Tower section** - a cross section view of the tower location.
- **Tower tip** - a cross section view of the tower tip.
- **Tower profile** - a longitudinal section view of the tower location.
- **Camera top** - a top-like camera view showing the tower location in an aerial image.
- **Camera oblique 1** - an oblique camera view looking in backward direction from the tower location along the powerline.
- **Camera oblique 2** - an oblique camera view looking in forward direction from the tower location along the powerline.

The tool is useful for various processing steps, for example:

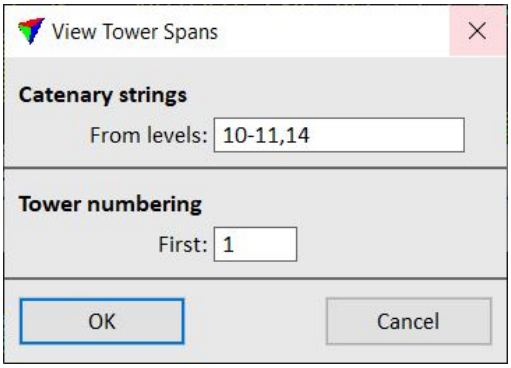
- for manually classifying laser points very close to wires or towers, such as noise points.
- for validating the automatically detected wires.
- for classifying points on towers.
- for placing tower models.

To view tower spans:

1. Activate a tower string element using the Activate Powerline tool.

2. Select the **View Tower Spans** tool.

   This opens the **View Tower Spans** dialog:

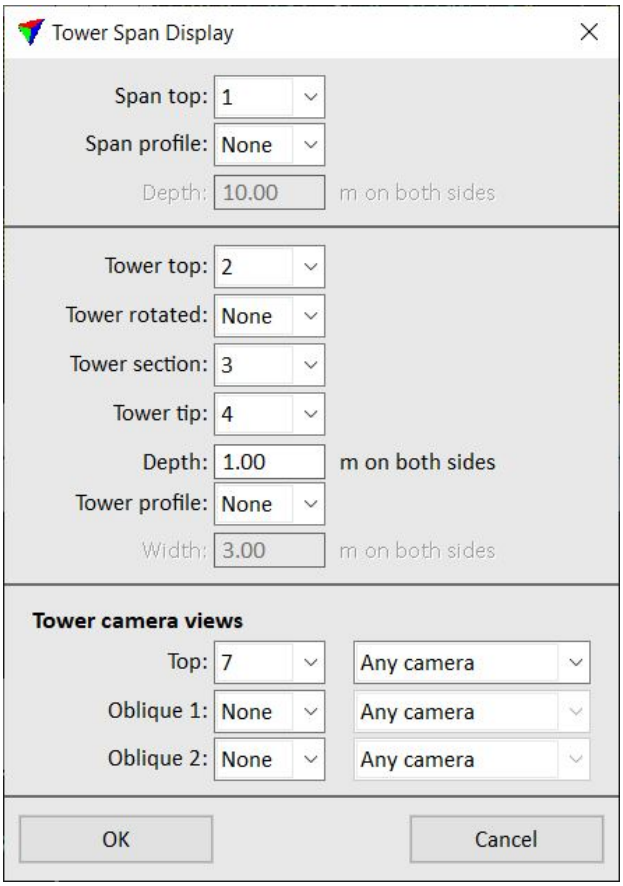


3. Type the number(s) of CAD file level(s) that contain catenary string elements into the **From levels** field.

4. Type a number for the first tower. The numbering is only applied to the list of towers displayed by this tool.

5. Click OK.

   This opens the **Tower Span Display** dialog:



6. Select views that are used for the different display options.

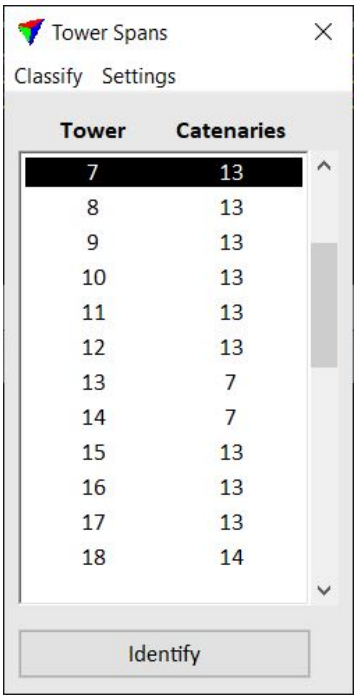
   A **Depth** value determines the depth of a section in section/profile views. It also defines the area of data within which points are classified using Above curve or Below curve commands.

   A **Width** value determines the width of a tower profile view in longitudinal direction left and right of the tower location.

   Select **None** if you do not want to use a display option in any view.

7. Click OK.

   This activates the **Tower Spans** dialog:



The dialog contains a list of all tower locations that are represented by vertices of the active tower string. For each tower, the number of catenaries starting from this tower is shown.

8. Open the CAD file views that you want to use to display data. Arrange the views on the screen.

9. Select a line in the list of towers.

   This updates the display in the open views according to the view setup.

   You can change the view setup by using the **Display** command from the **Settings** pulldown menu. This opens the Tower Span Display dialog described above.

   The commands from the **Classify** pulldown menu can be used to classify points in section views.

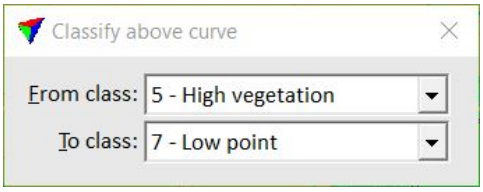
Above curve

Above curve command classifies points above a given curve and within the section view depth. The curve is defined by three manually placed data points.

To classify points above a curve:

1. Select **Above curve** command from the **Classify** pulldown menu.

   This opens the **Classify above curve** dialog:



2. Select a point class in the **From class** field from which to classify points.

3. Select a point class in the **To class** field into which to classify points.

4. Define the first, second, and third point of the curve with data clicks in a section view. The points are displayed temporarily as small white points and the curve is shown as a temporary drawing after the second point has been placed.

   You can undo the placement of the points step-by-step with reset clicks.

   After the third point is placed, the points above the curve and within the depth of the section are classified.

You can undo the classification by using the Undo or From list commands of TerraScan.

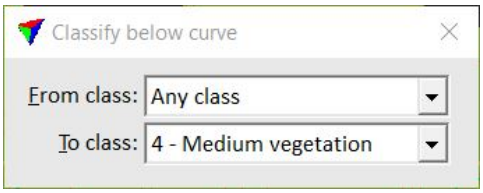
Below curve

Below curve command classifies points below a given curve and within the section view depth. The curve is defined by three manually placed data points.

To classify points below a curve:

1. Select **Below curve** command from the **Classify** pulldown menu.

   This opens the **Classify below curve** dialog:



2. Select a point class in the **From class** field from which to classify points.

3. Select a point class in the **To class** field into which to classify points.

4. Define the first, second, and third point of the curve with data clicks in a section view. The points are displayed temporarily as small white points and the curve is shown as a temporary drawing after the second point has been placed.

   You can undo the placement of the points step-by-step with reset clicks.

   After the third point is placed, the points below the curve and within the depth of the section are classified.

You can undo the classification by using the Undo or From list commands of TerraScan.

Roads and Railroads

The tools for processing data of roads and railroads have been developed a lot since more and more data became available from Mobile Mapping System (MMS) surveys. This development is still ongoing, so there will be additions and improvements for the tool sets in the future.

Most of the tools are intended to be used with dense point clouds of high positional accuracy. Such point clouds are usually produced by Mobile Laser Scanning (MLS) systems. However, some of the tool described in this Chapter are applicable to Airborne Laser Scanning (ALS) point clouds as well. Some tools benefit from images which are collected by one or several cameras as part of a modern MMS or ALS system.

The processing of point clouds from MLS systems is a complex task if a high accuracy and quality for the end products shall be achieved. This includes the calibration of the scanner system, the matching of drive paths, the classification of the points into classes that support the extraction of the required information, and finally the extraction of the required information itself.

The general workflow for processing MLS data for road and railroad projects can be outlined as follows:

1. *System calibration*: fine-tuning of the calibration values provided by system manufacturers. This is usually done based on laser data that is collected at a specific calibration site. The process is done with TerraScan and TerraMatch and the workflow is described in the TerraMatch Users' Guide.

2. *Project setup*: import and modify raw trajectory information, creation of a TerraScan project, import of raw laser data. This is done with tools of TerraScan.

3. *Drive path matching*: improving the internal and absolute accuracy of the project data. This involves TerraScan and (optionally) TerraPhoto, but mainly tools of TerraMatch are used and the workflow is described in the TerraMatch Users' Guide.

4. *Laser data classification*: cutting off overlap between drive paths, apply classification routines and possibly other automatic/manual processing steps. This depends on the purpose for which the data shall be used.

5. *Extraction of information*: this may include the analysis of the current situation, for example, on a road surface or along a road/rail track; or the detection and/or vectorization of specific features, such as paint markings, road breaklines, rails, overhead wires, or potentially dangerous objects.

The tools described in this Chapter are related to the last point in the workflow outline above.

Road data processing

TerraScan provides three options for road breakline extraction. There is the Find Automatic Breaklines tool for the automatic extraction of the road crown. The Find Road Breaklines tool can be used to extract the crown of a road and road edges semi-automatically based on approximate 2D lines. Finally, there is a special processing workflow that speeds up the digitization of any lines along a corridor. This workflow can be applied to MLS and high-density ALS data, and involves the Write section points routine and the Import Road Breaklines tool.

Draw Slope Arrows tool and Color by Slope display option can be used for water flow analysis on the road surface, for checking the superelevation of road lanes, and for detecting damage on the road surface, such as ruts.

Write Alignment Elevations tool exports road surface elevation values along an alignment element into text files. This may be required for further road surface analysis in other software products, such as the computation of the International Roughness Index (IRI).

Further functionality is implemented in TerraScan as macro actions, such as Compute section parameters and Find paint lines.

Road/Railroad data processing

There are some tools in TerraScan that are useful for both application fields, roads and railroads.

Place Railroad String tool is a useful tool for many purposes. It allows faster digitization of linear elements than any other CAD digitization tool.

One of them is the Draw Sight Distances tool that is applicable to ALS and MLS data. Line-of-sight analysis based on laser point clouds has the unique advantage that all objects in the road environment including vegetation are considered. Another tool, Draw Sign Visibility, analyzes the visibility of signs or signals from certain viewer positions.

Find Poles is a tool for detecting and vectorizing poles along a railroad or road.

Label Alignment Curvature and Label Clearance are tools for analyzing the radius of curved roads/rails and clearance distances below bridges or other overhead structures.

Fit Geometry Components tool derives geometry components from a surveyed centerline of roads, railroads, and possibly other corridor projects. Geometry components are required for design tasks, especially for the data exchange between different software products.

Railroad data processing

TerraScan has a few tools which are dedicated to railroad processing. They include classification and vectorization tools suited for ALS and/or MLS data.

Fit Railroad String tool is intended to be used for ALS data of railroads. It fits an approximate rail track centerline to the classified laser points on the rails. The resulting 3D line element follows the rail track centerline more accurately.

There are two tools for the automatic detection and vectorization of rails and overhead wires from MLS data. Find Rails tool creates vector lines along rail tracks based on classified laser points, a rail track cross section profile, and an alignment element. Find Wires tool is used for the vectorization of all kinds of overhead wires along rail tracks, tram tracks, etc. It creates vector lines and classifies laser points on wires. The automatic wire detection is usually followed by manual improvements of the wire lines which can be done with the Check Wire Ends tool.

Road toolbox

The tools in the **Road** toolbox are used to place breaklines of roads, to analyze the slopes on the road surface and sight conditions, to place labels for curvatures, to write elevation values along an alignment, and to start the fit geometry components module.

TO	USE TOOL	
Find road breaklines automatically		Find Automatic Breaklines
Find multiple breaklines along a road		Find Road Breaklines
Import breaklines back to world coordinate system		Import Road Breaklines
Draw slope arrows perpendicular and along alignment		Draw Slope Arrows
Draw sight distance values along road		Draw Sight Distances
Display locations with line of sight to a sign polygon		Draw Sign Visibility
Label smallest distance point inside polygons		Label clearance
Label alignment curvature at regular intervals		Label Alignment Curvature
Write alignment elevations at regular steps		Write Alignment Elevations
Fit geometry components to match surveyed alignment		Fit Geometry Components

Draw Sight Distances

Draw Sight Distances tool determines how far a viewer sees along a road, railroad, or other corridor, and produces labels for sight distances. It's basically a tool for line-of-sight analysis based on point clouds.

The path of the viewer along the corridor is defined by a line element. It should run along a lane of the road or a rail track at the elevation of the ground. It can be produced, for example, by drawing the trajectory line into the CAD file and draping it to the ground points using the Drape Linear Element tool. The viewer height is defined with a constant value in the tool's settings.

The target positions for the line-of-sight analysis are also defined by a line element draped on the ground elevation. The target line can be the same element as the viewer line or another line element that is a little bit longer than the viewer line. The viewing angle which determines the area for potential obstacle search is defined as a constant value in the tool's settings.

Potential obstacles for the viewer are represented in the point cloud. The points should be classified into a ground class by using preferably the Hard surface and/or Ground classification routines and above-ground classes. To get a reliable result from the line-of-sight analysis, any points below the ground, from moving objects, and noise above the road surface should be classified into separate classes. These classes can be excluded from the process.

The process checks if there is any point in the point cloud close to a straight line from a viewer position to a target position. If there is a point, the closest point to the viewer position determines the sight distance.

There are rules and regulations for required sight distances along a road. The distances mainly depend on the speed allowance and vary from country to country. There are different sight distance requirements for safely stopping a car or for safely overtaking another car. Example: For safely stopping a car, a viewer with a height of 1.10 m above lane center and a speed of 80 km/h must see a target of 0.40 m above the road surface in a distance of 120 m. For safely overtaking a car, the same viewer must see a target of 0.60 m above the road surface in a distance of 320 m. The viewer path needs to be cut into separate line elements according to speed limits in order to do a precise sight distance analysis.

Draw Sight Distances tool can run on loaded points as well as on TerraScan project points. It creates text elements as labels for sight distances. In addition, it can draw line elements for short sight distances into the CAD file. These line element can then be used to classify points close to them using the By centerline classification routine. The classification helps to identify obstacles in the point cloud.

To draw sight distances:

1. Create a line element along the lane centerline as viewer traveling path and (optional) another line element as target path.

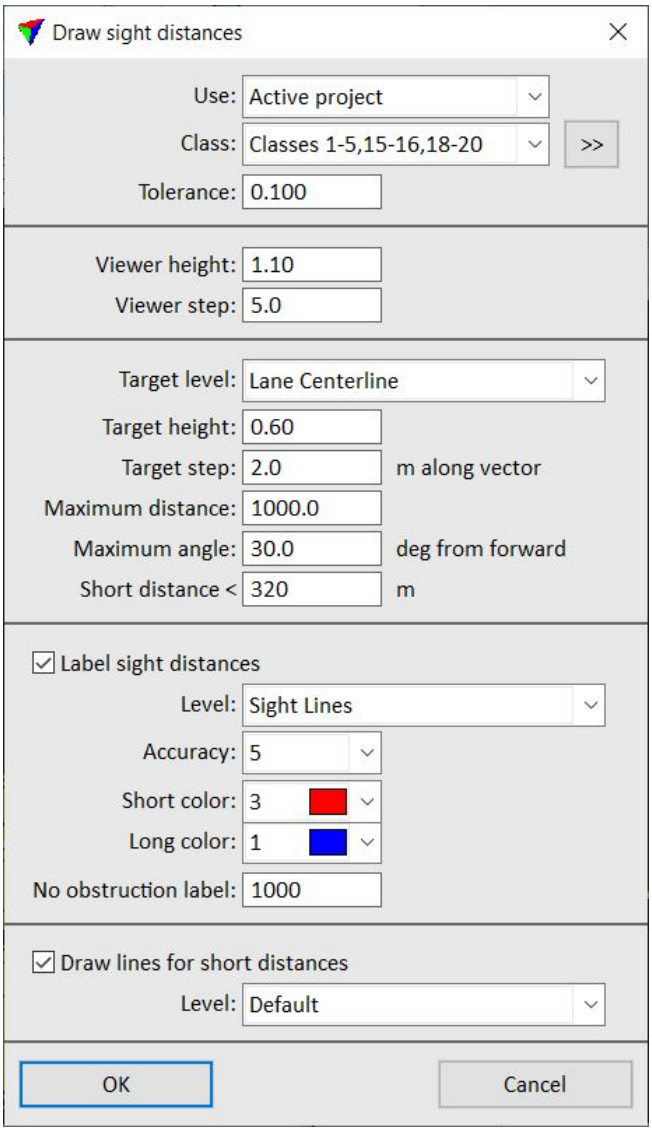
2. Drape the line element(s) to the ground on the road surface.

3. (Optional) Load points into TerraScan if you want to run the tool on loaded points.

4. Select the line element that defines the viewer traveling path.

5. Select **Draw Sight Distances** tool.

   This opens the **Draw sight distances** dialog:



6. Define settings an click OK.

   This starts the sight distance analysis process. The size of the text elements is determined by the active text size settings in the CAD file. The color, line weight, and line style of lines for short distances are determined by active symbology settings in the CAD file.

SETTING	EFFECT
Use	Laser points used for the sight distance analysis process: • **Loaded points** - points loaded into TerraScan. • **Active project** - points referenced by the active TerraScan project. *Not UAV*
Class	Point classes that are included in the sight analysis. This should include all points on potential obstacles. The list contains the active classes in TerraScan.

SETTING	EFFECT
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Class** field.
Tolerance	Radius around a straight line between viewer and target position within which a laser point may be considered a sight obstacle.
Viewer height	Height of the viewer above the viewer path element.
Viewer step	Distance between locations along the viewer path element where the software analyzes the sight of the viewer.
Target level	CAD file level on which the line element is drawn that defines the target path.
Target height	Height of a target object above the target path element.
Target step	Distance between locations along the target path element where the software analyzes the sight towards a target object.
Maximum distance	Maximum distances from the viewer considered in the sight analysis.
Maximum angle	Angle of sight forward from the viewer. Defines the area that is analyzed regarding sight obstacles.
Short distance	Defines the maximum value of a critical sight distance. Sight distances smaller or equal to the given value can be labeled with a different color in order to highlight the locations.
Label sight distances	If on, text elements are drawn into the CAD file that label the sight distance at each viewer position.
Level	Level on which sight distance labels are drawn.
Accuracy	Accuracy of sight distance labels. Values are rounded to the given accuracy, e.g. to the closest 5 m value.
Short color	Color of short sight distance labels. Applied to all distances smaller or equal to the given **Short distance** value.
Long color	Color of sight distance labels if the distance is longer than the given Short **distance** value.

SETTING	EFFECT
No obstruction label	Label of viewer positions for which no obstruction is found.
Draw lines for short distances	If on, lines are drawn from viewer to target positions if the distance is smaller or equal to the given **Short distance** value.
Level	Level on which lines for short distances are drawn.

You can undo the creation of sight distance labels and lines by using the **Undo** command of the CAD platform.

Draw Sign Visibility

Draw Sign Visibility tool determines from what points a target object, such as a traffic sign, traffic light, advertisement plate, etc., are visible. With other words, the tool also finds points on objects that may obstruct the sight to a target object.

The process requires a shape, ellipse, or complex shape element that represents the target object. The element must be drawn in a way that its front side faces towards the viewer. This means, for example, that the element must be drawn in counterclockwise direction in MicroStation. You can digitize the shape element in a section view created with the Draw Vertical Section tool. The centerline of the section has to be at the target object's location.

Further, the process requires a linear element drawn in driving direction that defines the path of the viewer. For instance, a trajectory line draped on the ground elevation by the Drape Linear Element tool can be used for this purpose. The viewer height is defined with a constant value in the tool's settings.

Potential obstacles for the visibility of a target object are represented in the point cloud. The points should be classified into a ground class by using preferably the Hard surface and/or Ground classification routines and above-ground classes. To get a reliable result from the visibility analysis, any points below the ground, from moving objects, and noise in the air should be classified into separate classes. These classes can be excluded from the process.

Draw Sign Visibility tool can run on loaded points as well as on TerraScan project points. The process checks if there is any point in the point cloud close to a straight line from a viewer position to the target object. It creates point elements as markers for each analyzed location. The color of the marker indicates whether the target object is visible or not from the analyzed location. In addition, the process can draw line elements for locations from which the visibility is obstructed into the CAD file. These line element can then be used to classify points close to them using the By centerline classification routine. The classification helps to identify obstacles in the point cloud.

To analyze sign visibility:

1. Create a line element along the lane centerline as viewer traveling path.

2. Drape the line element(s) to the ground on the road surface.

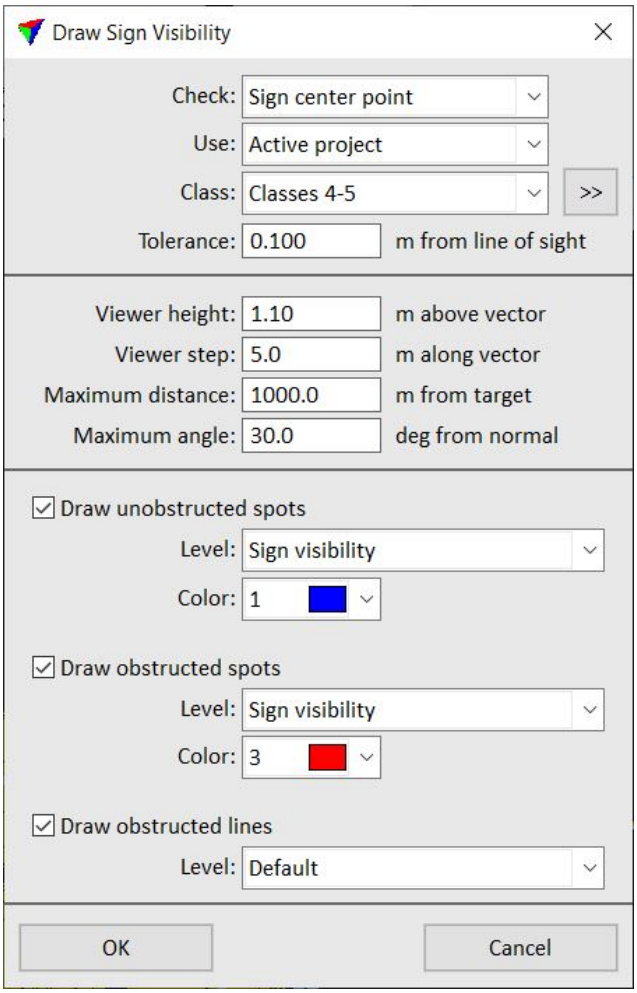
3. Digitize a shape element that represents the target object.

4. (Optional) Load points into TerraScan if you want to run the tool on loaded points.

5. Select the line element that defines the viewer traveling path.

6. Select **Draw Sign Visibility** tool.

   This opens the **Draw Sign Visibility** dialog:



7. Define settings an click OK.

8. Identify the target object by placing a data click close to the shape element in the section view.

9. Accept the selection with another data click.

   This starts the visibility analysis process and draws the visibility markers into the CAD file. An information dialog shows the success of the process.

SETTING	EFFECT
Check	Determines the area on the target object that

SETTING	EFFECT
	is checked for visibility: • **Sign center point** - the center point of the target object shape. • **All sign vertices** - the vertices of the target object shape and thus, the complete target object area.
Use	Laser points used for the visibility analysis process: • **Loaded points** - points loaded into TerraScan. • **Active project** - points referenced by the active TerraScan project. *Not UAV*
Class	Point classes that are included in the visibility analysis. This should include all points on potential obstacles. The list contains the active classes in TerraScan.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Class** field.
Tolerance	Radius around a straight line between viewer and target position within which a laser point may be considered an obstacle.
Viewer height	Height of the viewer above the viewer path element.
Viewer step	Distance between locations along the viewer path element where the software analyzes the visibility of the target object and places a visibility marker.
Maximum distance	Maximum distances from the target object considered in the visibility analysis.
Maximum angle	Angle off from the normal direction of the target object shape. Defines the area that is analyzed regarding visibility obstacles.
Short distance	Defines the maximum value of a critical sight distance. Sight distances smaller or equal to the given value can be labeled with a different color in order to highlight the locations.
Draw unobstructed spots	If on, point elements are drawn into the CAD file that mark locations of unobstructed visibility. The markers are drawn on the given **Level** and with the given **Color**.

SETTING	EFFECT
Draw obstructed spots	If on, point elements are drawn into the CAD file that mark locations of obstructed visibility. The markers are drawn on the given **Level** and with the given **Color**.
Draw obstructed lines	If on, lines are drawn from viewer position to target object if the visibility is obstructed. The lines are drawn on the given **Level** and using the active symbology settings of the CAD platform.

You can undo the creation of sign visibility markers and lines by using the **Undo** command of the CAD platform.

Draw Slope Arrows

Draw Slope Arrows tool computes the slope along an alignment element of a road. The computation can be done for the side slope of road lanes (representing the superelevation of the lane) or for the longitudinal slope of the road. The tool draws arrows which show the direction of the slope and text labels that show the gradient of the slope.

The tool requires a line string as alignment element. This is usually the approximate center line of the road which can be derived, for example, from the trajectory lines. The alignment element determines the longitudinal direction of the road as well as the horizontal location of the slope arrows. The elevation of the alignment element does not effect the slope arrows.

The elevation of the slope arrows is fitted to laser points on the road surface. Thus, these laser points should be classified into a separate class by using preferably the Hard surface classification routine. The gradient of a slope arrow is computed from the elevation values of the start and end points of the arrow element.

The tool requires points loaded into TerraScan. However, the same process can be performed for a TerraScan project using the Compute slope arrows macro action and then, reading the slope arrows from text files using the Read / Slope arrows command.

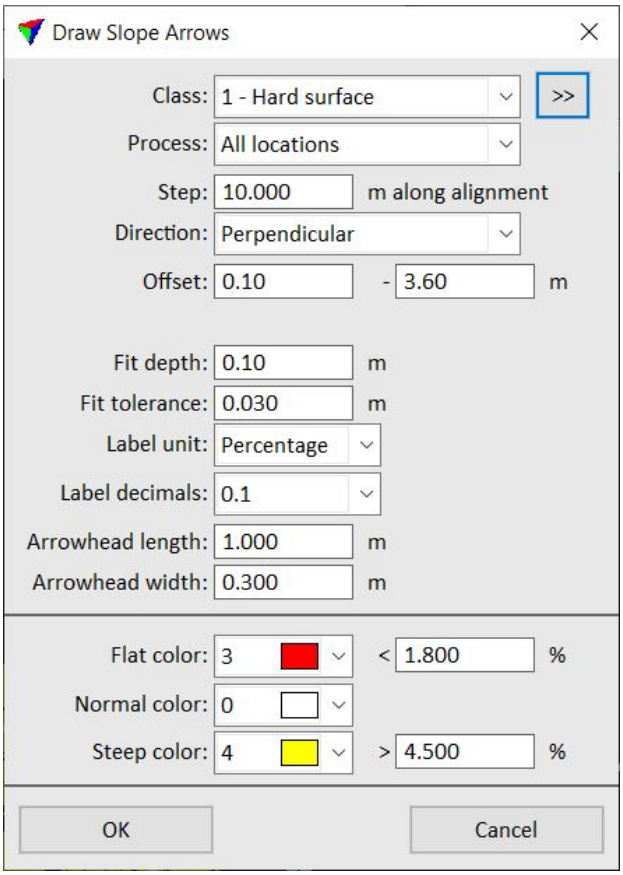
To draw slope arrows:

1. Load laser points into TerraScan. Only points on the road surface are required.

2. Select the alignment element with any **Selection** tool.

3. Select **Draw Slope Arrows** tool.

   This opens the **Draw Slope Arrows** dialog:



4. Define settings and click OK.

   This starts the process. The software draws arrows and text elements along the alignment element wherever it finds laser data. The level, line weight, line style, and text size of the arrow and label elements are determined by the active symbology and text size settings in the CAD file.

   An information dialog shows the number of created slope arrows.

SETTING	EFFECT
Class	Point class that contains points on the road surface. Used for fitting the elevation of slope arrows. The list contains the active classes in TerraScan.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Class** field.
Process	Area to process: • **All locations** - slope arrows are created wherever there is point cloud data available. • **Inside active block** - slope arrows are created only inside the active block. This excludes areas covered by neighbour points that are loaded in addition to the points of an active project block.

SETTING	EFFECT
Step	Distance between locations where the software places a slope arrow.
Direction	Direction of the slope arrows relative to the alignment element or the road direction: • **Longitudinal** - in road direction. • **Perpendicular** - perpendicular to the road direction.
Offset	Defines the horizontal distance of the start and end point of an arrow relative to the alignment element. This is only one value for **Longitudinal** arrows and two values for **Perpendicular** arrows. The two offset values also determine the length of slope arrows with **Perpendicular** direction. Positive offset values create slope arrows to the right side of the alignment element, negative values to the left side.
Length	Length of slope arrows with longitudinal direction. This is only active if **Direction** is set to **Longitudinal**.
Fit depth	Depth of a section in the point cloud data where the software fits the arrow to the points on the road surface.
Fit tolerance	Tolerance value for fitting the arrow to the points. Relates to the noise in the data.
Label unit	Unit for expressing the slope gradient: **Degree** or **Percentage**.
Label decimals	Number of decimals for slope labels.
Arrowhead length	Length of the arrow head as part of the slope arrow.
Arrowhead width	Width of the arrow head.
Flat color	Color of a slope arrow if the slope gradient is less or equal to the given value.
Normal color	Color of a slope arrow if the slope gradient is between the given flat and steep values.
Steep color	Color of a slope arrow if the slope gradient is larger than the given value.

You can undo the creation of slope arrows by using the **Undo** command of the CAD platform.

Find Automatic Breaklines

Find Automatic Breaklines tool is used for fully-automatic breakline detection along roads. The tool creates 3D breaklines based on dense loaded laser points. The laser points on the road surface should be classified into a separate class by using preferably the Hard surface classification routine.

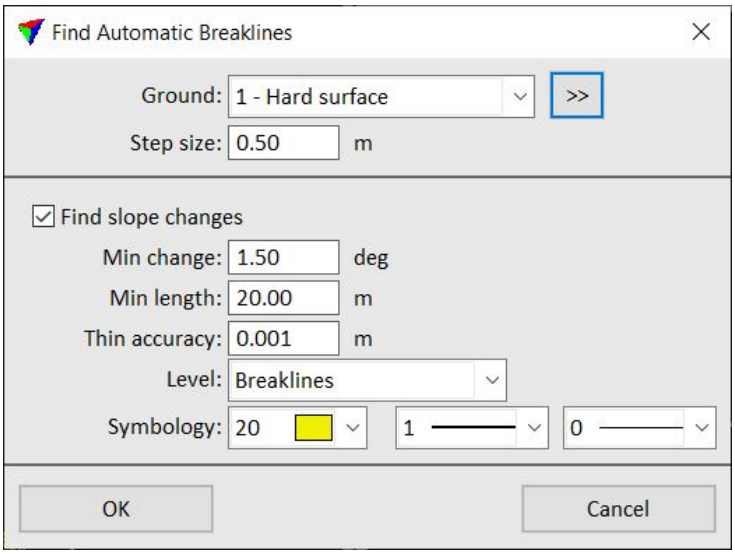
The detection works for breaklines along slope changes, for example along the crown of a straight road. After automatic breakline detection, you probably need to check and manipulate the breaklines manually.

To find road breaklines automatically:

1. Load laser points into TerraScan. Only points in the class for road breakline detection are required.

2. Select **Find Automatic Breaklines** tool.

   This opens the **Find Automatic Breaklines** dialog:



3. Define settings and click OK.

   This starts the breakline detection process. The software draws breakline elements if it finds a slope change in the loaded laser points.

SETTING	EFFECT
Ground	Point class that contains points on the road surface. Used for breakline detection. The list contains the active classes in TerraScan.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Ground** field.

SETTING	EFFECT
Step size	Distance between locations where the software tries to find a slope change in the laser data in order to insert a vertex for a breakline element.
Find slope changes	If on, the software detects slope changes in the laser data.
Min change	Minimum change in slope gradient. Given in degree.
Min length	Minimum length of a breakline element.
Thin accuracy	Defines the degree of thinning applied to a breakline element. A vertex is removed, if the location of the line does not change more than the given value.
Level	CAD file level on which the breakline elements are drawn.
Symbology	Color, line weight, and line style of the breakline elements. Uses the active color table and standard line weights and styles of the CAD file.

Find Road Breaklines

Find Road Breaklines tool is used for semi-automatic breakline detection along roads. The tool requires 2D line elements that run approximately along road breakline locations, for example close to the road center for crown of the road detection and close to the road edge for edge of pavement or curb stone detection. Based on that, the software searches the best 3D breakline close by.

The semi-automatic detection works for the following road breakline types:

- edge of pavement - runs along the edge of the road pavement.
- curb stone - runs along the curb stone edge of a road/sidewalk. A curb stone is defined as edge with a vertical elevation jump at the same XY location.
- crown of the road - runs along the crown of the road formed by a small slope change.
- planar surface
- section centerline - runs along the center of cross sections along the road.
- section breakline - runs along the edge of cross sections along the road.

The parameters for 3D breakline creation are defined for each breakline type.

The process requires dense laser points loaded in TerraScan. The laser points on the road surface should be classified into a separate class by using preferably the Hard surface classification routine. Further, the breakline placement along pavement edges benefits from color values assigned to the laser points.

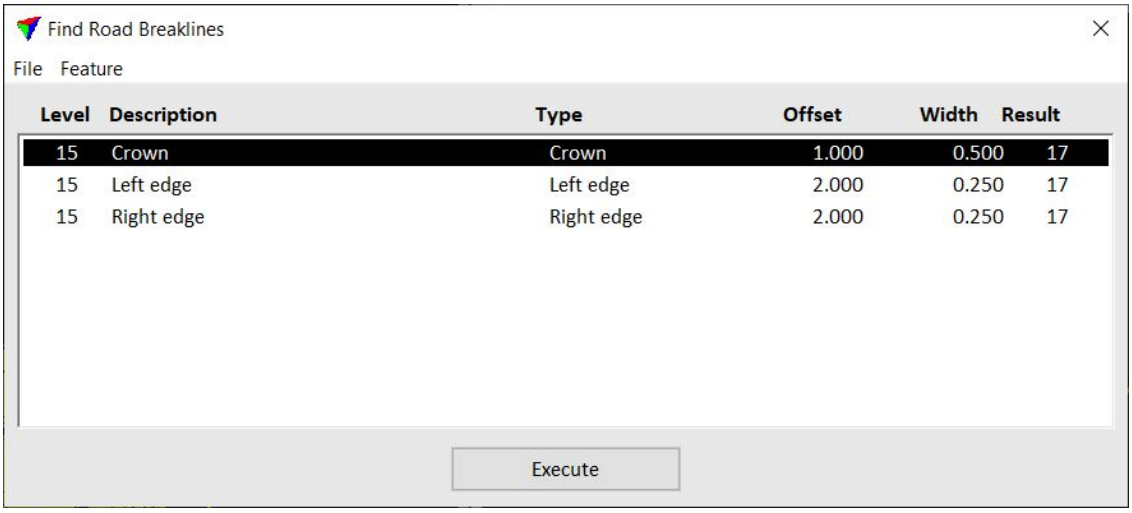
After automatic breakline detection, you probably need to check and manipulate the breaklines manually.

To find road breaklines:

1. Load laser points into TerraScan. Only points in the class for road breakline detection are required.

2. Select **Find Road Breaklines** tool.

   This opens the **Find Road Breaklines** dialog:

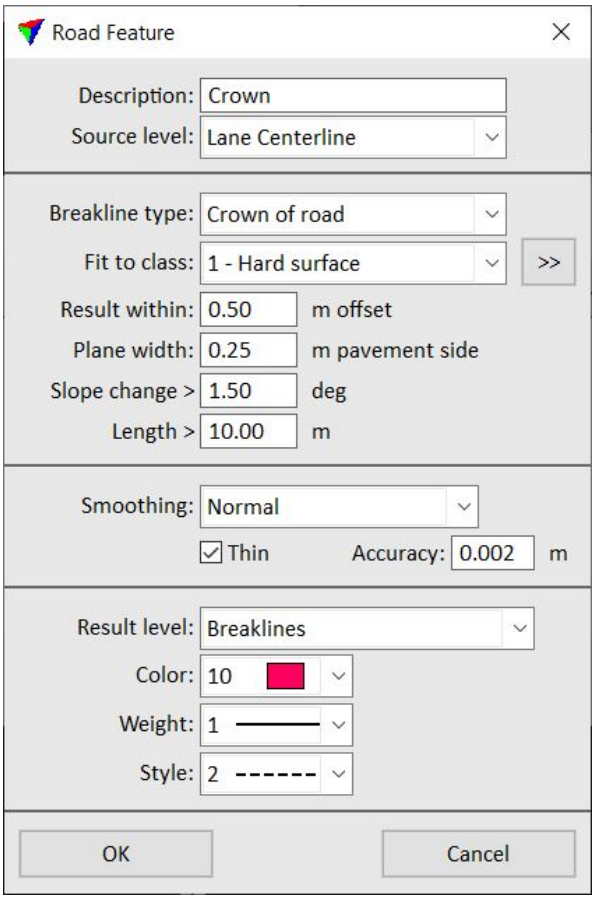


Level	Description	Type	Offset	Width	Result
15	Crown	Crown	1.000	0.500	17
15	Left edge	Left edge	2.000	0.250	17
15	Right edge	Right edge	2.000	0.250	17

3. Select **Add** command from the **Feature** pulldown menu in order to define a new feature for road breakline detection.

   You can modify an existing feature by selecting the feature and using the **Edit** command from the **Feature** pulldown menu. To delete a feature, select the **Delete** command from the **Feature** pulldown menu.

   Add and **Edit** commands open the **Road Feature** dialog. The settings in the **Road Feature** dialog partly depend on the selected breakline type.



4. Define settings and click OK.

   The feature is added to the list in the **Find Road Breaklines** dialog.

5. Repeat steps 3 to 4 for all road breakline features you want to detect.

6. (Optional) Save the road feature definitions into a text file using the **Save as** command from the **File** pulldown menu. You can save changes to an existing text file by selecting the **Save** command from the **File** pulldown menu.

7. Click **Execute** in order to run the breakline detection.

   This starts the breakline detection process. The software draws breakline elements if it finds the 3D location of the breakline features in the loaded laser points.

SETTING	EFFECT
Description	Descriptive name of the road feature.
Source level	CAD file level, on which the line elements are drawn that define the approximate location of the breakline feature.

SETTING	EFFECT
Breakline type	Type of the breakline feature: • **Crown of the road** - runs along a small slope change. • **Left curb stone** - runs along a curb stone on the left side of a road. • **Right curb stone** - runs along a curb stone on the right side of a road. • **Left edge of pavement** - runs along the edge of pavement on the left side of a road. • **Right edge of pavement** - runs along the edge of pavement on the right side of a road. • **Planar surface** • **Section centerline** - runs along the center of cross sections along the road. • **Section breakline** - runs along the edge of cross sections along the road.
Fit to class	Point class that contains points on the road surface. Used for breakline detection. The list contains the active classes in TerraScan.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Fit to class** field.
Result within	Maximum horizontal offset between the approximate line element and the true breakline location. This is not available if **Breakline type** is set to **Left curb stone** or **Right curb stone**.
Plane width	Width of a plane next to the breakline location. One value applies for the left and right side for **Crown of the road** and **Planar surface** features. There are two values for **Left/Right edge of pavement** features, one for the pavement side and another for the outside-road side. This is not available if **Breakline type** is set to **Section centerline** or **Section breakline**.
Slope change	Minimum change in slope gradient. Given in degree. This is only available if **Breakline type** is set to **Crown of road**.
Length	Minimum length of a breakline element. This is only available if Breakline **type** is set to **Crown of road**.

SETTING	EFFECT
Use color	If on, the software uses RGB color values assigned to laser points in order to find the breakline location. This is only available if **Breakline type** is set to **Left/Right edge of pavement**.
Fit tolerance	Tolerance value for fitting the breakline element into the laser points. Relates to the noise in the data. This is only available if **Breakline type** is set to **Section centerline**.
Step	Distance between consecutive cross sections along the road used for fitting the line element to the laser points. This is only available if **Breakline type** is set to **Section centerline**.
Percentile	This is only available if **Breakline type** is set to **Section centerline**.
Max offset	Maximum horizontal offset between the approximate line element and the true breakline location. This is only available if **Breakline type** is set to **Section breakline**.
Smoothing	Defines the degree of smoothing applied to a breakline element: • **None** - no smoothing. • **Normal** - a bit smoothing. • **Aggressive** - maximum smoothing.
Thin	If on, a vertex is removed, if the location of the line does not change more than the given value.
Result level	CAD file level on which the breakline elements are drawn.
Color	Color of the breakline elements. Uses the active color table of the CAD file.
Weight	Line weight of the breakline elements. Uses the standard line weights of the CAD file.
Style	Line style of the breakline elements. Uses the standard line styles of the CAD file.

Fit Geometry Components

Fit Geometry Components tool starts the component fitting module of TerraScan. The module creates design geometry built from the geometry components lines, arcs, and clothoids. The aim is to create a geometry from these components that forms the best match to a surveyed alignment of a road, a railroad, or a pipeline. The module finds the best fit for both horizontal and vertical geometry.

The module and the creation of geometry components serves different purposes:

- **Data exchange** - view the current geometry of a road/railroad/pipeline in design software such as Bentley InRoads, Bentley Track, etc and/or export the geometry into LandXML or Tekla 11/12 format. Design software may only accept certain geometry components for linear features.
- **Road surface analysis** - find long span deformations of road surfaces.
- **Object design comparison** - compare existing object geometry components with design recommendations.

The tool starts from a 3D centerline element that can be created, for example, based on ALS or MMS data.

To start the component fitting module of TerraScan:

1. Use the any **Selection** tool in order to select a centerline element that represents the surveyed object.

2. Select the **Fit Geometry Components** tool.

   This starts the module and opens the **Components Fitting** window.

The processing workflow for component fitting and the commands of the window are explained in detail in Chapter Fit Geometry Components.

Import Road Breaklines

Not Spatix

Import Road Breaklines tool converts linear elements from an artificial coordinate system into geographic coordinates. It is used in combination with digitized lines based on TerraScan section points which are produced by the Write section points macro action.

The line elements are digitized in an artificial coordinate system in a separate CAD file. The artificial coordinate system is defined by:

- X axis - scaled stations along an alignment element.
- Y axis - offset from an alignment element.
- Z axis - scaled elevation values of the original point cloud data.

The tool converts these artificial XYZ coordinates back to Easting, Northing, Elevation coordinates. It utilizes the same alignment element and inverse scaling factors as were used for producing the section points.

To import road breaklines:

1. Attach the section CAD file as a reference to the normal project CAD file.

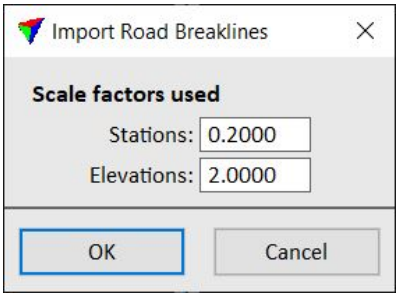
2. Open two top views, one showing the location of the road and the alignment element used for creating section points, and the other one showing the digitized lines in the attached reference CAD file.

3. Select the alignment element using any **Selection** tool.

4. Draw fence around the lines in the reference CAD file.

5. Select **Import Road Breaklines** tool.

   This opens the **Import Road Breaklines** dialog:



6. Define settings. The values must be the same as used for creating the section points with the Write section points macro action in order to compute correct coordinate values for the breakline elements.

7. Click OK.

   This converts the lines inside the fence from the artificial coordinates to the original coordinates and draws them into the master CAD file.

SETTING	EFFECT
Stations	Scale factor along the alignment element. Used for decompressing the digitized lines to their normal length.
Elevations	Scale factor for elevation values. Used for resolve the exaggeration of elevation values of the digitized lines.

At the moment, reference files and thus, the **Import Road Breaklines** tool can only be handled in MicroStation. There is not yet a comparable concept in Spatix.

Label Alignment Curvature

Label Alignment Curvature tool computes the radius of curves of 3D line elements. It places text elements that show the horizontal or vertical curvature radius.

For roads, the curvature radius determines the best value for the side slope inside a curve. Therefore, the tool supports the analysis of road surface properties.

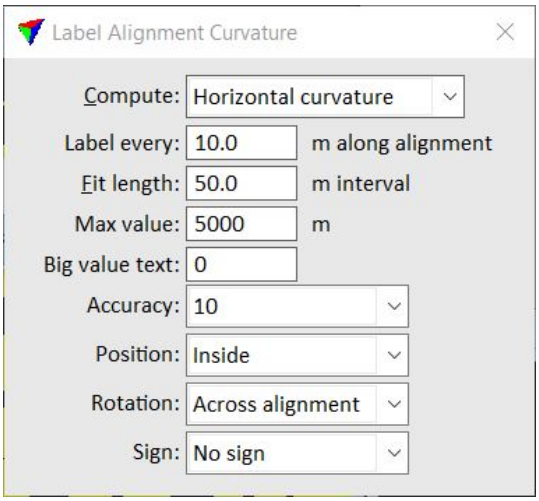
The tool requires an alignment element, which can be the centerline of a road, railroad, etc. derived from trajectory lines or any other representative line element.

To create labels for the radius of curves:

1. Select the alignment element with any **Selection** tool.

2. Select **Label Alignment Curvature** tool.

   This opens the **Label Alignment Curvature** dialog:



3. Define settings.

4. Place a data click inside a CAD view.

   This creates the curvature labels. The software draws text elements along the alignment element. The level, color, and text size of the labels are determined by the active symbology and text size settings in the CAD file.

   An information dialog shows the number of created curvature labels.

SETTING	EFFECT
Compute	Curvature to compute: **Horizontal curvature** or **Vertical curvature**.
Label every	Distance between locations along the alignment element where the software places a curvature label.

SETTING	EFFECT
Fit length	Length of an interval along the alignment element from which the software computes the curvature radius. From each labeling point, the software uses half of the given length forward and backward along the element and fits a circle. The radius of the circle determines the curvature radius at this labeling point.
Max value	Maximum curvature radius that is labeled.
Big value text	Text used for radius values larger than the given **Max value**.
Accuracy	Accuracy of curvature labels. Values are rounded to the given accuracy, e.g. to the closest 10m value.
Position	Determines the placement location of the labels relative to the alignment element: • **On alignment** - on the alignment element. • **Left** - left of the alignment according to digitization direction. • **Right** - right of the alignment according to digitization direction. • **Inside** - on the inside of a curve. • **Outside** - on the outside of a curve.
Rotation	Determines the placement rotation of the labels relative to the alignment element: • **Along alignment** - reading direction is parallel to the alignment. • **Across alignment** - reading direction is perpendicular to the alignment.
Sign	Sign added in front of the curvature radius value: • **No sign** - no sign is added. • **Left negative** - a minus sign is added to left-hand curves. • **Right negative** - a minus sign is added to right-hand curves.

You can undo the creation of curvature labels by using the **Undo** command of the CAD platform.

Label Clearance

Label Clearance tool labels the smallest distance between two point classes within a given area. The area is defined by one or more selected polygons.

The tool expects that a distance from ground value is computed for the points in the source class for clearance analysis. The value can be computed by using the Compute distance command for loaded points or the Compute distance macro action. Within each polygon, the tool finds the point with the smallest distance value in the source class. It classifies the point, and draws a line marker and a text element into the CAD file. The vertical line marker marks the location of the smallest distance. The text element shows the corresponding distance value.

The readability of the text element is determined by a given rotation setting. It is either readable in top views or in section views. If trajectory information is loaded into TerraScan, the text is automatically rotated in the direction of the system movement. If no trajectory information is available, the text can be rotated according to the drawing direction of the polygon that defines the area for clearance analysis. For example, if the first edge of the polygon is drawn in East-West direction, the text element is rotated to be readable in East-West direction.

The **Label Clearance** tool runs on points loaded into TerraScan.

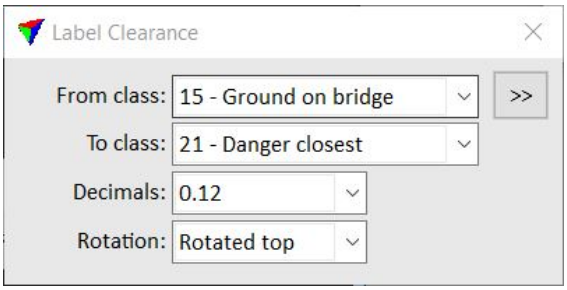
To label the clearance:

1. Draw a polygon around each location for which to label the clearance.

2. Select all polygons for which to label the clearance.

3. Select the **Label Clearance** tool.

   This opens the **Label Clearance** dialog:



4. Define settings.

5. Confirm the polygon selection with a data click inside a view.

   This classifies the point with the smallest distance value inside each polygon. In addition, it creates a text element showing the distance value and a vertical line element showing the location of the smallest distance. The text and line elements are drawn on the active level using active symbology settings of the CAD file.

SETTING	EFFECT
From class	Point class(es) from which to detect the smallest distance value.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **From class** field.
To class	Target class into which the point with the smallest distance value is classified. The list contains the active classes in TerraScan.
Decimals	Amount of decimals shown in the label of the smallest distance. Zero to three decimals are available.
Rotation	Determines the readability of the label: • **Top** - the text element is readable in a top view with north direction being up. • **Rotated top** - the text element is readable in a top view. It is rotated horizontally in drive/flight direction (if trajectory information is loaded into TerraScan) or in the direction of the first edge drawn for the polygon. • **Cross section** - the text element is readable in a cross section view. The viewing direction is in drive/flight direction (if trajectory information is loaded into TerraScan) or in the direction of the first edge drawn for the polygon.

You can undo the labelling of clearance by using the **Undo** command of the CAD platform (text and line placement) and the Undo command from the **Point** pulldown menu of TerraScan (point classification).

Write Alignment Elevations

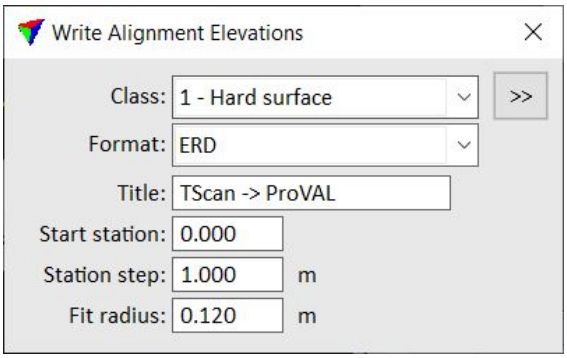
Write Alignment Elevations tool writes elevation values derived from point cloud data along an alignment. The process is based on the selection of two alignments, one defines the alignment for which to write elevation values, the other provides the stationing. The first alignment element can be, for example, a line that runs along a part of a road and is located on a lane, the second alignment element could be the centerline of the whole road. However, both alignments can also be derived from the same vector element.

The software fits a plane equation to the laser points at regular steps along the alignment and derives an elevation value. The values are written into a report. Supported report formats are **S X Y Z** (Station, Easting, Northing, Elevation) and **ERD**. The **ERD** file format can be read into *ProVAL*, a software for road profile analysis.

To write elevations along an alignment:

1. Select **Write Alignment Elevations** tool.

   This opens the **Write Alignment Elevations** dialog:



2. Define settings.

3. Select the first alignment element for which you want to write elevation values. You can select the element by placing a data click close to it.

4. Select the second alignment element from which you want to derive the stationing. You can select the element by placing a data click close to it.

   This derives the elevation values and opens the **Alignment elevations** window, a TerraScan report window.

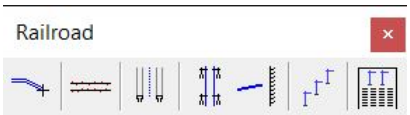
   You can save the report as text file or send it directly to a printer by using corresponding commands from the **File** pulldown menu of the report window. The size of the report window can be adjusted by commands from the **View** pulldown menu.

SETTING	EFFECT
Class	Point class(es) from which the elevation values are derived.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Class** field.
Format	Report format: • **S X Y Z** - Station Easting Northing Elevation • **ERD** - ERD file format including a header and the elevation values.
Title	Text used as title in an ERD file. This is only active if **Format** is set to **ERD**.
Start station	Station value of the first station along the alignment.
Station step	Interval size for deriving elevation values.

SETTING	EFFECT
Fit radius	Radius around each station within which the points are used to compute the elevation value.

Railroad toolbox

The tools in the **Railroad** toolbox are used to place lines, to fit lines to classified points along rails, to find rails and overhead wires automatically, and to check end points of overhead wires.



TO	USE TOOL	
Place approximate railroad centerline		Place Railroad String
Fit railroad centerline to laser points		Fit Railroad String
Find rails using a rail section template		Find Rails
Find overhead wires		Find Wires
Check end points of overhead wires		Check Wire Ends
Find poles		Find Poles
Output pole information to a text file		Output Poles

Check Wire Ends

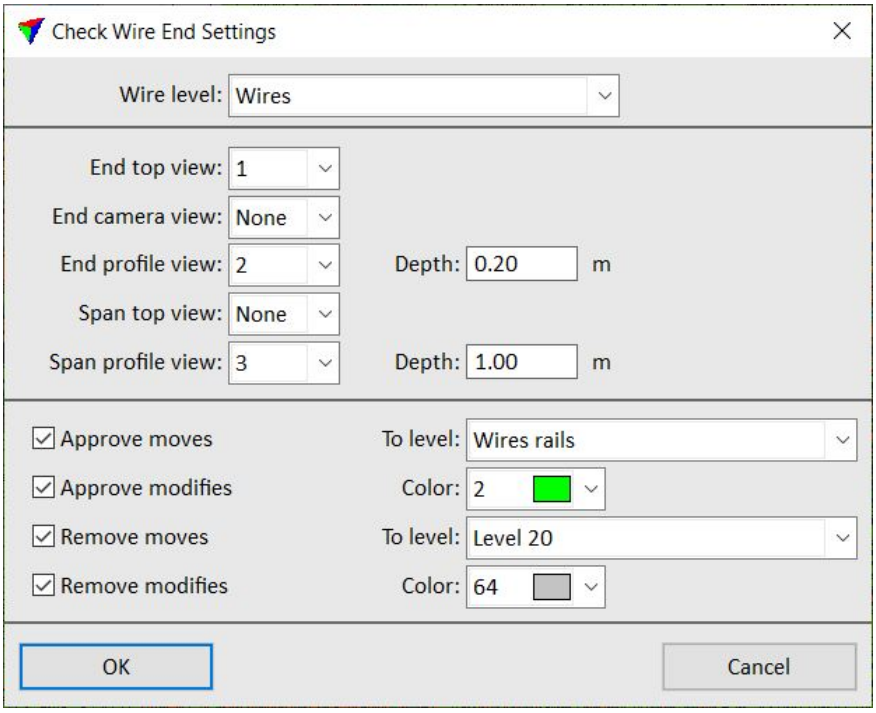
Check Wire Ends tool can be used to check automatically detected wires in an organized way. The tool opens the **Check wire ends** dialog which contains user controls for manipulating wires and wire end points. It is intended to be used after automatic wire detection by Find Wires tool.

The dialog shows a list that contains all wires and their end points. If a line in the list is selected, the software updates the display in a number of CAD file views. The views must be open and defined in the tools settings. The tool can update different view types, such as top, section, and camera views which show either a wire completely or the end point of a wire.

To check wire end points:

1. Select **Check Wire Ends** tool.

   This opens the **Check Wire End Settings** dialog:



2. Define settings and click OK.

   This opens the Check wire ends dialog.

SETTING	EFFECT
Wire level	CAD file level on which the lines for wires have been drawn. All lines on this level are added to the check list.
End top view	A top view showing the end of a wire is displayed in the given view.

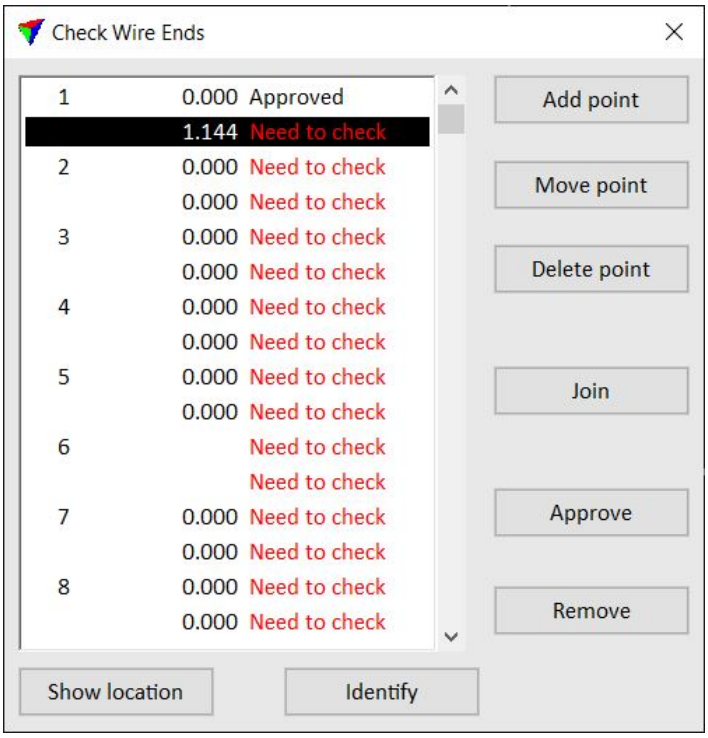
SETTING	EFFECT
End camera view	A camera view showing the end of a wire in displayed in the given view. This view can be used to display images if a mission, camera, and image list are loaded into TerraPhoto.
End profile view	A longitudinal section of the end of a wire is displayed in the given view.
Span top view	A rotated top view showing a wire completely in a horizontal section is displayed in the given view.
Span profile view	A longitudinal section showing a wire completely is displayed in the given view.
Approve moves	If on, the line of an approved wire is moved to the level defined in the **To level** list. If off, the level of approved wires remains unchanged.
Approve modifies	If on, the line color of an approved wire is modified to the given **Color**. The list contains the active color table of the CAD file. If off, the color of approved wires remains unchanged.
Remove moves	If on, the line of a removed wire is moved to the level defined in the **To level** list. If off, the level of removed wires remains unchanged.
Remove modifies	If on, the line color of an removed wire is modified to the given **Color**. The list contains the active color table of the CAD file. If off, the color of removed wires remains unchanged.

Check Wire Ends

The **Check Wire Ends** dialog shows a list that contains all wires and their end points. For each wire, there is a number and two end points. The status of each wire end point is *Need to check* by default.

Further, the dialog contains buttons that can be used to manipulate wire lines, to change the status of a wire in the list, and to display wire end locations. You can add intermediate vertices to wire lines, move the end points of wire lines, delete points from wire lines, and join wire lines in order to bridge gaps in the automatically detected wires.

If an end point of a wire is moved close to an end point of another wire which is an potential end point for a join, the horizontal and vertical distances between the end points are shown in the dialog.



To show the location of a wire end point, select a line in the **Check Wire Ends** dialog. Click on the **Show location** button and move the mouse pointer into a view. This highlights the selected wire end point with a cross.

To identify a wire end point, click on the **Identify** button and place a data click close to a wire end point in a view. This selects the corresponding line in the **Check Wire Ends** dialog.

After checking a wire end point and possibly improving its location, click on the **Approve** button. This changes the status of the selected end point to **Approved**. If both end points of a wire are approved, the wire line is moved to another level and/or the color is changed, if the settings in the **Check Wire End Settings** dialog are defined accordingly.

If you want to delete a wire, you click on the Remove button. This removes the selected wire from the list and the wire line is moved to another level and/or the color is changed, if the settings in the **Check Wire End Settings** dialog are defined accordingly. The **Remove** button does not delete the wire line from the CAD file.

To modify wire lines and their end points, you can use the other buttons of the dialog as described below. You can undo the modification of wires by using the **Undo** command of the CAD platform.

To add a vertex to a wire:

1. Select the wire in the list.

2. Click on the **Add point** button and move the mouse pointer into a view, preferably a section view.

   This dynamically displays the new vertex and wire line at the mouse pointer location.

3. Define the location of the new vertex by a data click.

You can continue with step 3. The software lets you place only intermediate vertices for the selected wire.

To move an end point of a wire:

1. Select the wire end point in the list.
2. Click on the **Move point** button and move the mouse pointer into a view.

   This dynamically displays the new end point and the wire line at the mouse pointer location.
3. Define the location of the new end point by a data click.

   You can continue with step 3. The software lets you move only the selected wire end point.

To delete a point of a wire:

1. Select the wire in the list.
2. Click on the **Delete point** button and move the mouse pointer into a view.

   This dynamically highlights the point on the wire closest to the mouse pointer location.
3. Delete the point by a data click.

   You can continue with step 3. The software lets you delete end points and intermediate vertices of the selected wire.

To join wires:

1. Select a wire in the list.
2. Click on the **Join** button and move the mouse pointer into a view.

   This dynamically displays possible connection lines for the selected wire at the mouse pointer location.
3. Move the mouse pointer close to the end point of the wire to which you want to join the selected wire.
4. Confirm a connection line by a data click.

   This joins two wire lines. If the status of the effected wires was already **Approved**, it is set back to **Need to check**.

Manual changes of the wire lines do not effect the classification of laser points. If you want to refine the classification of laser points on wires, you can run a **By centerline** classification using the wire lines as centerlines with offset and elevation difference settings of ± a few centimeters.

Find Poles

Limited in Spatix

Find Poles tool is used for the automatic detection of pole objects based on point clouds. This makes sense if you need to detect and possibly vectorize a large amount of identical poles or other objects from a point cloud. It requires a few preparation steps, such as pre-classification of the point cloud and the definition of a sample object for each pole type to detect. Therefore, it is probably not the fastest solution for detecting and vectorizing just a few poles.

The steps for defining a sample object are described in the Object library category of TerraScan **Settings**.

Any overlapping strips in the laser data must be matched and overlap should be cut off before running the object detection. It is required to do a classification of the point cloud in order to separate points on potential objects from other points. This includes ground classification, computation of distance from ground, and above-ground classification by distance. Furthermore, it is required to assign groups in order to detect objects.

The object detection can be supported by an alignment element. A trajectory line drawn in the CAD file can be used as alignment. The object definition defines whether an object is located left or right of the alignment or, with other words, whether the scanner system was driven on the left side or right side of the object. It is recommended to use an alignment in order to speed up object detection significantly. In addition, the alignment element can be used to classify point groups by centerline. This classifies potential objects within an offset from the alignment into a separate class and thus, reduces the amount of data for the object detection.

In the object detection process, the software compares point groups in the project data set to point groups in the object library by testing different 3D rotations. If a match is found, it classifies the point group into the target class specified in the object definition. In addition, it can place cell elements in the CAD file. An example workflow for creating a cell element is provided in Object library category of TerraScan **Settings**. Cells are stored in cell libraries. A cell library must be attached to the CAD file before cells from the library can be used. The tool creates a copy of the orginal cell element for each detected object. The cell is shifted to the correct object location and rotated according to the object rotation. The cell is further clipped at ground elevation, if the corresponding setting for the object item is switched on. The extent of the cell is not adjusted to the point cloud. If no cell library is attached or if the cell for an object is not available, the process only classifies points.

The tool is most confident in detecting poles along roads or rail tracks that have been driven during the data collection. All scanners of a system should see a pole from the same side. Thus, most of the points are on the sides of the pole facing towards the road. Poles on side roads/rail tracks or on bridges away from the data collection path are not detected automatically.

The tool requires laser points loaded into TerraScan. However, the same process can be performed for a TerraScan project using the Find poles macro action and then, reading the pole cells from text files using the Read / Poles command.

To detect poles automatically:

1. Load points into TerraScan.

2. Select Scan Settings tool and open **Object library** category.

3. Click inside the rectangle field next to an object item in order to activate/deactivate object items for detection. Activate object items to detect in the point cloud. Deactivate all other object items.

4. Close the **TerraScan settings** dialog.

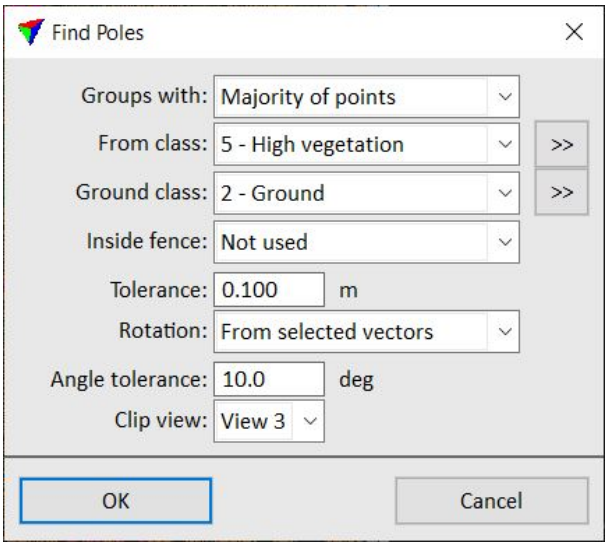
5. Draw a vertical section in one of the CAD file views.

6. (Optional, to place cells for poles) Open the **Cell Library** window of MicroStation and attach the cell library that contains the cell element(s) for the objects in the point cloud. *(Not Spatix)*

7. (Optional, but highly recommended) Select the alignment element.

8. Select **Find Poles** tool.

   This opens the **Find Poles** dialog:



9. Define settings and click OK.

   This starts the detection process. The software classifies point groups into the target class specified in the object definition and (optionally) places cell elements in the CAD file. The cells are only placed if a cell library is attached to the CAD file and if a cell with the name specified in the object definition is available. The cell is placed on the active level in the CAD file.

SETTING	EFFECT
Groups with	Determines which groups are used for detection: • **One or more points** - groups with one or more points in the source class. • **Majority of points** - groups with more than 50 % of points in the source class. • **All points** - groups with all points in the source class.

SETTING	EFFECT
From class	Point class that contains points on static parts of objects. These points of a group are compared to the sample object in the object library.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **From class** field.
Ground class	Point class that represents the ground. Used for clipping a cell at ground elevation in the **Clip view**.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Ground class** field.
Inside fence	Determines how a fence or selected polygon(s) effect the classification: • **Not used** - fence or selected polygons are ignored. • **One or more points** - groups are classified if one or more points are inside. • **Average xy** - groups are classified if the average xy point is inside. • **Majority of points** - groups are classified if more than 50 % of points are inside. • **All points** - groups are classified if all points are inside.
Tolerance	Maximum distance between a point in the sample object and a point from the object to be detected in a project point cloud. The elevation tolerance is further determined by multiplying this value with the **At center** and **At edge** factors given in the object item definition. For dense MLS point clouds, 0.1 m is the recommended setting. A smaller value may improve the result but slow down the process significantly. A larger value should be used for point clouds with a lower density. For low-altitude dense ALS point clouds of powerlines, 0.25 m may be a good estimate for detecting powerline towers. The larger the value, the higher the probability that objects are detected even though the point group differs from the sample object group.

SETTING	EFFECT
Rotation	Horizontal rotation of an object relative to the alignment: • **Free** - any rotation of the object is possible. This slows down the detection process significantly. • **From selected vectors** - the object is rotated only within the given **Angle tolerance** relative to the alignment. This is only active if an alignment element has been selected before the tool was started.
Angle tolerance	Maximum allowed angular difference between an object and the alignment. The larger the angle, the slower the process. Use a smaller value (e.g. around 3.0 deg) for objects that have the same rotation angle relative to the alignment, such as towers along a straight powerline. This is only active if **Rotation** is set to **From selected vectors**.
Clip view	CAD file view used for applying a clip mask to the object's cell element. The view must be a vertical section view. If **Clip to ground** setting is switched on for an object in the object item definition, the cell elements are clipped at the elevation of the **Ground class**.

You can undo the placement of cell elements for objects by using the **Undo** command of the CAD platform. You can undo the classification of points for objects by using the undo command of TerraScan.

At the moment, cells and thus, the cell placement capability of the **Find Poles** tool does only work in MicroStation. There is not yet any corresponding element type in Spatix.

Find Rails

Find rails tool is used for the automatic vectorization of rails based on MLS point clouds.

The software looks at consecutive cross sections of laser data along an alignment element. For each cross section, it tries to find the position where a user-defined cross section profile of the track matches the best number of laser points.

The vectorization process starts from an alignment element which represents the approximate centerline of a rail track. Any digitized centerline can be used as alignment. You can use, for example, Draw into design command for trajectories and apply a lever arm correction in order to derive a centerline from the trajectory. The lever arms are the three components of the vector between the IMU and the center of the rail track.

Alternatively, trajectories can be used directly for the rail detection. They must be imported with the correct system definition values for IMU misalignment. See Scanner systems category of TerraScan **Settings** for more information. In addition, they must be projected on the ground and to the center of the rail track. This can be established by applying a lever arm correction to the original trajectories by using the Add lever arm command. In the vectorization process based on trajectories, the software uses the roll angle of the trajectory positions as cant (superelevation) angle of the rail track.

The tool further requires a cross section profile defined in TerraScan **Settings**. The profile includes the two rails of a track, possibly places where there are no laser point (shadow parts of rails), and the location of lines that the software creates in the vectorization process. The creation of a rail track cross section is described in Rail section templates category of TerraScan **Settings**.

Any overlapping strips in the laser data should be matched and overlap should be cut off before running the rail vectorization. The By centerline classification routine should be used for an approximate classification of the points on rails. You can use, for example, the lever arm-corrected trajectory drawn in the CAD file for the classification by centerline.

The **Find Rails** tool runs on points loaded into TerraScan. It creates line string elements at the location(s) defined in the cross section profile.

To vectorize rails automatically:

1. Load points into TerraScan. Only points on and close to the rails are required.

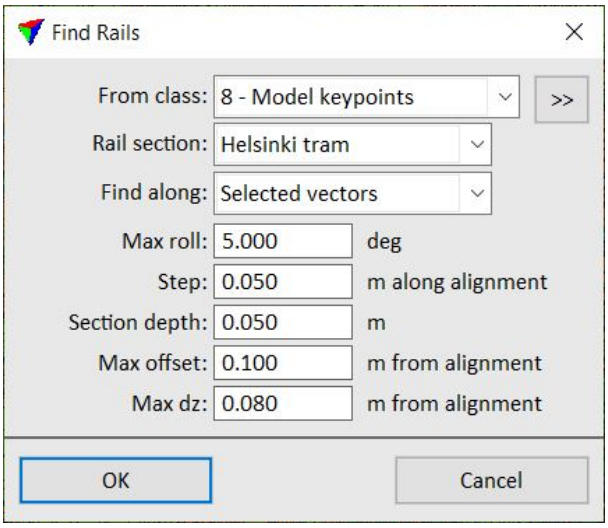
2. Select the alignment element with the **Selection** tool of your CAD platform.

   OR

2. Load trajectories into TerraScan using the Manage Trajectories tool.

3. Select **Find Rails** tool.

   This opens the **Find Rails** dialog:



4. Define settings and click OK.

This starts the vectorization process. The software draws line strings wherever it is able to fit the rail track cross section to the point cloud data. The level, color, line weight, and line style of the lines are determined by the active level and symbology settings of the CAD file.

Depending on the amount of laser data, the accuracy of the alignment element, and how well the software can fit the rail track cross section to the laser data, the process may take some time. It is recommended to test the settings for the tool with small data samples.

SETTING	EFFECT
From class	Point class that contains points on and close to the rails. Used for fitting the rail track cross section. The list contains the active classes in TerraScan.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **From class** field.
Rail section	Name of the rail track cross section. The list contains all sections that are defined in Rail section templates category of TerraScan **Settings**.
Find along	Defines the alignment element used for the vectorization: • **Trajectories** - active trajectories in TerraScan. • **Selected vectors** - a selected line string element.
Trajectories	Trajectory numbers that are used for the vectorization. Separate several numbers by comma. Type 0-65535 for using all trajectories. This is only active if **Find along** is set to **Trajectories**.
Max roll	Maximum value of the cant angle (= rail track superelevation). This is only active if **Find along** is set to **Selected vectors**.
Step	Distance between locations along the alignment where the software tries to fit the rail track cross section to the loaded points.
Section depth	Depth of a section in the point cloud data where the software fits the rail track cross section to the laser points.
Max offset	Maximum horizontal distance between the alignment and a line element that the software should draw as result of the vectorization process (usually a line on the rails or the track centerline).

SETTING	EFFECT
Max dz	Maximum vertical distance between the alignment and a line element that the software should draw as result of the vectorization process (usually a line on the rails or the track centerline).

You can undo the vectorization of rails by using the **Undo** command of the CAD platform.

Find Wires

Find wires tool is used for the automatic detection of overhead wires based on dense point clouds. The tool can be used for any kind of overhead wires, such as rail or tram wires, in contrast to the Detect Wires tools which is exclusively for the detection of powerline wires.

The detection process starts from classified laser points and, optionally, from an alignment element which runs in the direction of the wires. Any overlapping strips in laser point clouds should be matched and overlap should be cut off before running the wire extraction.

If data was captured by an MLS system mounted on a survey train, the overlap of parallel strips should be cut off in a way that points from a more distant drive path can be used for the wire detection. This leads to a more reliable result since wires are raised by the survey train in the closest drive path. As an alternative to cutting off overlap, the By section template classification routine can be used for classification of points from the closest drive path into a separate class.

Further, points should be classified into ground and above ground points. One of the above-ground point classes should contain the points on wires (e.g. the high vegetation class) and is then used as source class for the wire detection.

The **Find Wires** tool runs on points loaded in TerraScan. It classifies points on wires into a separate class and creates line string elements that are fitted to the points on wires. The software stops each wire at a small distance from its end points. The wire ends can be placed more accurately by using the Check Wire Ends tool.

The tool requires points loaded into TerraScan. However, the same process can be performed for a TerraScan project using the Find wires macro action and then, reading the wire lines from text files using the Read / Wires command.

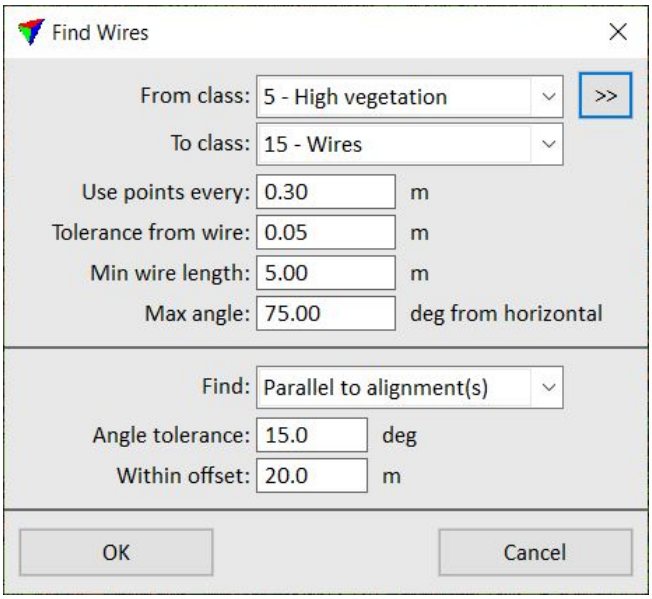
To detect wires automatically:

1. Load points into TerraScan. Only points on the wires are required.

2. (Optional) Select an alignment element with any **Selection** tool if you want to detect wires running parallel or perpendicular to the alignment.

3. Select **Find Wires** tool.

   This opens the **Find Wires** dialog:



4. Define settings and click OK.

   This starts the detection process. The software draws line strings wherever it is able to fit a line to the laser data. The level, color, line weight, and line style of the lines are determined by the active level and symbology settings of the CAD file.

SETTING	EFFECT
From class	Point class that contains points on the wires. Used for fitting the lines. The list contains the active classes in TerraScan.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **From class** field.
To class	Target class into which points on detected wires are classified.
Use points every	Distance between locations along a wire where the software tries to fit the line element to the laser points.
Tolerance from wire	Distance around a wire within which the software uses points for fitting the line element.
Min wire length	Minimum length of a line element at a wire location.
Max angle	Maximum vertical angle off from horizontal of a line element at a wire location.
Find	Defines what wires the software is searching for: • **All wires** - wires in all directions.

SETTING	EFFECT
	• **Parallel to alignment(s)** - wires that run parallel to the selected alignment element(s). • **Perpendicular to alignment(s)** - wires that run perpendicular to the selected alignment element(s).
Angle tolerance	Maximum horizontal angular difference between the alignment and a line element at a wire location. This is only active if an alignment element is selected and if **Find** is set to **Parallel to alignment(s)** or **Perpendicular to alignment(s)**.
Within offset	Maximum horizontal distance between the alignment and a line element at a wire location. This is only active if an alignment element is selected and if **Find** is set to **Parallel to alignment(s)** or **Perpendicular to alignment(s)**.

You can undo the detection of wires by using the **Undo** command of the CAD platform (vectorization) and the Undo command from the **Point** pulldown menu of TerraScan (classification).

Fit Railroad String

Fit Railroad String tool can be used to fit a manually placed railroad centerline to classified laser points. It is intended to be used after an approximate railroad centerline has been placed by using, for example, the Place Railroad String tool.

After placing an approximate track centerline, you can continue as follows:

1. Classify points on rails more accurately by using Railroad classification routine with the approximate centerline as alignment element.

   This classifies points with a specific elevation pattern and within a given offset (half of the rail width) from the alignment.

2. Use **Fit Railroad String** tool in order to fit the centerline to the classified points on the rails.

   OR

1. Classify ground using the Ground classification routine and drape the centerline to the ground elevation using the Drape Linear Element tool.

2. Classify points on the rails more accurately by using the By centerline classification routine with appropriate offset and elevation difference values.

3. Use **Fit Railroad String** tool in order to fit the centerline to the classified points on the rails.

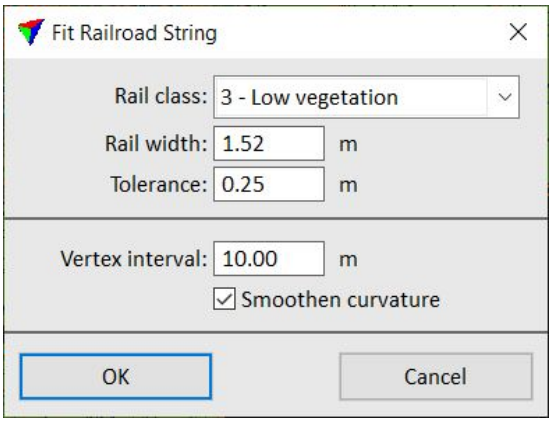
The tool uses points on rails within an offset distance in order to find the best location for the centerline. The offset depends on the width of the rail track and the initial accuracy of the centerline elements. The offset is defined in the tool's dialog as (0.5 * Rail width) ± Tolerance. The elevation of the fitted centerline is derived from the elevation values of the laser points.

To fit railroad centerlines:

1. Select the centerline element(s) using any **Selection** tool.

2. Select the **Fit Railroad String** tool.

   This opens the **Fit Railroad String** dialog:



3. Define settings and click OK.

   The application compares each selected line element with points in the given **Rail class** within the given offset from the centerline. It creates a new line string element for the fitted centerline which is drawn on the active level using the active symbology settings of the CAD file.

SETTING	EFFECT
Rail class	Point class that contains points on rails.
Rail width	Rail track width, distance from the center of one rail to the center of the other rail.
Tolerance	Tolerance value for the offset between centerline and rails. This should be big enough to compensate some locational inaccuracy in the initial centerline and in laser points. However, it should be less than half of **Rail width**.
Vertex interval	Maximum distance between vertices of the fitted centerline. Normally between 5.0 and 25.0 meters.
Smoothen curvature	If on, the fitted centerline is smoothed by balancing angular direction changes between consecutive vertices. Normally, this should be switched on.

Output Poles

Not Spatix

Output Poles tool creates a tabulator-delimited text file for selected pole cells. The cells must have been placed by the Find Poles tool. The text file contains the attributes of the pole cell, such as the name of the pole cell, XY coordinates of the pole, elevation coordinate of the pole base point, angle off from the alignment element, and lean off from vertical.

To output pole cells:

1. Select pole cells with any **Selection** tool.
2. Select **Output Poles** tool.

   This opens the **Pole output file** dialog, a standard Windows dialog for saving files.
3. Define a location and name for the output file. You may add an extension to the file name, such as .CSV or .TXT.
4. Click **Save** in order to save the text file.

Place Railroad String

Place Railroad String tool can be used for the digitization of line strings. The tool integrates three types of functionality: it draws a line element and allows view panning and rotation. Thus, it enables faster digitization compared with other CAD platform tools.

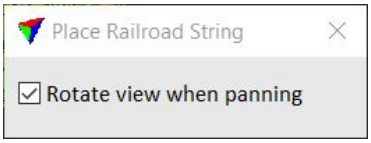
Initially, the tool was implemented for the manual placement of an approximate centerline between two rails based on ALS data or aerial images. The centerline can be used to classify points on rails more accurately. However, the tool is very useful for digitizing any kind of line string.

The **Tentative** mouse button can be used to snap to a point loaded into TerraScan while drawing a line string with the tool. This may be useful for digitizing a 3D line element directly. However, in many cases it is more convenient to digitize a 2D line first and then, adjust it to the point cloud elevation by using the Drape linear element tool.

To place a line string:

1. Select the **Place Railroad String** tool.

   The **Place Railroad String** dialog opens:



2. Define the location of the first point on the line string with a data click.

The application draws a dynamic rectangle whenever you move the mouse pointer inside the view. If you place a data click outside the rectangle, the application pans the view in the direction of the data click. If **Rotate view when panning** is switched on in the tool's dialog, the view is also rotated in the direction of the data click. If you place a point inside the rectangle, you add a new vertex to the line string.

3. Digitize the complete line string.

4. After placing the last vertex, click on the reset button in order to finish the line string.

   The lines string is drawn on the active level and using the active symbology settings of the CAD file.

To prepare for railroad line string placement:

1. Classify potential points on rail into a separate point class using By intensity or Railroad classification routines.

   This initial classification probably includes a number of points which are not points on rails. However, the should provide a visual impression of the railroad track location.

2. Switch on the display of the points on rails and switch off the display of all other point classes in a top view.

3. (Optional) Define a custom line style in your CAD platform.

   A recommended line style consists of two lines which are the railroad width apart from each other. This supports the placement of parallel lines for the rails and allows viewing the railroad string as a centerline of the track or as a pair of lines for the rails simply by switching line styles on or off in a view.

4. Select level, color, and the custom line style as the active symbology in the CAD file.

5. Select the **Place Railroad String** tool and digitize the railroad string according to the instructions above.

Trees toolbox

Tools in the **Trees** toolbox are used to place, modify, and output tree cells. Tree cells can be used to map trees of different species based on point clouds and (optional) images.

TO	USE TOOL	
Create tree cells from point groups		Create Tree Cells

TO	USE TOOL	
Place a single tree cell manually		Place Tree Cell
Modify tree cell information		Modify Tree Cells
Output tree cell information		Output Tree Cells

At the moment, cells and thus, the tools of the **Tree** toolbox do only work in MicroStation. There is not yet any corresponding element type in Spatix.

Create Tree Cells

Not Spatix

Create Tree Cells tool places cells for trees into the CAD file. It fits the cell element to the point cloud following two alternative methods. One method uses the points on the tree crown for fitting, the other method points from the tree trunk. The tool requires some preparation steps:

- (For **Highest point** method only) Creation of 3D cell elements in a MicroStation cell library. A tree cell should represent the stem and the crown of a tree. An example library is provided with the TerraScan installation. The library is stored in the \CELL folder of the Terra applications installation directory, for example C:\TERRA64\CELL\KARTTALI.CEL. It can be used, for example, to test the tools of the Trees toolbox.
- Definition of tree species and usage of cells in Tree species category of TerraScan **Settings**.
- Classification of ground in the point cloud.
- Grouping and tree classification based on groups of the point cloud. Trees may be classified using By parameters routine, By best match routine, or Trees routine. Any other way of classifying point groups into separate tree species classes is valid.

Create Tree Cells tool fits the cell elements to the points in a corresponding point class. Therefore, there should be a cell element, a species definition, and a separate point class for each tree species that you want to map.

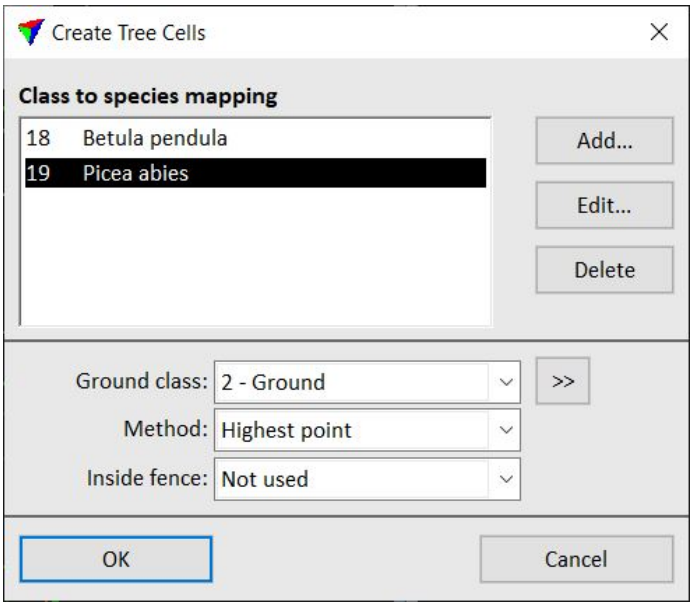
The tool lets you define rules for mapping trees. The rules rely on the tree species definitions in the **Settings**. Each rule defines the point class for tree detection and the common and scientific names of a tree species. You can **Add**, **Edit**, and **Delete** rules by using the corresponding buttons in the tool dialog.

To create tree cells:

1. (Optional) Draw a fence around the area in which you want to create tree cells.

2. Select **Create Tree Cells** tool.

   This opens the **Create Tree Cells** dialog:

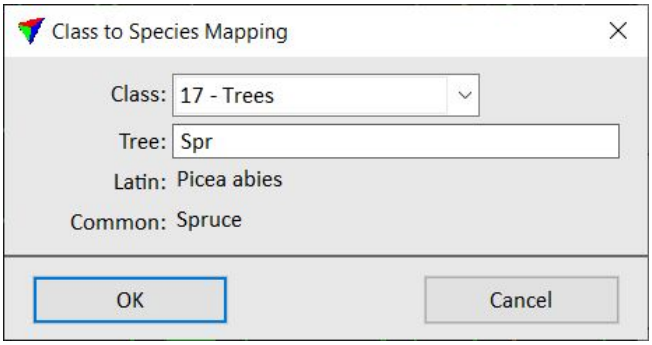


SETTING	EFFECT
Add	Add a new rule for mapping a tree species.
Edit	Modify the selected rule for mapping a tree species.
Delete	Delete the selected rule.
Ground class	Point class(es) that define the base point of trees.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Ground class** field.
Method	Method of cell placement: • **Highest point** - tree cell is matched to the tree crown. The cell defined in the Tree species category of TerraScan **Settings** is used. The cell provides the XY location of the tree crown center. • **Trunk** - tree cell is matched to the tree trunk. The cell is created automatically from the points in the point class that is used for mapping the tree. The cell provides the exact XY location of the tree trunk on the ground.
Inside fence	Determines how a fence or selected polygon(s) effect the cell placement: • **Not used** - fence or selected polygons are ignored. • **One or more points** - a cell is placed if one or more points of the tree group are inside.

SETTING	EFFECT
	• **Average xy** - a cell is placed if the average xy point of the tree group is inside. • **Majority of points** - a cell is placed if the majority of points of the tree group is inside. • **All points** - a cell is placed if all points of the tree group are inside.

3. Click on the **Add** button.

   This opens the **Class to species mapping** dialog:



SETTING	EFFECT
Class	Point class containing points of one tree species.
Tree	Tree species. Start typing the common or Latin (scientific) name of a tree species. If the name is found in Tree species category of TerraScan **Settings**, the **Latin** and **Common** names are displayed after you typed a few characters.

4. Define settings and click OK.

5. Repeat steps 3 and 4 for all tree species that you want to map.

6. Define additional settings in the **Create Tree Cells** dialog.

7. Click OK in order to start the placement of cells.

   The software places the cell elements according to the species mapping and the settings for tree cells.

Single tree cells can be placed manually by using the Place Tree Cell tool. Use the Modify Tree Cells tool to check and approve automatically created tree cells.

Modify Tree Cells

Not Spatix

Modify Tree Cells tool lets you check and modify tree cell elements that have been placed with the Create Tree Cells tool. The tool opens a list of selected tree cells. Based on the list, you can display the tree cells, check their attributes and size, and possibly modify the tree species name and the cell's height, width, and trunk width. You can also move the cell completely to another location.

The work can be supported by images. This requires a mission and image list in TerraPhoto. More information can be found in the TerraPhoto User Guide.

For displaying a selected tree, the tool supports automatic display update in three CAD file views, one top view, one section view, and one camera view. Images can be displayed in the camera view.

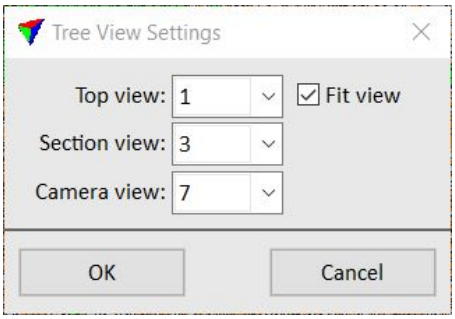
The list shows a flag for each tree. By default, the flag is set to *Check*. After you checked a tree, you may change it to *Approved*.

To merge two point groups:

1. Select tree cells with any **Selection** tool.

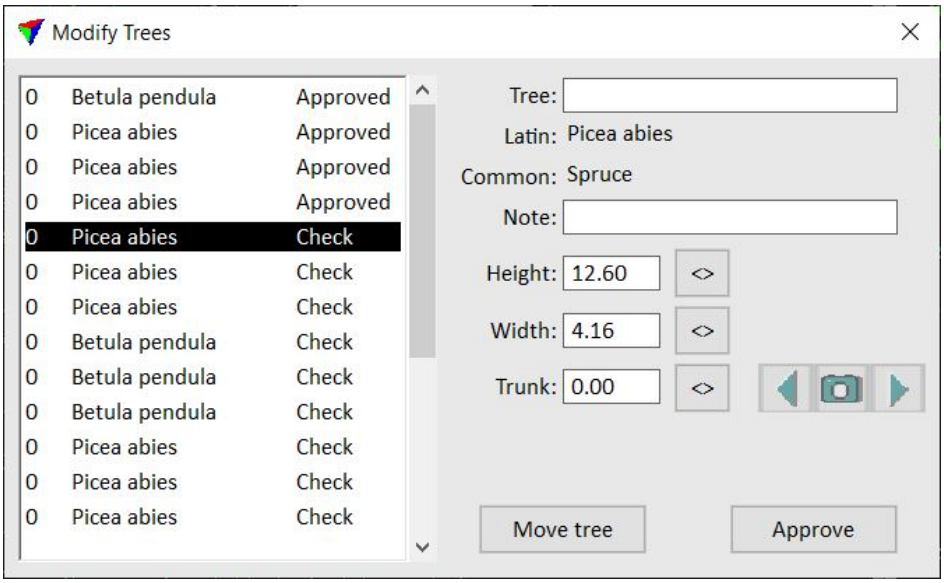
2. Select **Modify Tree Cells** tool.

   This opens the **Tree View Settings** dialog:



3. Select CAD file views in which you want to display the tree cells and click OK.

   This opens the **Modify Trees** dialog:



The dialog contains a list of all selected tree cells. For a selected tree in the list, the attributes are shown on the right side of the dialog.

SETTING	EFFECT
Tree	Tree species. Start typing the common or Latin (scientific) name of a tree species. If the name is found in Tree species category of TerraScan **Settings**, the **Latin** and **Common** names are displayed after you typed a few characters.
Note	Free text field for adding a note to a tree.
Height	Height of the tree represented by the cell height. You can change the height by typing a new value or using the button right of the field.
Width	Width of the tree crown represented by the cell width. You can change the width by typing a new value or using the button right of the field.
Trunk	Width of the tree trunk at its base point. You can change the width of the trunk by typing a new value or using the button right of the field.
	Activates the manual modification of the cell. After clicking the button, move the mouse pointer into a top or section view. The extend of the tree is dynamically displayed. Define the new height, width, or trunk width with a data click.
	Click on the camera button in the middle of the button group in order to identify an image for display. Move the mouse pointer into a view. The image footprint closest to the mouse

SETTING	EFFECT
	pointer is dynamically displayed. Select an image for display with a data click. Click on the arrow buttons left and right in the button group in order to select the previous or next image from the currently displayed image in the images list.
Move tree	Moves the tree cell to another location. After clicking the button, move the mouse pointer into a top or section view. The cell of the tree is dynamically displayed at the mouse pointer location. Define the new location with a data click.
Approve | Set to check	After checking a tree cell and possibly modifying it, click on the **Approve** button. This changes the status of the selected model to *Approved*. If an approved tree is selected in the list, the button changes to **Set to check**. Click on the button in order to change the status of the selected model to *Check*.

Output Tree Cells

Not Spatix

Output Tree Cells tool creates a tabulator-delimited text file for selected tree cells. The text file contains the attributes of the tree cell, such as the scientific name of the tree species, XY coordinates of the tree depending on the cell placement method (**Highest point** - center of the tree crown or **Trunk** - trunk base point on the ground), elevation coordinate of the tree XY point on the ground, height, crown width, and trunk base width.

The text file is suited for importing tree information into a database. Tree cells can also be drawn into a CAD file by reading the file with the Read / Tree cells command from the TerraScan window.

To output tree cells:

1. Select tree cells with any **Selection** tool.
2. Select **Output Tree Cells** tool.

   This opens the **Tree output file** dialog, a standard Windows dialog for saving files.
3. Define a location and name for the output file. You may add an extension to the file name, such as .CSV or .TXT.
4. Click **Save** in order to save the text file.

Place Tree Cell

Not Spatix

Place Tree Cell tool lets you place a tree cell manually. The cell placement requires the following preparation steps:

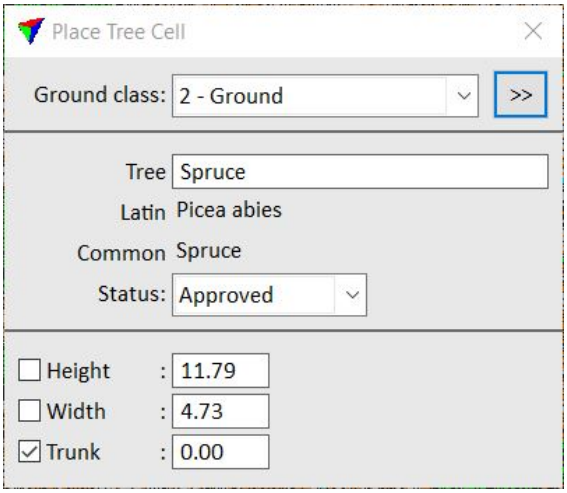
- Creation of 3D cell elements in a MicroStation cell library. A tree cell should represent the stem and the crown of a tree. An example library is provided with the TerraScan installation. The library is stored in the \CELL folder of the Terra applications installation directory, for example C:\TERRA64\CELL\KARTTALI.CEL. It can be used, for example, to test the tools of the Trees toolbox.
- Definition of tree species and usage of cells in Tree species category of TerraScan **Settings**.
- (Optional) Classification of ground in the point cloud.

A tree cell can be placed, for example, based on a vertical section view of a tree in a point cloud. The base point of the tree can be fixed to the ground elevation if the ground is classified in the point cloud.

To place a tree cell manually:

1. Select **Place Tree Cell** tool.

   This opens the **Place Tree Cell** dialog:



2. (Optional) Select a **Ground class**.

3. Start typing the common or Latin name of a tree species in the **Tree** field.

4. Define additional settings in the **Create Tree Cell** dialog, if necessary.

5. Define the base point of the tree cell with a data click in a top or section view.

   If the data click is placed in a another view than a section view (typically in a top view), the base point elevation is fitted to the given **Ground class**(es) and the XY location is defined by the data click. If the data click is placed in a section view, the base point elevation is determined by the data click and the XY location by the center of the vertical section.

6. Move the mouse pointer inside a section view.

The height of the tree cell is dynamically displayed.

7. Define the highest point of the tree cell with a data click in a section view.

   The width of the tree cell is dynamically displayed.

8. Define the width of the tree cell with a data click in a top or section view.

   The trunk width of the tree cell is dynamically displayed.

9. Define the trunk width of the tree cell with a data click in a section view.

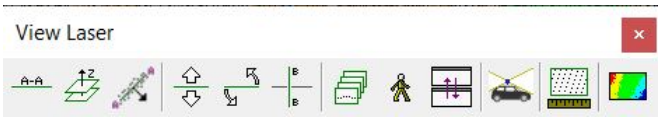
   The software places the cell element according to the settings for tree species.

SETTING	EFFECT
Ground class	Point class(es) that may define the base point of a tree.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Ground class** field.
Tree	Tree species. Start typing the common or Latin (scientific) name of a tree species. If the name is found in Tree species category of TerraScan **Settings**, the **Latin** and **Common** names are displayed after you typed a few characters.
Status	Sets the status of the tree cell: **Check** or **Approved**. The status flag is used by the Modify Tree Cells tool.
Height	If on, the height of the tree cell is fixed to the given value and can not be changed dynamically anymore.
Width	If on, the width of the tree cell is fixed to the given value and can not be changed dynamically anymore.
Trunk	If on, the trunk width of the tree cell is fixed to the given value and can not be changed dynamically anymore.

Tree cells can be placed in a more automatic way by using the Create Tree Cells tool.

View Laser toolbox

The tools in the **View Laser** toolbox are used to create and modify section views, create views along a path, create and move in perspective views, open Street View© images, measure point density, and to update distance coloring.



TO	USE TOOL	
Rotate view to show vertical cross section		Draw Vertical Section
Rotate view to show horizontal cross section		Draw Horizontal Section
Show points on a plane in a section view		Draw Plane Section
Move forward or backward in section view		Move Section
Rotate a section view around its center		Rotate Section
Cut perpendicular section from section view		Cut Section
Travel along path and display views		Travel Path
Setup and move in perspective view		Travel View
Define automatic synchronization of views		Synchronize Views
Open a Street View© image in a browser		Show Street View
Measure point density		Measure Point Density
Recompute distance colors and update views		Update Distance Coloring

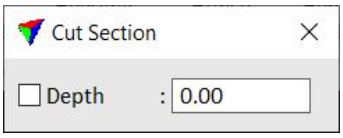
Cut Section

Cut Section tool creates a vertical section view which is perpendicular to another vertical section view. In addition to just rotating the section by 90 degree, the cut section tool allows you to define another depth for the new section view.

To cut a perpendicular section view:

1. Create a vertical section view using the Draw Vertical Section tool.
2. Select the **Cut Section** tool.

   This opens the **Cut Section** dialog:



3. Define the position of the new section's center line with a data click in the section view.

   The center line of the new section is defined by the given position perpendicular to the center line direction of the source section.

4. Define the section view depth by placing a data click or by typing a value in the **Depth** field of the **Cut Section** dialog.

5. Identify a view for displaying the new section with a data click inside the view.

   The selected view is rotated to show the new section.

SETTING	EFFECT
Depth	Display depth of a section on both sides of the center line. If on, the depth is fixed to the given value.

Draw Horizontal Section

Draw Horizontal Section tool creates a top view which shows laser data and vector elements in a limited elevation range (display or viewing depth).

Horizontal sections are useful, for example, to display the exact XY location of vertical objects, such as building walls or poles in MLS data sets. You should open at least one top view and one (vertical) section view before starting to create horizontal sections.

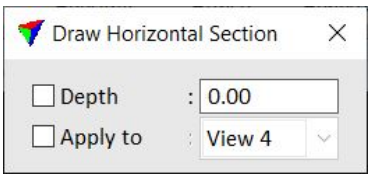
To create a horizontal section view:

1. In the top view, zoom to the area of interest that you want to display in the horizontal section.

2. Use the Draw Vertical Section tool in order to create a vertical section view of the area of interest.

3. Select the **Draw Horizontal Section** tool.

   The **Draw Horizontal Section** dialog opens:



4. Define the center elevation of the horizontal section with a data click in the vertical section view.

5. Define the horizontal section display depth (= visible elevation range) with a data click in the vertical section view or by typing a value in the **Depth** field of the **Draw Horizontal Section** dialog.

6. If **Apply** to is not switched on in the **Draw Horizontal Section** dialog, identify the view for displaying the horizontal section with a data click.

   The selected view is rotated to a top view and displays the defined elevation range.

SETTING	EFFECT
Depth	Display depth or visible elevation range of a horizontal section view up and down from the center elevation. If on, the display depth is fixed to the given value.
Apply to	If on, the horizontal section is automatically displayed in the selected view.

The tool does not apply any XY adjustment to the horizontal section view based on the vertical section location.

Draw Plane Section

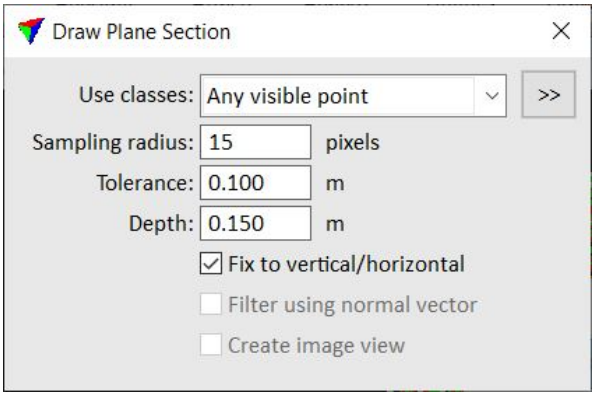
Draw Plane Section tool lets you set up a section that contains points on a plane. The user defines one or several circular sample locations in the point cloud from which the software derives a plane equation. Then, the software checks which points fit to the same plane and draws them into a section view. The tool can be used for drawing a section of any 3D plane.

The tool may be useful, for example, for manual 3D digitization work in a point cloud. In order to digitize the planar part of traffic signs, you can draw a plane section of the sign plate and digitize its boundary in the section view.

To draw a plane section:

1. Start **Draw Plane Section** tool.

   This opens the **Draw Plane Section** dialog:



2. Define settings.

3. Place a data click in a view that displays the point cloud. This can be a top or section view.

This computes the plane equation from the points inside the sample radius and highlights the are in which the points fit to this plane.

4. (Optional) Place more data clicks in the same view in order to refine the plane selection.

5. Place a data click into another view in order to draw the plane section.

   This displays the points that fit to the plane in the section view.

SETTING	EFFECT
Use classes	Point class(es) that must fit to the plane.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Use classes** field.
Sampling radius	Radial distance from a data click. The plane equation is derived from the points that are inside the sample radius.
Tolerance	Distance by which points are allowed to differ from a perfect plane. Points that are within the tolerance distance are consider as points belonging to the plane.
Depth	Display depth of the plane section on both sides of the section's center line.
Fix to vertical/horizontal	If on, the software forces the plane to be exactly vertical or horizontal if all points of the plane are within two degree from vertical/horizontal.
Filter using normal vector	If on, only points of planar dimension are used for drawing the plane section. This requires the computation of normal vectors for the points.
Create image view	If on, an image is displayed in the background of the plane section view. This requires that TerraPhoto is available, and that a mission and image list are loaded. See TerraPhoto User Guide for more information.

Draw Vertical Section

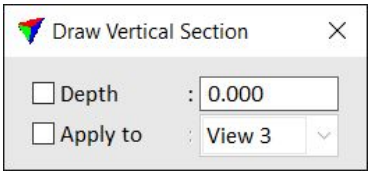
Draw Vertical Section tool creates a 3D section view from a location defined by a center line of the section and its depth.

A vertical section view is simply a rotated CAD file view which displays all visible CAD file elements and laser points inside the given slice of space. This makes it well-suited for viewing laser points and for placing 3D vector elements.

To create a vertical section view:

1. Select the **Draw Vertical Section** tool.

   The **Draw Vertical Section** dialog opens:



2. Define the start or left point of the section center line with a data click in a top view.

3. Define the end or right point of the section center line with a data click in a top view.

4. Define the section view depth with a data click in a top view or by typing a value in the **Depth** field of the **Draw Vertical Section** dialog.

5. If **Apply to** is not switched on in the **Draw Vertical Section** dialog, select the view for displaying the section with a data click inside this view.

   The selected view is rotated to show the vertical section. The application automatically computes the required elevation range so that all laser points inside the given section space are displayed.

SETTING	EFFECT
Depth	Display depth of a section on both sides of the center line. If on, the depth is fixed to the given value.
Apply to	If on, the section is automatically displayed in the selected view.

Measure Point Density

Measure Point Density tool displays the average number points per squared master unit. You can measure the point density in a rectangular or circular area, inside selected polygons, or from the whole data set. The measurement can be based on points loaded into TerraScan or points residing in the active project.

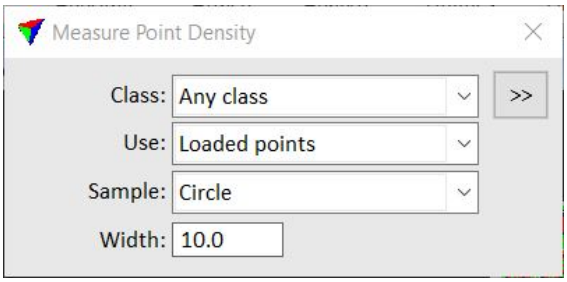
The point density values are displayed in the information bar at the bottom of the CAD platform interface. The values include the amount of points per sample area and the average point density.

If the point density is measured in selected polygons, the tool places a text element inside each polygon. The text element shows the point density. It is drawn with the active symbology settings of the CAD file.

To measure the point density from all loaded points:

1. Select the **Measure Point Density** tool.

   This opens the **Measure Point Density** dialog:



2. Define settings.

3. If **Sample** is set to **All points** or **Selected polygons**, place a data click anywhere in a CAD file view (**Loaded points**) or inside the project area (**Project points**).

   This displays the average point density of loaded points or points in the project.

   OR

4. If **Sample** is set to **Rectangle** or **Circle**, define the center point of the sample area with a data click.

   This displays the average point density inside the sample area.

SETTING	EFFECT
Class	The point density is computed for points of any class or of a specific class. The list contains the active class definitions in TerraScan.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Class** field.

SETTING	EFFECT
Use	Defines which points are used for the density measurement: • **Loaded points** - density is measured from points loaded in TerraScan. • **Project points** - density is measured from binary files referenced by the active project in TerraScan. *Not UAV*
Sample	Sample area for the density measurement: • **All points** - area covered by all points. • **Rectangle** - rectangular area. • **Circle** - circular area. • **Selected polygons** - one or more selected shape elements.
Width	Defines the width of a **Rectangle** or the diameter of a **Circle** depending on the setting in the **Sample** field. Given in master units of the CAD file.
Decimals	Number of decimals for labeling the point density inside polygons. This is only active if **Sample** is set to **Selected polygons**.

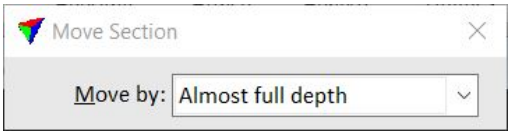
Move Section

Move Section tool lets you move stepwise forward or backward in section views. The tool is most useful in views created by Draw Vertical Section and Draw Horizontal Section tools.

To move sections forward or backward:

1. Start **Move Section** tool.

   This opens the **Move Section** dialog:



2. Move the mouse pointer into a section view.

   The area covered by a vertical section is highlighted by a rectangle in all top views.

3. Place a data click in order to move the section forward.

   OR

4. Place a reset click in order to move the section backward.

SETTING	EFFECT
Move by	Step size: • **Half of view depth** - the section is moved by half of the section's depth. If the section depth is 1 m, the section is moved 0.5 m with each mouse click. • **Almost full depth** - the section is moved by almost its full depth. This is the recommended setting if you zoom in/out in section views using the mouse wheel. Zoom by mouse wheel can lead to little inaccuracies between consecutive sections, so that points may be missed by classification tools when moving with full view depth. • **Full view depth** - the section is moved by its full depth. If the section depth is 1 m, the section is moved 1 m with each mouse click.

Rotate Section

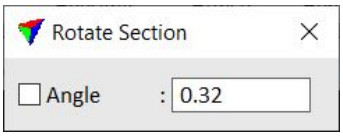
Rotate Section tool rotates a vertical section view stepwise around its center point.

The direction and angle of rotation can be determined by data clicks inside the section view or by a fixed value in the tool's dialog.

To rotate a section view:

1. Select the **Rotate Section** tool.

   This opens the **Rotate Section** dialog:



2. Move the mouse pointer into a section view.

   The area covered by a vertical section is highlighted by a rectangle in all top views.

3. Place a data click inside a vertical section view.

   If the data click is placed on the right side of the section view's center, the section is rotated counterclockwise. If the data click is placed on the left side, the rotation direction is clockwise.

   The angle of rotation is determined by the distance of a data click from the center of the section view or by a fixed value in the **Angle** field of the **Rotate Section** dialog.

SETTING	EFFECT
Angle	Rotation angle applied to a view with each data click. If on, the rotation is fixed to the given value. Positive values rotate in counterclockwise direction, negative values in clockwise direction.

Show Street View

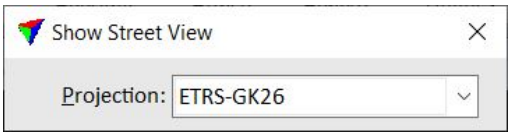
Show Street View tool opens a Google Street View© image in a browser. The Street View© image is determined by a mouse click. This requires that the projection system of the data set is selected in the tool's dialog.

The Street View images category of TerraScan **Settings** defines display settings for the images in the browser, as well as a key for accessing Street View© images. The access key can be changed to a company's own key, if necessary.

To show a Street View image:

1. Select the **Show Street View** tool.

   This opens the **Show Street View** dialog:



2. Select the correct projection system.

3. Place a data click inside a top view in order to determine the viewer location.

4. Place another data click inside the top view in order to determine the viewing direction and the target location.

   The standard browser opens the Google Street View© image that is closest to the viewer location and looks in the direction of the target location.

SETTING	EFFECT
Projection	Projection system of the data set. The list contains all projection systems that are activated in Coordinate transformations / Built-in projection systems, Coordinate transformations / US State Planes, and Coordinate transformations / User projection systems categories of TerraScan **Settings**.

Synchronize Views

Synchronize Views tool defines dependencies between CAD file views. The display in a dependent view is automatically updated, if the master view display changes. This is useful if you want to view the same location using two different types of content. For example, you may want to see an orthophoto and laser points side by side in two different top views.

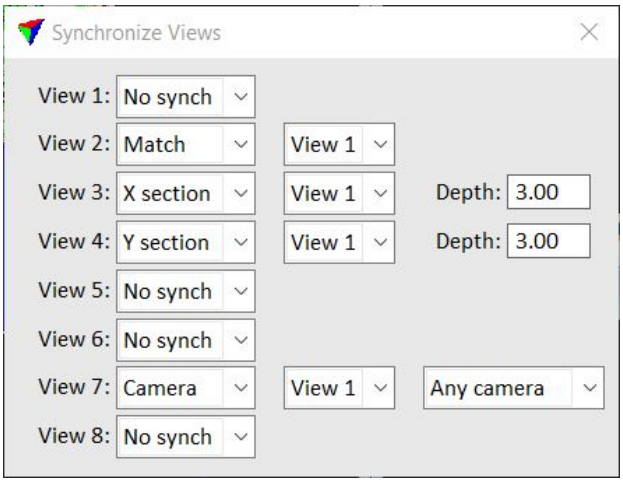
Synchronize Views tool can define the following dependencies:

- **No synch** - view works normally and does not depend on other views. This is the default setting.
- **Match** - dependent view shows the same area using the same rotation or perspective as the master view.
- **X section** - dependent view shows a cross section along the screen X axis of the master view.
- **Y section** - dependent view shows a cross section along the screen Y axis of the master view.
- **Front 3D** - dependent view is a 45 degree oblique view looking forward and down to the area displayed by the master view.
- **Side 3D** - dependent view is a 45 degree oblique view looking right and down to the area displayed by the master view.
- **Camera** - dependent view displays a camera view. This is useful to display images that are referenced by a TerraPhoto image list. See TerraPhoto User Guide for more information. The software chooses the image for display which best matches the viewing direction (for example: top view - nadir images, section view - oblique images) and sees the four corner points of the view. A camera view works best if there is one image covering the whole view.

To set up synchronized views:

1. Select the **Synchronize Views** tool.

   This opens the **Synchronize Views** dialog:



2. Define the dependencies by selecting a dependency type from the lists.

3. Define the master view for each dependency.

4. If required, define additional settings.

Whenever you move, pan, zoom, or redraw a master view, the dependent views are updated automatically.

SETTING	EFFECT
Depth	Depth of a dependent section view.
Camera	Name of a camera in a TerraPhoto **Mission**. Only images captured by this camera are displayed. This is only available for dependency type **Camera** and if a **Mission** is loaded in TerraPhoto.

Synchronization stays active if the **Synchronize Views** dialog is closed. If you want to release the view dependencies and stop synchronization, reopen the dialog and set all views to **No synch**.

Travel Path

Travel Path tool lets you view an animation along an alignment element. The tool provides an excellent way for traversing along the survey path and checking the data visually.

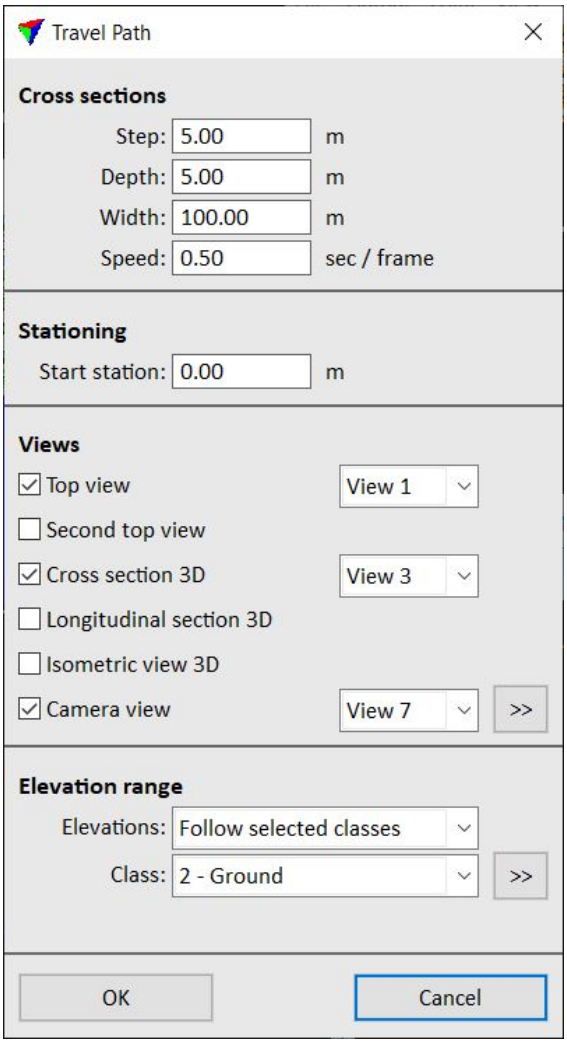
The alignment element can be any linear element. In most cases you create the element manually or draw, for example, a trajectory line into the CAD file by using the Draw into design command. Alternatively, you can use TerraScan's Draw from points command which draws an approximate flight path deduced from the order of loaded laser points.

You can define what kind of views you want to see while traveling along the alignment. Supported view types include top, cross section, longitudinal section, isometric and camera views.

To create an animation along an alignment:

1. Draw and select the alignment element.
2. Select the **Travel Path** tool.

   This opens the **Travel Path** dialog:



3. Define settings and click OK.

   The application constructs internal tables for the animation and then opens the **Travel Player** dialog.

SETTING	EFFECT
Step	Step along alignment between consecutive cross sections.
Depth	Full depth of each cross section. Each cross section covers a rectangular area defined by the **Depth** and **Width** values.
Width	Full width of each cross section. Each cross section covers a rectangular area defined by the **Depth** and **Width** values.
Speed	Speed for automatic animation display.
Start station	Defines the point on the alignment element from which the animation starts.

SETTING	EFFECT
Views	CAD file views that are used for displaying the animation: • **Top view** - displays data from the top. • **Second top view** - displays data from the top. • **Cross section 3D** - displays data in a cross section. • **Longitudinal section 3D** - displays data in a longitudinal section. • **Isometric view 3D** - displays data in an isometric view. • **Camera** - display data in a camera view. Click on the **>>** button in order to open the Travel Path Camera Settings dialog and define settings for the camera view.
Elevations	Method of elevation range computation which defines how the animation follows elevation changes in the data: • **Follow all points** - all points determine the visible elevation range. • **Follow selected classes** - points from selected classes determine the visible elevation range. Select a single class from the **Class** list. Click on the **>>** button in order to open the list of active classes and select several classes. • **Follow 3D alignment** - the alignment element determines the center elevation and the visible elevation range is determined by the **Minimum dz** and **Maximum dz** values given relative to the alignment element. • **Fixed** - a fixed elevation range defined by **Minimum z** and **Maximum z** values is used for the whole animation.

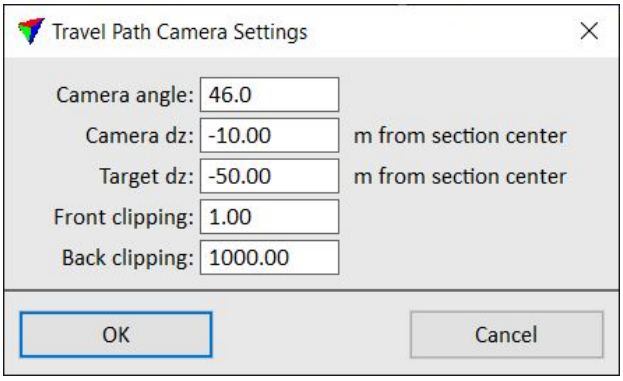
The **Travel Player** dialog contains the following commands and tools for traveling along the alignment element:

MENU/TOOL	COMMAND/TOOL NAME	EFFECT
File	**Using mouse**	Update a cross section view dynamically as you move the mouse pointer along the alignment. If you place a data click, the tool updates all views.
File	**To start**	Move to the start of the alignment and update all views.

MENU/TOOL	COMMAND/TOOL NAME	EFFECT
File	**To end**	Move to the end of the alignment and update all views.
	Play backward	Start automatic animation display backward along the alignment.
	Step backward	Move stepwise backward along the alignment.
	Stop	Stop automatic animation display.
	Step forward	Move stepwise forward along the alignment.
	Play forward	Start automatic animation display forward along the alignment.

Travel Path Camera Settings

The **Travel Path Camera Settings** dialog lets you define settings for a camera view.



SETTING	EFFECT
Camera angle	Field-of-view angle of the camera.
Camera dz	Altitude of the camera position. Defined as elevation difference from the section center (= alignment element).
Target dz	Altitude of the target position. Defined as elevation difference from the section center (= alignment element).
Front clipping	Distance up to which the content of the view is clipped in the foreground. Data is displayed in the range between **Front** and **Back clipping**.

SETTING	EFFECT
Back clipping	Distance after which the content of the view is clipped in the background. Data is displayed in the range between **Front** and **Back clipping**.

Travel View

Travel View tool lets you setup perspective views by defining a viewer position, viewer height above a reference surface, and vertical viewing angle. Further, the tool provides user controls for navigating in the perspective view.

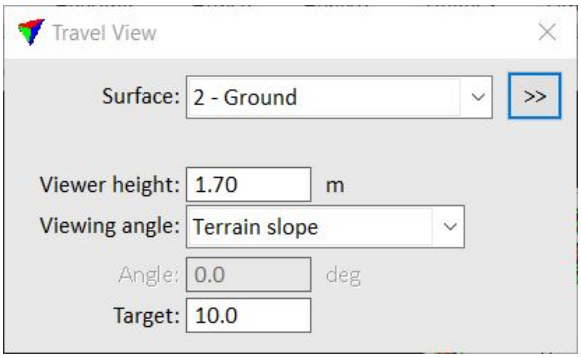
To setup and travel in a perspective view:

1. Open and setup a CAD file view that you want to use as perspective view.

   The setup may include display settings for the point cloud, vector data and imagery, if available.

2. Select the **Travel View** tool.

   This opens the **Travel View** dialog:



3. Define settings.

SETTING	EFFECT
Surface	Point class that defines the reference surface for the viewer. Used as the base elevation level for calculating the viewer position. Alternatively, select **Fixed elevation** and define the viewer's base point elevation in the **Base Z** field.
>>	Opens the **Select Classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Surface** field.
Base Z	Reference elevation for computing the viewer position. This is only active if **Surface** is set to **Fixed elevation**.

SETTING	EFFECT
Viewer height	Height of the viewer above the reference elevation.
Viewing angle	Defines the vertical viewing angle: • **Horizontal** - the viewer looks horizontally forward. • **Terrain slope** - the viewer looks according to the terrain slope in view direction. • **Angle down** - the viewer looks down by the given **Angle**. • **Angle up** - the viewer looks up by the given **Angle**.
Target	Maximum distance to define a target position.

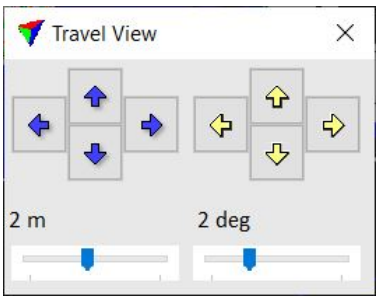
4. Define the viewer position by placing a data click inside a CAD file top view.

   This displays the viewer position and the viewing angle dynamically if the mouse pointer is moved.

5. Define the target position by placing another data click inside the CAD file top view.

6. Select another CAD file view as target view.

   This displays the point cloud in the target view which is changed into a perspective view. The software opens the **Travel View** dialog for navigating in the perspective view:



   Move the viewer forward/backward, left/right by using the blue arrow buttons on the left side of the dialog. Adjust the moving distance per button click with the slider on the left side. The distance can be adjusted to fixed values between 0.1 and 100 meters.

   Turn the viewing direction up/down, left/right by using the yellow arrow buttons on the right side of the dialog. Adjust the turning angle per button click with the slider on the right side. The angle can be adjusted to fixed values between 0.5 and 90 degree.

   After at least one data click on the buttons of the **Travel View** dialog, you can also use the <Arrow-up> and <Arrow-down> keys to move forward/backward, and the <Arrow-left> and <Arrow-right> keys to turn the viewing direction to the left/right.

   The other **Travel View** dialog is still available and settings can be adjusted. The new settings are applied when the next navigation step is done. Even if the **Travel View** dialogs are closed, the tool is still active and you can setup new perspective views by starting from step 4.

Update Distance Coloring

Update Distance Coloring tool recomputes distances of laser points to other laser points, surfaces or vector elements, and updates views in which distance coloring is active. A distance for each point must be computed first by using the Compute distance command.

You need to use this tool only if:

- you have classified points to or from classes involved in distance computation.
- you have transformed point XY coordinates or elevations.
- you have modified CAD file elements involved in distance computation.

For more information about distance coloring, see Color by Distance.

Waveform Processing

Not UAV

TerraScan is able to read waveform information from LAS 1.3 and 1.4 files, WDP files (external waveform data storage for LAS files) and from TopEye.TEW 1.15 (MarkII) files. It uses the waveform information for processing tasks. It is not possible to write out files that include waveform information.

TerraScan UAV does not include an waveform capabilities.

Waveform capabilities

If waveform data is available, you can perform the following processing steps:

- View Waveform for a point in a graph and export waveform information of a point into a text file.
- Extract echo properties for laser points:
  - **Echo length** - relative length (millimeter) of a return signal compared to a typical return from a hard surface.
  - **Echo normality** - difference in shape of a return signal compared to a typical return from a hard surface.
  - **Echo position** - difference in position of a peak of a return signal compared to a typical return from a hard surface.
- Classify laser points By echo length.
- Extract Echoes in problem areas using a specific echo extraction logic:
  - **Last possible** - for example in areas with dense low vegetation where the default extraction logic did not provide ground points.
  - **All possible** or **All distinct** - for example in places where points on some feature are missing, such as powerline wires.
  - **First possible**.

Waveform processing principles

In a TerraScan project, the block binary files must be saved as *LAS* or *FastBinary* files in order to enable the use of waveform data for processing. Waveform-related attributes, such as echo length, echo normality, and echo position can only be stored in *FastBinary* format.

The waveform files are linked to laser points via the trajectory files. The **Trajectory information** dialog contains an input field **Waveform** which defines the file(s) used for reading waveform information. Once a laser point is assigned to a trajectory (by the line number) and the trajectory is linked to a waveform file, the software is able to find the waveform information for any laser point using the time stamp and the echo number stored for the laser point.

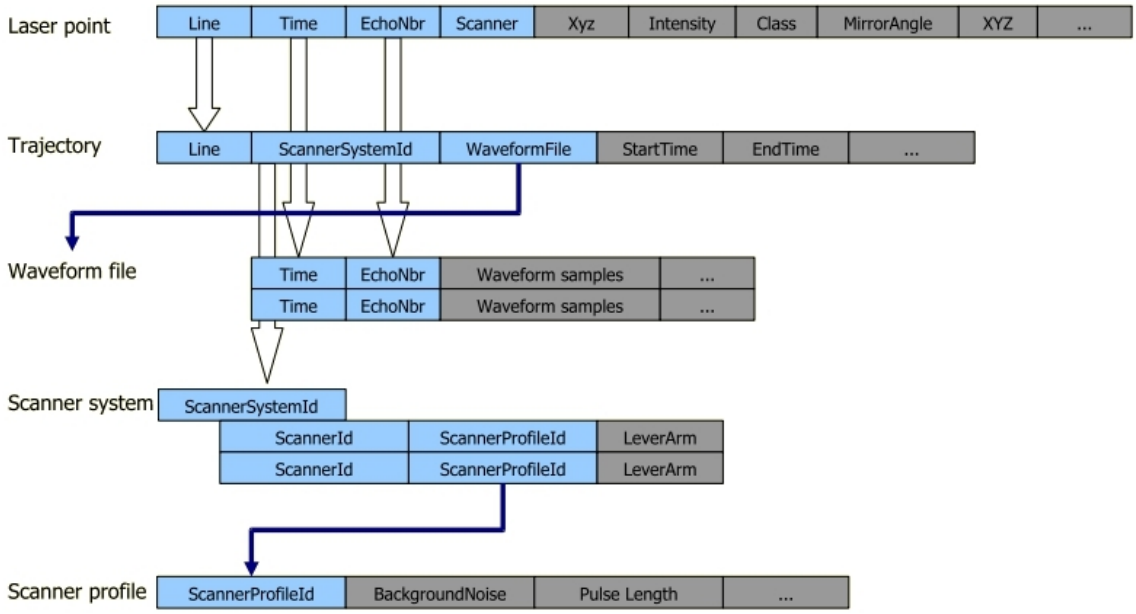
For the extraction of echo properties and of additional points, the software also needs a scanner waveform profile. The profile stores properties of typical returns from a single hard surface. These properties include:

- the background noise level
- the pulse length at 50% of peak strength
- the pulse length at 35% of peak strength
- the shape of the return pulse
- the system-derived point position relative to the return pulse

The scanner waveform profile can be extracted from laser point samples on hard, flat, open ground surfaces. There should be only-echo returns and some intensity variation within the sample area. The sample areas must not be located at the edges of scan lines. The scanner waveform profile is then automatically computed from the sample laser points.

Finally, the scanner waveform profile must be referenced by a scanner system definition which in turn must be linked to the trajectory files.

The following figure illustrates the method how TerraScan finds waveform information for a laser point.



TerraScan expects waveform data stored as 16-bit unsigned integer values. Some software for generating waveform data uses signed integer values which leads to problems when TerraScan reads the waveform data. Depending on the software information provided in the header of an LAS file, TerraScan excludes values > 32767 in order to avoid errors caused by signed integer values in the waveform files.

Workflow summary

1. Load trajectories into TerraScan using the Manage Trajectories tool.

2. Link trajectories with waveform files using the Edit information or Link to waveform files commands of the **Trajectories** dialog.

3. Create a TerraScan project using the Define Project tool, storage format must be **FastBinary** for storing echo properties, or **LAS**.

4. Import points into the project using the Import points into project command of the **Project** dialog, deduce line numbers from trajectories.

   This enables the display of waveform information using the View Waveform tool.

5. Draw polygons around sample areas of single, open, hard surfaces that contain only-echo returns and some variation in intensity values. Sample areas should not be too close to scan corridor edges.

6. Classify points inside the polygons into a separate class using Inside fence command or By polygons classification routine.

7. Create a scanner waveform profile using the user controls in Scanner waveform profiles category of TerraScan **Settings**.

You have to repeat steps 5 to 7 for all scanners or lines collected with different pulse rates.

8. Link the scanner waveform profiles with scanner system definitions using the user controls in Scanner systems category of TerraScan **Settings**.

9. Link the trajectories with scanner system definitions using the Edit information command of the **Trajectories** dialog.

   This enables the extraction of echo properties using the Extract echo properties command of the **Project** dialog or the Extract echo properties command for loaded points, and the extraction of additional points using the Extract Echoes tool.

Waveform toolbox

Tools in the **Waveform** toolbox are used to view waveform information and to extract additional echoes from the waveform information.

TO	USE TOOL	
View waveform data		View Waveform
Extract echos from waveform		Extract Echoes

Extract Echoes

Not UAV

Extract Echoes tool extracts additional points from a return signal.

When scanner system software is generating laser points, it follows a certain logic. It may generate a point from the strongest, first, or last return but usually, it extracts one point from a multiple-return signal. In general, the extraction method of system software works well for laser point clouds.

However, in some places, the generated points might not be optimal. Examples are missing returns from wires or from ground below dense vegetation. In both cases, the system software might extract a point from the return signal, but possibly not the point of biggest interest for certain applications. The **Extract Echoes** tool can be used at such places in order to extract additional points from return signals.

There are several extraction methods available:

- **First possible** - looks only at rising start of the return signal.
- **Last possible** - looks only at trailing end of the return signal. This should be used, for example, to extract additional ground points.
- **All distinct** - constant fraction discriminator, more reliable result than All possible method.
- **All possible** - Gaussian decomposition, can generate multiple points from overlapping signals. This should be used, for example, to extract additional points on wires.

The extraction of additional points should only be performed in limited areas where the extraction method of the system software did not provide optimal results. The following methods can be used to limit the processing area for point extraction:

- Place a fence in a section view in order to specify a 3D slice of space where to generate new points. The section depth defines the XY area and the fence the elevation range for point extraction.
- Draw a fence or select polygon(s) to specify a 2D area where to generate points. In this case, the new points can be located at any elevation, only the XY area is defined.

The process creates new points only if it finds returns in the waveform that match the settings for the extraction. For example, if there is no part of the laser beam that penetrated to the ground because of dense vegetation, the Last possible method will probably not generate a point on the ground level.

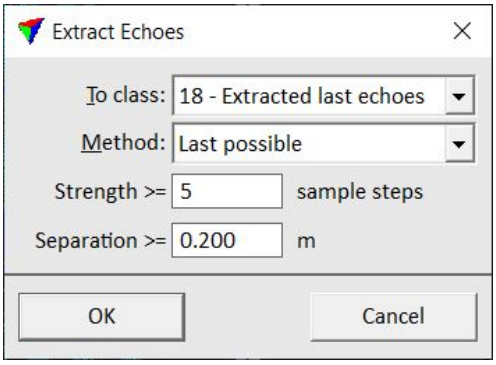
The extraction of additional points requires that trajectories are active and laser points are loaded in TerraScan. The points must be linked to the trajectories and the trajectories must reference the waveform files and the scanner systems. The scanner waveform profile must be available and linked to the scanner system. See Waveform processing principles and Workflow summary for more information.

To extract additional points:

1. Draw a fence or polygon(s) around the area(s) for which you want to extract points as described above. Select the polygon(s).

2. Select **Extract Echoes** tool.

   The **Extract Echoes** dialog opens:



3. Define settings and click OK.

   This generates new points if the software finds return signals in the waveforms that match the settings. The points are added to the laser points in TerraScan memory.

4. Use Save points as commands in order to save the laser points into a file.

The new points are created as inactive points. You must save the points with setting **Points** set to **All points** in the **Save points** dialog. Otherwise, the additional points extracted by **Extract Echoes** tool are not stored.

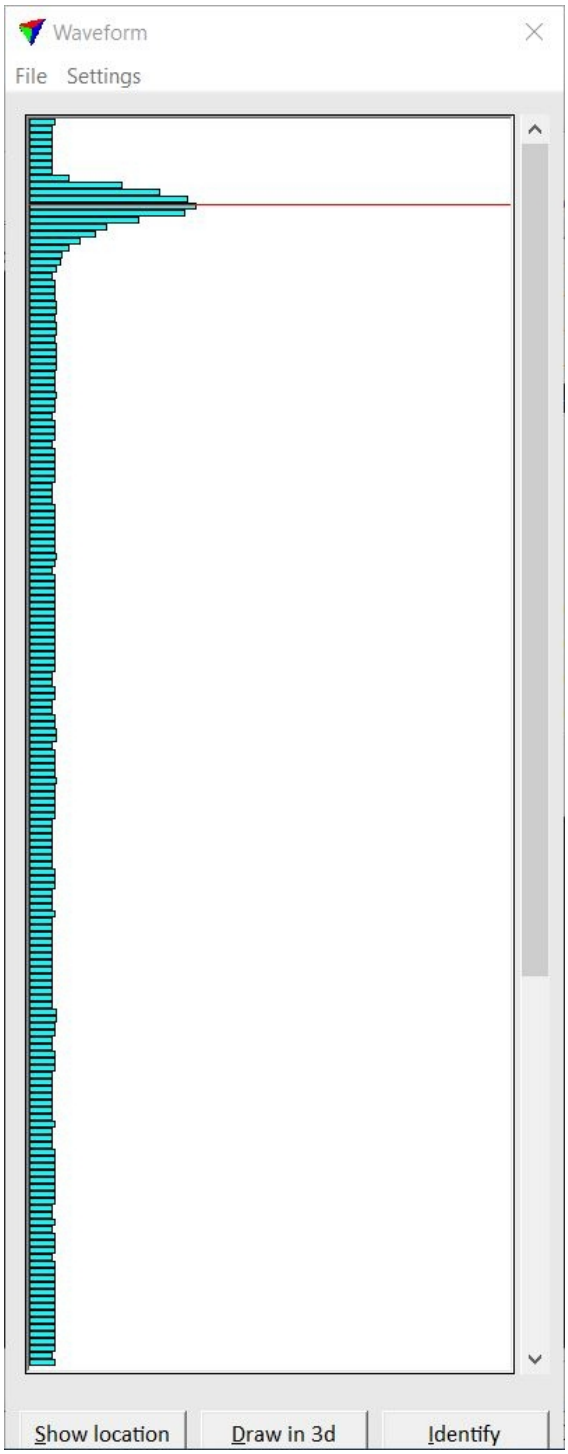
SETTING	EFFECT
To class	Target class for extracted points. The list contains the active classes in TerraScan.
Method	Method of point extraction. See explanations above.
Strength	Required number of photons in addition to the background noise. Only if the return signal is stronger than the background noise plus the given value, a points is extracted.
Separation	Minimum distance along the waveform between an existing point and a new point.

View Waveform

Not UAV

View Waveform tool opens the Waveform dialog that displays the waveform shape of single laser points. The dialog contains commands for identifying a point, showing a point's location, drawing the waveform vector into the CAD file, saving the waveform as text file, and changing the display settings of the waveform graph.

The waveform graph represents the waveform of a return signal by bars of constant height and varying length. The height of a bar corresponds to a 30 centimeters distance of light travel. The length of a bar indicates how many photons returned to the scanner from an object. Short bars of approximately the same length (= small waveform sample values) represent the background noise, longer bars more or less strong returns from objects. A red line in the graph indicates the location of the selected laser point.



Viewing the waveform requires that trajectories are active and laser points are loaded in TerraScan. The points must be linked to the trajectories and the trajectories must reference the waveform files. See Waveform processing principles and Workflow summary for more information.

To view the waveform of a laser point:

1. Select **View Waveform** tool.

   The **Waveform dialog** opens.

2. Click on the **Identify** button of the dialog.

3. Define a laser point with a data click inside a view.

   This displays the waveform graph for the laser point closest to the data click.

   Click on the **Show location** button and move the mouse pointer inside a CAD file view in order to highlight the point for which the waveform is show. A data click inside a view centers the highlighted point in the view.

To draw the waveform vector into the CAD file:

1. Identify a laser point for waveform graph display.

2. (Optional) Center the point in a cross section view. This may be best for viewing the waveform vector.

3. Click on the **Draw in 3d** button.

   This draws the waveform vector into the CAD file. The vector is represented by a cell element that contains lines of different colors. *Red* color is used for the strongest return, *yellow*, *green*, *cyan*, *blue* for other returns of decreasing strength, and *gray* for background noise.

To save the waveform of a point into a text file:

1. Identify a laser point for waveform graph display.

2. Select **Save As text** command from the **File** pulldown menu.

   This opens the **Save waveform as text** dialog, a standard dialog for saving files.

3. Define a location and file name for saving the text file and click **Save**.

   This saves the text file.

To save the waveform of multiple points into a text files:

1. Draw a fence or polygon around the points for which you want to export the waveform. Select the polygon.

2. Select **Save inside fence** command from the **File** pulldown menu.

   This opens the **Browse For Folder** dialog, a standard dialog for selecting a storage folder.

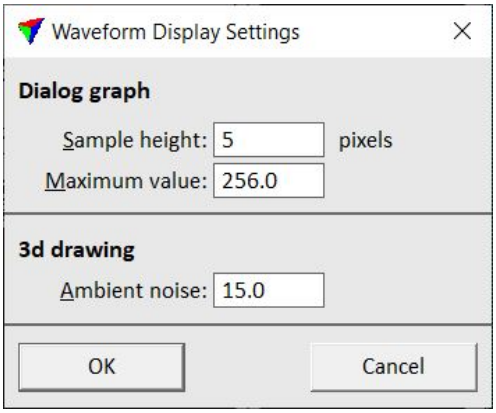
3. Select a folder for saving the text files and click **OK**.

   This saves a text file for each point inside the fence/selected polygon. An information dialog shows the number of saved text files out of the number of points. The text files are named automatically as WAVEFORM_<timestamp>_<echo type>.TXT.

To change the settings for waveform display:

1. Select **Display settings** command from the **Settings** pulldown menu.

   This opens the **Waveform Display Settings** dialog:



2. Define settings and click OK.

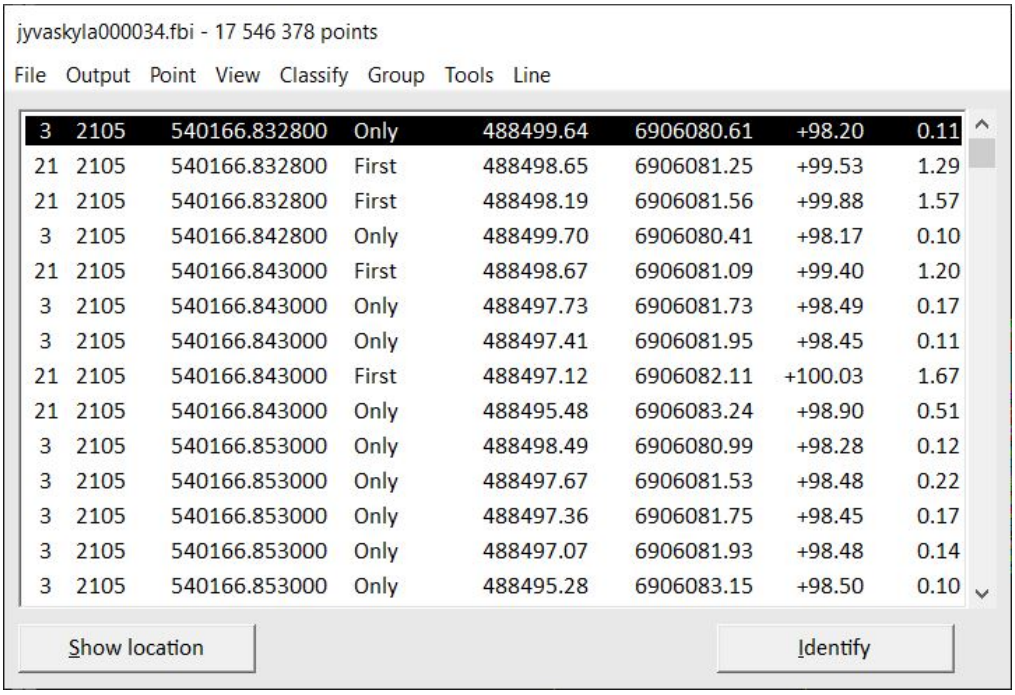
   This applies the new settings for the display.

SETTING	EFFECT
Sample height	Height of a bar in the waveform graph. Given in screen pixels.

SETTING	EFFECT
Maximum value	Defines the maximum length of a bar that can be displayed in the graph. The value effects the scale of the length of the bars.
Ambient noise	Limit value for background noise. If a waveform sample value is smaller than or equal to the given value, a 3d vector line is drawn in gray.

TerraScan Window

The **TerraScan window** is opened when TerraScan is loaded and if the **Open main window** option in the Operation category in TerraScan **Settings** is switched on.



The **TerraScan** window contains pulldown menu commands that are used to process point cloud data loaded into RAM. If data is loaded by any tool or command for loading points, the software reads the points into RAM. As long as the **TerraScan** window is open, the points remain in memory and can be displayed and processed.

If set to **Small dialog** size, the **TerraScan** window is minimized to the menu. If set to a larger size by commands from the View pulldown menu, the window shows the list of loaded points. The list contains attributes of each point that are set to be visible in the Fields dialog.

To show the location of a point, select a line in the **TerraScan** window's list of points. Click on the **Show location** button and move the mouse pointer into a view. This highlights the selected point with a square. You can show the location of several points by pressing the <Shift> or <Ctrl> keys while selecting lines in the list.

To identify a point, click on the **Identify** button and place a data click close to a point in a view. This selects the corresponding line in the **Main** window's list of points.

If the **TerraScan** window is accidentally closed, it can be re-opened with the key-in command:

scan app mainwin

However, points are unloaded from memory if the **TerraScan** window is closed.

The mouse button(s) defined in the CAD platform can be used to snap to a point. This may be helpful, for example, for dynamically rotating a view around a location in the point cloud or for digitizing

vector data based on the point cloud. Snapping to points can be disabled in the Snapping category of TerraScan **Settings**.

Classify pulldown menu

Commands from the **Classify** pulldown menu are used to classify laser points and to detect planes or trees from laser points.

TO	USE COMMAND
Start an automatic classification routine for points	Routine
Classify points inside a fence	Inside fence
Classify points inside a 3D fence	3D fence
Classify points captured along a trajectory interval	Trajectory interval
Detect plane areas from laser points	Detect plane
Detect trees from laser points	Detect trees
Add points within a specific area to ground class	Add point to ground

3D fence

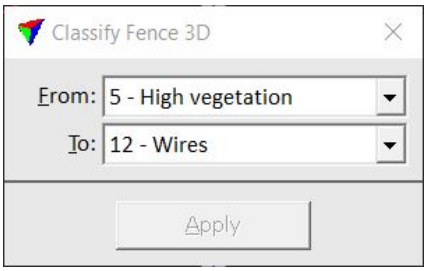
Not Spatix

3D fence command lets you classify points inside a 3D fence from one class into another class. Only points that are displayed on screen are effected by the classification. The 3D fence is defined in two steps. Usually, a fence or selected polygon is drawn in a top view first. Then the tool waits for the selection of a second view, which is automatically turned into a front view covering the same area as defined by the fence. In this front view a second fence can be drawn to define the final 3D fence content.

To classify points inside a 3D fence:

1. Draw a fence or select a polygon around points to classify in a top view.
2. Select **3D fence** command from the **Classify** pulldown menu.
3. Select a second view by placing a data click inside the view.

   This turns the second view into a section view, displays the points that are located in the fence, and opens the **Classify Fence 3D** dialog:



4. Select classes in the **From** and **To** fields.

5. Draw a fence in the section view. The fence tool is already started by the command.

6. When at least 3 vertices for the fence are defined by data clicks, the **Apply** button in the **Classify Fence 3D** dialog becomes active. Click **Apply** to finish the fence.

   This classifies the points that are displayed in the top view and located inside the 3D fence.

Add point to ground

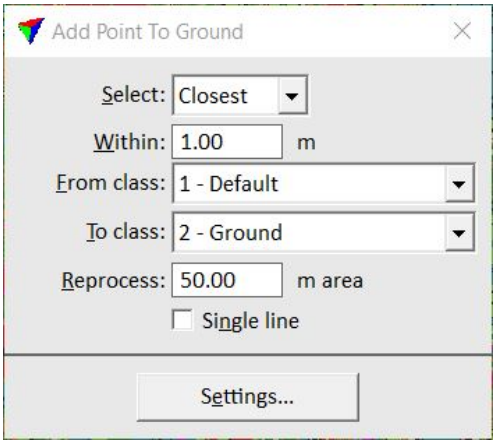
Add point to ground command lets you classify points inside a certain area to the ground class. This may be useful to correct classification errors effectively in areas where the automatic ground classification does not provide a good result.

The command is used most likely after an automatic Ground classification has been performed. It classifies additional points to the ground class based on an initial point and additional settings defined in the command's settings dialog. The source class from which the points are added to ground has to be visible in the view in which the classification is started.

To add points to the ground:

1. Select **Add point to ground** command from the **Classify** pulldown menu.

   This opens the **Add Point to Ground** dialog:

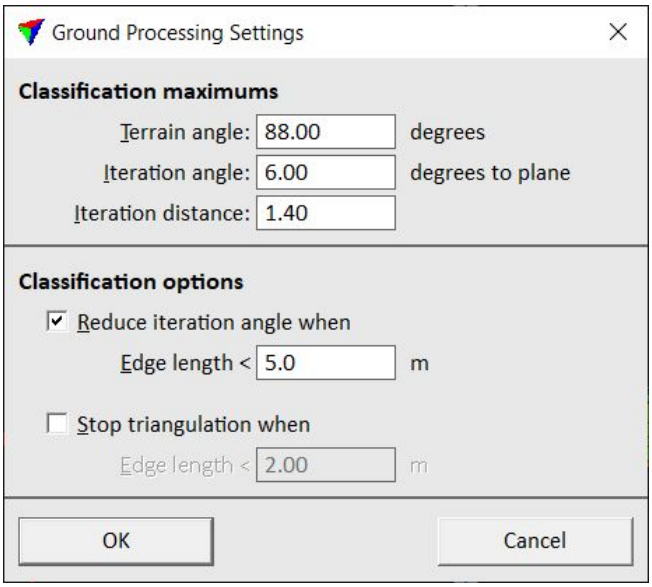


2. Define settings for the search for additional ground points.

SETTING	EFFECT
Select	Defines which point is selected as initial point for the ground classification: • **Closest** - the point closest to the data click within the search radius. • **Highest** - the highest point within the search radius. • **Lowest** - the lowest point within the search radius.
Within	Search radius around the mouse click to find the initial point for starting the ground classification.
From class	Source class from which points are classified into ground.
To class	Target class for classified ground points.
Reprocess	Area within which points are classified. The value defines the radius of a circular area around the initial point.
Single line	If on, only points from one line at a time are classified. This may result in ground levels per line.

3. (Optional) Click on the **Settings** button in order to change settings for the ground detection parameters.

   This opens the **Ground Processing Settings** dialog:



   The settings are the same as for the automatic ground classification. See Ground classification routine for a detailed description of the settings.

   Apply the settings by clicking OK.

4. Click inside a view to define the initial location for adding ground points.

   This classifies visible points from the source class to the ground class according to the given settings and within the defined reprocessing area.

Detect plane

Not Lite

Detect plane command detects points on a plane inside a fence or selected polygon from one point class. It classifies the points into another class and optionally draws a 3D rectangle around the points on the plane.

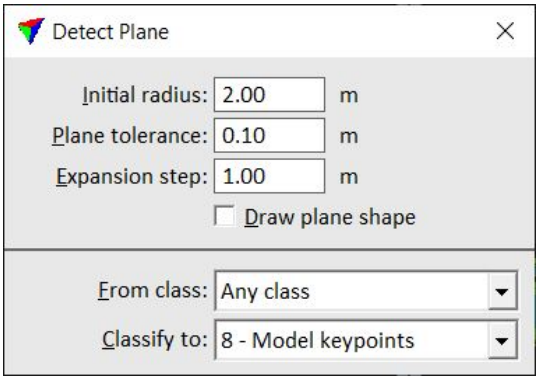
The tool works only in top views. It does not detect close-to-vertical or vertical planes.

To detect a plane from laser points:

1. Draw a fence or select a polygon around the area where the plane(s) are located.

2. Select **Detect plane** command from the **Classify** pulldown menu.

   This opens the **Detect Plane** dialog:



3. Define settings.

4. Click inside the fence to define a start point for the plane detection.

   The software highlights points that are found on a plane. In addition, the angle of the plane and the time spent for plane detection is shown in the message center at the bottom of the CAD platform interface.

5. Accept the points with another data click.

   This classifies the points on the plane and (optionally) draws a rectangle around the plane. The rectangle element is drawn on the active level using the active symbology settings of the CAD file.

SETTING	EFFECT
Initial radius	Start radius for plane detection.

SETTING	EFFECT
Plane tolerance	Defines the distance how close points must match a fitted plane equation.
Expansion step	Maximum gap between points belonging to the same plane.
Draw plane shape	If on, a 3D rectangle is drawn around the points on the detected plane.
From class	Source class from which points are used for plane detection.
Classify to	Target class for points on the detected plane.

Detect trees

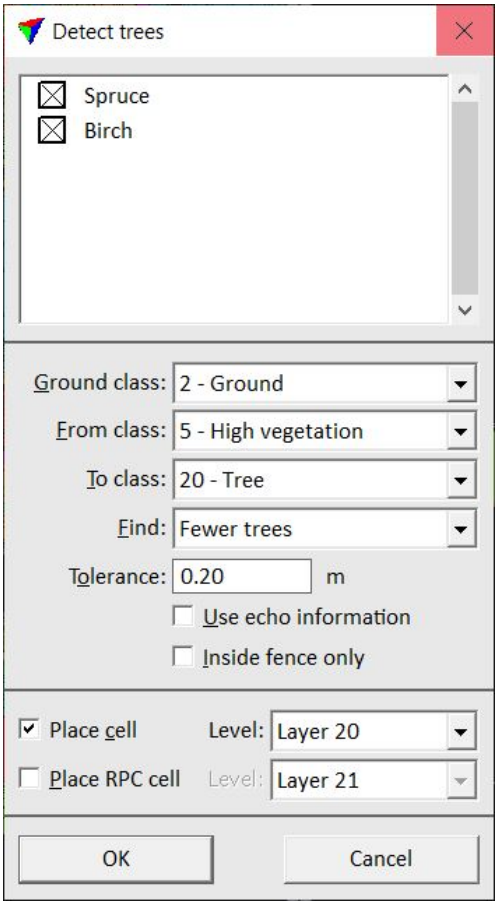
Not Lite, Limited in Spatix

Detect trees command detects trees from the laser point cloud automatically based on tree shape definitions. This requires the classification of the laser points into ground, vegetation and optionally building points as well as the definition of tree types in TerraScan **Settings**. See Tree types category for information on how to define tree types.

For detected trees, either MicroStation cells or RPC cells can be placed to represent trees in 3D visualizations. MicroStation cells must be defined in the MicroStation **Cell Library** to be placed correctly. RPC cells are replaced by RPC files when a view is rendered and if the software finds the RPC file at the given location. Settings for cell names and RPC files can be found in the Tree types category of TerraScan **Settings** as well.

RPC files are purchased by Archvision (www.archvision.com). For more information about RPC cells and visualization options, see TerraPhoto User Guide or the MicroStation Online Help.

The automatic tree detection is based on tree crown shapes that it tries to detect in the point cloud. It is possible to distinguish trees with clearly different crown shapes, such as very slim shapes of many coniferous trees from very round shapes of some deciduous trees. The tool is not able to detect different tree types or the true amount of trees in tree groups or a dense forest. Thus, the aim of the tree detection is mainly visualization for which the real amount of trees in tree groups or forests does not play a role.



To detect trees from laser points:

1. Select **Detect trees** command from the **Classify** pulldown menu.

   This opens the **Detect trees** dialog.

2. Select one or more tree types listed in the upper part of the dialog.

3. Define settings for tree detection.

4. Click OK.

   The software starts the detection process. It classifies points from detected trees into the given target class and (optionally) places cells and/or RPC cells on each tree location. A process window shows the progress of the detection. Depending on the amount of ground and vegetation points loaded into TerraScan and given settings, the process might take some time.

SETTING	EFFECT
Ground class	Point class representing the ground level.
From class	Source point class from which trees are detected.
To class	Target point class for points from detected trees.
Find	Determines how many trees are detected: • **More trees** - higher amount of trees is detected. • **Normal level** - normal amount of trees is detected. • **Fewer trees** - lower amount of trees is detected.
Tolerance	Positional accuracy tolerance for laser points.
Use echo information	If on, echo information is used for determining what is likely to be a tree.

SETTING	EFFECT
Inside fence only	If on, the detection area is limited to a fence area. Requires that a fence is drawn or a polygon is selected in the CAD file.
Place cell	If on, MicroStation cells are places at detected tree locations.
Place RPC cells	If on, RPC cells are placed at detected tree locations.

RPC cells can be also placed manually based on laser points and aerial images using the Place Rpc Tree tool in TerraPhoto.

At the moment, cells and thus, the cell placement options of the **Detect trees** tool only works in MicroStation. There is not yet any corresponding element type in Spatix.

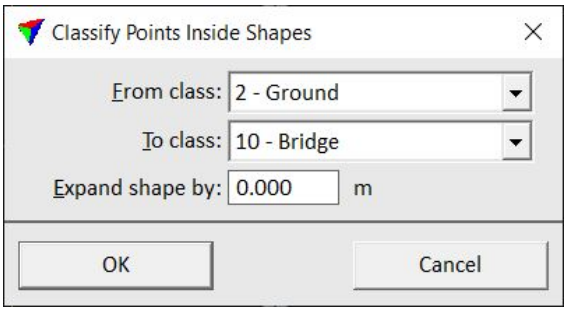
Inside fence

Inside fence command classifies points inside a fence or selected polygon from one class into another class. Only points that are displayed on screen are effected by the classification.

To classify points inside a fence:

1. Draw a fence or select a polygon around points to classify.
2. Select **Inside fence** command from the **Classify** pulldown menu.

   This opens the **Classify Points Inside Shapes** dialog:



3. Select classes in the **From class** and **To class** fields and click OK.

   This classifies the points that are displayed and located inside the fence.

Routine

Routine sub-menu contains commands for calling automatic classification routines. They can be used to classify points loaded in TerraScan. Most of the classification routines are also available as macro actions in order to use them in batch processing.

The different routines are explained in detail in Chapter Classification Routines.

Trajectory interval

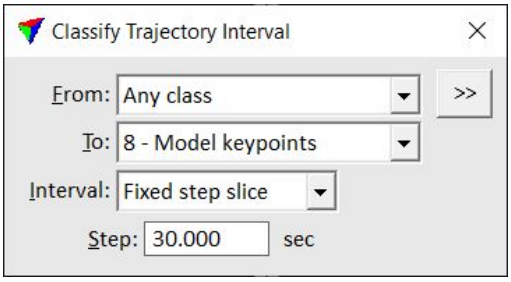
Trajectory interval command classifies points that were captured within a given interval along a trajectory. The interval can be defined by start and end position on a trajectory or by start position and duration in seconds.

The command relies on trajectories stored in the active trajectory folder of TerraScan. In addition, the trajectory number must be assigned to the loaded points.

To classify points of trajectory intervals:

1. Select **Trajectory interval** command from the **Classify** pulldown menu.

   This opens the **Classify Trajectory Interval** dialog:



3. Define settings.

4. Move the mouse pointer inside a view.

   The trajectory closest to the mouse pointer is dynamically displayed.

5. Identify the trajectory to classify with a data click.

6. Move the mouse pointer to the start position of the interval to classify.

   The trajectory position closest to the mouse pointer is dynamically displayed.

7. Identify the start position with a data click.

   If **Interval** is set to **Fixed step slice**, the software classifies the points captured within the given **Step** time interval to the target class. The data click defines the center point of the interval.

8. If **Interval** is set to **Free**, move the mouse pointer to the end position of the interval to classify.

   The trajectory position closest to the mouse pointer is dynamically displayed.

9. Identify the end position with a data click.

   This classifies the points captured during the time interval defined by the start and end position to the target class.

SETTING	EFFECT
From	Source class(es).
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **From** field.
To	Target class.
Interval	Determines how the trajectory interval is defined: • **Free** - by two data clicks along a trajectory line. • **Fixed step slice** - by a given interval duration.
Step	Duration of the interval to classify. This is only active if **Interval** is set to **Fixed step slice**.

Group pulldown menu

Commands from the **Group** pulldown menu are used to assign groups to laser points and to classify points based on the grouping.

TO	USE COMMAND
Assign a group number to points	Assign groups
Test parameters for separating point groups into different classes	Test parameters
Start an automatic classification routine for point groups	Classify
Clear group numbers from selected classes	Clear by class
Copy the group attribute from closest neighbour point	Copy from closest
Split groups based on classification	Split groups by class
Check groups in a systematic way	Inspect groups

Any functionality related to grouping is still under development and commands/tool may change in future versions of TerraScan.

Assign groups

Assign groups command assigns a group number to points of one or more classes. The grouping is done based on different methods, such as plane fitting, watershed algorithm, or 3D spacing between points.

The group assignment relies on a distance from ground value that must be computed for the points before the command is started. Use Compute distance command for loaded points or Compute distance macro action with setting **Compare to** = **Ground** in order to compute the distance value.

The group number can be used for the visualization of points and for classifying points. There are various automatic classification routines that rely on grouping. The tools from the Groups toolbox can be used to manipulate the groups manually.

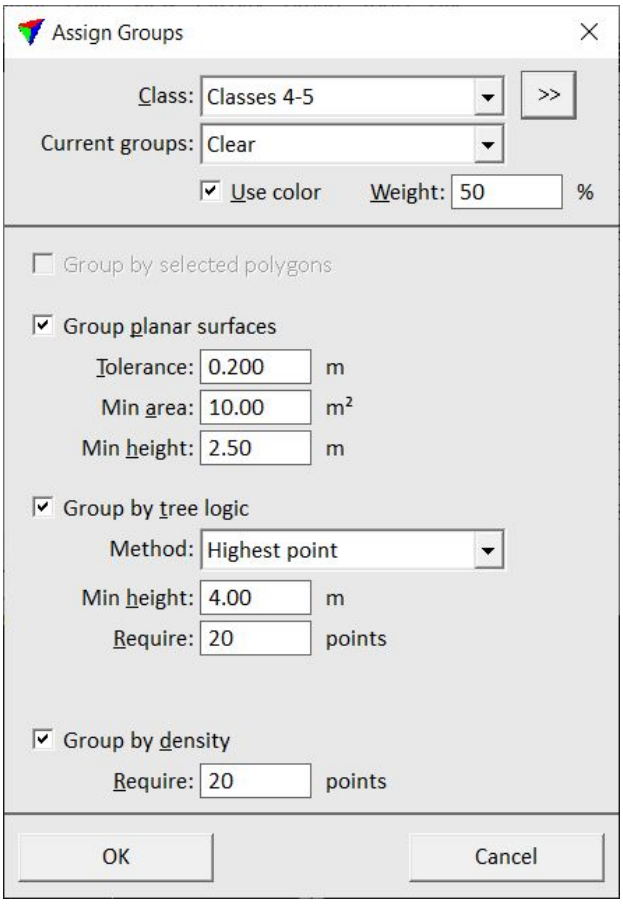
The group number can be stored in TerraScan FastBinary files. Switch on the **Group** attribute in the Attributes to store dialog of a TerraScan project in order to store the group number for project block files.

To assign a group number to points:

1. Compute the distance from ground using the Compute distance command or corresponding macro action.

2. Select **Assign groups** command from the **Group** pulldown menu.

   This opens the **Assign Groups** dialog:



2. Define settings and click OK.

   This assigns a group number to points in the selected class(es) that fit the requirements. All other points get group number 0.

SETTING	EFFECT
Class	Point class(es) included in the search for groups.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Class** field.
Current groups	Determines how current group numbers of points are handled: • **Clear** - existing group numbers are deleted before assigning new numbers. • **Keep** - existing group numbers are kept.
Use color	If on, color values assigned to the points are used for grouping. The **Weight** of color values is determined by the given percentage value.
Group by selected polygons	If on, group numbers are assigned based on selected polygons. Points inside a polygon get

SETTING	EFFECT
	the same group number. This is only active if one or more shape elements are selected in the CAD file.
Group planar surfaces	If on, points that fit to planes are grouped. Points fitting to the same plane get the same group number.
Tolerance	Determines how much points may differ from a plane. Point that are within the given vertical distance from a plane are grouped. This is only active if **Group planar surfaces** is switched on.
Min area	Minimum area of a planar surface for grouping points. This is only active if **Group planar surfaces** is switched on.
Min height	Minimum height above ground of a planar surface for grouping points. This is only active if **Group planar surfaces** is switched on.
Group by tree logic	If on, points on trees are grouped Point that belong to the same tree get the same group number.
Method	Method of grouping trees: • **Highest point** - based on a watershed algorithm starting from the locally highest point. This is suitable for airborne point clouds. • **Trunk** - based on approximately circular tree trunks visible in the point cloud. This requires a high amount of points from trunks and is suitable for very high-density airborne point clouds, for mobile data and point clouds from static scanners.
Require	Minimum number of points that form a single tree group. This is only active if **Group by tree logic** is switched on and **Method** is set to **Highest point**.
Min height	Minimum height above ground of a group. This is only active if **Group by tree logic** is switched on.
Max diameter	Maximum diameter of a tree trunk as an approximate estimate. This is only active if **Group by tree logic** is switched on and **Method** is set to **Trunk**.
Min trunk	Minimum length of the tree trunk. This is only active if **Group by tree logic** is switched on and **Method** is set to **Trunk**.

SETTING	EFFECT
Group by density	If on, points are grouped based on their distance to each other. Close-by points get the same group number.
Require	Minimum amount of points that form a single group. This is only active if **Group by density** is switched on.

Classify

Classify sub-menu contains commands for calling automatic classification routines for groups of points. They can be used to classify points loaded in TerraScan. Most of the classification routines are also available as macro actions in order to use them in batch processing.

Classifying groups requires that a group number is assigned to the points. See Assign groups command for more information.

The different routines are explained in detail in Chapter Classification Routines.

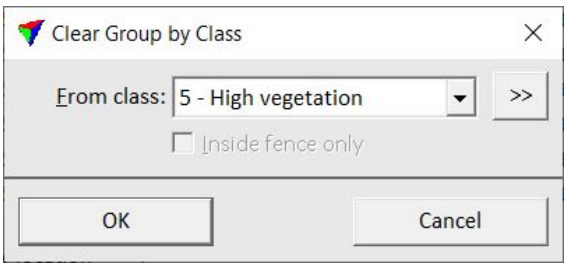
Clear by class

Clear by class command clears the group number from points in selected classes. The group number is set to 0.

To clear group numbers:

1. Select **Clear by class** command from the **Group** pulldown menu.

   This opens the **Clear Group by Class** dialog:



2. Define settings and click OK.

   This sets the group number of points in the selected class(es) to 0.

SETTING	EFFECT
From class	Point class(es) included in the search for groups.

SETTING	EFFECT
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **From class** field.
Inside fence only	If on, only loaded points that are located inside a fence or selected polygon are effected. Requires a fence or selected polygon in the CAD file.

Group numbers can also be cleared when assigning new group numbers.

Copy from closest

Copy from closest command assigns a group number to points that do not yet have a group assignment (non-grouped points). It copies the group number from the closest grouped point in given classes.

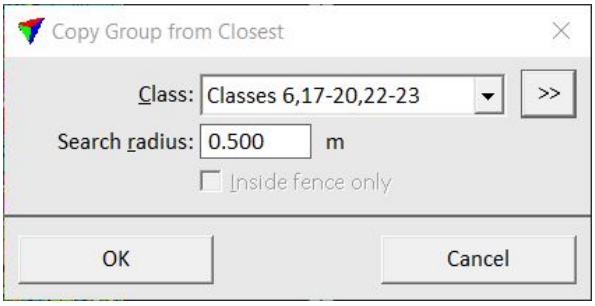
Copying the group number may be useful for group assignment and classification of very dense point clouds. The grouping and classification of groups processes can be speed up by the following workflow:

- Thin points and move some of the points to an 'Unnecessary density' class.
- Run Assign groups and classification processes, exclude 'Unnecessary density' class.
- Copy class to 'Unnecessary density' points from closest classified point.
- Copy group to 'Unnecessary density' points from closest grouped point.

To copy the group number from closest points:

1. Select **Copy from closest** command from the **Group** pulldown menu.

   This opens the **Copy Group from Closest** dialog:



2. Define settings and click OK.

   This assigns the number of the closest group to non-grouped points.

SETTING	EFFECT
Class	Point class(es) effected by the process.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Class** field.
Search radius	Distance around a point within which the software checks for points with a group assignment.
Inside fence only	If on, only loaded points that are located inside a fence or selected polygon are effected. Requires a fence or selected polygon in the CAD file.

Inspect groups

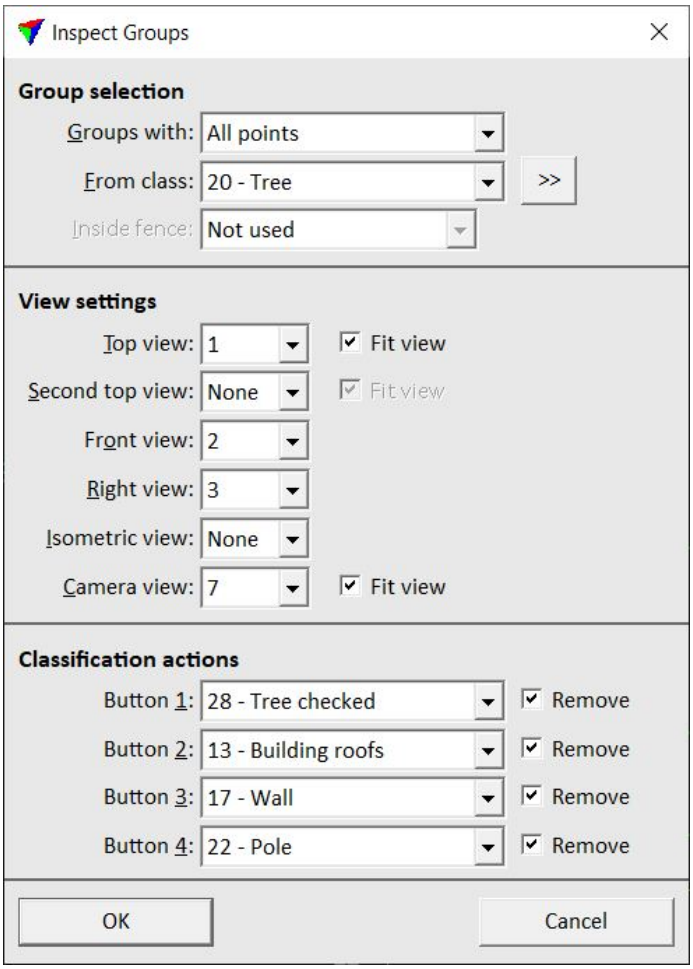
Inspect groups command supports the systematic check of groups. It provides a list of groups from which you can select one group after the other.

The tool includes view settings that define CAD file views displaying the selected group in different view orientations. The selected element is automatically centered in these views. It becomes the active displayset and thus, it is displayed in views for which **Points** is set to **Displayset only** in the Display mode dialog. In addition, the tool allows the definition of up to four classification buttons for modifying the class of a group.

To inspect groups:

1. Select the **Inspect groups** command from the **Groups** pulldown menu.

   This opens the **Inspect Groups** dialog:

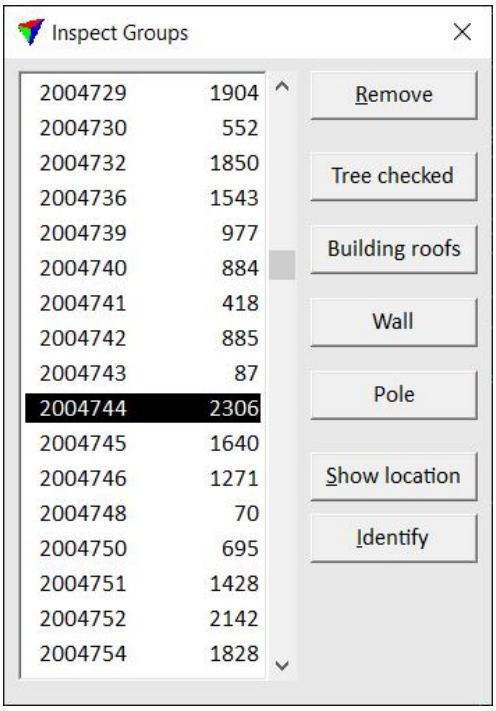


SETTING	EFFECT
Groups with	Determines which groups are inspected: • **One or more points** - groups with one or more points in the source class. • **Majority of points** - groups with the majority of points in the source class. • **All points** - groups with all points in the source class.
From class	Point class(es) that are included in the group inspection.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **From class** field.
Inside fence	Determines how a fence or selected polygon(s) effect the classification: • **Not used** - fence or selected polygons are ignored. • **One or more points** - groups are classified if one or more points are inside.

SETTING	EFFECT
	• **Average xy** - groups are classified if the average xy point is inside. • **Majority of points** - groups are classified if the majority of points is inside. • **All points** - groups are classified if all points are inside.
Top view	View window that displays the point cloud in a top view.
Second top view	View window that displays the point cloud in a top view.
Front view	View window that displays the point cloud in a front section view. The cross section is 90 degree rotated compared with the **Right view**.
Right view	View window that displays the point cloud in a right section view. The cross section is 90 degree rotated compared with the **Front view**.
Isometric view	View window that displays the point cloud in an isometric view.
Camera view	View window that displays the point cloud in a camera view. The view can display images that are referenced by an active image list in TerraPhoto.
Fit view	If on, the selected group is automatically fitted and centered in the view.
Button 1...4	If on, a target class can be selected for classifying the group into another class after inspection.
Remove	If on, the selected group is removed from the list in the **Inspect Groups** list dialog.

3. Define settings and click OK.

   This opens another **Inspect Groups** dialog that contains the list of groups:



4. Select a line in the list of elements.

   This centers the selected group in all view windows defined in the tool's **View settings**. You can click on the classification buttons of the dialog to classify a group into another target class.

SETTING	EFFECT
Remove	Removes the selected group from the list. The group itself is not deleted. The button is inactive if **Remove** is switched on in the tool's **Classification action** settings.
<Class>	Up to four classification buttons defined in the tool's **Classification action** settings. The button shows the name of the target class.
Show location	Select a line in the list, click on the button and move the mouse pointer inside a view window. This highlights the selected group in the view.
Identify	Click on the button and identify a group with a data click in a view window. This selects the corresponding line in the list.
	Moves one image backward in the active image list and displays the new image in the camera view. This is only visible if a **Camera view** is set in the tool's **View settings**.
	Click on the button and move the mouse pointer inside a view window. The image closest to the mouse pointer is highlighted.

SETTING	EFFECT
	Select an image for the camera view display with a data click. This is only visible if a **Camera view** is set in the tool's **View settings**.
	Moves one image forward in the active image list and displays the new image in the camera view. This is only visible if a **Camera view** is set in the tool's **View settings**.

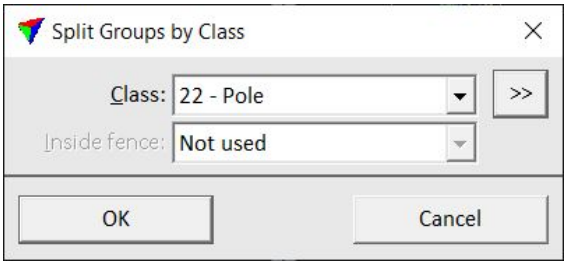
Split by class

Split by class command can be used to split groups if points in the group belong to different classes. This may happen, for example, if different objects are close to each other, such as poles and vegetation. Grouping may result in one group for a pole and close-by vegetation.

To split groups by class:

1. Select **Split groups by class** command from the **Group** pulldown menu.

   This opens the **Split Groups by Class** dialog:



2. Define settings and click OK.

   This assigns a new group number to points in the selected class(es), if points from another class are in the same group.

SETTING	EFFECT
Class	Point class(es) that are split from groups containing points of several classes and become an own group.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Class** field.
Inside fence	Determines how a fence or selected polygon(s) effect the classification: • **Not used** - fence or selected polygons are ignored.

SETTING	EFFECT
	• **One or more points** - groups are classified if one or more points are inside. • **Average xy** - groups are classified if the average xy point is inside. • **Majority of points** - groups are classified if the majority of points is inside. • **All points** - groups are classified if all points are inside.

Test parameters

Test parameters command can be used to test parameters for the differentiation of point groups. The testing is done based on sample groups which need to be defined before running the command. As a result of the testing, a parameter file can be saved. This is then used to classify points with the By parameters routine for point groups.

There are several parameters which may be used to distinguish groups:

- **Transparency** - number of returns or density of points.
- **Intensity** - intensity values. Requires intensity values stored for points.
- **Scanner 1|2|3 intensity** - intensity values of different scanners/wavelengths. Requires data from a multiple-wavelength scanner.
- **Color brightness** - brightness values.
- **Color red|green|blue** - color values in red, green, blue channel. Requires color values stored for points.
- **Color N** - near-infrared value. Requires a near-infrared value stored for points.
- **Complex|Planar|Linear ratio** - dimension ratio. Requires normal vector and dimension computed for the points.
- **Crown sharpness** - shape of a tree crown.
- **Height** - height from ground.
- **Width to height** - width to height ratio of a group.
- **NDVI** - normalized differential vegetation index.

A typical use case for **Test parameters** is the recognition of tree types from multi-spectral point clouds. The point cloud may be produced by a multi-wavelength scanner, from near-infrared stereo image pairs or by assigning color values from near-infrared images to a LiDAR point cloud. The workflow for classifying points of trees of different types into separate classes can be outlined as follows:

1. Classify the ground.
2. Compute distance from ground for above ground points.
3. Classify by distance from ground. Points on trees are classified into High vegetation class.
4. Assign groups to points in High vegetation class.
5. Classify trees using the Best match routine.
6. Classify sample trees into separate classes. There should be a few sample groups for each tree type.

7. Test parameters based on the sample groups. Depending on the source data, there may be a combination of 2-4 parameters that give the best probability of separating the tree types.
8. Save a parameter file.
9. Classify the data set using the parameter file.

To test parameters and save a parameter file:

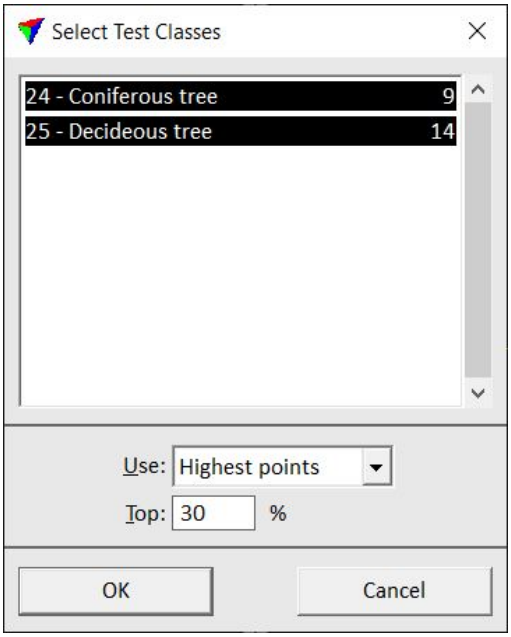
1. Compute the distance from ground using the Compute distance command or corresponding macro action.

2. Assign groups using the Assign groups command or corresponding macro action.

3. Define sample objects by classifying groups manually or automatically into separate classes.

4. Select **Test parameters** command from the **Group** pulldown menu.

   This opens the **Select Test Classes** dialog:



   The upper part of the dialog shows a list with all classes for which groups are assigned and the amount of sample groups for each class.

5. Select classes you want to use for testing. Define additional settings.

SETTING	EFFECT
Use	Determines which points from the selected classes are used for deriving parameters: • **All points** - all points. • **Only** | **First** | **Last echoes** - only points of the given echo type. • **Highest points** - the highest points are used. The value given in the **Top** field determines the percentage of highest points measured from the ground to the top point.

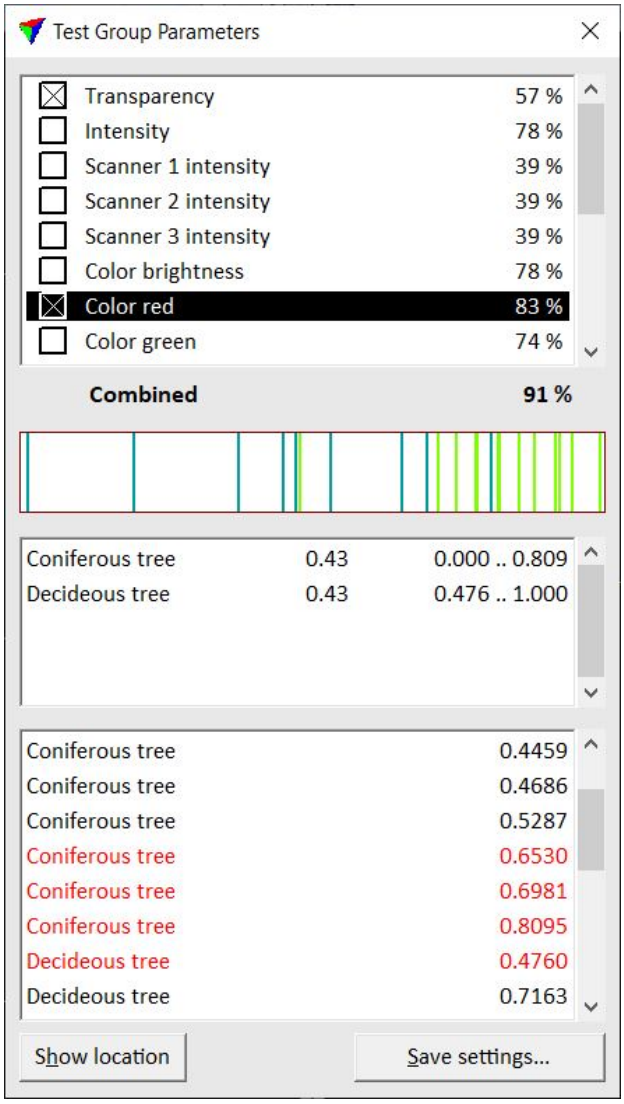
6. Click OK.

   This opens the **Test Group Parameters** dialog.

   The upper part of the dialog shows a list of parameters and their certainty of distinguishing the sample classes from each other. The parameter with the highest certainty percentage is selected automatically. The combined percentage value below the list shows the final certainty of differentiation if several parameters are selected.

   The central part of the dialog illustrates how good the sample classes can be distinguished by a selected parameter. The black field represents a statistical value from 0.0 to 1.0. The colored bars in the field show where the sample groups are located within the range of 0.0 to 1.0. If the sample groups overlap, the differentiation is less certain as if groups are clearly separated. The color of the bars is determined by the class color settings in TerraScan. The field below also shows for each sample group where it is located in the range between 0.0 and 1.0. In addition, a weight factor is displayed.

   The lower part of the dialog contains a list of all sample groups and their statistical value of a selected parameter. Critical sample groups that overlap with groups from other classes are displayed in red. A selected sample group can be centered in a view by using the **Show location** button and placing a data click inside the view.



7. Select different parameters and check how well they distinguish the sample classes. Switch on all parameters that you want to use in the final parameter set.

8. Click on the **Save settings** button.

   This opens the **Group parameter file** dialog, a standard dialog for saving files.

9. Define a location and name for storing the parameter file and click **Save**.

   This saves the parameters into a text file.

File pulldown menu

Commands from the **File** pulldown menu are used to open, save and close points.

TO	USE COMMAND
Read points of a project block	Open block
Read points of a project block inside a fence	Open inside fence
Read points from a file	Read points
Read points from files in a directory	Read directory
Read points as read-only reference	Read reference points
Save modified points	Save points
Save or export points to a file	Save points As
Change the cloud type setting for loaded points	Cloud type
Close loaded points	Close points

Close points

Close points command removes loaded points from memory.

If points have been modified, a dialog opens to ask if you want to save changes before closing the points.

- Click **Yes** to save points. If points can not be saved into the original file, the **Save points** dialog opens. See Save points As command for a description of the dialog's settings.
- Click **No** to unload points without saving changes.
- Click **Cancel** to close the dialog without removing the points from memory.

Cloud type

Cloud type sub-pulldown menu lets you select the type of the point cloud loaded in TerraScan. Selecting the correct type is recommended in order to optimize the processing speed for many automatic routines. The command can be used to change the cloud type if it has not been selected correctly when reading points.

The following point cloud types are available:

- **Airborne lidar** - point cloud captured by an airborne laser scanner system. The system is carried by flying vehicle.
- **Mobile lidar** - point cloud captured by a mobile laser scanner system. The system is carried by a ground-based vehicle.
- **Stationary lidar** - point cloud captured by a static terrestrial laser scanner system. The scanner is mounted on a static device such as a tripod.
- **Airborne photo** - photogrammetric point cloud generated from airborne images. The images are captured from a flying vehicle.
- **Mobile photo** - photogrammetric point cloud generated from mobile images. The images are captured from a ground-based vehicle.

- **Stationary photo** - photogrammetric point cloud generated from static terrestrial images. The images are captured by a static camera mounted on a tripod.
- **Boat sonar** - point cloud captured by a sonar mounted on a boat.
- **Mixed** - point cloud produced by different devices.

Open block

Not UAV

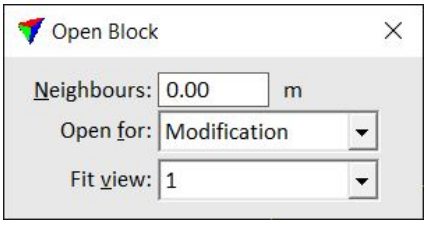
Open block command loads points linked to a TerraScan project block into memory. You select the project block geographically by clicking inside the boundary of the block. The software reads the binary file that is linked to the selected project block.

You can open a block for modification or for viewing only. This setting has an effect if you use project file locking. See File locking for more information. The work with projects in TerraScan is described in detail in Chapter Working with Projects.

To open points of a project block:

1. Select **Open block** command from the **File** pulldown menu.

   This opens the **Open Block** dialog:



2. Define settings.

3. Move the mouse pointer into the project block you want to open.

   A block boundary is highlighted dynamically if the mouse pointer is inside the block area.

4. Place a data click inside the block.

   This reads the binary file linked to the selected block.

SETTING	EFFECT
Neighbours	Width of an overlap area around the active block for which the application loads points from neighbouring blocks.
Open for	Block opening mode: **Viewing only** or **Modification**.
Fit view	View(s) to fit in order to display the area of all loaded points.

SETTING	EFFECT
Load reference points	If on, points from a reference project are loaded. This is only active if a reference project is set in the **Project information** dialog. See Edit project information and Compare with reference commands for more information.

The points in a block may include classes that are not defined in the active class definition in TerraScan. In this case, a temporary class is added to the active class definition for any missing class. The temporary class gets a default description according to its class number, for example "class 0" for a temporary class number 0. Any temporary class is deleted if the points are unloaded. You can store a temporary class permanently by using the Define Classes tool.

Open inside fence

Not UAV

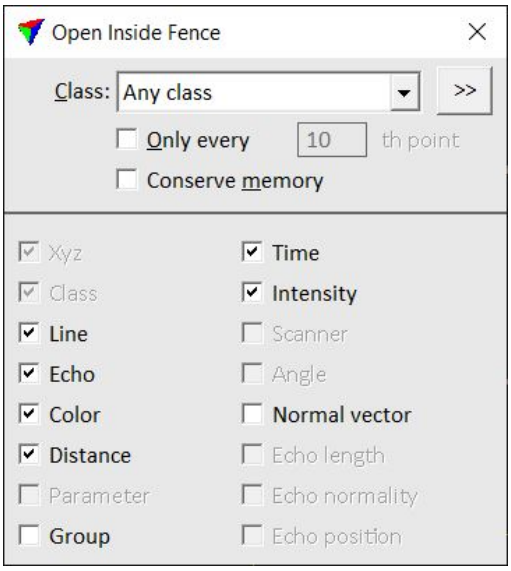
Open inside fence command loads laser points from TerraScan project blocks inside a fence or selected polygon(s). The points are opened for read-only access. After modifications, the points can be saved into a new file using Save points As command.

To open points inside fence:

1. Draw a fence *(MicroStation only)* or polygon(s) around area(s) for which to load the points. Select the polygon(s).

2. Select **Open inside fence** command from the **File** pulldown menu.

   This opens the **Open Inside Fence** dialog:



3. Define settings.

4. Move the mouse pointer into the view.

The fenced area is highlighted dynamically if the mouse pointer is inside the view.

5. Place a data click inside the view.

   This reads the points from the binary files which fall inside the fenced area.

SETTING	EFFECT
Class	Class(es) that are loaded for the fenced areas.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Class** field.
Only every	If on, only every **n th** point is loaded where **n** is the given number.
Conserve memory	If on, the software first determines how many points will be loaded and thus, how much memory needs to be allocated for the exact number of points. This slows down the loading process but it is less likely to run out of memory.
Attributes	Attributes that are loaded for laser points. Switch on attributes that you want to read in. Only attributes that are stored in the point file(s) are available for loading. Point coordinates and the class number are always required.

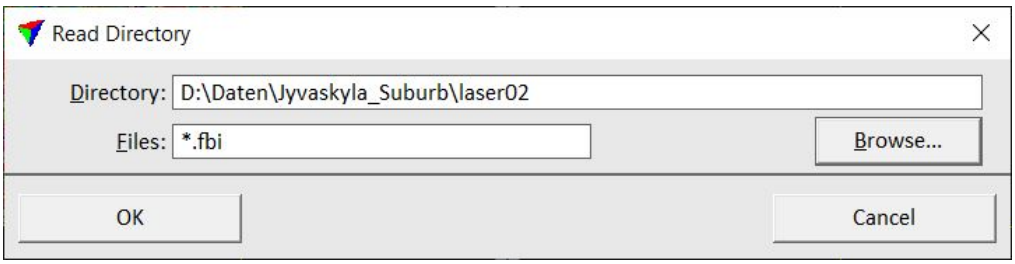
Read directory

Read directory command loads points from all files in a given directory into TerraScan. Basically, it performs the same action as the Read points command but for several files at the same time.

To load reference points from files into memory:

1. Select **Read directory** command from the **File** pulldown menu.

   This opens the **Read Directory** dialog:



2. Define settings and click OK.

   This opens the Read points dialog.

3. Define settings and click OK.

   This loads the points into the memory and displays them on the screen.

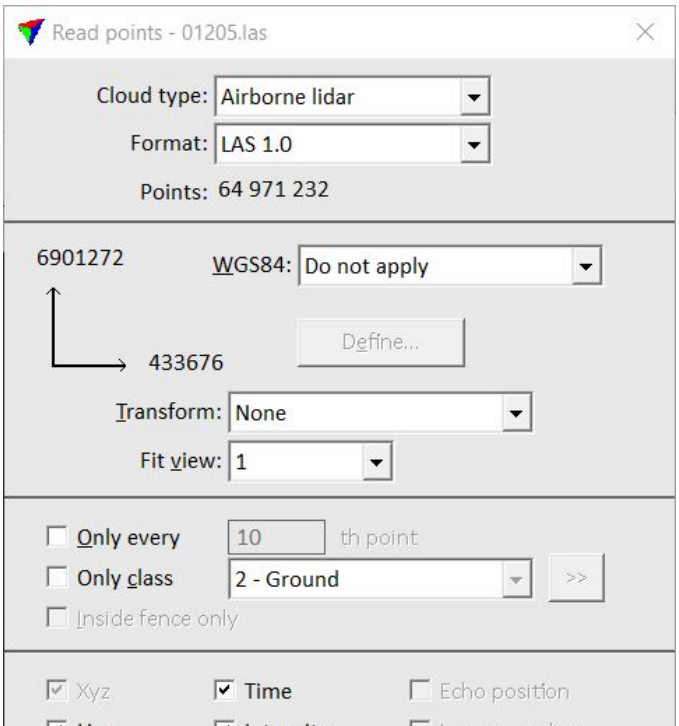
SETTING	EFFECT
Directory	Path to the folder from which files are loaded.
Browse	Opens a standard dialog for selecting a folder. Select the folder from which to read files and click OK. The path to the folder is written to the **Directory** field.
Files	Determines which files are loaded from the directory: • ***.*** - all files. • ***.fbi** - all FastBinary files. Type the extension of files you want to load after the point character. • **pt*.fbi** - all FastBinary files whose name start with "pt". Use the * character to replace a free number of characters in the file name or extension. • **name.las** - only the LAS file with the given "name".

The points in a block may include classes that are not defined in the active class definition in TerraScan. In this case, a temporary class is added to the active class definition for any missing class. The temporary class gets a default description according to its class number, for example “class 0” for a temporary class number 0. Any temporary class is deleted if the points are unloaded. You can store a temporary class permanently by using the Define Classes tool.

Read points

Read points command loads points from files into TerraScan for visualization or processing tasks. It performs exactly the same action as the Load Airborne Points tool.

More information about file formats that can be read into TerraScan can be found in Section Supported file formats and File formats / User point formats category of TerraScan **Settings**.



You can load several files of the same file format together in one reading process. The file format is automatically recognized if it is known by the software. The points of the selected file(s) are loaded into TerraScan memory. You can add more points by loading additional files. If the memory is full, the software shows an error message and the reading process stops.

To load points from files into memory:

1. Select **Read points** command from the **File** pulldown menu.

   This opens the **Read points** dialog, a standard dialog for selecting files.

2. Select files and click **Open**.

   This opens the **Load points** dialog.

3. Select the correct **Cloud type**. This is recommended in order to optimize the processing speed for many automatic routines.

4. Define other settings and click OK.

   This loads the points into the memory and displays them on the screen.

SETTING	EFFECT
Cloud type	Type of the point cloud related to the way of how the point cloud is produced: • **Airborne lidar** - point cloud captured by an airborne laser scanner system. The system is carried by flying vehicle.

SETTING	EFFECT
	• **Mobile lidar** - point cloud captured by a mobile laser scanner system. The system is carried by a ground-based vehicle. • **Stationary lidar** - point cloud captured by a static terrestrial laser scanner system. The scanner is mounted on a static device such as a tripod. • **Airborne photo** - photogrammetric point cloud generated from airborne images. The images are captured from a flying vehicle. • **Mobile photo** - photogrammetric point cloud generated from mobile images. The images are captured from a ground-based vehicle. • **Stationary photo** - photogrammetric point cloud generated from static terrestrial images. The images are captured by a static camera mounted on a tripod. • **Boat sonar** - point cloud captured by a sonar mounted on a boat. • **Mixed** - point cloud produced by different devices. The cloud type can be changed for loaded points by using the Cloud type command.
Format	Format of the point file. This is automatically recognized by the software. For user-defined text file formats, it might be necessary to select the correct format.
Points	Amount of points in all selected files.
Coordinate preview	Coordinate values of the first point found in the point file. This helps to decide whether a coordinate transformation needs to be applied.
WGS84	Transformation from WGS84 coordinates into projection system coordinates applied during the reading process. The list contains projection systems which are active in Coordinate transformations / Built-in projection systems, Coordinate transformations / US State Planes, or Coordinate transformations / User projection systems categories of TerraScan **Settings**.
Transform	Transformation applied to points during the reading process. The list contains transformations that are defined in Coordinate transformations /

SETTING	EFFECT
	Transformations category of TerraScan **Settings**.
Fit view	Defines which CAD file view is fitted to the extent of the loaded points: • **None** - no view is fitted. • **All** - all open views are fitted. • **1...8** - the selected view is fitted. Only numbers of open views are available.
Only every	If on, only a selection of points is loaded. The software reads every n th point from the file(s), where n is the given number.
Only class	If on, only points from the selected class(es) are loaded.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Only class** field.
Inside fence only	If on, only points inside a fence or selected polygon are loaded.
Attributes	Attributes that are loaded for laser points. Switch on attributes that you want to read in. Only attributes that are stored in the point file(s) are available for loading. **All on** and **All off** buttons switch the selection of all attributes on or off. Point coordinates and the class number are always required.
Line numbers	Defines, how line numbers are assigned to the points during the loading process: • **Use from file** - line numbers from source files are used. • **Assign constant** - the number given in the **First number** field is assigned to all points. • **From file name** - the last numerical sequence in a file name is used as line number. • **From folder name** - the last numerical sequence in the name of the folder containing the point files is used as line number. • **Deduce using time** - numbers are assigned based on trajectories loaded into TerraScan. The same process can be performed for by the Deduce using time command or the corresponding macro action.

SETTING	EFFECT
	• **Increase by xy jump** - the line numbers increase from the given **First number** if the xy distance is bigger than the value given in the **By distance** field. • **Increase by time jump** - the line numbers increase from the given **First number** if a jump in time stamps occurs. This requires that trajectory information is available in TerraScan. • **Increase by file** - the line numbers increase from the given **First number** for each separate file. • **Increase by file name** - the line numbers increase from the given **First number** for each file with another file name. Files with the same name get the same number. • **Increase by directory** - the line numbers increase from the given **First number** for each file stored in another source folder. Files from the same source folder get the same number. The **Line numbers** settings are only available if the **Line** attribute is switched on for loading.
Scanner number	Defines, how scanner numbers are assigned to the points during the loading process: • **Use from file** - scanner numbers from source files are used. • **Assign constant** - the number given in the **First number** field is assigned to all loaded files. • **Increase by file** - the scanner numbers increase from the given First number for each separate file. • **From file name** - the first numerical sequence in a file name is used as scanner number. • **From folder name** - the first numerical sequence in the name of the folder containing the point files is used as scanner number. • **From line number** - the line number is used as scanner number. The **Scanner number** settings are only available if the **Scanner** attribute is available and switched on for loading.
Default	Point that is assigned to all points if no class attribute is stored in the point file. This is only

SETTING	EFFECT
	active if text file formats are selected for loading.

The point file(s) may include classes that are not defined in the active class definition in TerraScan. In this case, a temporary class is added to the active class definition for any missing class. The temporary class gets a default description according to its class number, for example “class 0” for a temporary class number 0. Any temporary class is deleted if the points are unloaded. You can store a temporary class permanently by using the Define Classes tool.

Read reference points

Read reference points command loads points from files into TerraScan. Basically, it performs the same action as the Read points command but the loaded reference points are read-only. By default, the highest possible line number 65535 is assigned to reference points.

The reference points can be used for a direct comparison of old and new data sets. They are required for the Fit to reference command which can be used, for example, to fit two photogrammetric point clouds to each other.

To load reference points from files into memory:

1. Select **Read reference points** command from the **File** pulldown menu.

   This opens the **Read reference points** dialog, a standard dialog for opening files.

2. Select files and click **Open**.

   This opens the **Load reference** dialog, basically the same dialog as described for the Read points command. The line number is fixed to 65535 in this dialog.

3. Define settings and click OK.

   This loads the points into the memory and displays them on the screen.

Save points

Save points command saves all loaded points to the same binary file from which they were read in or into which they were saved earlier.

To prevent an original file being overwritten by a file which does not include all original information, the **Save points** command is disabled if points have been loaded incompletely. This includes the following cases:

- any attribute was switched off.
- only points of selected classes were loaded.
- only points inside a fenced area were loaded.
- only every n th point was loaded.

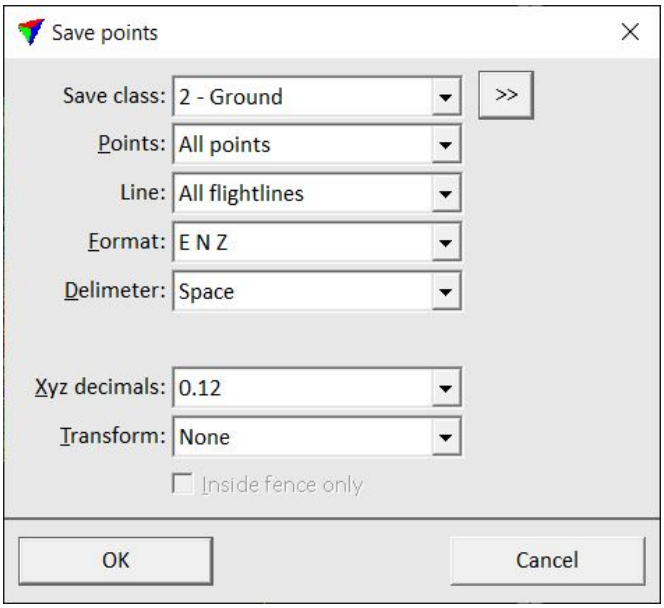
Save points As

Save points As command writes points into a new file. The output file format can be any of the Supported file formats implemented in TerraScan or a user-defined point file format. See File formats / User point formats for more information.

To write points into a file:

1. Select **Save points As** command from the **File** pulldown menu.

   This opens the **Save points** dialog:



2. Define settings for the output file.

3. Click OK.

   This opens the **Save points** dialog, a standard dialog for saving files.

4. Define a location and name for the output file and click **Save**.

   This creates the output file.

SETTING	EFFECT
Classes	Selection of classes that are written into the output file. Contains the list of active classes in TerraScan. Use the <Shift> or <Ctrl> keys to select several classes.
Select all	Selects all classes in the list.
Deselect all	Deselects all classes in the list.
Points	Defines what points are written into the new file:

SETTING	EFFECT
	• **All points** - all loaded points. This may include inactive points, for example from neighbour blocks or points extracted from waveform information. • **Active block** - points inside the active block area. This is only active if there are inactive points in memory.
Line	Line(s) that are written into the new file: points of **All lines** or points of the selected line number.
Format	Format of the output file. The list of formats includes all implemented file formats as well as any user-defined formats.
Attributes	Opens the **Attributes to save** dialog. Switch on attributes that you want to write into the output file. Only attributes that can be stored in the selected format are available. If specific attributes are required for the selected output format they are switched on by default. If color values are stored for the points, an additional selection of the amount of color channels is required. See more information here.
Delimiter	Defines the delimiter for text files: **Space**, **Tabulator**, or **Comma**. This is only active if some of the implemented text file formats are selected as **Format**.
Surface	Class(es) from which a surface is computed. This is only active if **Format** is set to **E N Z dZ** or if a user-defined format includes **dZ**, where **dZ** is the elevation difference between a point and the surface.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Surface** field.
Xyz decimals	Defines the number of decimals for coordinate values. This is only active if an implemented text file format is selected as **Format**.
Transform	Transformation applied to points during the writing process. The list contains transformations that are defined in Coordinate transformations / Transformations category of TerraScan **Settings**.

SETTING	EFFECT
Inside fence only	Only points that are inside a fence or a selected polygon are written into the output file.

Line pulldown menu

Commands from the **Line** pulldown menu are used to manipulate laser points based on line information or to create and modify lines.

TO	USE COMMAND
Assign line number to laser points from trajectory information	Deduce using time
Assign line number to laser points from scan pattern	Deduce from order
Start a new line from a selected laser point	Start new at selection
Modify the numbering of lines	Modify numbering
Draw approximate flight path	Draw from points
Draw a flight path from a raw trajectory file	Draw from file
Apply a boresight angle correction	Adjust laser angles
Remove points from long range measurements	Cut long range
Remove points from overlapping lines	Cut overlap
Fit a point cloud to control measurements or to a reference point cloud	Fit using targets
Shift a point cloud interactively	Translate
Rotate a point cloud interactively	Rotate

The **Line** pulldown menu is not available if laser data is loaded into TerraScan using the Load Ground Points tool. In this case it is replaced by the **Measurement** pulldown menu which offers special commands for laser data from static terrestrial scanners.

Adjust laser angles

Adjust laser angles command applies a positional correction to laser points. The command is normally used to fix misalignment of lines in the laser data. The software computes an XYZ correction for a point as true angular rotation of the vector from the scanner to the point. The user can then decide, whether the full correction is applied or only a partial horizontal/vertical correction along the XY or Z axis.

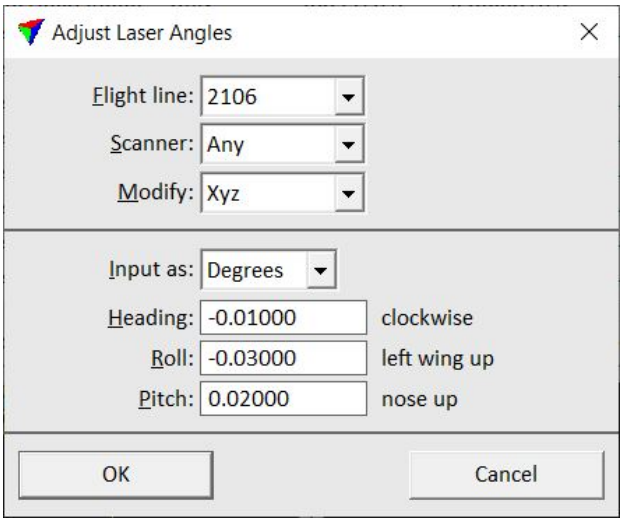
Adjust laser angles requires trajectory information in order to determine the scanner position of each laser point. Therefore, trajectories must be imported into TerraScan and the line numbers of laser points must match the numbers of trajectories. See Deduce using time command for more information about matching line numbers of points and trajectories easily.

Time stamps for laser points are not mandatory. If time stamps are present, the application can derive the laser scanner position for each laser point more accurately. If time stamps are missing, the application uses a perpendicular projection from a laser point to the trajectory in order to determine the scanner position.

To adjust laser angles:

1. Select **Adjust laser angles** command from the **Line** pulldown menu.

   This opens the **Adjust Laser Angles** dialog:



2. Define settings and click OK.

   The application derives the laser scanner position for each point and applies a positional correction according to the settings.

SETTING	EFFECT
Flight line	Line to which the correction is applied. Select Any in order to apply the correction to all loaded points.
Scanner	Scanner to which the correction is applied. This can be used for data from multi-scanner systems. Select Any in order to apply the correction to all loaded points.
Modify	Coordinate axis to modify: • **Xyz** - applies a full 3D correction. • **Xy** - applies a horizontal shift relative to trajectory and movement direction. • **Z** - applies a vertical shift relative to trajectory and movement direction.

SETTING	EFFECT
Input as	Unit of misalignment correction values: • **Degrees** - angles given in decimal degrees. • **Radians** - angles given in radians. • **Ratio** - angles given as a ratio over a value corresponding to a full circle.
Heading	Heading angle correction, positive values increase in clockwise direction.
Roll	Roll angle correction, positive values increase in left wing up direction.
Pitch	Pitch angle correction, positive values increase in nose up direction.

Cut long range

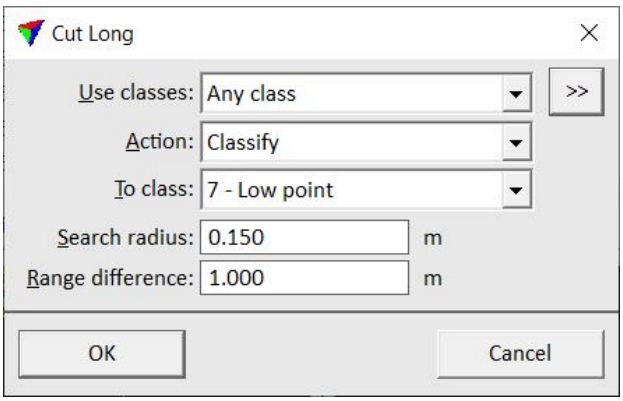
Not Lite

Cut long range command cuts off long range measurements if there are shorter measurements from the same line closeby. This is suited for point clouds from mobile/backpack/handheld scanners which may see the same location multiple times in a single line. The main purpose of the process is to reduces the noise level in the data set as longer measurements may be less accurate.

To cut long range points:

1. Select **Cut long range** command from the **Line** pulldown menu.

   This opens the **Cut Long** dialog:



2. Define settings and click OK.

   This starts the process.

SETTING	EFFECT
Use classes	Point class(es) that are considered in the process.

SETTING	EFFECT
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Use classes** field.
Action	Action to perform on points that are cut off: • **Classify** - classify points into one target class. • **Delete** - remove points from the data set. • **Set overlap bit** - sets the overlap bit for points in **LAS 1.4** files.
To class	Target class for points that are cut off. This is only active if **Action** is set to **Classify**.
Search radius	Radius from a point within which the software searches for shorter range points. A point is cut off if another point with a shorter range is found within the given radius.
Range difference	Maximum allowed range difference of two points. Only if the range difference is smaller, the longer range point is cut off.

Cut overlap

Not Lite

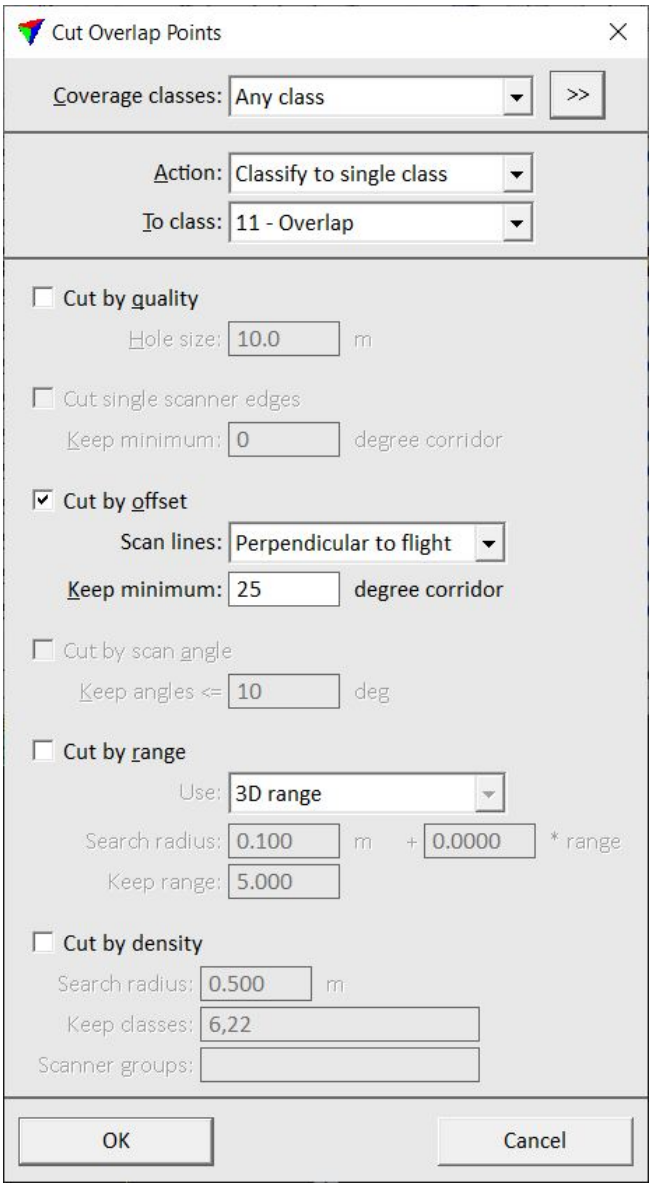
Cut overlap command removes laser points from locations where laser data from multiple lines overlap. The points of overlapping areas can be classified to a specific class or classes, or they can be deleted from the data set. In **LAS 1.4** files, the overlap bit can be set instead of classifying or deleting points.

There are several methods for cutting off overlap. If you want to apply several methods to a data set, run the one method after another in separate processing steps in order to avoid incorrect results.

General procedure to cut overlap:

1. Select **Cut overlap** command from the **Line** pulldown menu.

   This opens the **Cut Overlap Points** dialog:



2. Define settings and click OK.

   This classifies/removes points from overlapping areas.

SETTING	EFFECT
Coverage classes	List of point classes to consider when determining if a line covers an area. Use 0-255 to cover all classes.
Select	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Coverage classes** field.
Action	Action to perform on points in overlapping areas: • **Add constant to class** - add a given value to the class number.

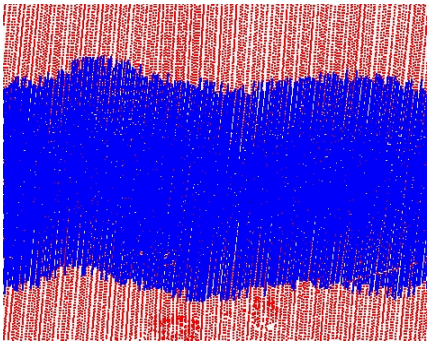
SETTING	EFFECT
	• **Classify to single class** - classify points into one target class. • **Delete** - remove points from the data set. • **Set overlap bit** - sets the overlap bit for points in **LAS 1.4** files.
Add	Value to add to class numbers if **Action** is set to **Add constant to class**.
To class	Target class if **Action** is set to **Classify to single class**.

Cut by quality

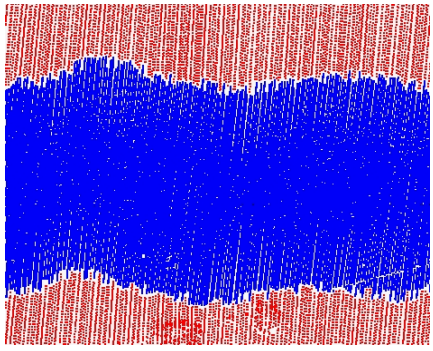
Cut by quality method removes lower quality laser points from locations where there is laser data from a higher quality line. The method is suited if:

- there is laser data from two different flying altitudes. Data of a lower flight line is normally more accurate than data of a higher line.
- the hardware was not working at the best level for some lines.
- the GPS trajectory is weak for some lines.
- there is laser data from single crossing lines while the majority of lines follows another direction.

You can define the quality of lines as an attribute of imported trajectories. See Manage Trajectories tool and Edit information command from the **Trajectory** window for more information. If you do not have imported trajectories, you can assign quality values to different line number sequences in the Default flightline qualities category of TerraScan **Settings**.



Before cut



After cut by quality

Point color: red points are from a lower quality line, blue points from a higher quality line

SETTING	EFFECT
Cut by quality	If on, cut by quality method is applied.
Hole size	Approximate maximum diameter of hole. If a larger area is not covered by points of a higher

SETTING	EFFECT
	quality line, points from the lower quality line are kept.

Cut single scanner edges

Cut single scanner edges method is designed for removing overlap in data sets collect by airborne dual-scanner systems. It removes edges of lines that are captured by one scanner only.

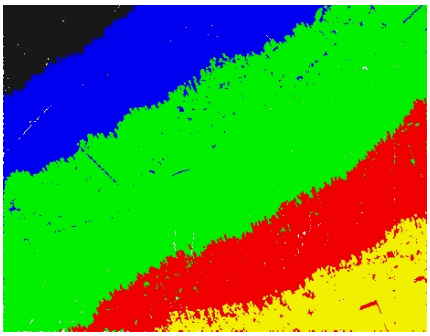
SETTING	EFFECT
Cut single scanner edges	If on, cut single scanner edges method is applied.
Keep minimum	Minimum central part of a line and scanner to keep. A value of 20 would keep a +10.. -10 degree corridor of each line and scanner.

Cut by offset

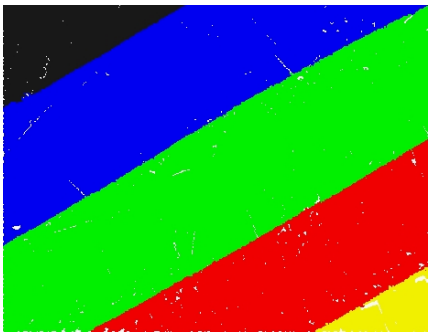
Cut by offset method removes laser points from the edges of lines if the same location is covered more vertically from another line. This method serves two purposes:

- Removing edges of lines produces a more uniform point density and point pattern.
- The magnitude of error sources grows with increasing scan angle. Removing edges of lines removes less accurate points and keeps the more accurate central part of a line.

The method is suited for cutting off overlap between parallel lines in an airborne data set. It requires trajectory information and matching line numbers of points and trajectories. See Deduce using time command for more information about matching line numbers of points and trajectories easily. If time stamps are not available for laser points, the application uses a perpendicular projection from a laser point to the trajectory.



Before cut



After cut by offset

Point color: different colors represent points of different lines

SETTING	EFFECT
Cut by offset	If on, cut by offset method is applied.
Scan lines	Scan line pattern of the data set: • **Perpendicular to flight** - parallel or zigzag scan lines. • **Elliptical** - elliptical scan lines.
Keep minimum	Minimum central part of a line to keep. A value of 20 would keep a +10.. -10 degree corridor of each line.

Cut by scan angle

Cut by scan angle method removes points with a scan angle that is larger than a given value. It removes the edges of lines similar to the **Cut by offset** method but does not rely on trajectory information.

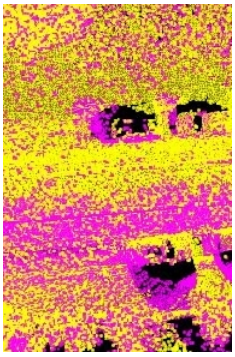
SETTING	EFFECT
Cut by scan angle	If on, cut by scan angle is applied.
Keep angles <=	Points with scan anlges smaller or equal to the given value are kept.

Cut by range

Cut by range method removes laser points from the edges of a line if the same location is covered by points from a shorter measurement distance. This method is designed for laser data from mobile ground-based systems.

The software searches for points inside a sphere and cuts off points resulting from long measurements if points from a shorter range within the search radius are present. The range can be defined as 3D range or offset range.

The method requires trajectory information and matching line numbers of points and trajectories. See Deduce using time command for more information about matching line numbers of points and trajectories easily.



Before cut *After cut by 3D range* *After cut by offset range*

Point color: different colors represent points of different lines

SETTING	EFFECT
Cut by range	If on, cut by range method is applied.
Use	Method to use for cut by range: **3D range** or **Offset range.**
Search radius	Radius of a sphere within which the software searches for closer range points from another line. The radius can be automatically increased as a factor of the range. By increasing the radius at longer ranges, the method is more eager to keep points from more distant lines.
Keep range	Range from scanner within which all points are kept.

Cut by density

Cut by density method is developed for removing overlap in merged point clouds from different sensors. The merged point cloud is a result of several point clouds with significantly different point densities. Examples are point clouds from mobile and airborne laser scanners, or photogrammetric airborne and terrestrial laser point clouds. Some areas may be covered by both sensors but other areas just by one sensor. In overlap areas, the method keeps the data from the sensor with the higher point density and removes more sparse data from the other sensor.

The cut overlap method requires scanner numbers assigned to the points. The data from different sensors is identified by a unique scanner number.

SETTING	EFFECT
Cut by density	If on, cut by density method is applied.
Search radius	Determines the area in which the local density is computed.
Keep classes	Point class(es) from which all points are kept even if the density is lower than in data from another sensor. Examples: roofs from airborne laser data, hard surface areas from mobile laser data.
Scanner groups	If a range of scanner numbers is given, the corresponding sensors are treated as a group. For example, if there is data from two mobile laser scanners with numbers 1 and 2, and the scanners are calibrated, use **1-2** in order to define a group for these 2 scanners. Separate

SETTING	EFFECT
	different groups by semicolon, for example **1-2;3-4**. If the field is empty, each sensor is treated individually.

Deduce from order

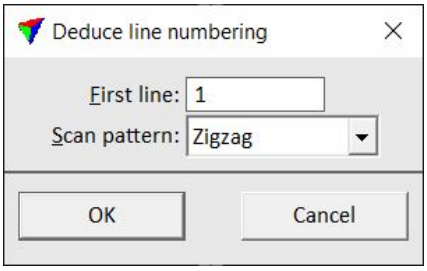
Deduce from order command assigns a line number to the laser points according to the scan pattern of the scanner system. There are two scan pattern supported:

- **Rotating** - the scanner system produces points in a circular pattern for each line.
- **Zigzag** - the scanner system produces points in a linear zigzag pattern for each line.

To deduce line numbers from the scan pattern:

1. Select **Deduce from order** command from the **Line** pulldown menu.

   This opens the **Deduce line numbering** dialog:



2. Type a number for the first line in the **First line** field.

3. Select the **Scan pattern** type.

4. Click OK.

   This assigns line numbers to the laser points according to the scan pattern. An information dialog shows the resulting line numbers.

Deduce using time

Deduce using time command assigns line numbers to laser points based on time stamps and imported trajectory information. The process looks at the time stamp of each laser point, finds a trajectory which covers that time and assigns that trajectory number to the laser point.

The command is the easiest way to make sure that the line numbers of laser points match the trajectory numbering.

The process of deducing line numbers works only correct if the time stamps of laser points and trajectories are unique. This may cause a problem, for example, if time stamps are stored in **GPS**

second-of-week format and if you have loaded multiple flight sessions from the same week day but different weeks. If both, laser points and trajectories store time stamps in **GPS standard time** format, the time information is unique and there are no roll-over problems between different weeks.

Matching time stamps of laser data and trajectories are required for a number of processing steps, especially in TerraMatch but also in TerraScan.

The same process can be done during the import of points into a TerraScan project or as a macro step. See Import points into project command and Deduce line numbers macro action for more information.

To deduce line numbers from trajectories:

1. Load trajectories into TerraScan.
2. Select **Deduce using time** command from the **Line** pulldown menu.

   This assigns the trajectory numbers as line number attributes to laser points. An information dialog shows the number of points that were effected by the process.

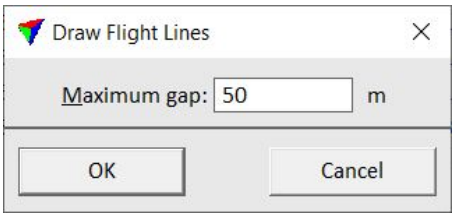
Draw from points

Draw from points command draws an approximate survey path as a linear element into the CAD file. This works if the laser points are in the original scan order as the command deduces the path from laser points only.

To draw an approximate flight path:

1. Choose **Draw from points** command from the **Line** menu.

   This opens the **Draw Flight Lines** dialog:



2. Define a **Maximum gap** value and click OK.

   This computes the approximate survey path(s) and draws one or several linear elements into the CAD file. The line elements are drawn on the active level using the active symbology settings of the CAD file.

SETTING	EFFECT
Maximum gap	A new line element starts if the distance between two consecutive laser points exceeds this value.

Fit using targets

Fit using targets command can be used to georeference point clouds based on control measurements. The process derives a transformation from source and target points. In the point cloud, the following source objects are applicable:

- **ball targets** - ball objects with a constant and known diameter, requires empty space above the ball object, ball must be located lower than the scanner.
- **square signals** - four black and white squares in chessboard pattern, visible in the RGB colors of a point cloud and placed on vertical surfaces, such as walls.
- **manual entry** - interactive definition of source points.

The target points can be defined as follows:

- **control points** - coordinates of control points in a text file, format of the text file is PointID X Y Z.
- **manual entry** - interactive definition of target points.

The transformation for fitting the source points to the target points may include:

- an XYZ shift and a 2D rotation.
- a rubbersheet correction that forces the point cloud to match the control point locations with residuals to be 0. Warps the data to match the control points. The correction values are computed based on a TIN model where the modification for places outside the TIN area approach 0.

The georeferencing method is best suited for point clouds that are produced by handheld devices, static scanners or cameras (photogrammetric point clouds), SLAM data sets, also from indoor facilities.

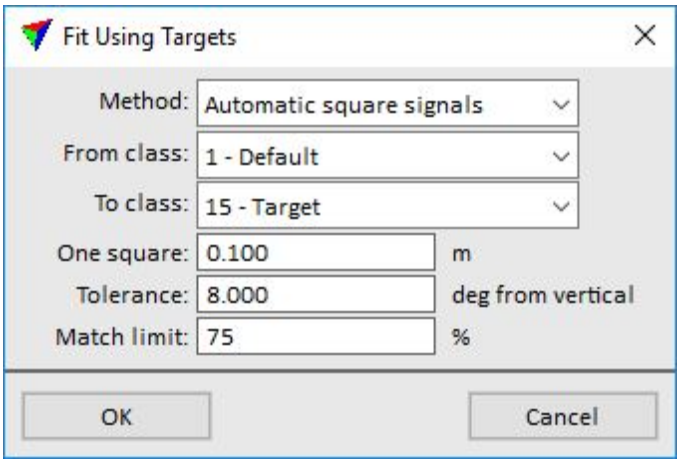
The transformation can also be applied to trajectories that are loaded in TerraScan and linked to the point cloud loaded in TerraScan. This may be useful for SLAM data sets that include a trajectory solution in a local coordinate system. As a result, the trajectory positions are georeferenced as well.

If the difference between the original coordinate values of the point cloud and the target coordinates is very large, it may be useful to shift the point cloud to the approximate location of the geographic coordinate system before applying the **Fit using targets** command. This can be done by defining a linear transformation in the TerraScan **Settings** and applying this transformation when reading the point cloud into TerraScan. The same must be applied to trajectories, if available.

To fit a point cloud to targets:

1. Select **Fit using targets** command from the **Line** pulldown menu.

   This opens the **Fit Using Targets** dialog:



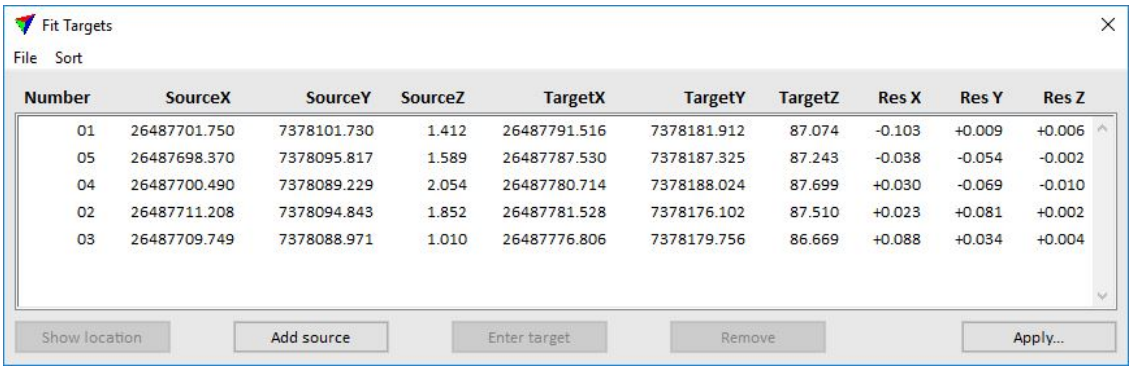
2. Define settings and click OK.

   If the **Method** is set to an automatic target detection method, the software searches for the targets in the point cloud. It opens the Fit targets window. The window contains commands for completing the fitting task.

SETTING	EFFECT
Method	Fitting method: • **Manual entry** - source and target points for computing the transformation are defined manually. No additional settings are required for this method and the Fit targets window is opened without any entries. • **Automatic ball targets** - ball objects are available in the point cloud and represent control point locations. • **Automatic square signals** - square signal marker objects are available in the point cloud and represent control point locations. The signals must be located on vertical surfaces and visible in RGB colors stored for the point cloud.
From class	Point class in which the software searches for the target objects, such as balls or signal markers.
To class	Point class into which points from the target objects are classified.
Diameter	Diameter length of a ball target object. This is only active if **Method** is set to **Automatic ball targets**.
One square	Size of one square in a square signal marker. Typically, a chess-board pattern of four black and white squares forms a signal. This is the size of one black or white square. This is only active if **Method** is set to **Automatic square signals**.

SETTING	EFFECT
Tolerance	For ball targets, maximum distance of points from the target objects to be considered as returns from the target object. For square signals, maximum angle off from vertical surfaces.
Require	Minimum amount of points that are returns from a target. Only targets with at least this amount of points are considered. This is only active if **Method** is set to **Automatic ball targets**.
Match limit	Defines how easily the software finds a square signal based on color values. This needs to be changed if the software finds too many wrong targets or not enough correct targets in the point cloud. This is only active if **Method** is set to **Automatic square signals**.

Fit targets



Number	SourceX	SourceY	SourceZ	TargetX	TargetY	TargetZ	Res X	Res Y	Res Z
01	26487701.750	7378101.730	1.412	26487791.516	7378181.912	87.074	-0.103	+0.009	+0.006
05	26487698.370	7378095.817	1.589	26487787.530	7378187.325	87.243	-0.038	-0.054	-0.002
04	26487700.490	7378089.229	2.054	26487780.714	7378188.024	87.699	+0.030	-0.069	-0.010
02	26487711.208	7378094.843	1.852	26487781.528	7378176.102	87.510	+0.023	+0.081	+0.002
03	26487709.749	7378088.971	1.010	26487776.806	7378179.756	86.669	+0.088	+0.034	+0.004

COMMAND/BUTTON	EFFECT
Show location	Shows the location of a selected target object. Select an entry in the targets list and click **Show location**. Move the mouse pointer inside a view. The target location is highlighted by a small square and a cross. You can center the target object in the view by placing a data click inside the view.
Add source	Add a new target object manually. Required, if **Manual entry** was selected as **Method** in the **Fit Using Targets** dialog or if the automatic detection of target objects fails.
Enter target	Define target coordinates for a target object manually. Required, if **Manual entry** was selected as **Method** in the **Fit Using Targets**

COMMAND/BUTTON	EFFECT
	dialog or if there are no target coordinates provided in the control point text file.
Remove	Delete a selected target object from the list and thus, from the transformation computation. Should be done for false findings of the automatic target object search before reading a control point text file.
Apply	Apply a transformation to loaded data.
File / Read targets xyz	Read control point coordinates from a text file and link them with the source coordinates.
File / Save source xyz	Write a text file with source point coordinates.
File / Save transformation	Write the transformation parameters into the Transformation category of TerraScan **Settings**. A transformation of **Type 3D translate & rotate** and with the given **Name** is created.
File / Save rubbersheet	Write the rubbersheet correction values into a text file.
Sort	Sort the target object list by **Increasing** | **Decreasing X** | **Y**.

To add a new target object manually:

1. Create a narrow cross section of the target object in the point cloud by using the Draw vertical section tool.
2. Click **Add source** button in the **Fit Targets** window.
3. Place a data click in the cross section where you see the center point of the target object.

   This adds the source coordinates of the new target object to the list in the **Fit Targets** window.

To enter target coordinates for a target object manually:

1. Select the target object in the list in the **Fit Targets** window.
2. Click **Enter target** button in the **Fit Targets** window.
3. Place a data click in any view at the target location. You may use a snapping methods to snap, for example, to the drawing of a control point in the CAD file in order to get exact XYZ coordinates.

   This adds the target coordinates to the target object selected in the list.

To continue with fitting a point cloud to target objects:

3. Check the targets that are listed in the **Fit Targets** window. If necessary, add additional target objects or remove false findings by using the corresponding buttons of the window.

4. Select **Read targets xyz** command from the **File** pulldown menu.

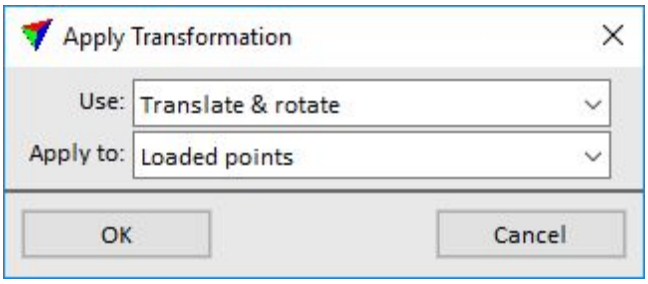
   This opens the **Target coordinates file** dialog, a standard dialog for selecting a file.

5. Select the control points text file and click **Open**.

   The software links the control point coordinates stored in the text file with the source coordinates. Coordinates that produce the smallest residuals are linked to each other.

6. Click **Apply** button in the **Fit Targets** window.

   This opens the **Apply Transformation** dialog:



7. Define settings and click OK.

   This transforms the loaded point cloud and possibly also the active trajectory files.

SETTING	EFFECT
Use	Transformation type: • **Translate & rotate** - applies a translation and then a 2D rotation to the point cloud. • **Translate & rotate + rubbersheet** - additional rubbersheet correction.
Apply to	Determines which data is modified: • **Loaded points** - point cloud loaded in TerraScan. • **Loaded points + trajectories** - point cloud loaded in TerraScan, and trajectories that are in the active trajectory folder and linked with the loaded points.

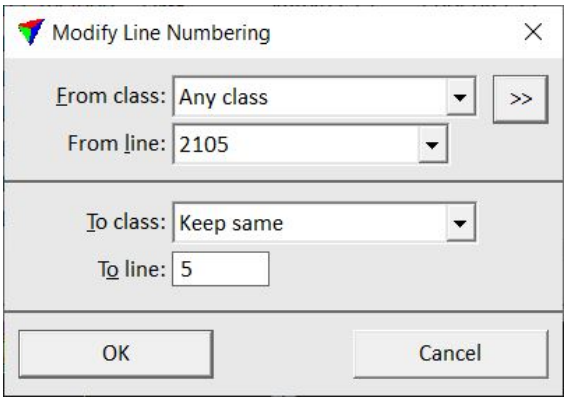
Modify numbering

Modify numbering command modifies the line number of points in a single or all lines. The modification can be done for all points or a selection of point classes. Additionally, points of the renumbered line(s) can be classified to another class at the same time.

To modify line numbering:

1. Select **Modify numbering** command from the **Line** pulldown menu.

   This opens the **Modify Line Numbering** dialog:



2. Define settings and click OK.

   This modifies the numbering for the laser points in the selected classes.

SETTING	EFFECT
From class	Point class(es) for which the new line number is assigned.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **From class** field.
From line	Line number to be changed. This can be a specific line number between 0 and 65535, or **Any line**.
To class	Target class for laser points of the renumbered line(s). Select Keep **same** if you do not want to classify the points.
To line	New line number assigned to the points. This can be a specific line number between 0 and 65535.

The command can be used to classify points of a specific line into another class. If the line numbers are the same in the **From line** and **To line** fields, the line attribute of points is not effected.

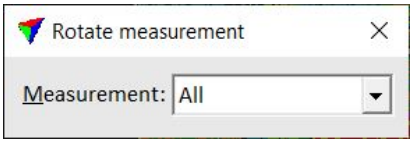
Rotate

Rotate command lets you rotate a point cloud interactively. The point cloud is rotated around a center point and only in horizontal direction.

To rotate a point cloud:

1. Select **Rotate** command from the **Line** pulldown menu.

This opens the **Rotate measurement** dialog:



2. Define settings.

3. Place a data click inside a view in order to define the center point of the rotation.

   If the mouse pointer is moved, a temporary line illustrates the base line for the rotation.

4. Place another data click inside the view in order to define the origin point of the rotation (= end point of the base line).

   If the mouse pointer is moved, a temporary line illustrates the direction and angle of the rotation.

5. Place another data click inside the view in order to define the destination point of the rotation (= rotation direction and angle).

   This rotates the point cloud to the new position.

SETTING	EFFECT
Measurement	Points that are effected by the rotation: points from all lines or only from one specific line. The list contains all line numbers that are available in the loaded point cloud.

Start new at selection

Start new at selection command starts a new line numbering at the location of a selected laser point. A new line number is assigned to all laser points that are recorded after the selected point. The line number of all following points is increased by 1.

To start a new line numbering at a selection:

1. Select a laser point in **TerraScan** window using the **Identify** button or by selecting a row in the point list.

2. Select **Start new at selection** command from the **Line** pulldown menu.

   This assigns a new line number to all laser points recorded later than the selected point.

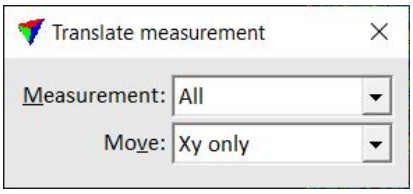
Translate

Translate command lets you shift a point cloud interactively. The shift can be horizontally, vertically, or both.

To translate a point cloud:

1. Select **Translate** command from the **Line** pulldown menu.

   This opens the **Translate measurement** dialog:



2. Define settings.

3. Place a data click inside a view in order to define the origin point of the translation.

   If the mouse pointer is moved, a temporary line illustrates the translation line.

4. Place another data click inside the view in order to define the target point of the translation.

   This shifts the point cloud to the new position.

SETTING	EFFECT
Measurement	Points that are effected by the translation: points from all lines or only from one specific line. The list contains all line numbers that are available in the loaded point cloud.
Move	Dimension of the translation: • **Xyz** - 3D shift. • **Xy only** - horizontal shift. • **Z only** - vertical shift.

Output pulldown menu

Commands from the **Output** pulldown menu are used to create output alignment reports, to create surface models, to export points as lattice or raster files, and to draw points into the CAD file.

TO	USE COMMAND
Output report from alignment	Output alignment report
Create a surface model	Create surface model
Export laser data into a lattice file	Export lattice model
Export laser data into a colored image	Export raster image
Draw points into the CAD file	Write to design file
Draw points as line strings into the CAD file	Draw as line strings

Create surface model

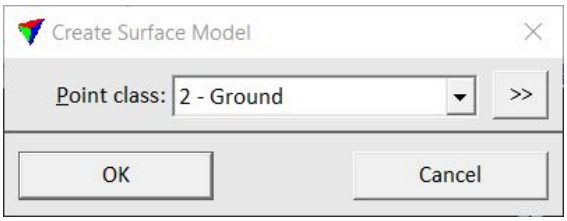
Create surface model command passes points in a selected class to TerraModeler. TerraModeler creates a triangulated surface model from the points.

The command requires TerraModeler running on the same computer. It starts the application if it is not yet running.

To create a triangulated surface model:

1. Select **Create surface model** command from the **Output** pulldown menu.

   This starts TerraModeler if it is not yet running and opens the **Create Surface Model** dialog:



2. Select a **Point class** to use for surface creation and click OK.

   TerraModeler opens the **Triangulate surface dialog**.

3. Define settings and click OK.

   This opens the TerraModeler **Surface settings** dialog.

4. Enter a descriptive name for the new surface, define other settings if required, and click OK.

   TerraModeler creates the surface model.

   For more detailed information about surface creation and display in TerraModeler, see the TerraModeler User Guide.

The command performs a similar action as the Create Editable Model tool. However, the surface model created by the command is not editable, which means that the surface model is not updated if the points in the source class are modified.

Draw as line strings

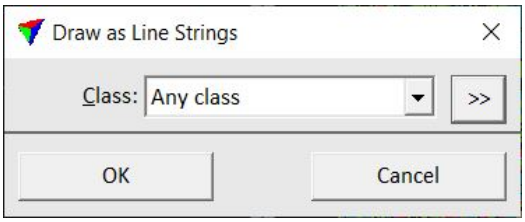
Draw as line strings command draws points loaded in TerraScan as line string element(s) into the CAD file. The points are connected in the order they appear in the file or in the point list in the **TerraScan** window.

The line strings may be separated by a line number assigned to the points. The command connects only points of the same line number with a single line string.

To write points as line strings into the CAD file:

1. Select **Draw as line strings** command from the **Output** pulldown menu.

   This opens the **Draw as Lines Strings** dialog:



2. Select a point class and click OK.

   This draws the line strings into the CAD file using active symbology settings of the CAD file.

SETTING	EFFECT
Class	Point class(es) to draw into the CAD file. The list includes the active classes in TerraScan.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Class** field.

Export lattice model

Export lattice model command creates a grid file with uniform distances between points from one or more selected point classes. For each grid point, the lattice model file stores XY coordinates and one of the following value types:

- elevation values
- point count/density values
- analytical values, such as average intensity, horizontal or vertical distance, surface roughness

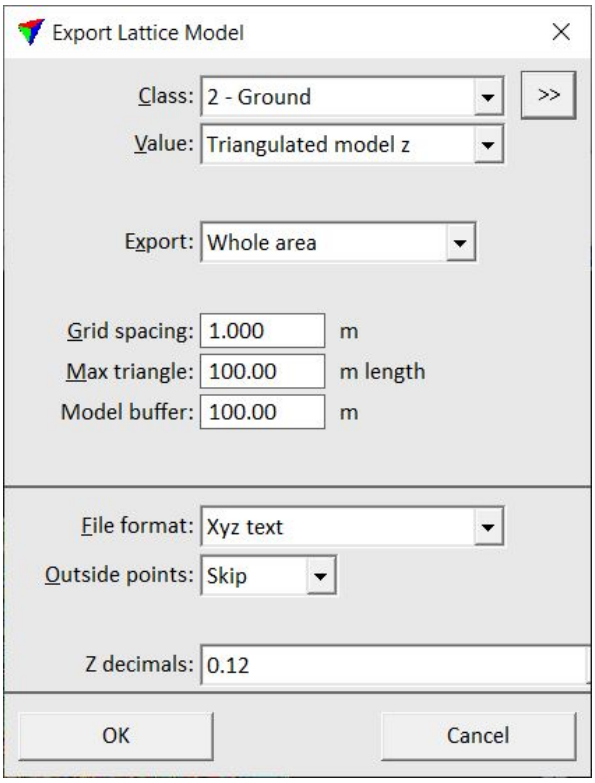
There are several formats supported to store lattice models as raster or text files. The command requires the selection of at least one polygon that defines the lattice model boundary. If several polygons are selected, the software creates a separate lattice model file for each polygon. Text elements placed inside the polygon(s) can be used as file names for the lattice model files. A polygonal area is always expanded to a rectangular area of a lattice model. The cells outside the polygon are filled with a defined “outside” value or can be skipped from output in XYZ text files.

To export a lattice model:

1. (Optional) Draw rectangle(s) around areas from which you want to create lattice model(s). Place text elements inside the rectangles. Select rectangle(s) and text(s).

2. Select **Export lattice model** command from the **Output** pulldown menu.

   This opens the **Export Lattice Model** dialog:



3. Select settings and click OK.

   This starts the lattice model file creation.

   If **File naming** is set to **Enter name for each**, the **Export lattice model** dialog opens, a standard dialog for saving files.

4. Define a location and name for the lattice file and click **Save**.

   Repeat step 4 for each lattice model.

SETTING	EFFECT
Class	Source class(es) for lattice model creation.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Class** field.
Value	Value stored for each grid point of a lattice model: • **Triangulated model z** - elevation value calculated from a TIN model of the points in the source class(es). • **Highest hit z** - elevation value determined by the highest point in the source class(es). • **Average hit z** - elevation value calculated as average of points in the source class(es) falling inside the grid cell.

	• **Lowest hit z** - elevation value determined by the lowest point in the source class(es). • **Closest hit z** - elevation value determined by the point closest to the grid cell center in the source class(es). • **Point count** - amount of points falling inside the cell. • **Point density** - amount of points per squared master unit. • **Average intensity** - average intensity value of points in the source class(es) falling inside the grid cell. • **Distance to point** - average horizontal distance between a grid point and the two closest points in the source class(es). • **Dz from ground** - vertical distance between a grid point and a surface model created from the Ground class(es). • **Surface roughness** - difference of a grid point from a plane fitted to the closest points in the source class(es). Represents the local elevation variation of points in the source class(es). • **Biggest distance** - biggest distance value inside the grid cell. Requires that a distance value is computed for each point using the command for loaded points or corresponding macro action. • **Average distance** - average distance value of points inside the grid cell. Requires that a distance value is computed for each point using the command for loaded points or corresponding macro action. • **Smallest distance** - smallest distance value inside the grid cell. Requires that a distance value is computed for each point using the command for loaded points or corresponding macro action.
Ground	Source class(es) for creating a surface model. This is only active if **Value** is set to **Dz from ground**.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Ground** field.
Export	Area to be exported into lattice model file(s): • **Whole area** - the area covered by the loaded points is exported into a single lattice model file.

	• **Selected rectangle(s)** - the area inside selected polygon(s) is exported into separate files.
Expand by	Distance by which a selected polygon is expanded for the model export. The area covered by the expanded polygon is included in a lattice model file. This is only active if **Export** is set to **Selected rectangle(s)**.
Grid spacing	Defines the distance between grid points and thus, the resolution of the lattice model.
Max triangle	Maximum length of a triangle edge for TIN creation. Effects how big gaps are filled in the lattice model by interpolating grid point values from the TIN. This is only active if **Value** is set to **Triangulated model z**, or **Distance to point**.
Model buffer	Width of a buffer area around the actual model area that is considered for calculating grid point values. This is only active if **Value** is set to **Triangulated model z**, **Distance to point**, **Dz from ground**, or Surface **roughness**.
Fill gaps up to	Defines the size of gaps that are filled in the lattice model by deriving grid point values from closest points in the source class(es). This is only active if **Value** is set to **Highest**, **Average**, **Lowest hit z**, **Average intensity**, or **Surface roughness**.
File format	Format of the lattice model file: **ArcInfo**, **GeoTIFF**, **Intergraph GRD**, **Raw**, **Surfer ASCII** or **binary**, **Xyz text**.
Z unit	Unit of grid point values. Relevant for formats storing elevations as integers. This is only active if **File format** is set to **GeoTIFF**, **Intergraph GRD**, or **Raw**.
Outside points	Defines how the software handles grid cells that are not covered by points in the source class(es): **Skip** or **Output**. This is only active if **File format** is set to **Xyz text**.
Outside Z	Defines the value for grid cells that are inside the rectangular lattice model area but not covered by points in the source class(es). This is only active if **File format** is set to **ArcInfo**, **GeoTIFF**, **Surfer ASCII** and **binary**, **Xyz text**.
Z decimals	Determines the number of decimals stored for the grid point value. This is only active if **File**

	format is set to **ArcInfo**, **Surfer ASCII**, or **Xyz text**.
Create TFW files	If on, the software creates external georeference files for GeoTIFFs. This is only active if **File format** is set to **GeoTIFF**.
File naming	Defines how lattice model files are named: • **Enter name for each** - a name for each lattice model file has to be defined manually. • **Selected text elements** - selected text elements inside the lattice model area are used as file names. This is only active if **Export** is set to **Selected rectangle(s)**.
Directory	Directory for storing lattice model files. Click on the **Browse** button in order to select a folder in the **Browse For Folder** dialog. This is only active if **File naming** is set to **Selected text elements**.
Extension	Extension of lattice model files. There may be a specific extension required for a certain file format, such as .GRD for ArcInfo or .TIF for GeoTIFF. This is only active if **File naming** is set to **Selected text elements**.

Lattice models can be produced in batch mode by using

- Export lattice models command for project blocks.
- Export lattice macro action.
- Produce lattice models command in TerraModeler.

Export raster image

Export raster image command generates a raster image where pixel values are derived from laser point attributes. The source data is points loaded in TerraScan.

The raster image can be created as Windows bitmap (.BMP) or GeoTIFF (.TIF). The color of a pixel is determined using laser points whose coordinate values fall inside the pixel. The coloring attribute can be chosen as:

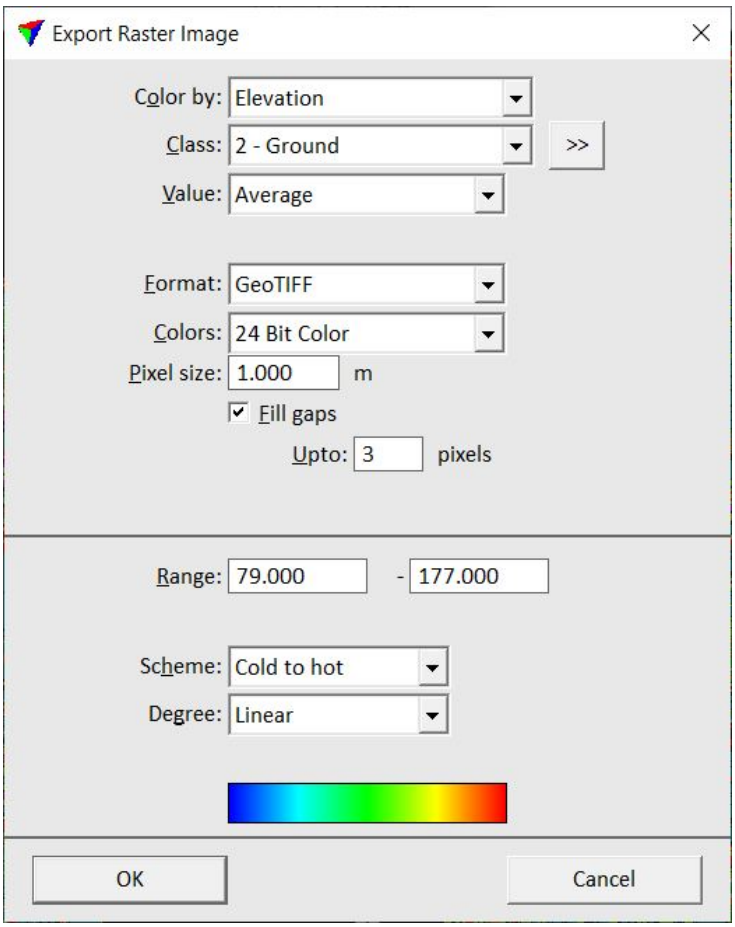
- **Elevation** - laser point elevation.
- **Elevation difference** - elevation difference between laser points of two different classes.
- **Intensity hits** - laser point intensity.
- **Intensity footprint** - average intensity of points within a footprint area overlapping the pixel.
- **Point color** - color values stored for laser points.

- **Point class** - laser point class.
- **Road intensity** - intensity of laser points computed by directional sampling along road alignment vectors.
- **Slope** - slope gradient based on normal vector computation.

To create a colored raster image:

1. Select **Export raster image** command from the **Output** pulldown menu.

   This opens the **Export Raster Image** dialog:



2. Define settings. You may define your own coloring scheme by using the **Define** button.

3. Click OK.

   This starts the generation of the raster file and opens the **Export raster image** dialog, a standard dialog for saving files.

4. Define a location and name for the output file and click **Save**.

   This creates the raster image.

SETTING	EFFECT
Color by	Coloring attribute. See description above.

SETTING	EFFECT
Class	Point class(es) to use for creating the raster file. If **Color by** is set to **Elevation difference**, two classes have to be selected.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Class** field.
Alignments	Level on which the road alignment vectors are located. This is only active if **Color by** is set to **Road intensity**.
Value	Determines the value for each pixel: • **Lowest** - smallest value of points inside the pixel area. • **Average** - average value of points inside the pixel area. • **Highest** - highest value of points inside the pixel area. This is only active if **Color by** is set to **Elevation**, **Elevation difference**, or **Intensity hits**.
Sampling	Computation of raster values based on directional sampling: • **Road intensity** - distance along the road alignment element(s). • **Slope** - radius around a point.
Format	Format of the output file: **Windows BMP**, **GeoTIFF**, or **GeoTIFF + TFW**.
Colors	Color depth of the output file for elevation or intensity coloring: • **24 Bit Color** - true color image. • **256 Colors** - 256 color image. • **Grey Scale** - 8 bit gray scale image.
Pixel size	Size of each pixel in the output file.
Fill gaps	If on, gaps up to the given number of pixels are filled in places where there are no laser points inside a pixel area.
Attach as reference	If on, the output image is attached as a raster reference. This requires TerraPhoto to run.
Draw placement rectangle	If on, the output image boundary is drawn as a shape into the CAD file. The shape is drawn on the active level using the active symbology settings of the CAD file.

SETTING	EFFECT
Range	Defines the value range that is covered by the color scheme for **Elevation** and **Intensity** coloring. Should be set to the general elevation or intensity range covered in the laser data to ensure that all values are represented by the complete color scheme.
Unit	Defines the unit for **Slope** coloring: **Degree** or **Percentage**.
Scheme	Type of coloring scheme for elevation or intensity coloring: • **Cold to hot** - varies from blue for low pixel values via cyan, green, and yellow to red for high pixel values. This is a common coloring scheme for elevation coloring. • **Hot to cold** - varies from red for low pixel values via cyan, green, and yellow to blue for high pixel values. • **Selected colors** - a user-defined coloring scheme can be created by clicking on the **Define** button. The dialog for color scheme definition depends on the selection of the color depth in the **Colors** field, 24-Bit Color or 256 Colors. • **Black to white** - varies from black for low pixel values to white for high pixel values. This is only active if **Colors** is set to **Grey scale**. This is the common coloring scheme for intensity coloring. • **White to black** - varies from white for low pixel values to black for high pixel values. This is only active if **Colors** is set to **Grey scale**.
Degree	Determines how the color changes in color schemes are computed. **Warm** and **Hot** move a coloring scheme towards the red-yellow color range, **Cool** and **Cold** towards the blue-cyan color range. For gray scale images, **Light** moves the gray scale towards white and light gray, **Dark** towards black and dark gray. **Linear** defines a linear distribution of colors.

Raster images can be produced in batch mode by using the Export raster images command for project blocks.

Output alignment report

Output alignment report command generates a report with information at given intervals along an alignment element. The command requires the definition of an alignment report format in Alignment reports category of TerraScan **Settings**.

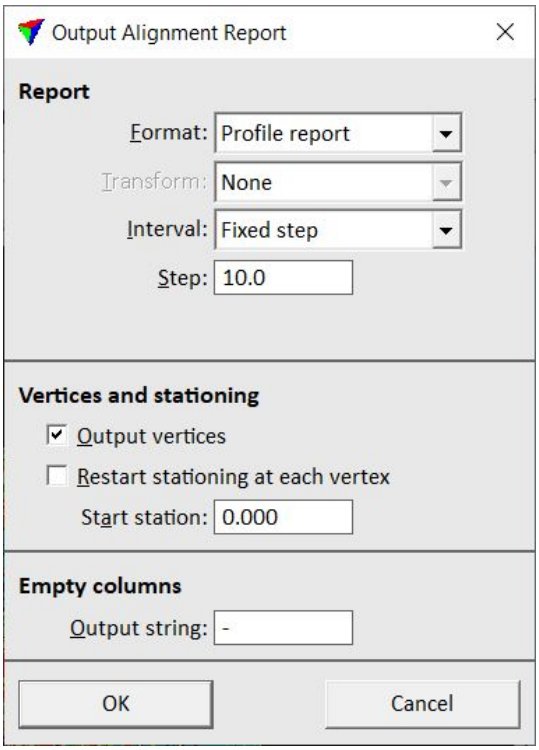
Alignment elements can be any linear element type such as line strings, shapes, circles, etc.. The report can be seen as a table where each row corresponds to an alignment station and each column contains a specific type of information.

To create an alignment report:

1. Select an alignment element.

2. Select **Output alignment report** command from the Output pulldown menu.

   This opens the **Output Alignment Report** dialog:



3. Define settings and click OK.

   This opens the **Alignment report** dialog. The dialog contains the list of stations along the alignment element and the information for each station according to the alignment report definition.

   You can save the report as space-delimited text file by using the **Text file** command, or as tabulator-delimited text file by using the **Table file** command from the **Output** pulldown menu of the dialog.

   To show the location of a station, select a line in the **Alignment report** dialog. Click on the **Show location** button and move the mouse pointer into a view. This highlights the selected station with a cross.

To identify a station, click on the **Identify** button and place a data click close to a station in a view. This selects the corresponding line in the Alignment **report** dialog.

SETTING	EFFECT
Format	Alignment report format to use. Defines the report content.
Transform	Transformation applied to coordinates in the report. The list contains transformations that are defined in Coordinate transformations / Transformations category of TerraScan **Settings**. This is only active if coordinates are included in the report.
Interval	Distance between two consecutive stations along the alignment element: • **Fixed step** - fixed distance along the whole alignment element. • **Triangle edges** - distance is defined by intersection points between surface triangle edges and the alignment element. This requires a surface model loaded in TerraModeler.
Surface	Name of the surface model. This is only active if **Interval** is set to **Triangle edges**.
Step	Distance between consecutive stations along the alignment element. Given in CAD file master units. This is only active if **Interval** is set to **Fixed step**.
Min step	Minimum distance between consecutive stations along the alignment element. Given in CAD file master units. This is only active if **Interval** is set to **Triangle edges**.
Output vertices	If on, the report includes the information from each location of a vertex of the alignment element in addition to the station locations.
Restart stationing at each vertex	If on, station values restart from **Start station** at each vertex location.
Start station	Station value at the start vertex of the alignment element.
Output string	Text string written into the report if a column has no valid value.

Write to design file

Write to design file command draws points loaded in TerraScan permanently into the CAD file. The command uses drawing rules assigned to point classes. See Define Classes tool for information about class definitions.

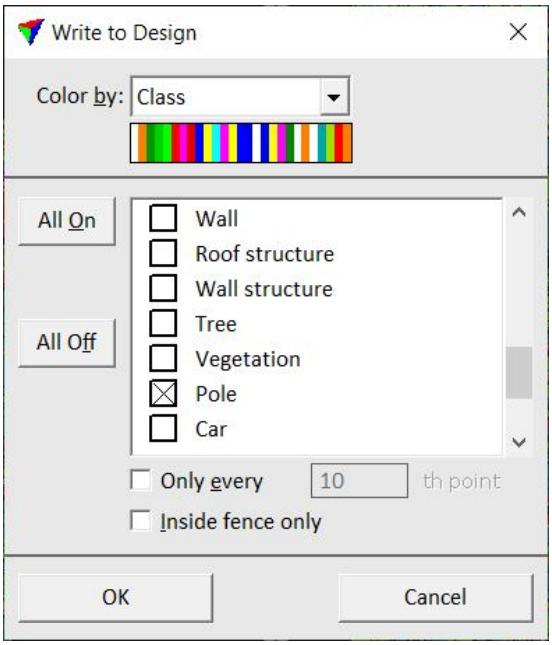
The points drawn into the CAD file can be colored by the following methods:

- **Active symbology** - the active color of the CAD file is used for all points.
- **Class** - colors are defined in the active class file of TerraScan. See also Color by Class display mode for points.
- **Elevation** - colors are determined by the elevation values of points. The coloring scheme can be changed by clicking on the **Colors** button. See also Color by Elevation display mode for points.
- **Line** - colors are determined by the line numbers of points. The colors can be changed by clicking on the **Colors** button. See also Color by Line display mode for points.
- **Intensity** - colors are determined by the intensity values of points. The coloring scheme can be changed by clicking on the **Colors** button. See also Color by Intensity display mode for points.
- **Point color** - colors are determined by color values assigned to points.

To write points to the CAD file:

1. Select **Write to design file** command from the **Output** pulldown menu.

   This opens the **Write to Design** dialog:



2. Define settings and click OK.

   This draws the points into the CAD file.

SETTING	EFFECT
Color by	Coloring method.

SETTING	EFFECT
Color	Opens the dialog for selecting colors or a coloring scheme. This is only active if **Color by** is set to **Elevation**, **Line** or **Intensity**.
Classes	Point class(es) to draw into the CAD file. The list includes the active classes in TerraScan. You can switch all classes on or off by using the **All On** and **All Off** buttons.
Only every	If on, only every n th point is drawn into the CAD file where n is the given number of points.
Inside fence only	If on, only points inside a MicroStation fence or selected polygon are drawn into the CAD file.

You can undo the drawing of points by using the **Undo** command of the CAD platform.

Point pulldown menu

Commands from the **Point** pulldown menu are used to undo processing steps performed on loaded points, to manipulate point attributes, to select points and to delete points.

TO	USE COMMAND
Undo a classification action	Undo
Undo classification actions from a list	From list
Edit attributes of one or more laser points	Edit selected
Select laser points of a specific class	Select by class
Select laser points with specific attribute values	Find
Delete points	Delete

Delete

Delete commands can be used for deleting points from memory:

- Selected points - deletes the point(s) selected in the list of points.
- By point class - deletes all points in given class(es).
- By line - deletes all points of selected line(s).
- Inside fence - deletes all visible points inside a fence or selected polygon.
- Outside fence - deletes all visible points outside a fence or selected polygon.
- Using centerline - deletes all points which are more than a given distance away from a selected centerline element. Any linear CAD element type can serve as a centerline element.

If you delete points with the commands from the **TerraScan** window, the points are only deleted from memory. You need to save the points in order to remove points permanently from a file on a hard disk.

For LAS 1.4 files, you can decide whether to remove points or set the withheld bit. If **Action** is set to **Set withheld bit** in any dialog for deleting points, the point is kept and only the withheld bit is set.

To delete selected points:

1. Select points in the list of points of **TerraScan** window. You need to switch the display size of the **TerraScan** window to Medium dialog, Large dialog, or Wide dialog in order to see the list of loaded points and the selected points.

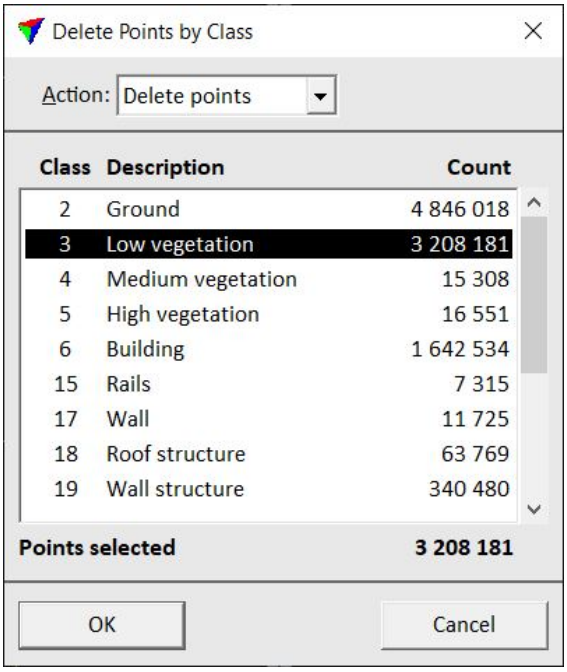
2. Select **Delete / Selected points** from the **Point** pulldown menu.

   This opens a dialog which asks for confirmation of the delete action. Click **Yes** in order to delete the points and **No** in order to cancel the process.

To delete points by point class:

1. Select **Delete / By point class** from the **Point** pulldown menu.

   This opens the **Delete Points by Class** dialog:



2. Select one or more class(es) of which you want to delete the points.

3. Click OK.

   This opens a dialog which informs about the number of deleted points.

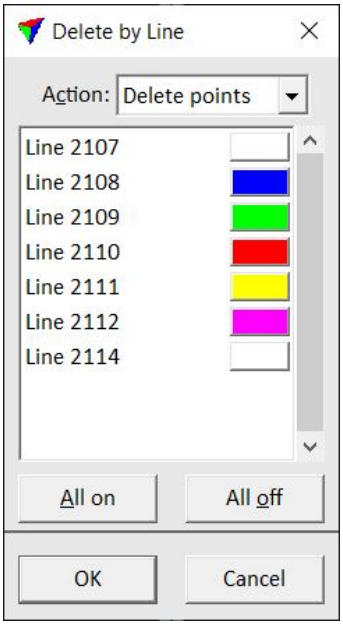
SETTING	EFFECT
Action	Defines what to do with the points:

SETTING	EFFECT
	• **Delete** - points are removed from the point cloud. This is the normal action. • **Set withheld bit** - the classification bit for withheld (deleted) points is set. This applies only to points stored in LAS 1.4 files.

To delete points by line:

1. Select **Delete / By line** from the **Point** pulldown menu.

   This opens the **Delete by Line** dialog:



2. Select one or more line(s) from which you want to delete the points.

3. Click OK.

   This opens a dialog which informs about the number of deleted points.

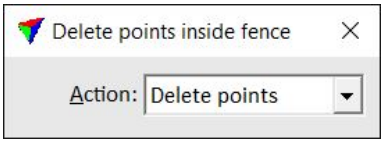
SETTING	EFFECT
Action	Defines what to do with the points: • **Delete** - points are removed from the point cloud. This is the normal action. • **Set withheld bit** - the classification bit for withheld (deleted) points is set. This applies only to points stored in LAS 1.4 files.
All on	Selects all lines in the list for deleting points.
All off	Deselects all lines in the list for deleting points.

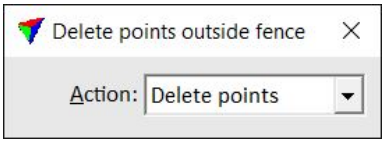
To delete points inside or outside a fence:

1. Draw a fence or a polygon around the area for which you want to delete points. Select the polygon.

2. Select **Delete / Inside fence** or **Delete / Outside fence** from the **Point** pulldown menu.

   This opens the **Delete Inside/Outside Fence** dialog:





3. Select an **Action**.

4. Click inside the view where the fence is defined.

   This deletes points inside or outside the fence. It effects only points that are visible in the view where the fence is defined. See Display mode for more information about point display settings.

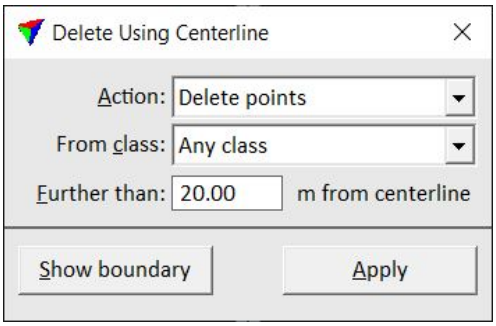
SETTING	EFFECT
Action	Defines what to do with the points: • **Delete** - points are removed from the point cloud. This is the normal action. • **Set withheld bit** - the classification bit for withheld (deleted) points is set. This applies only to points stored in LAS 1.4 files.

To delete points using a centerline:

1. Select a centerline element.

2. Select **Delete/ Using centerline** from the **Point** pulldown menu.

   This opens the **Delete Using Centerline** dialog:



3. Define settings.

4. (Optional) Show the distance by clicking on the **Show boundary** button. The boundary up to which points are deleted is displayed if the mouse pointer is moved inside a view.

5. Click **Apply**.

This opens a dialog which asks for confirmation of the delete action. Click **Yes** in order to delete the points and **No** in order to cancel the process. Another dialog shows the number of deleted points.

SETTING	EFFECT
Action	Defines what to do with the points: • **Delete** - points are removed from the point cloud. This is the normal action. • **Set withheld bit** - the classification bit for withheld (deleted) points is set. This applies only to points stored in LAS 1.4 files.
From class	Point class from which to delete points.
Further than	Defines an offset distance from the centerline within which points are kept. Points outside the distance are deleted.

You can delete points of a given class or line in batch mode by using the Delete by class or Delete by line macro actions.

Edit selected

Edit selected command lets you modify attributes for one or more points selected in the list of the **TerraScan** window. You need to switch the display size of the **TerraScan** window to Medium dialog, Large dialog, or Wide dialog in order to see the list of loaded points and to select points for editing.

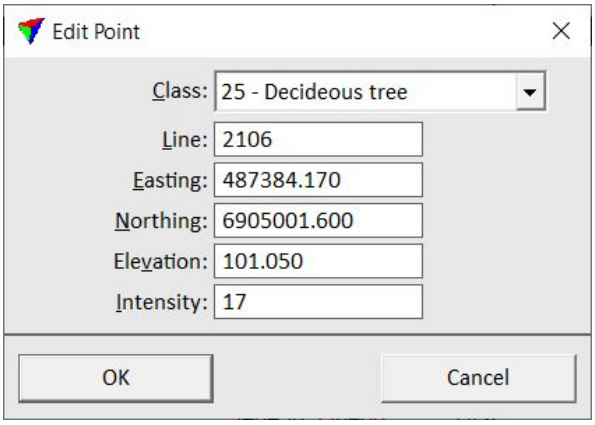
For a single point, you can modify the class, line number, coordinate values, and the intensity value. For several points, you can edit the class (code), line number, elevation value, and apply an elevation change (dz).

To edit attributes of one point:

1. Select a point in the list of **TerraScan** window.

2. Select **Edit selected** command from the **Point** pulldown menu.

   This opens the **Edit Point** dialog:



3. Define new values for attributes that you want to change.

4. Click OK.

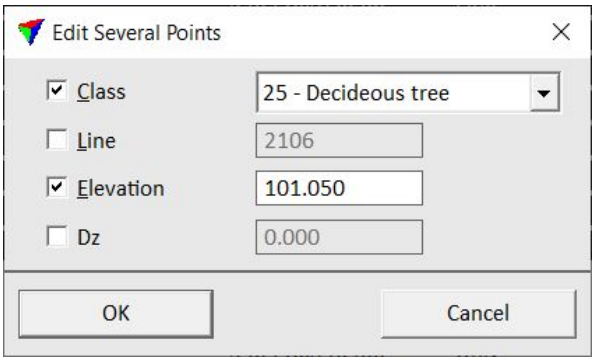
   This changes the attribute values for the selected point.

To edit attributes of several points:

1. Select several points in the list **TerraScan** window.

2. Select **Edit selected** command from the **Point** pulldown menu.

   This opens the **Edit Several Points** dialog:



3. Switch on the attribute new values for attributes that you want to change.

4. Click OK.

   This changes the attribute values for all selected points.

Edit selected actions can not be undone.

Find

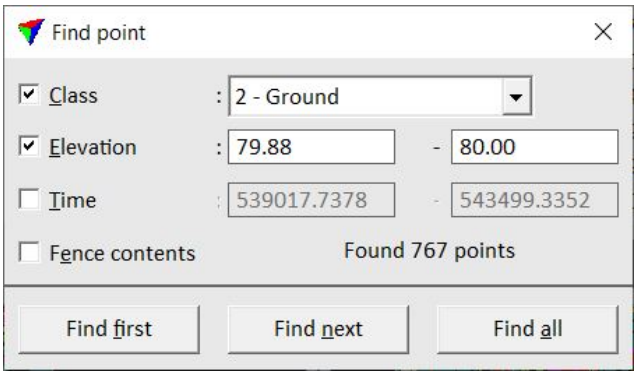
Find command selects points with specific attribute values. The attributes including class, elevation range, and time range. In addition, the selection of points can be limited to a fence or selected polygon area. The command selects either all points with the given attribute values, or only the first and next point in the list of points.

You need to switch the display size of the **TerraScan** window to Medium dialog, Large dialog, or Wide dialog in order to see the list of loaded points and the selected points.

To select points with given attribute values:

1. Select **Find** command from the **Point** pulldown menu.

   This opens the **Find point** dialog:



2. Switch on the attribute(s) and define values for points to be selected.

3. (Optional) Switch on **Fence contents** to limit the search to an area defined by a fence or selected polygon.

4. If you want to select the first point in the point list for which the given values apply, click **Find first**. The next point in the list with corresponding values can be selected by clicking **Find next**.

   OR

5. If you want to select all points for which the given values apply, click **Find all**.

   This selects the points in the list of points in **TerraScan** window. The number of points is displayed in the **Find** point dialog.

From list

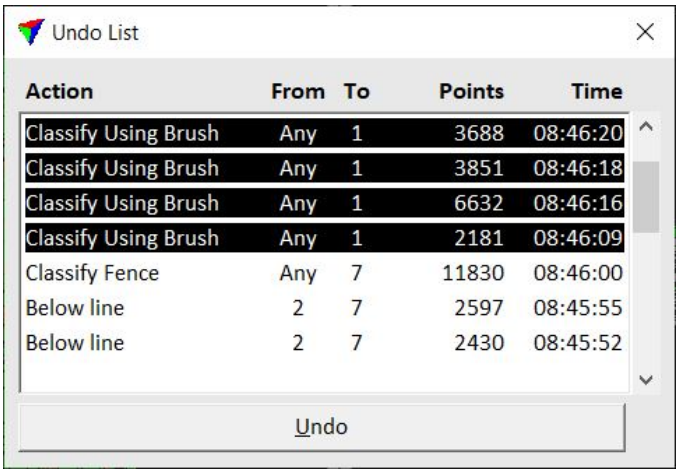
From list command lets you undo one or several processing steps. This is the preferred way to undo more than one step. The number of actions for undo is determined in Undo and backup category of TerraScan **Settings**.

The command opens the dialog which shows a list of all processing steps that fit into the memory reserved for the undo buffer. For each step, the action, time, and possibly additional information is displayed. You can undo a number of steps by selecting the earliest step.

To undo several actions:

1. Select **From list** command from the **Point** pulldown menu.

   This opens the **Undo List** dialog:



2. Select the step up to which you want to make an undo.

3. Click **Undo**.

   This makes all selected steps undone.

Select by class

Select by class command selects points of the same class. It requires the selection of one sample point in the **TerraScan** window. You need to switch the display size of the **TerraScan** window to Medium dialog, Large dialog, or Wide dialog in order to see the list of loaded points and to select a sample point.

To select points by class:

1. Select one point of the class to be selected in **TerraScan** window.

2. Select **Select by class** command from the **Point** pulldown menu.

   This selects all points of the same class as defined by the sample point.

Undo

Undo command lets you undo modifications of points. Processing steps that can be undone include:

- Manual classification steps
- Automatic classification steps
- Macro actions applied to loaded points
- Smoothen points and thin points commands
- Adjust to geoid command
- Changes to line numbering
- Automatic detection of wires

More than one action can be undone by using the command several times or by using the From list command. The number of actions for undo is determined in Undo and backup category of TerraScan **Settings**.

Tools pulldown menu

Commands from the **Tools** pulldown menu are used to process loaded laser points or to perform actions based on laser points. There are also commands for opening the macro window, starting addon tools, converting a geoid model, reading in text files for drawing certain types of vector data into the CAD file.

TO	USE COMMAND
Open toolbars of the **TerraScan** toolbox	Toolboxes
View statistics about points	Show statistics
Open the Macro window	Macro
Start an addon tool	Addon
Draw bounding box for fitting views	Draw bounding box
Draw points into a profile drawing	Draw into profile
Draw points into alignment cross sections	Draw into sections
Draw polygons around groups of points	Draw polygons
Smooth points	Smoothen points
Remove unnecessary point density	Thin points
Adjust the elevations of points to a geoid model	Adjust to geoid
Extract a local geoid model from a geoid file	Convert geoid model
Transform loaded points into a new projection system	Transform loaded points
Transform a known points file into a new projection system	Transform known points
Check the z accuracy of the point cloud data	Output control report
Assign color values to points	Assign color to points
Assign a distance value to points	Compute distance
Assign a normal vector value to points	Compute normal vectors
Assign color values from images to points	Extract color from images
Assign echo properties to laser points	Extract echo properties
Compare laser points with points from a reference project	Compare with reference
Sort points	Sort

TO	USE COMMAND
Draw building models from text files	Read / Building models
Draw paint markings from text files	Read / Paint lines
Draw poles from text files	Read / Poles
Draw polygons from text files	Read / Polygons
Draw section parameter values from text files	Read / Section parameters
Draw slope arrows from text files	Read / Slope arrows
Draw trees from text files	Read / Tree cells
Draw wires from text files	Read / Wires

Addon

Addon sub-menu consists of commands that call custom functions in TerraScan. New commands and functions can be added to TerraScan using a custom DLL.

An example is provided by the **View histogram** function which opens the **Laser intensity histogram** window. It displays the distribution of intensity values of all loaded points or of selected classes or scanners. The window also shows the average, median, and spread values of the points that are in the selected class and captured by the selected scanner.

See Chapter DLL Interface for more information about custom DLLs in TerraScan.

Adjust to geoid

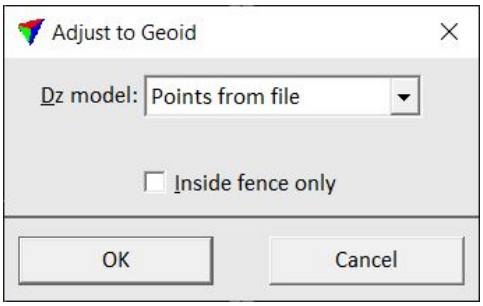
Adjust to geoid command adjusts the elevation values of a laser data set to a local elevation model. The elevation model can be defined by a text file, a TerraModeler surface, or a selected linear chain.

The theory of geoid adjustment and the use of the command in TerraScan are explained in detail in section Geoid adjustment.

To run elevation adjustment on loaded points:

1. Choose **Adjust to geoid** command from the **Tools** pulldown menu.

   This opens the **Adjust to Geoid** dialog:



2. Select the input model type in the **Dz model** field.

3. Click OK.

   If **Points from file** is selected as the **Dz model**, the **Geoid dz file** dialog opens, a standard dialog for opening a file.

4. Select the text file and click **Open**.

   This starts the geoid adjustment for the loaded points. When the process has finished, an information dialog shows the number of adjusted points and the variation of adjustment values.

SETTING	EFFECT
Dz model	Input model for geoid corrections: **Points from file**, **Selected linear chain**, a specific surface model that is active in TerraModeler.
Extend	Distance from a selected linear element by which the linear chain is extended for elevation value corrections. This is only active if **Dz model** is set to **Selected linear chain**.
Inside fence only	If on, geoid adjustment effects only loaded points that are located inside a fence or selected polygon. Requires a fence or selected polygon in the CAD file.

Assign color to points

Assign color to points command assigns a color value to laser points of one or more classes. This might be useful for laser point classes where color extraction from images results in poor coloring (e.g. thin features like wires) or to emphasize certain classes.

The color assigned to a point can be defined in different ways:

- **Constant color** - a constant RGB color is assigned to each point.
- **From intensity** - a text file storing intensity values is used to assign an RGB color value to points. The text file can be created in the Coloring Scheme dialog, opened from the Display mode dialog.
- **Color mixture** - a mixture of up to three colors is randomly assigned to laser points. A color mixture can be defined in Color mixtures category of TerraScan **Settings**.

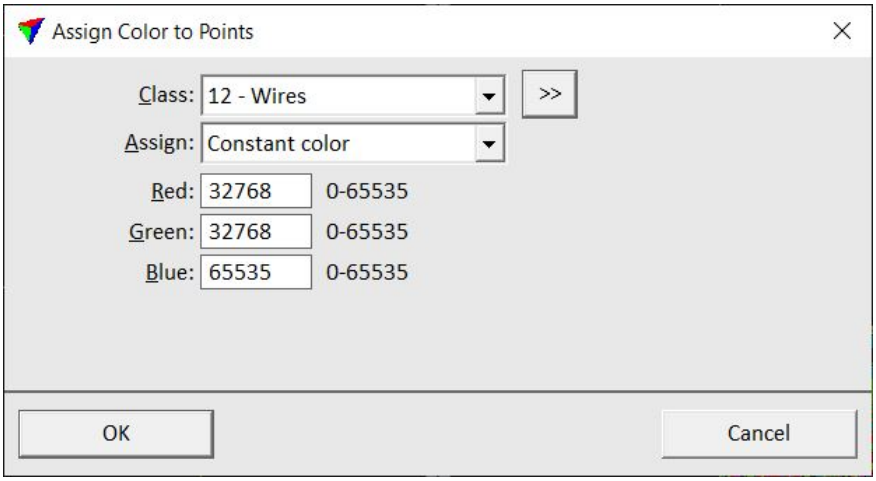
- **Group color** - points assigned to a group get a color value. The color per group is selected randomly, as for the Color by Group display mode for points. Non-grouped points do not get a color value.
- **Multi scanner intensity** - intensity values of three laser channels are assigned as RGB colors to points. This requires data of a scanner that records three intensity channels for each point. The RGB values then represent the strength of a return signal in each intensity channel. It might be necessary to optimize the intensity values in order to get better results from the color assignment.

Color values can be stored in TerraScan FastBinary and Binary files, and in LAS 1.2+ files.

To attach a color value to laser points:

1. Select **Assign color to points** from the **Tools** pulldown menu.

   This opens the **Assign Color to Points** dialog:



2. Define settings and click OK.

   This assigns the color to the loaded points in the selected class(es).

SETTING	EFFECT
Class	Point class(es) effected by the color assignment.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Class** field.
Assign	Color source: **Constant color**, **Color mixture, Group color**, **From intensity** or **Multi scanner intensity**.
Red | Green | Blue	RGB color values for a constant color definition. The color is defined with 16-bit values which can range from 0 to 65535. This is only active if **Assign** is set to **Constant color**.

SETTING	EFFECT
Color scheme	Location and name of a color scheme file storing intensity values. This is only active if **Assign** is set to **From intensity**.
Mixture	Name of the color mixture. The list contains all mixtures defined in Color mixtures category of TerraScan **Settings**. This is only active if **Assign** is set to **Color mixture**.
Red scanner | Green scanner | Blue scanner	Number of laser scanner channels. This is only active if **Assign** is set to **Multiple scanner intensity**.
Max distance	Maximum search radius around a point for the two closest neighbouring points from the other two scanners. This is only active if **Assign** is set to **Multiple scanner intensity**.
Default value	Determines the values for the two missing color channels for a point for which not enough neighbouring points from other scanners are found within the **Max distance**: • **Minimal value** - the value of 1 is used for the missing color channels. This results in a more colorful display and makes it easier to recognize features in the colored point cloud. This is recommended if colors are assigned for classification purposes. • **Grey color value** - the value of the single scanner is used for all channels which results in a grey color value.

Compare with reference

Compare with reference command compares two laser data sets from the same location. It classifies locations where the two data sets differ from each other. This is useful to locate places where buildings or other objects have been built or destroyed, trees have grown or ground has changed.

The command can be used for simple change detection analysis based in loaded laser points. The newer data set is stored in the active project, the older data set has to be defined as a reference project. See New project for information about defining a reference project.

The process can distinguish between ground and object comparison. Ground comparison is based on a triangulated ground surface calculated from the given classes in the active and the reference laser points. For object comparison, a search radius is defined around each active point within which the software expects a corresponding reference point on the same object.

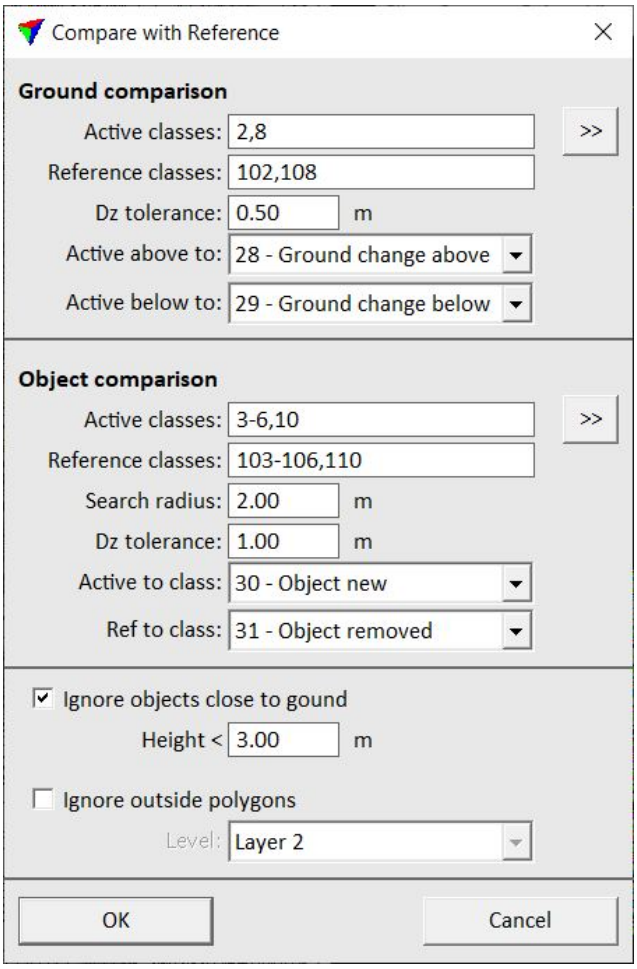
It is required that different classes are set for ground and object comparison, even if no classification is applied for one or the other comparison. However, the same class numbers can be used in active and reference projects.

To compare laser points with a reference project:

1. Select Open block command from the **File** pulldown menu. In the **Open Block** dialog, switch on the **Load reference points** option.

2. Select **Compare with reference** command from the **Tools** pulldown menu.

   This opens the **Compare with Reference** dialog:



3. Define settings and click OK.

   This classifies points for which there are no corresponding points in the other project.

SETTING	EFFECT
Active classes	List of classes for ground comparison in active laser points.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source

SETTING	EFFECT
	classes from the list that are then used in the **Active classes** field.
Reference classes	List of classes for ground comparison in reference laser points.
Dz tolerance	Elevation tolerance for ground comparison. Points within this tolerance are considered as corresponding points.
Active above to	Points in the active file that are below the ground surface in the reference file are classified into this class.
Active below to	Points in the active file that are above the ground surface in the reference file are classified into this class.
Active classes	List of classes for object comparison in active laser points.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Active classes** field.
Reference classes	List of classes for object comparison in reference laser points.
Search radius	Xy radius around each active point to search for corresponding reference points.
Dz tolerance	Elevation tolerance for object comparison. Points within this tolerance are considered as corresponding points.
Active to class	Active points for which there are no corresponding reference points are classified into this class.
Ref to classes	Reference points for which there are no active points are classified into this class.
Ignore objects close to ground	If on, points with a distance above the ground smaller than the given **Height** value are ignored for object comparison.
Ignore outside polygons	If on, points outside polygons on the given **Level** are ignored for comparison.

Compute distance

Compute distance command computes a distance value for points. The distance can be computed based on several references:

- 3D vector elements in the CAD file, such as planar shapes, linear elements or surface elements.
- the closest overlapping line in a point cloud.
- the average between overlapping lines.
- ground class(es) or other class(es).
- any surface model loaded in TerraModeler.
- powerline wires that have been produced with the Detect wires tool.
- range which is the distance to the scanner position at the point of time when a point was captured. This requires trajectory information available in TerraScan.
- built-in vegetation indexes **Normalized** and **Visual band differences**. This requires true-color (Visual band difference) and additionally infra-red color information (Normalized difference) stored for the points. The distance value expresses how likely a point belongs to vegetation (larger positive value) or not (larger negative value). Distance values range between -1 and +1. The vegetation index distance values may support the ground classification of photogrammetric point clouds.
- road bumps and potholes represented by vector elements, such as lines or polygons.

The distance value can be used for visualization of points and for further processing steps. The following processing commands rely on a distance computed from ground:

- Classify by distance / Classify groups by distance
- Assign groups and corresponding macro action
- Classify groups by best match

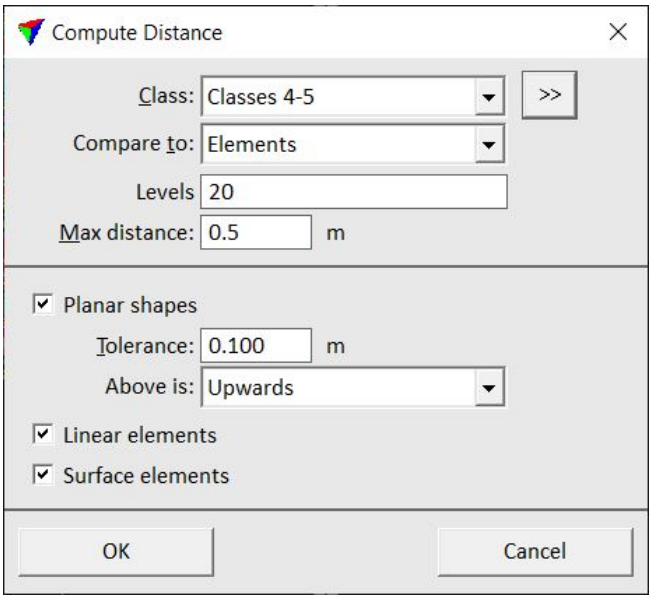
The distance value can be stored in TerraScan FastBinary files.

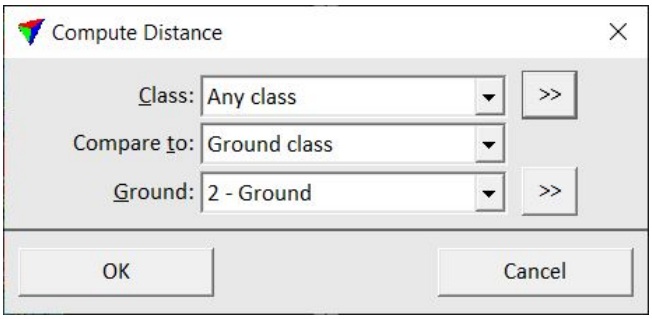
To compute distance values:

1. If you want to compute distance values for road bumps and potholes, select the vector elements that represent these road damages.

2. Select **Compute distance** command from the **Tools** pulldown menu.

   This opens the **Compute Distance** dialog:





2. Define settings and click OK.

   This computes the distance values. An information dialog shows the number of effected points when the process is finished.

SETTING	EFFECT
Class	Point class(es) for which dimensions and normal vectors are computed.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Class** field.
Compare to	Method for computing the distance: • **Elements** - distance between a point and vector elements drawn in the CAD file. • **Closest line 3D** - 3D distance between a point and its closest neighbour point. The computation takes the normal vector direction of points into account. • **Closest line dz** - vertical distance between a point and its closest neighbour point in XY of another line.

SETTING	EFFECT
	• **Line average Z** - vertical distance from a point to the average elevation between overlapping lines. The computation is based on TINs created from points of one line. • **Ground class** - vertical distance from a point to the ground class(es). A temporary TIN is created from points in the ground class(es) in order to compute the distance. • **Another class** - vertical distance from a point to points in the given **Other class**. • **Wires** - distance from a point to the closest powerline wire. Line strings for wires must be produced with the Detect wires tool. • **<TIN>** - distance from a point to a surface model loaded in TerraModeler. • **Range 3D** - 3D distance from the scanner position to a point. • **Range xy** - horizontal distance from the scanner position to a point. • **Range offset** - left/right distance from the scanner position to a point. • **Range forward** - forward/backward distance from the scanner position to a point. • **Range dz** - vertical distance from the scanner position to a point. • **Normalized difference** - built-in vegetation index. • **Visual band difference** - built-in vegetation index. • **Road bumps & potholes** - distance to road surface damages represented by vector elements.
Levels	Numbers of CAD file levels on which the elements are drawn. Level ranges and several levels can be define with minus and comma, for example: 1-5,10,22. This is only active if **Compare to** is set to **Elements**.
Max distance	Maximum distance of points from elements for which a distance value is computed. This is only active if **Compare to** is set to **Elements**.
Planar shapes	If on, planar shapes are considered as reference elements. This is only active if **Compare to** is set to **Elements**.
Tolerance	Defines a stroking tolerance for curved element types. This determines what shapes are considered as planar. This is only active if

SETTING	EFFECT
	Compare to is set to **Elements** and **Planar shapes** is switched on.
Above is	Defines which direction is considered as positive distance from planar shapes: • **Upwards** - points above a shape. • **Clockwise side** - points in clockwise direction of a shape. This is only active if **Compare to** is set to **Elements** and **Planar shapes** is switched on.
Linear elements	If on, linear elements are used for computing a distance. This is only active if **Compare to** is set to **Elements**.
Surface elements	If on, surface elements are used for computing a distance. This is only active if **Compare to** is set to **Elements**.
Max distance	Maximum distance between overlapping lines for which a distance value is computed. This is only active if **Compare to** is set to **Closest line 3D** or **Closest line dz**.
Ground	Point class(es) used as reference ground level. This is only active if **Compare to** is set to **Ground class**.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Ground** field. This is only active if **Compare to** is set to **Ground class**.
Other class	Point class(es) used as reference class(es). This is only active if **Compare to** is set to **Another class**.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Other class** field. This is only active if **Compare to** is set to **Another class**.
Max distance	Maximum distance between a point and reference points in another class. The distance value is not stored for a point if it is larger than the given maximum value. This is only active if **Compare to** is set to **Another class**.
Wire levels	Number(s) of CAD file level(s) on which powerline wires are drawn. Several level number can be separated by comma, a range

SETTING	EFFECT
	of level numbers can be defined by a minus sign, for example 10,14,22 or 3-5. This is only active if **Compare to** is set to **Wires**.
Find using	Distance definition: • **Vertical distance to wire** - vertical distance from a wire. • **3D distance to wire** - 3D distance from a wire. • **Falling tree logic** - each point is considered as the tip of a tree with its trunk at the xy location of the point and the elevation of the base point on the ground. The shortest distance of the point "falling like a tree" to a wire is computed. This is only active if **Compare to** is set to **Wires**.
Within offset	Maximum horizontal distance from a wire within which a distance value is computed. Basically, this value defines the corridor left and right of the wires for computing distance values. This is only active if **Compare to** is set to **Wires**.
Ground	Point class(es) used as ground level. This is only active if **Compare to** is set to **Wires** and **Find using** to **Fallen tree logic**.
Limit	Threshold value that separates vegetation from non-vegetation by using a vegetation index. This is only active if **Compare to** is set to **Normalized difference** or **Visual band difference**.
Max offset	Maximum horizontal distance of a point from the element that represents a bump or pothole. Only point within the given offset are considered in the distance computation. This is only active if **Compare to** is set to **Road bumps & potholes**.
Fit length	This is only active if **Compare to** is set to **Road bumps & potholes**.
Fit depth	This is only active if **Compare to** is set to **Road bumps & potholes**.

Compute normal vectors

Compute normal vectors command can be used to compute and store two additional attributes for laser points, a dimension and a normal vector.

The software determines the dimension of each point by analyzing the point and its closest neighbour points. There are three types of dimensions:

- **Linear** - points form a linear feature.
- **Planar** - points form a planar surface.
- **Complex** - random group of points.

The normal vector is computed for points of planar dimension. It is strongly recommended to have trajectory information available for computing normal vectors for mobile ground-based laser data. However, the process also runs without trajectory information which may give good results for airborne data sets.

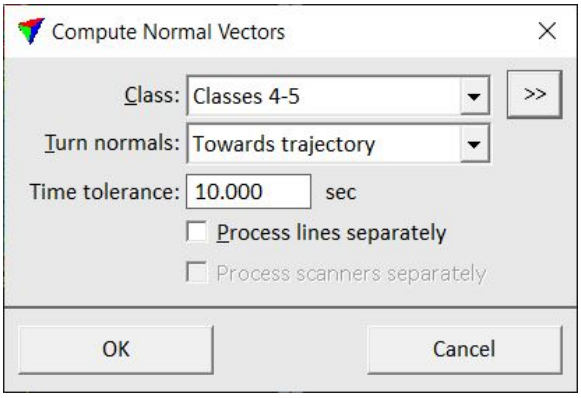
The attributes can be used for the visualization of points and for classifying points. For instance, points on a road surface in an MLS data set can be classified By normal vector in order to detect locally flat places. In ALS data, the normal vector can be utilized to analyze roof structures.

The dimension and the three components (XYZ) of the normal vector can be stored in TerraScan FastBinary files.

To compute normal vectors:

1. Select **Compute normal vectors** command from the **Tools** pulldown menu.

   This opens the **Compute Normal Vectors** dialog:



2. Define settings and click OK.

   This starts the computation process. It assigns a dimension value to all laser points and a normal vector to all points of planar dimension. An information dialog shows the number of points for which a normal vector has been computed.

SETTING	EFFECT
Class	Point class(es) for which dimensions and

SETTING	EFFECT
	normal vectors are computed.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Class** field.
Turn normals	Defines a target point for the normal vector direction: • **Towards trajectory** - turned towards the trajectory position. This requires trajectory information available in TerraScan. This is the preferred method for airborne and mobile point clouds. • **Towards center point** - turned towards the center point of the point cloud. This is suitable for static terrestrial point clouds. • **Upwards** - turned to upward direction. Suited for airborne point clouds if no trajectory is available. • **Towards vectors** - turned towards a selected vector element. This is suitable for mobile point clouds of tunnels or other inside scans if no trajectory is available. It requires that a 3D line string is drawn inside along the tunnel center and selected before the tool is started.
Time difference	Maximum difference in time stamps of points that are considered for calculating the normal vector.
Process lines separately	If on, the dimension and normal vector computation is done for each line separately. This is recommended if the data of different lines do not match to each other.
Process scanners separately	If on, the dimension and normal vector computation is done for each scanner separately. This is recommended if the data of different scanners do not match to each other.

Convert geoid model

Convert geoid model command converts a source geoid model into a geoid text file that can be used by Terra application tools. Supported source geoid models are:

- **Denker** - European geoid model
- **EGM96**, **EGM2008** - global geoid models
- **GSIGEOME** - Japanese geoid model

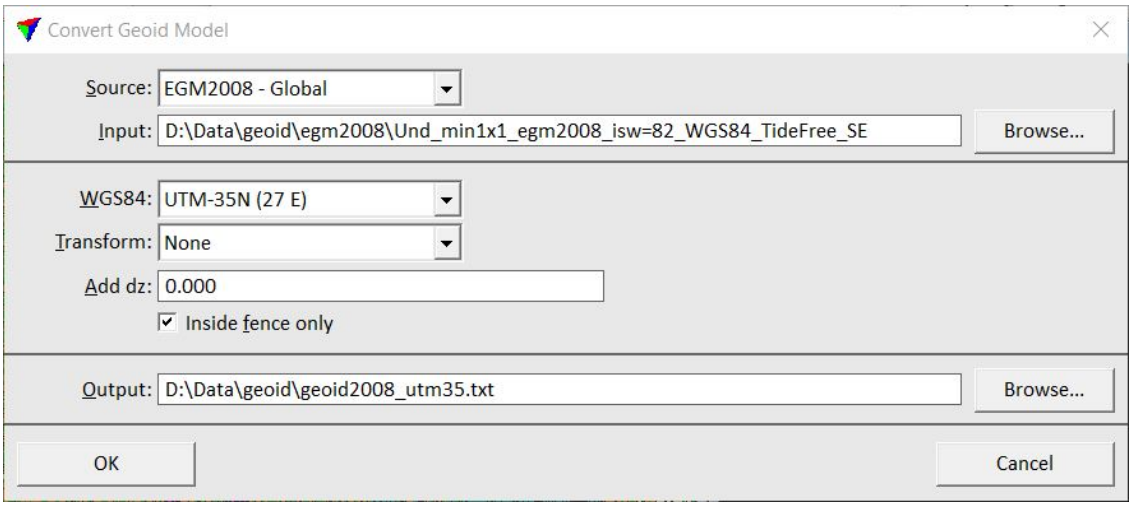
- **HBG03**, **HBG18** - Belgian geoid models
- **Norway** - Norwegian geoid model file format (such as "href2016b_nn2000_euref89.txt")
- **RAF98**, **RAF18** - French geoid models
- **SA2010** - South African geoid model

The process can include a coordinate conversion from WGS84 latitude/longitude coordinates into a projection system, another transformation as well as the addition of an elevation difference. A projection system has to be activated and a transformation has to be defined in TerraScan **Settings** before they can be used in the geoid model conversion. See **Settings** categories Coordinate transformations / Built-in projection systems, Coordinate transformations / Transformations, Coordinate transformations / US State Planes, and Coordinate transformations / User projection systems for more information.

To convert a geoid model:

1. Select **Convert geoid model** command from the **Tools** pulldown menu.

   This opens the **Convert Geoid Model** dialog:



2. Define settings and click OK.

   This converts the source geoid model into the output text file.

SETTING	EFFECT
Source	Source geoid model.
Input	Storage location and name of the source geoid model file.
WGS84	Conversion from WGS84 coordinates into a given projection system. The list contains all projection systems that are activated in Coordinate transformations / Built-in projection systems, Coordinate transformations / Transformations, Coordinate transformations / US State Planes, and Coordinate

SETTING	EFFECT
	transformations / User projection systems categories of TerraScan **Settings**.
Transform	Transformation applied to the geoid model coordinates. The list contains all transformation defined in the Coordinate transformations / Transformations category of TerraScan **Settings**.
Add dz	Defines a value that is added to the elevation difference values of the geoid model.
Inside fence only	If on, the output file is written only for geoid model points inside a fence or selected polygon. Requires a fence or selected polygon in the CAD file. This is recommended if you convert a global or country-wide geoid model for a smaller project area.
Output	Storage location and name of the output text file.

Draw bounding box

Draw bounding box command draws a bounding box around points loaded in TerraScan. The command draws a graphical group of twelve 3D line elements which enclose all laser points. The line element are drawn on the active level using the active symbology settings of the CAD file.

The bounding box may be useful, for example, if you want to use the **Fit View** tool of the CAD platform to fit rotated views to the extend of the laser data. As an alternative, the TerraScan command Fit view or the **Fit** button in the Display mode dialog can be used to fit views to the extend of loaded laser points.

You can undo the drawing of a bounding box by using the **Undo** command of the CAD platform.

Draw into profile

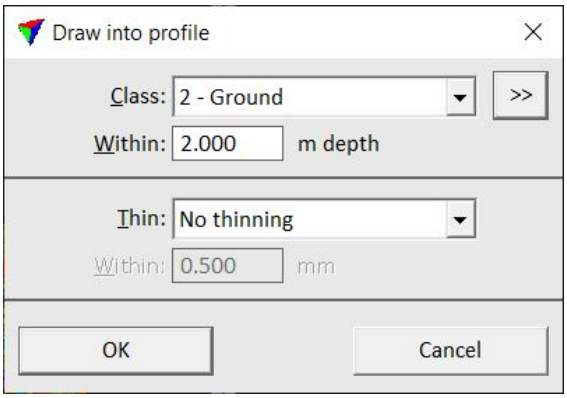
Draw into profile command draws laser points into a profile as permanent elements. This command draws all points from one or more given classes into the profile which are within the given depth from the profile alignment.

You have to create the profile using Draw Profile tool of TerraModeler before this tool can be used.

To draw laser points into a profile:

1. Select **Draw into profile** command from the **Tools** pulldown menu.

   This opens the **Draw into profile** dialog:



2. Define settings and click OK.

3. Identify the profile cell element with a data click.

   This draws the points as permanent CAD file elements into the profile.

SETTING	EFFECT
Class	Source class(es) from which points are drawn into the profile.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Class** field.
Within	Depth distance from the alignment element of the profile.
Thin	Determines how points are thinned for being drawn into the profile: • **No thinning** - points are not thinned. • **As point cloud** - appropriate thinning when drawing 3D objects such as trees or powerline towers. • **As terrain surface** - appropriate thinning when drawing a ground surface.
Within	Determines how close a point must be to the alignment in order to be drawn into the profile. This is only active if **Thin** is set to **As point cloud** or **As terrain surface**.

Draw into sections

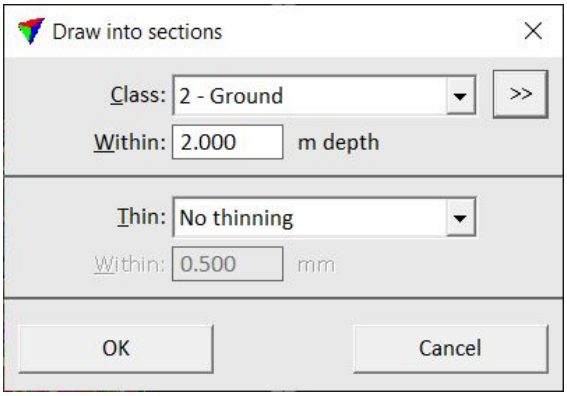
Draw into sections command draws laser points into a cross section as permanent elements. This command draws all points from one or more given classes into the cross section which are within the given depth from the cross section center line.

You have to create the cross section(s) using Draw Alignment Sections tool of TerraModeler before this tool can be used.

To draw laser points into cross sections:

1. Select **Draw into sections** command from the **Tools** pulldown menu.

   This opens the **Draw into sections** dialog:



2. Define settings and click OK.

3. Identify the cross section cell element with a data click.

   This draws the points as permanent CAD file elements into the cross section.

SETTING	EFFECT
Class	Source class(es) from which points are drawn into the cross section.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Class** field.
Within	Depth distance from the center line of the cross section.
Thin	Determines how points are thinned for being drawn into the cross section: • **No thinning** - points are not thinned. • **As point cloud** - appropriate thinning when drawing 3D objects such as trees or powerline towers. • **As terrain surface** - appropriate thinning when drawing a ground surface.
Within	Determines how close a point must be to the alignment in order to be drawn into the cross section. This is only active if **Thin** is set to **As point cloud** or **As terrain surface**.

Draw polygons

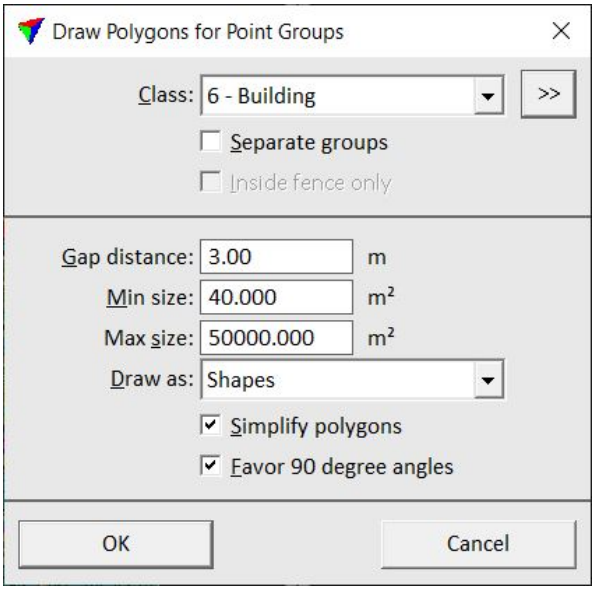
Draw polygons command draws 3D shapes or line strings around groups of points within one or more point classes. The elevation of the vertices for the elements is derived from the elevation values of the laser points.

Optionally, grouping of points can be taken into account for drawing polygons. The requires the assignment of group numbers to points by using the Assign groups menu command or macro action.

To draw polygons around groups of points:

1. Select **Draw polygons** command from the **Tools** pulldown menu.

   This opens the **Draw Polygons for Point Groups** dialog:



2. Define settings and click OK.

   This draws the shape or line string elements around point groups into the CAD file. The elements are drawn on the active level using the active symbology of the CAD file.

SETTING	EFFECT
Class	Source class(es) of laser points. Shapes or line strings are drawn around point groups of the given class(es).
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Class** field.
Separate groups	If on, a separate polygon is drawn around points belonging to one group.

SETTING	EFFECT
Inside fence only	If on, only the area inside a fence or selected polygon is considered for processing. Requires that a fence is drawn or a polygon is selected in the CAD file.
Gap distance	Maximum gap between points. Points which are less than the given distance away from each other are considered to belong to the same group.
Min size	Minimum size of a polygon to be drawn.
Max size	Maximum size of a polygon to be drawn.
Draw as	Definition of the element type that is created: **Shapes** or **Line strings**.
Simplify polygons	If on, unnecessary vertices of the polygons are removed.
Favor 90 degree angles	If on, 90 degree angles are enforced for close-to 90 degree angles.

Extract color from images

Extract color from images command extracts color values from raster images and assigns the values to laser points. The color sources can be orthophotos attached as TerraPhoto raster references or raw images in an active TerraPhoto image list. In addition, a color point file can be used to balance colors of the raw images before the color values are assigned to the laser points. The command requires TerraPhoto or TerraPhoto Lite running on the same computer.

TerraScan can extract up to 10 color channels for each point. The maximum amount of color channels can only be stored in the TerraScan FastBinary format. LAS 1.4 format and later can store up to 4 color channels, LAS 1.2 format and later and TerraScan Binary files up to 3.

The color for a laser point is derived by resampling the color values of all the pixels inside a circular area around the point. There are different methods of color value extraction from raw images which are either suitable for airborne or mobile data sets.

The process can involve the computation and storage of image numbers. The number of the image used for extracting the color can be stored for each laser point. This requires the storage of points in TerraScan FastBinary format. The image number stored as laser point attribute is required for advanced coloring methods for mobile point clouds.

Extract color from images command is also available for TerraScan projects and thus, can be performed for all or selected blocks of a project.

To extract color from attached orthophotos or raw images:

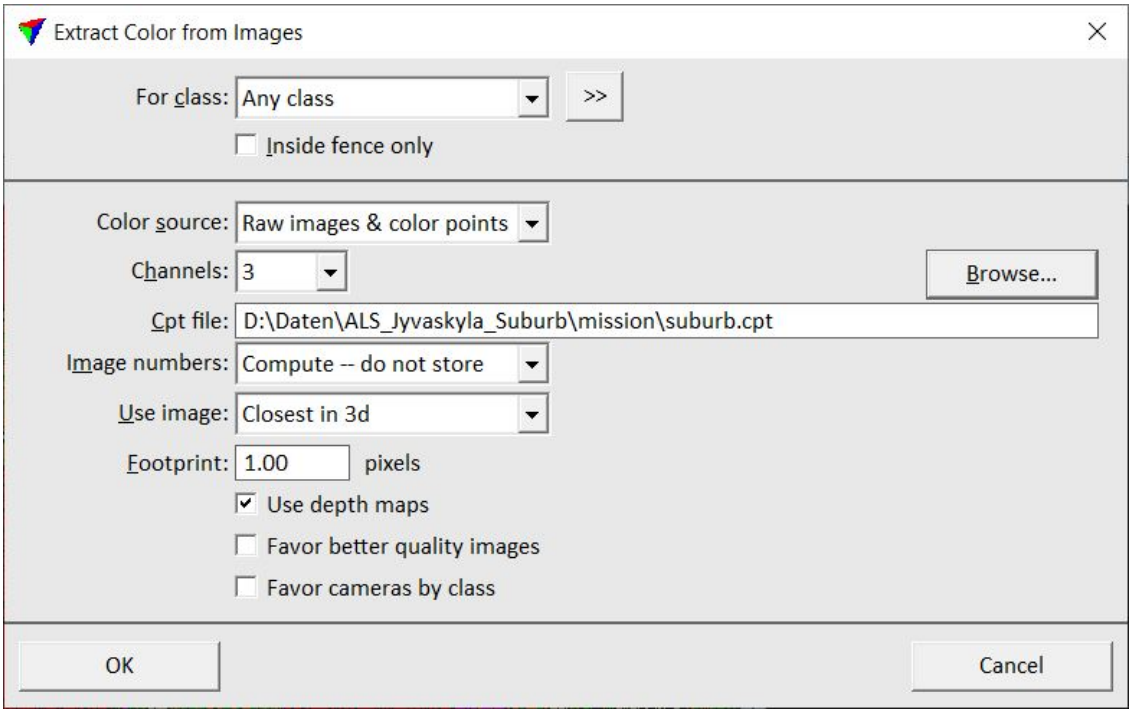
1. Attach reference images in TerraPhoto's Manage Raster References tool.

OR

1. Load an image list into TerraPhoto.

2. Select **Extract color from images** command from the **Tools** pulldown menu.

   This opens the **Extract Color from Images** dialog:



3. Define settings and click OK.

   This derives color values for the laser points from the defined source images.

SETTING	EFFECT
For class	Laser point class(es) for which colors are extracted.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **For class** field.
Inside fence only	If on, color values are extracted for laser points inside a fence or selected polygon.
Color source	Source files for color extraction: • **Ortho images** - colors are extracted from attached TerraPhoto raster references. • **Raw images** - colors are extracted from raw images in an active image list in TerraPhoto. • **Raw images & color points** - colors are extracted from raw images and from a color point file.

SETTING	EFFECT
Channels	Amount of color channels to be extracted.
Cpt file	Location and name of a color point file. This is only active if **Color source** is set to **Raw images & color points**.
Image numbers	Method how the software handles the computation and storage of image numbers from raw images: • **Compute -- do not store** - image numbers are computed but not stored for laser points. • **Compute and store** - image numbers are computed and each laser point stores the number of the image from which it gets the color. • **Use stored** - stored image numbers of laser points are used for color extraction.
Use image	Method how the software determines the raw image for extracting a color for a laser point: • **Closest in 3d** - closest camera XYZ position. Optimized for airborne data sets. • **Closest in xy** - closest camera XY position. Optimized for airborne data sets. • **Closest in time** - closest time stamp. Optimized for airborne data sets. • **Mobile -- closest in time** - closest time stamp. Optimized for mobile data sets. • **Mobile -- ground surface** - best ground surface visibility. Optimized for mobile data sets. • **Mobile -- closest in 3d** - closest camera XYZ position. Optimized for mobile data sets.
Footprint	Radius of a circular area around each laser point within which pixel color values are resampled. Given in meters for a method optimized for airborne data sets and in pixels for mobile data sets. This is the only setting that is active if **Color source** is set to **Ortho images**.
Max distance	Maximum distance between a raw image and a laser point. Images outside that distance are not considered for color extraction. This is only active if **Use image** is set to any method optimized for mobile data sets.
Max time diff	Maximum time difference between a raw image and a laser point. Images outside that difference are not considered for color

SETTING	EFFECT
	extraction. This is only active if **Use image** is set to **Mobile -- closest in time**.
Use depth maps	If on, depth maps files are included in the color extraction process. See **TerraPhoto User Guide** for more information about depth maps.
Favor better quality images	If on, the quality attribute stored for raw images in an image list is considered in the color extraction process.
Favor cameras by class	If on, the settings in the TerraPhoto mission file related to favoring cameras for coloring points are considered in the color extraction process.

Extract echo properties

Extract echo properties command extracts information from waveform data and assigns it as attributes to the laser points. The command requires that waveform data and a scanner waveform profile are available. The processing steps for preparing the extraction of waveform-related information are described in detail in Chapter Waveform Processing.

The command can extract the following attributes:

- **Echo length** - relative length (millimeter) of a return signal compared to a typical return from a hard surface.
- **Echo normality** - difference in shape of a return signal compared to a typical return from a hard surface.
- **Echo position** - difference in position of a peak of a return signal compared to a typical return from a hard surface.

The echo length can be used for the visualization of points and for classifying points. For instance, a classification By echo length prior to ground classification can improve the result of the Ground routine especially in areas of low vegetation.

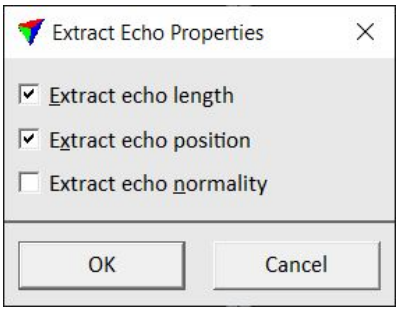
The echo properties can be stored in TerraScan FastBinary files.

Extract echo properties command is also available for TerraScan projects and thus, can be performed for all or selected blocks of a project.

To extract echo properties:

1. Select **Extract echo properties** command from the **Tools** pulldown menu.

   This opens the **Extract Echo Properties** dialog:



2. Select what properties you want to extract by switching the corresponding options on.

3. Click OK.

   This starts the extraction process. It assigns the extracted attributes to all laser points for which waveform information is available. Depending on the amount of points, the process may take some time. An information dialog shows the number of effected points.

Fit to reference

Fit to reference command applies a systematic correction to a data set. The correction may include translation in XYZ direction and rotation around XYZ axes. Alternatively, a rubbersheet correction (correction that changes over time) can be computed to fit data in elevation.

The correction is computed based on reference points loaded into TerraScan. See Read reference points command for more information. The correction is applied to the active points loaded in TerraScan.

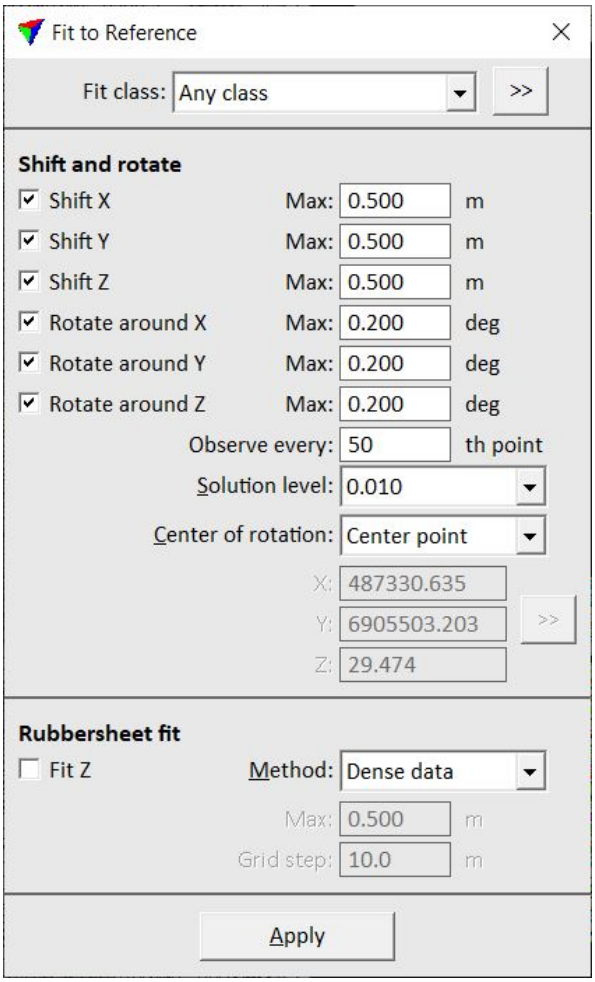
The command is primarily developed for change detection of volumes in mines based on photogrammetric point clouds. The detection and computation of volume changes requires that the point clouds of, for example, two days match where there was no change. Thus, the command can be used to fit two photogrammetric point clouds in preparation of volume computation. While a systematic shift can be done with all (ground) points of the point cloud, it is recommended to exclude areas that have changed for the rubbersheet fit. This can be done by drawing polygons around such areas and classify points inside the polygons into a separate class.

To fit points to reference points:

1. Load points and reference points into TerraScan.

2. Select **Fit to reference** command from the **Tools** pulldown menu.

   This opens the **Fit to Reference** dialog:



3. Define settings and click OK.

   This fits the active points to the reference points.

SETTING	EFFECT
Fit class	Class(es) used for computing the fit.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Fit class** field.
Shift X	If on, a translation in X direction is computed and applied up to the given **Max**imum value.
Shift Y	If on, a translation in Y direction is computed and applied up to the given **Max**imum value.
Shift Z	If on, a translation in Z direction is computed and applied up to the given **Max**imum value.
Rotate around X	If on, a rotation around the X axis is computed and applied up to the given **Max**imum value.

SETTING	EFFECT
Rotate around Y	If on, a rotation around the Y axis is computed and applied up to the given **Max**imum value.
Rotate around Z	If on, a rotation around the Z axis is computed and applied up to the given **Max**imum value.
Observe every	Determines how many points are used for computing the translation and rotation correction.
Solution level	Level of accuracy for the correction values. The selection list provides some accuracy levels depending on the CAD file master and secondary units.
Center of rotation	Defines the center point for rotating the point cloud: • **Center point** - center point of the active point cloud. • **User point** - user-defined point.
X | Y | Z	XYZ coordinate values of the center of rotation. Define the point by typing values in the text fields or by using the **>>** button next to the coordinate fields. This is only active if **Center of rotation** is set to **User point**.
>>	Lets you define the point of rotation with a data click inside a CAD file view. You may use the snapping functionality of the CAD platform to define the point exactly based on an element drawn in the CAD file. This is only active if **Center of rotation** is set to **User point**.
Fit Z	If on, a rubbersheet correction in Z direction is computed.
Method	Method of computing rubbersheet correction values: • **Dense data** - grid-based suited for dense point clouds. • **Sparse data** - based on point-to-point spacing suited for sparse point clouds.
Max	Maximum rubbersheet correction value applied to the active point cloud.
Grid step	Defines the grid step size for computing correction values. This is only active if **Method** is set to **Dense data**.
Spacing	Defines the maximum space between correction values. This is only active it **Method**

SETTING	EFFECT
	is set to **Sparse data**.

Macro

Not Lite

Macro command opens the **Macro** window which lets you create a macro for automated batch processing. The creation and use of macros in TerraScan is described in detail in Chapter Macros.

Output control report

Output control report command creates a report of elevation differences between laser points and ground control points or a surface model. This can be used, for example, to check the elevation accuracy of a laser data set and to calculate a correction value for improving the elevation accuracy of the laser points.

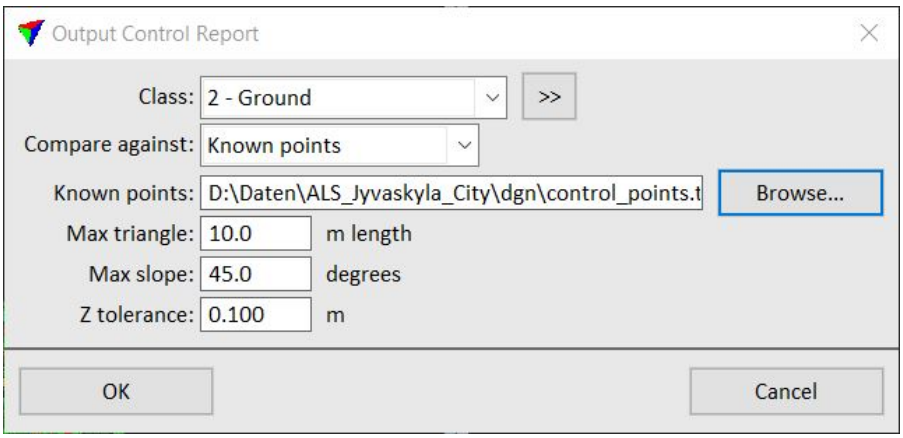
The ground control points have to be stored in a space-delimited text file in which each row has three or four fields: (optionally) identifier, easting, northing and elevation. The identifier field is usually a number but it may include non-numeric characters as well.

The surface model has to be an active model in TerraModeler. The output report shows statistical information about the elevation difference between points and surface model, such as points used for computing the values, average difference and magnitude, standard deviation, and RMS value. It does not show elevation difference values for single point locations.

To create a control report:

1. Select **Output control report** command from the **Tools** pulldown menu.

   This opens the **Output Control Report** dialog:



2. Define settings and click OK.

This calculates the elevation differences and opens the **Control report** window. The content of the report depends on the source file used for the comparison, ground control points in a text file or a surface model. The results and the further usage of a comparison with ground control points is described in detail in Section Systematic elevation correction.

SETTING	EFFECT
Class	Point class(es) used for the elevation value comparison.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Class** field.
Compare against	Source file for elevation value comparison: • **Known points** - text file that contains coordinates of ground control points. • <**surface model**> - name of a specific surface model active in TerraModeler.
Known points	Location and name of the file that stores the coordinates of the ground control points. This is only active if **Compare against** is set to Known **points**.
Max triangle	Maximum length of a triangle edge. The software creates a triangle from the closest 3 laser points around a ground control point location. If the triangle edge length exceeds the given value, the control point is not used for the report.
Max slope	Maximum terrain slope for which an elevation difference is computed.
Z tolerance	Normal elevation variation (noise level) of laser points. This value is used only when computing the terrain slope so that small triangles do not exceed the **Max slope** value.

Read / Building models

Not Spatix

Read / Building models command reads text files that have been created in an automatic building vectorization process. It is used to draw the buildings as 3D vector models into the CAD file. See Vectorize buildings macro action for more information about the creation of building text files.

The building models are drawn as MicroStation cell elements into the CAD file. The settings in Building vectorization / Levels and Building vectorization / Model categories of TerraScan **Settings** determine level, color, and layout definitions of the models.

After drawing the building models into the CAD file, they can be checked and modified using dedicated tools of TerraScan. They are described in detail in Chapter 3D Building Models.

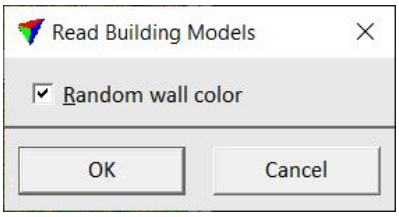
To read building models into the CAD file:

1. Select **Read / Building models** command from the **Tools** pulldown menu.

   This opens the **Read building models** dialog, a standard dialog for opening files.

2. Select building text files and click **Open**.

   This opens the **Read Building Models** dialog:



3. Select a wall coloring option and click OK.

   This reads the text files and draws the building models into the CAD file.

SETTING	EFFECT
Random wall color	If on, the walls are drawn by using colors chosen randomly from the active color table of the CAD file. If off, the color defined in Building vectorization / Model category of TerraScan **Settings** is used for all walls.

You can undo the action by using the **Undo** command of the CAD platform.

At the moment, cells and thus, the **Read / Building models** command only works in MicroStation. There is not yet any corresponding element type in Spatix.

Read / Paint lines

Read / Paint lines command reads text files that have been created in an automatic vectorization process for linear paint markings on a road surface. It is used to draw the paint markings as 3D line string elements into the CAD file. See Find paint lines macro action for more information about the creation of paint line text files.

The line string elements are drawn on the active level using the active symbology settings of the CAD file.

After drawing the paint lines into the CAD file, they should be checked with the help of, for example, the Inspect Elements tool of TerraScan or the **Validate linear elements** tool of TerraModeler.

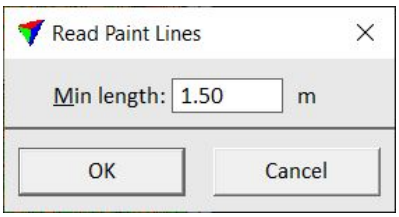
To read lines for paint markings into the CAD file:

1. Select **Read / Paint lines** command from the **Tools** pulldown menu.

   This opens the **Paint line files** dialog, a standard dialog for opening files.

2. Select paint line text files and click **Open**.

   This opens the **Read Paint Lines** dialog:



3. Define a minimum length value and click OK.

   This reads the text files and draws the paint lines into the CAD file.

SETTING	EFFECT
Min length	Minimum length of a paint line that is drawn into the CAD file. Any shorter line elements are ignored.

You can undo the action by using the **Undo** command of the CAD platform.

Read / Poles

Not Spatix

Read / Poles command reads text files that have been created in an automatic process for extracting pole objects. See Find poles macro action for more information about automatic object extraction in batch mode.

The poles are drawn as cell elements into the CAD file. The cells must be defined in a MicroStation cell library and the cell library must be attached to the CAD file. The cells are drawn on the active level using the active color of the CAD file.

To read poles into the CAD file:

1. Select **Read / Poles** command from the **Tools** pulldown menu.

   This opens the **Read poles** dialog, a standard dialog for opening files.

2. Select pole text files and click **Open**.

   This reads the text files and draws the cells into the CAD file.

You can undo the action by using the **Undo** command of the CAD platform.

At the moment, cells and thus, the **Read / Poles** command only works in MicroStation. There is not yet any corresponding element type in Spatix.

Read / Polygons

Read / Tree cells command reads text files that have been created in an automatic process for creating polygons. See Draw polygons macro action for more information about creating polygons in batch mode.

The polygons are drawn as 3D shape elements into the CAD file. The shape elements are drawn on the active level using the active color, line width, and line style settings of the CAD file.

To read tree cells into the CAD file:

1. Select **Read / Polygons** command from the **Tools** pulldown menu.

   This opens the **Polygon files** dialog, a standard dialog for opening files.

2. Select polygon text files and click **Open**.

   This reads the text files and draws the polygons into the CAD file.

You can undo the action by using the **Undo** command of the CAD platform.

Read / Section parameters

Read / Section parameters command reads text files that have been created in an automatic process for extracting road section parameters. It is used to draw the section parameter values into the CAD file. See Compute section parameters macro action for more information about section parameters and the creation of section parameter text files.

The section parameter values are drawn as text and linear elements into the CAD file. The settings in Road section parameters category of TerraScan **Settings** determine level, color, text size and unit definitions of the parameters.

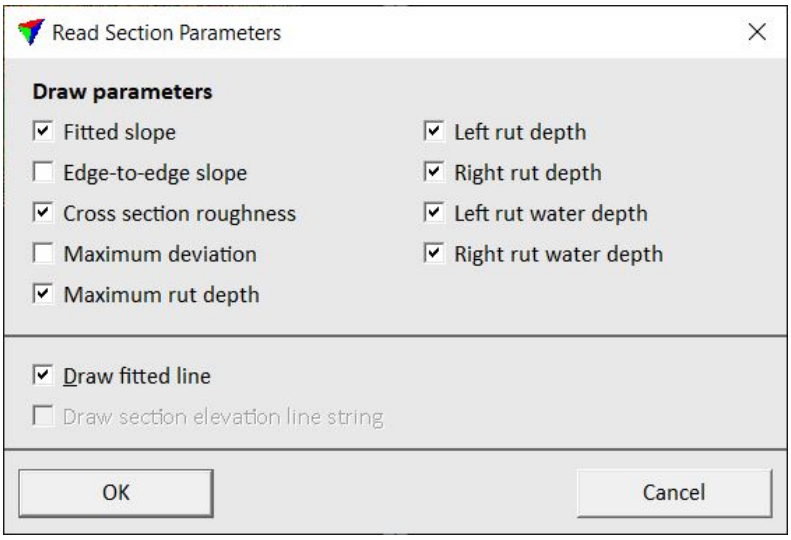
To read section parameters into the CAD file:

1. Select **Read / Section parameters** command from the **Tools** pulldown menu.

   This opens the **Section parameter files** dialog, a standard dialog for opening files.

2. Select section parameters text files and click **Open**.

   This opens the **Read Section Parameters** dialog:



3. Define parameters that you want to draw and click OK.

   This reads the text files and draws the section parameter values into the CAD file.

SETTING	EFFECT
Fitted slope	If on, the fitted slope value of a section is drawn as text element.
Edge to edge slope	If on, the edge-to-edge slope value of a section is drawn as text element.
Cross section roughness	If on, the roughness value of a section is drawn as text element.
Maximum deviation	If on, the maximum deviation value of a section is drawn as text element.
Maximum rut depth	If on, the maximum depth value of ruts at a section location is drawn as text element.
Left rut depth	If on, the depth value of the left rut at a section location is drawn as text element.
Right rut depth	If on, the depth value of the right rut at a section location is drawn as text element.
Left water depth	If on, the water depth value of the left rut at a section location is drawn as text element.
Right water depth	If on, the water depth value of the right rut at a section location is drawn as text element.
Draw fitted line	If on, the line of the fitted slope of a section is drawn as line element.
Draw section elevation line string	If on, the line following the elevation variation of a section is drawn as line string element.

You can undo the action by using the **Undo** command of the CAD platform.

Read / Slope arrows

Read / Slope arrows command reads text files that have been created in an automatic process for extracting the superelevation of road lanes. It is used to draw arrows and labels into the CAD file. The arrows point into the direction of the slope and the labels show the gradient of the slope. See Compute slope arrows macro action for more information about slope arrows and the creation of slope arrow text files.

The slope arrows are drawn as 3D line string elements and the gradient values as text elements into the CAD file. The elements are drawn on the active level using the active line width, line style, and text size settings of the CAD file. The color is determined by settings in the **Read slope arrows** dialog.

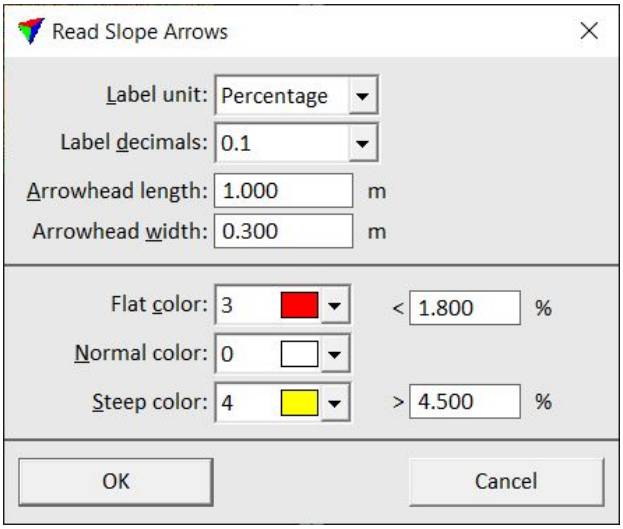
To read slope arrows into the CAD file:

1. Select **Read / Slope arrows** command from the **Tools** pulldown menu.

   This opens the **Slope arrow files** dialog, a standard dialog for opening files.

2. Select slope arrow text files and click **Open**.

   This opens the **Read Slope Arrows** dialog:



3. Define settings and click OK.

   This reads the text files and draws the slope arrows and values into the CAD file.

SETTING	EFFECT
Label unit	Unit for expressing the slope gradient: **Degree** or **Percentage**.
Label decimals	Number of decimals used in slope gradient text elements.
Arrowhead length	Length of the arrowhead in the arrow drawing. Given in CAD file units.

SETTING	EFFECT
Arrowhead width	Width of the arrowhead in the arrow drawing. Given in CAD file units.
Flat color	Color of slope arrows and labels if the gradient is smaller than or equal to the given value.
Normal color	Color of slope arrows and labels if the gradient value is larger than the value defined for **Flat color** and smaller than or equal to the value defined for **Steep color**.
Steep color	Color of slope arrows and labels if the gradient is larger than the given value.

You can undo the action by using the **Undo** command of the CAD platform.

Read / Tree cells

Not Spatix

Read / Tree cells command reads text files that have been created by the Output Tree Cells tool. It draws the tree cells on the active level into the CAD file. The cells must be defined in a cell library attached to the CAD file. See Create Tree Cells for more information about tree cell definition and an example cell library.

To read tree cells into the CAD file:

1. Select **Read / Tree cells** command from the **Tools** pulldown menu.

   This opens the **Read tree cells** dialog, a standard dialog for opening files.

2. Select tree cells text files and click **Open**.

   This reads the text files and draws the tree cells into the CAD file.

You can undo the action by using the **Undo** command of the CAD platform.

At the moment, cells and thus, the **Read / Tree cells** command only works in MicroStation. There is not yet any corresponding element type in Spatix.

Read / Wires

Read / Wires command reads text files that have been created in an automatic process for extracting overhead wires. See Find wires macro action for more information about automatic wire extraction in batch mode.

The wires are drawn as 3D line string elements into the CAD file. The line string elements are drawn on the active level using the active color, line width, and line style settings of the CAD file.

To read wires into the CAD file:

1. Select **Read / Wires** command from the **Tools** pulldown menu.

   This opens the **Read wires** dialog, a standard dialog for opening files.

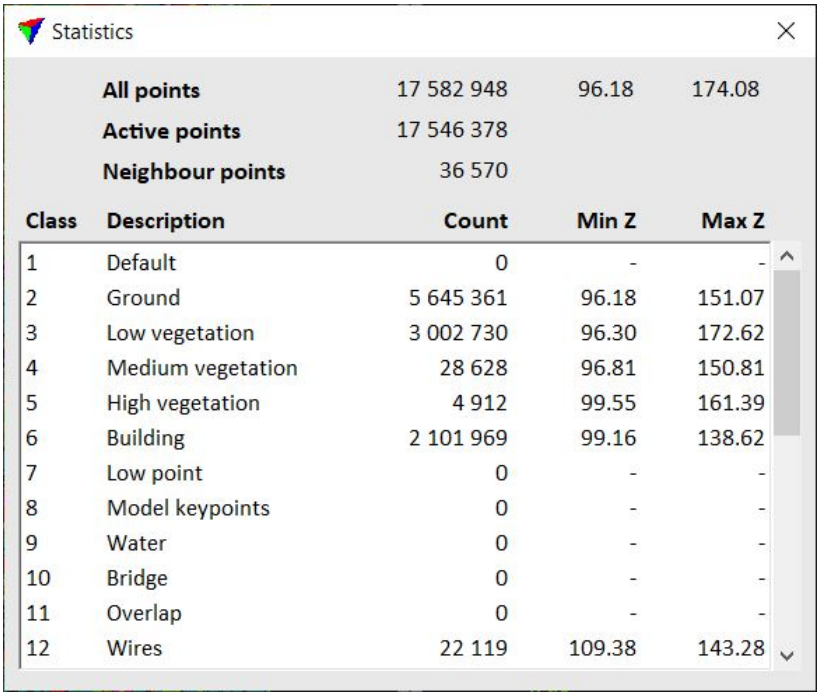
2. Select wire text files and click **Open**.

   This reads the text files and draws the wires into the CAD file.

You can undo the action by using the **Undo** command of the CAD platform.

Show statistics

Show statistics command displays basic statistics information about laser points. In the upper part of the **Statistics** window, the amount of all points, active points and neighbour points are listed, as well as the elevation range for all points. In the lower part, the separate classes are listed with the point count, minimum elevation and maximum elevation values for each class.



Statistics

	All points	17 582 948	96.18	174.08
	Active points	17 546 378		
	Neighbour points	36 570		
Class	**Description**	**Count**	**Min Z**	**Max Z**
1	Default	0	-	-
2	Ground	5 645 361	96.18	151.07
3	Low vegetation	3 002 730	96.30	172.62
4	Medium vegetation	28 628	96.81	150.81
5	High vegetation	4 912	99.55	161.39
6	Building	2 101 969	99.16	138.62
7	Low point	0	-	-
8	Model keypoints	0	-	-
9	Water	0	-	-
10	Bridge	0	-	-
11	Overlap	0	-	-
12	Wires	22 119	109.38	143.28

Smoothen points

Smoothen points command can be used to smooth attributes of laser points. The points are modified according to their closest neighbours. This results in a more homogeneous appearance of the point cloud. The following methods can be applied:

- **Xyz** - 3D smoothing process, points on vertical surfaces are smoothed in XY direction, point on horizontal surfaces in Z direction.
- **Elevation** - adjusts elevation values of laser points iteratively. A best fit plane equation is derived for this group of points and the elevation of the center point is adjusted to better match the plane

equation. Then, the application tries to decide what areas became smooth and what areas did not result in a smooth surface. Only the smooth areas are finally adjusted. Areas which still have significant elevation variation are restored to the original state.

- **Intensity** - averages intensity values of each point with its closest neighbours.
- **Color** - averages color values of each point with its closest neighbours.
- **Distance** - averages distances value of each point with a given amount of neighbour points.

You would normally run elevation smoothing on ground point class in order to:

- remove random variation in laser point elevations and produce a more accurate model.
- produce a smoother surface so that contours look nicer.
- produce a smoother surface so that profile drawings look nicer.

You should not use elevation smoothing if:

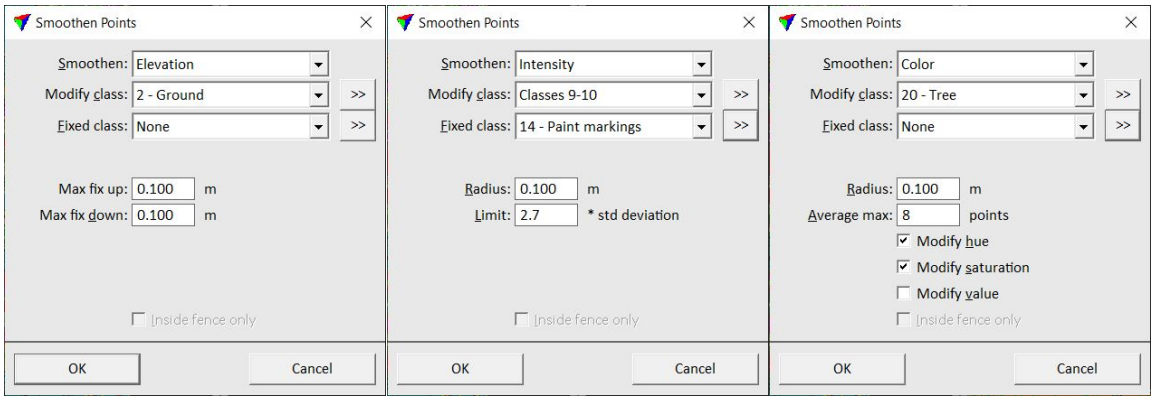
- the terrain is covered with low vegetation and there are no smooth surfaces.
- you intend to extract features which have only a small elevation change such as road curb stone lines. Those would be completely smoothed out.

You can include a point class into the smoothing process which is used in the smoothing process but is not modified. You can use this capability, for example, if you have classified points close to breakline features into a separate class (elevation smoothing) or if points on paint markings are classified into a separate class (intensity smoothing for points on road surfaces).

To smoothen points:

1. Select **Smoothen points** command from the **Tools** pulldown menu.

   This opens the **Smoothen Points** dialog:



2. Define values and click OK.

   This starts smoothing process.

SETTING	EFFECT
Smoothen	Smoothing method: **Xyz**, **Elevation**, **Intensity**, or **Color** values.
Modify class	Point class(es) included in and modified by the smoothing process.

SETTING	EFFECT
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Modify class** field.
Fixed class	Point class(es) included in the smoothing process but points in these classes are not modified.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Fixed class** field.
Max fix | up | down	Maximum horizontal/vertical up/down change to apply to a point. This is only active if **Smoothen** is set to **Xyz** or **Elevation**.
Radius	Radial distance from a point which defines the circular sample area from which the average intensity or distance values is derived. This is only active if **Smoothen** is set to **Intensity** or **Distance**.
Limit	Defines the maximum intensity value of points that are effected by the smoothing process. Given as factor of the standard deviation of intensity values in the sample area defined by the **Radius**. If a point differs more in intensity from its neighbours than the computed limit, it is not effected by the smoothing process. This is only active if **Smoothen** is set to **Intensity**.
Average max	Maximum amount of points that are included in the average distance computation. This is only active if **Smoothen** is set to **Distance**.
Modify hue	If on, the hue component of the HSV value of a point is modified. This is only active if **Smoothen** is set to **Color**.
Modify saturation	If on, the saturation component of the HSV value of a point is modified. This is only active if **Smoothen** is set to **Color**.
Modify value	If on, the value component of the HSV value of a point is modified. This is only active if **Smoothen** is set to **Color**.

SETTING	EFFECT
Inside fence only	If on, only points inside a fence or selected polygon are effected by the process. Requires a fence or a selected polygon in the CAD file.

Sort

Sort command sorts loaded laser points according to the selected attribute. The sub-menu includes the following options for sorting points:

- **By time stamp** - points are sorted by increasing time stamps. This is the recommmended order for processes that rely on trajectory positions, such as cut overlap, classify by range, and most of the TerraMatch processes.
- **By line and time** - points are sorted by increasing line numbers and time stamps.
- **By line, scanner and time** - points are sorted by increasing line numbers, scanner numbers and time stamps.
- **By xy location** - points are sorted geographically by increasing xy location. This is the recommended order for processes that rely on geometrical conditions between points in the point cloud. Especially for photogrammetric point clouds, this order is strongly recommended in order to speed up many automatic processes.
- **By increasing X** - points are sorted by increasing easting coordinate values.
- **By decreasing X** - points are sorted by decreasing easting coordinate values.
- **By increasing Y** - points are sorted by increasing northing coordinate values.
- **By decreasing Y** - points are sorted by decreasing northing coordinate values.
- **By increasing Z** - points are sorted by increasing elevation coordinate values.
- **By decreasing Z** - points are sorted by decreasing elevation coordinate values.

To sort laser points:

1. Select an option from the **Sort** sub-menu from the **Tools** pulldown menu.

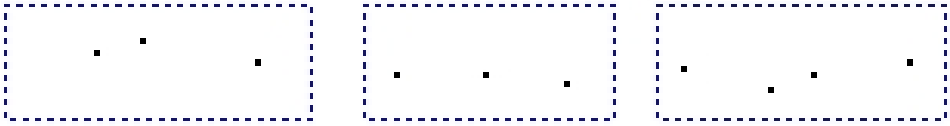
   This sort the points according to the selected attribute.

Thin points

Thin points command reduces unnecessary point density by removing some of the points which are close to each other or within a grid cell of a given size.

The thinning method which relies on point density tries to find groups of points where all the points are within the given horizontal distance and elevation difference from a central point in the group. Alternatively, the thinning can be done based on a 2D or 3D grid. Then, one point per grid cell is kept. This may be useful to thin a point cloud to a specific density, for example, to 1 point per square meter in selected areas. The 2D grid method is optimized for airborne data. The 3D grid method is developed for mobile data but not yet optimized for achieving a homogeneous point density.

Another setting of the command determines which point in each group or grid cell is kept. The **Keep** methods for groups that rely on the point cloud geometry are illustrated in the following figures:



Groups of points

Highest point in each group

Lowest point in each group

Central point in each group

Created average for each group

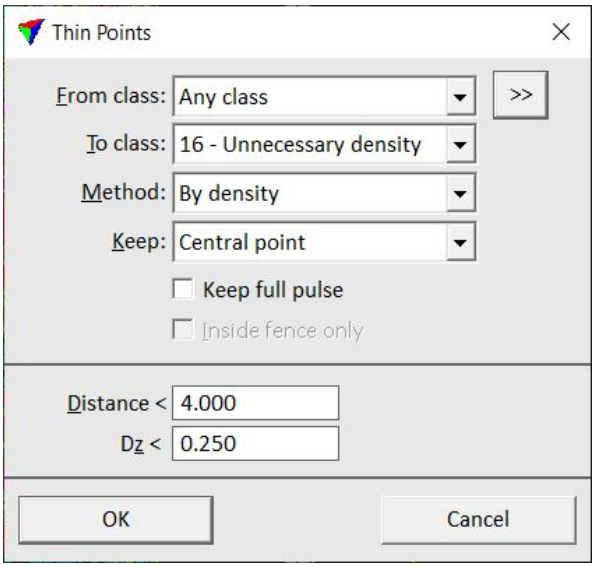
Additional options are to keep points with the highest or lowest intensity value, points with the earliest or latest time stamp, or points from a given class.

The removed points can be either deleted or classified into another point class.

To thin points:

1. Select **Thin points** command from the **Tools** pulldown menu.

   This opens the **Thin Points** dialog:



2. Define settings and click OK.

   This thins the point cloud.

SETTING	EFFECT
From class	Point class(es) from which to thin out unnecessary points.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **From class** field.
To class	Class into which removed points are classified. Alternatively, you can select the **Delete** option in order to delete the points completely from the point cloud.
Method	Thinning method: • **By density** - thinning based on distance and elevation difference between points. • **2D Grid** - thinning based on a 2D grid. One point per grid cell is kept. • **3D Grid** - thinning based on a 3D grid. One point per grid cell is kept. Developed for mobile point clouds. • **By order** - thinning based on point order. The common order of LiDAR point clouds is by time stamp, but any other order, for instance created by the Sort command, is possible.
Keep	Defines which point to keep in each group of points: • **Highest point** - point with highest elevation.

SETTING	EFFECT
	• **Median Z point** - point with median elevation value. • **Lowest point** - point with lowest elevation. • **Central point** - point closest to the center of the group. If **Method** is set to **2D Grid**, the point closest to the central XYZ location of points in a grid cell is kept. • **Create average** - substitute group by creating an average point. • **Highest intensity** - point with highest intensity value. • **Lowest intensity** - point with lowest intensity value. • **First in time** - point with earliest time stamp. • **Last in time** - point with latest time stamp. • **First echo** - point with the lowest return number. • **Last echo** - point with the highest return number. • **Random** - random selection of a point to keep. • points of a specific point class.
Keep full pulse	If on, all points of the same pulse are kept if the software decides to keep one point of a pulse.
Inside fence only	If on, only points inside a fence or selected polygon are effected by the process. Requires a fence or a selected polygon in the CAD file.
Distance	Horizontal distance limit between two points. This is only active if **Method** is set to **By density**.
Dz	Elevation difference limit between two points. This is only active if **Method** is set to **By density**.
Grid step	Size of a grid cell. This is only active if **Method** is set to **2D Grid** or **3D Grid**.

Toolboxes

The commands from the **Toolboxes** submenu open the TerraScan toolbars. The toolbars are part of the TerraScan toolbox. Alternatively, a toolbar can be opened by selecting **As Toolbar** command from the toolbox pop-up which is displayed after keeping the data button pressed for 1-2 seconds on a tool icon in the **TerraScan** toolbox.

Transform known points

Transform known points command transforms coordinate values of points in a text file. This can be used, for example, to transform known ground control points from one projection system into another.

A projection system has to be activated and a transformation has to be defined in TerraScan **Settings** before they can be used for transforming known points. See **Settings** categories Coordinate transformations / Built-in projection systems, Coordinate transformations / Transformations, Coordinate transformations / US State Planes, and Coordinate transformations / User projection systems for more information.

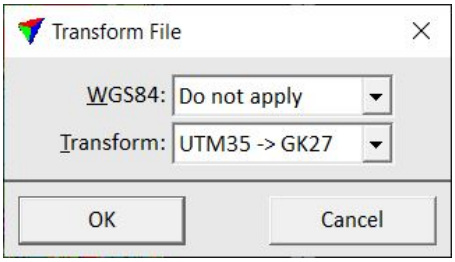
To transform a known points file:

1. Select **Transform known points** command from the **Tools** pulldown menu.

   This opens the **Source point file** dialog, a standard dialog for selecting a file.

2. Select the file that stores the known points' coordinates and click **Open**.

   This opens the **Transform File** dialog:



3. Define transformation settings and click OK.

   This opens the **Save transformed points** dialog, a standard dialog for saving a file.

4. Define a storage location and name for the output file and click **Save**.

   This saves the transformed known points into a new text file.

SETTING	EFFECT
WGS84	Conversion from WGS84 coordinates into a given projection system. The list contains all projection systems that are activated in Coordinate transformations / Built-in projection systems, Coordinate transformations / US State Planes, and Coordinate transformations / User projection systems categories of TerraScan **Settings**.
Transform	Transformation applied to the known points' coordinates. The list contains all transformation defined in the Coordinate transformations /

SETTING	EFFECT
	Transformations category of TerraScan **Settings**.

Transform loaded points

Transform loaded points command applies a transformation to points loaded in TerraScan. The transformation can be applied to all points or to points of selected class(es), selected lines, and/or inside a fence or selected polygon.

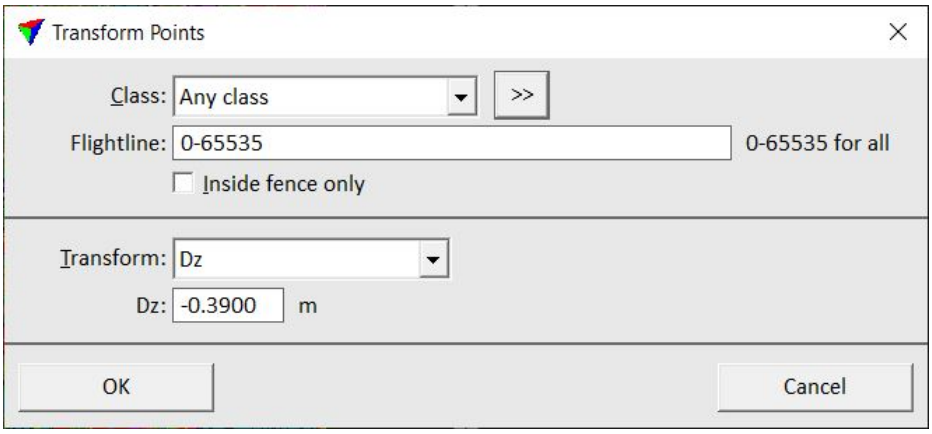
The transformation can be defined in different ways:

- **Dz** - fixed elevation difference value in order to apply a systematic elevation shift.
- **Dxyz** - text file storing difference values for easting, northing, and elevation. The format of the text file is X Y dX dY dZ.
- **Height from ground** - elevation of each point is replaced by its height above ground. Ground is defined by points in one or more point classes.
- **Height from TIN** - elevation of each point is replaced by its height above a TIN model. The TIN model is defined by an active surface model in TerraModeler.
- **Add TIN** - elevation values of a TIN are added to the original elevation values of loaded points. The corresponds to an Adjust to geoid action which uses a TIN as geoid model. The TIN is defined by an active surface model in TerraModeler.
- **Scale dZ from TIN** - scaled elevation difference values between the loaded points and a TIN are added to the original elevation values of loaded points. The TIN is defined by an active surface model in TerraModeler.
- A specific transformation defined in Coordinate transformations / Transformations category of TerraScan **Settings**.

To transform loaded points:

1. Select **Transform loaded points** command from the **Tools** pulldown menu.

   This opens the **Transform Points** dialog:



2. Define settings and click OK.

This modifies the coordinates of loaded laser points according to the selected transformation.

SETTING	EFFECT
Class	Point class(es) for which the transformation is applied.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Class** field.
Line	Line number(s) for which the transformation is applied. Use a comma or minus to separate several line numbers, for example 2-5,10.
Transform	Type of transformation.
Dz	Fixed value added to the original elevation values of loaded points. This is only active if **Transform** is set to **Dz**.
File	Text file that stores the XYZ correction values. This is only active if **Transform** is set to **Dxyz**.
Ground	Point class(es) defining the ground surface from which height values are derived. This is only active if **Transform** is set to **Height from ground**.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Ground** field. This is only active if **Transform** is set to **Height from ground**.
Surface	Surface model name that defines the TIN surface from which height values are derived. Requires an active surface model in TerraModeler. This is only active if **Transform** is set to **Height from TIN**, **Add TIN**, or **Scale dZ from TIN**.
Scale	Scale factor for exaggerating elevation differences between loaded points and a TIN model. This is only active if **Transform** is set to **Scale dZ from TIN**.

Both, the original and the final coordinates must fit inside the coordinate value ranges defined in Define Coordinate Setup tool.

View pulldown menu

Commands from the **View** pulldown menu are used to change the appearance of the **TerraScan** window as well as the display settings for laser points.

TO	USE COMMAND
Switch **TerraScan** window to small size	Small dialog
Switch **TerraScan** window to medium size	Medium dialog
Switch **TerraScan** window to large size	Large dialog
Switch **TerraScan** window to wide size	Wide dialog
Change the display of fields in **TerraScan** window	Fields
View header records of an LAS file	Header records
Fit a view to display all loaded points	Fit view
View points using elevation or intensity coloring	Display mode

Small dialog

Small dialog command changes the **TerraScan** window to a minimal size which consists of a title bar and pulldown menus only.

Medium dialog

Medium dialog command changes the **TerraScan** window to a medium size which consists of a title bar, the pulldown menus, and a medium size list displaying the attributes of loaded points.

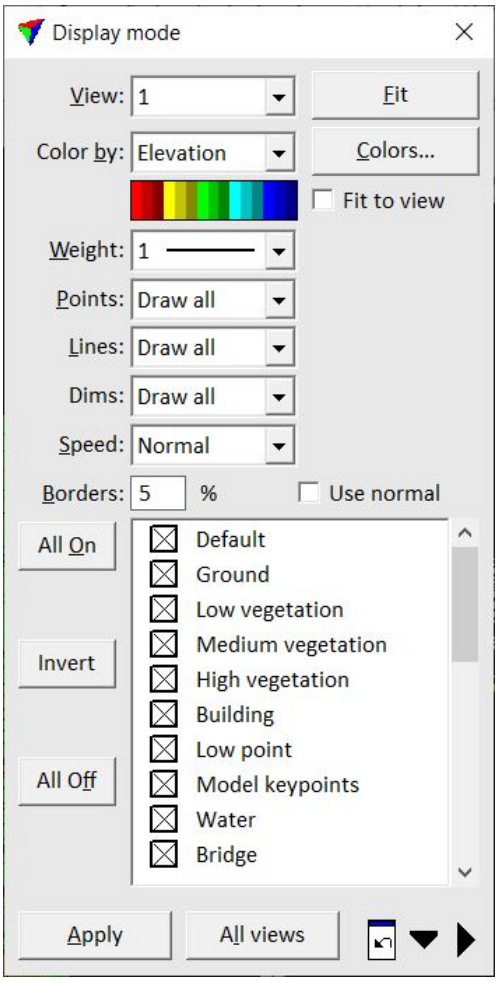
Large dialog

Large dialog command changes the **TerraScan** window to a large size which consist of a title bar, the pulldown menus, and a large size list displaying the attributes of loaded points.

Wide dialog

Wide dialog command changes the **TerraScan** window to a wide size which consist of a title bar, the pulldown menus, and a wide size list displaying the attributes of loaded points.

Display mode



Display mode command opens the **Display mode** dialog which contents settings for controlling the display of points loaded in TerraScan. See also Point display category of TerraScan **Settings** for setting default values and the general drawing method for points.

To set up the display of points:

1. Select **Display mode** command from the **View** pulldown menu.

   This opens the **Display mode** dialog.

2. Select a **View** for which you want to modify the point display.

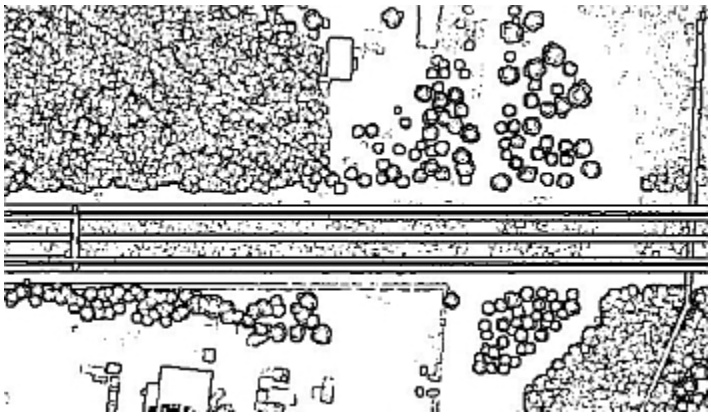
3. Define display settings.

4. Click **Apply** in order to update the display settings to the selected view.

SETTING	EFFECT	
View	View for which display settings are modified and applied.	
Fit	Fit the display in the selected **View** to the extend of the visible points.	
Color by	Attribute of points used for coloring:	
	• Class • Class + intensity • Color • Color + intensity • Density • Dimension • Distance • Echo • Echo length • Elevation • Group • Group + Intensity	• Image assignment • Intensity custom/auto • Line • Line + Intensity • Normal + intensity • Scanner • Scan direction • Shading • Slope • Time • Time + Intensity • Vegetation index

SETTING	EFFECT
	Coloring options are only available for display if the corresponding attribute is assigned to the loaded points.

Colors	Opens a specific dialog depending on the selected coloring attribute. See more information for each coloring attribute below.
Fit to view	If on, the software dynamically fits the coloring scheme for elevation coloring to the points in the current view extend. This improves the perceptibility of elevation differences when zoom in. This is only active if **Color by** is set to **Elevation**.
Display dynamics	If on, the point cloud is displayed in dynamic views, for example if the view is moved or rotated with CAD tools. This slows down the movement speed. If off, the point cloud is only drawn in static views. This is only active if **Color by** is set to **Shading**.
R | G | B	For each field, you can select the color channel for display. These fields are only active if **Color by** is set to **Color** or **Color + Intensity**. See Color by Color for more information.
Weight	Point size. Uses the standard line weights of the CAD file or class-specific size settings. See Define Classes command for information about class definitions.
Points	Visibility of points based on a Displayset: • **Draw all** - all points. • **Displayset only** - points of a defined display set. This requires that a displayset has been defined by the Add Points To Displayset tool.
Lines	Visibility of points based on the line number: **Draw all** or **Selected**. The **Select** button opens the **Display lines** dialog where you can select the visibility of specific lines. The **>** button lets you select a line for display interactively. The line closest to a data click inside a view is displayed, all other lines are switched off.
Dims	Visibility of points based on the dimension attribute: • **Draw all** - all points. • **Unknown only** - points of unknown dimension. • **Linear only** - points of linear dimension. • **Planar only** - points of planar dimension. • **Complex only** - points of complex dimension. This is only active if dimensions and normal vectors are computed for the points. See Compute normal vectors command for more information.
Speed	Method of drawing points: • **Fast** - sparse points - amount of displayed points depends on the zoom factor. If you zoom out, only a subset of points is drawn. This is the recommended setting for displaying a larger amount of points. • **Normal** - more points are drawn. The software decides based on the density of the point cloud. In sparse data sets, this already draws all points. • **Slow - all points** - all points are drawn at every zoom level. This is recommend, for example, for displaying sparse points spread over a larger area. It may slow down the display speed for a large amount of points.

Borders	The setting enhances the depth perception of the point cloud by drawing black borders around point groups that are in front of other points from the viewer's perspective. If set to a value > 0, the display of the point cloud includes the given percentage of black space on the screen. The larger the value, the more black borders are drawn and the more detailed the point cloud appears. 10% is a good value for many data sets and viewing directions. The maximum value is 50. The setting enforces the **Raster** display method. See also Point display category of TerraScan **Settings**.	
Use normal	If on, normal vectors are used for drawing points. The brightness of the points is detemined by the normal direction where 'foreside' points are drawn brighter and 'backside' points are drawn darker. The setting works only for points of planar dimension and in combination with constant coloring types for points, such as Color by = Class, Dimension, Echo, Line, etc. This is only active if normal vectors are computed for the points.	
List of classes	Switch the visibility of single classes on or off by clicking in the field next to the class name. The list contains the active classes of TerraScan.	
All On	Switch on the visibility of all classes.	
Invert	Invert the visibility of classes.	
All Off	Switch off the visibility of all classes.	

Apply	Apply current settings to the selected **View**.
All views	Apply current settings to all views.
	Undo the last change to display settings. This sets back the display to the previous state. The undo button works only for one last modification of display settings.
	Extend the length and/or width of the **Display mode** dialog. If the dialog is extended to a wide dialog, the class numbers are displayed in addition to the class names.

Color by Class

Color by Class setting displays the points according to their class attribute. To change the colors for classes, click the **Colors** button next to the **Class by** field. This opens the **Point classes** dialog which is described in detail for the Define Classes tool.

Color by Class + intensity

Color by Class + intensity setting combines class and intensity coloring. The color value is determined by the class while the brightness of the color indicates the intensity value.

The **Colors** button opens the **Point classes** dialog, the same dialog as for Color by Class. The brightness values for intensity cannot be changed.

Color by Color

Color by Color setting displays the points according to their color values. This requires that color values are assigned to the points. TerraScan can assign up to 10 color channels for each point. The maximum amount of color channels can only be stored in the TerraScan FastBinary format. LAS 1.4 format and later can store up to 4 color channels, LAS 1.2 format and later up to 3.

There are three channels available for displaying points by color. The color channel number can be selected from the list. Only channel with values <> 0 are selectable in the list. If colors have been extracted from 4-channel images, where the channels are 0=red (R), 1=green (G), 2=blue (B), and 3=near-infrared (NIR), the typical channel combinations for display are:

- R: 0, G: 1, B: 2 - true-color display, typical RGB color point display
- R: 3, G:1, B: 2 - infrared-color display
- same value for all channels - one-channel display

Color values for laser points can be extracted from raw images or raster attachments loaded into TerraPhoto, or assigned per class by setting values for red, green, and blue channels. See Extract color from images and Assign color to points for corresponding commands in TerraScan.

Color by Color + intensity

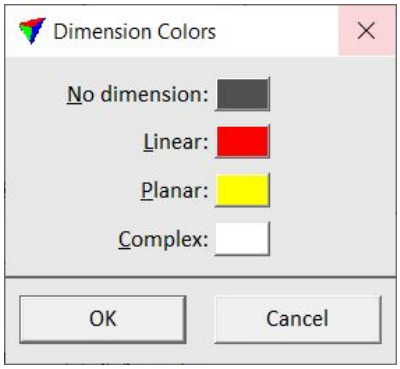
Color by Color + intensity setting combines color and intensity coloring. The color value is determined by the color values assigned to points while the brightness of the color indicates the intensity value.

Color by Density

Color by Density setting displays points according to the local point density. The denser the point cloud, the brighter the gray-scale value of the density coloring scheme. A new adjustment bar is added to the **Display mode** dialog which lets you adjust the coloring scheme more to the dark or bright side of the gray values.

Color by Dimension

Color by Dimension setting displays points according to their dimension attribute. The dimension of a point is computed together with its normal vector. See Compute normal vectors command for more information.



The **Colors** button opens the **Dimension colors** dialog.

The dialog lets you choose colors for displaying points with **No dimension** and for the three possible dimension types: **Linear**, **Planar**, and **Complex**. Click on the color field in order to select a color from the standard Windows dialog for color selection.

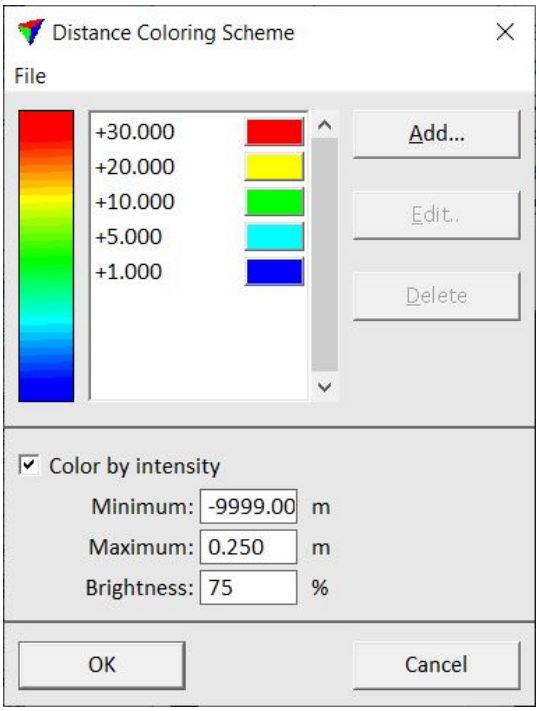
Color by Distance

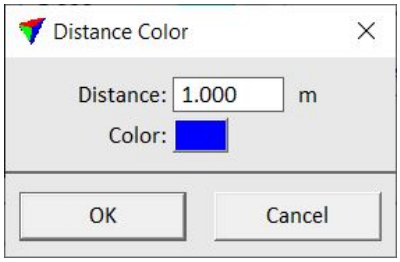
Color by Distance setting displays points based on the distance value. This requires that a distance value has been computed for points using the Compute distance command or macro action.

To color points by distance

1. Click on the **Colors** button next to the **Class by** field.

   This opens the **Distance Coloring Scheme** dialog:





2. Define settings.

A distance color scheme can be saved into a file by using the **File / Save as** command of the dialog. This creates a text file with the extension .DCL. You can open a distance color scheme file by using the **File / Open** command of the dialog.

SETTING	EFFECT
Add	Add a new distance value and color to the scheme. The button opens the **Distance Color** dialog. Type a **Distance** value in the text field. Click on the **Color** field in order to open the standard Windows dialog for selecting a color.
Edit	Modify a distance value and color. Select a value-color pair in the list of the dialog. Click the button which opens the **Distance Color** dialog. Type a **Distance** value in the text field. Click on the **Color** field in order to open the standard Windows dialog for selecting a color.
Delete	Delete a distance value-color pair from the scheme. Select the value-color pair and click the button.
Color by intensity	If on, points outside the distance range defined in the color scheme are displayed with intensity gray-scale values.
Minimum	Minimum distance value for intensity display. This is only active if **Color by intensity** is switched on.

SETTING	EFFECT
Maximum	Minimum distance value for intensity display. This is only active if **Color by intensity** is switched on.
Brightness	Determines the amount of white added to the intensity gray-scale scheme. A lower value makes the display darker, a higher value brighter. This is only active if **Color by intensity** is switched on.

3. Click OK in the **Distance Coloring Scheme** dialog.

   The software computes the distances.

4. Click **Apply** in the **Display mode** dialog in order to update the point display for the selected view.

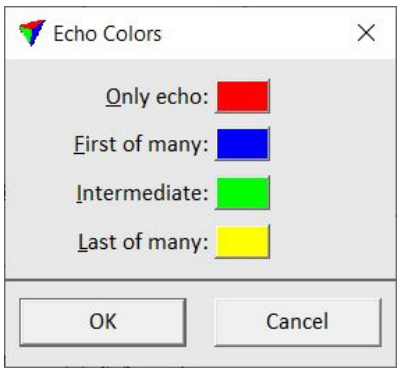
After changes in distance values of points, the distance coloring can be updated in a fast way by using the Update Distance Coloring tool.

Color by Echo

Color by Echo setting displays the points according to their return type.

The **Colors** button opens the **Echo Colors** dialog.

The dialog lets you choose colors for displaying points of different echo types: **Only**, **First of many**, **Intermediate**, and **Last of many**. Click on the color field in order to select a color from the standard Windows dialog for color selection.

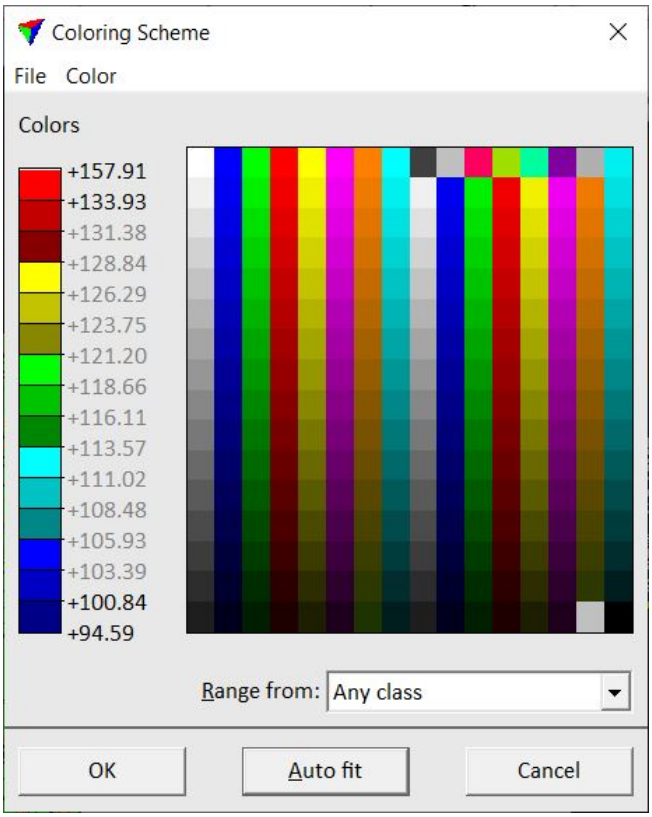


Color by Echo length

Color by Echo length setting displays points according to the length of the return pulse. The value is relative to a typical return pulse length on a hard surface. The echo length has to be extracted from waveform information. See Chapter Waveform Processing for more information.´

The **Colors** button opens the **Coloring Scheme** dialog, the TerraScan dialog for 256 Colors.

Color by Elevation



Color by Elevation setting displays the points according to their elevation value. A color scheme is applied where each color represents a certain range of elevation values. The default color scheme of TerraScan consists blue colors for low elevation values, green and yellow colors for intermediate values, and red colors for high elevation values.

A new setting **Fit to view** is added to the **Display mode** dialog. If on, the colors are adjusted dynamically to the elevation range of points that are currently visible in a view extend. This leads to a better visualization of points when views are zoomed or panned.

The **Colors** button opens the **Coloring Scheme** dialog, the TerraScan dialog for 256 Colors.

By default, the range for the color ramp includes the elevation values of all loaded points. This can be changed by selecting a class in the **Range from** list. The list contains the active classes in TerraScan. Then, only the elevation values of the selected class are applied to the color scheme.

The **Auto fit** button puts outliers in the lowest and highest elevation range. This results in a color scheme where outliers do not affect the color distribution in the coloring scheme. The auto fit action can be performed automatically whenever points are loaded into TerraScan if the corresponding setting in Point display category of TerraScan **Settings** is switched on.

Color by Group

Color by Group setting displays points according to their group number. This requires that a group number has been assigned to points. See Assign groups command for more information. The coloring mode uses colors of the active color table of the CAD file.

You can use the **Shuffle** button in order to change the color values for the groups.

Color by Group + Intensity

Color by Group & Intensity setting combines group and intensity coloring. The color value is determined by the group while the brightness of the color indicates the intensity value.

You can use the **Shuffle** button in order to change the color values for the groups.

Color by Image assignment

Color by Image assignment setting displays points according to their image number. This requires that an image number has been assigned to points. This can be achieved by extracting color from images for loaded points or project block binary files. The coloring mode uses colors of the active color table of the CAD file.

A new setting **Draw seamlines** is added to the **Display mode** dialog. If on, image seamlines are displayed temporarily.

Color by Intensity auto/custom

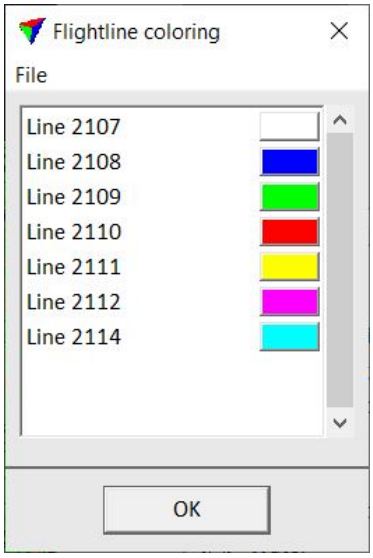
Color by Intensity custom/auto settings display the points according to their intensity value. The default gray scale of TerraScan stretches from dark gray for low intensity values to white for high intensity values.

Intensity auto uses a more homogeneous gray-scale coloring scheme. The intensity values are fitted to the gray values of the coloring scheme in a way that each gray value gets a similar amount of points. This results in a smoother intensity display, especially if intensity values are not normal-distributed.

Intensity custom uses the CAD file color table and fits the intensity values to the gray-scale color values. The **Colors** button opens the **Coloring Scheme** dialog, the TerraScan dialog for 256 Colors.

Color by Line

Color by Line setting displays the points according to their line number. By default, the soft ware uses the first seven colors from the active color table of the CAD file for coloring points.



The **Colors** button opens the **Line coloring** dialog.

The dialog lets you choose colors for displaying points of different line numbers. Select a line from the list. Click on the color field in order to select a color from the standard Windows dialog for color selection.

Color by Line & Intensity

Color by Line & Intensity setting combines line and intensity coloring. The color value is determined by the line while the brightness of the color indicates the intensity value. This can be used, for example, to check the horizontal accuracy of overlapping line.

The **Colors** button opens the **Line coloring** dialog, the same dialog as for Color by Line. The brightness values for intensity can not be changed.

Color by Normal + intensity

Color by Normal & Intensity setting combines slope and intensity coloring. The color value is determined by the normal vector computed for a point while the brightness of the color indicates the intensity value.

A new setting **Bright display** is added to the **Display mode** dialog. If on, the brightness of intensity values is increased. This makes the slope colors better visible in point clouds with rather dark intensity values.

Color by Scanner

Color by Scanner setting displays the points according to their scanner number. This is useful to distinguish points collected by different scanners of a multi-scanner system.

The **Colors** button opens the **Scanner coloring** dialog, the same dialog as for Color by Line.

Color by Scan direction

Color by Scan direction setting displays the points according to the scan direction. A point may be recorded in negative or positive scan direction, or as edge point. This requires that laser points are stored in LAS files. The display is based on the *scan direction* and *edge of flight line* bit fields present in LAS files.

Color by Shading

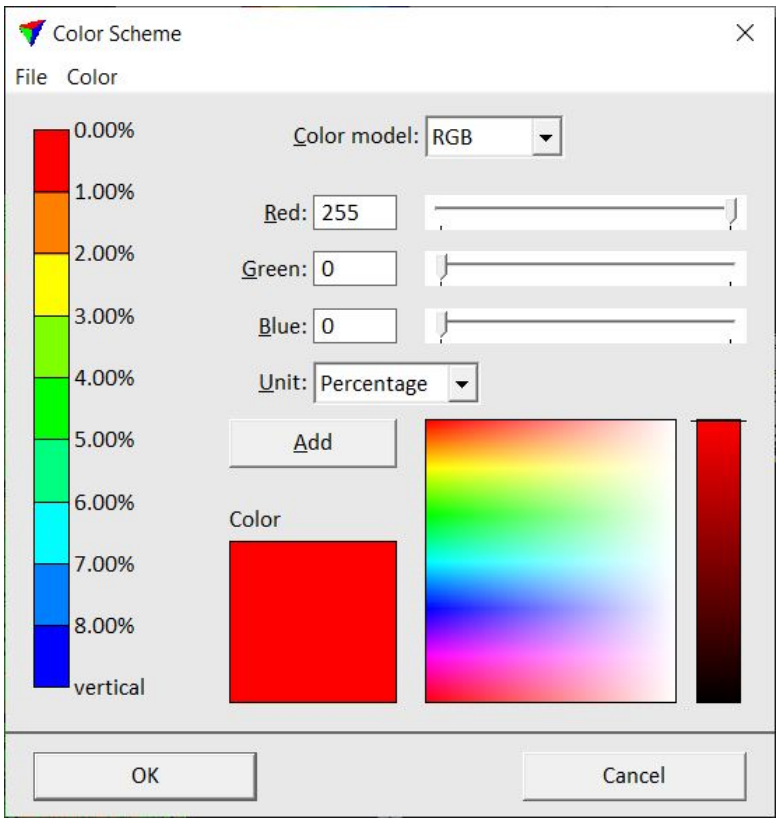
Color by Shading setting displays a triangulated surface of the points colored by class and shaded by triangle slope. This is useful, for example, to check ground classification results because error ground points show up clearly in the shaded surface display.

If **Color by Shading** is selected, three more settings are added to the **Display mode** dialog:

- **Azimuth** - direction of the light source. Zero is north and positive values increase clockwise.
- **Angle** - height above horizon of the light source.

- **Display dynamics** - if on, the point cloud is displayed in dynamic views, for example if the view is moved or rotated with CAD tools. This slows down the movement speed. If off, the point cloud is only drawn in static views.

Color by Slope



Color by Slope settings displays points according to their normal vector. This requires the computation of normal vectors.

The **Colors** button opens the **Color Scheme** dialog.

To define a new color scheme or modify a color scheme:

1. Select a **Unit** for expressing slope gradients: **Degree** or **Percentage**.
2. Follow the steps for color scheme definition.
3. Click OK.

Color by Time

Color by Time setting displays the points according to time stamp intervals. A new setting **Separation** is added to the **Display mode** dialog. It determines how long a time interval is for drawing points in one color. A time interval can be defined in seconds, minutes, hours, days, weeks or years.

The display option requires that times stamps are stored for the points.

Color by Time + Intensity

Color by Time + Intensity setting combines time and intensity coloring. The color value is determined by the time stamp interval while the brightness of the color indicates the intensity value.

Color by Vegetation index

Color by Vegetation index setting displays points according to a color channel computation. The display method is available if colors of multiple channels are extracted for the point cloud. The display mode provides two implemented methods for computing the vegetation index.

Normalized difference

The method assumes that channel 0 stores red (R) and channel 3 near-infrared (NIR) color values. The normalized difference value is computed with the following equation:

```
ND = (NIR - R) / (NIR + R)                    -1 <= ND <= +1
```

Visual band difference

The method assumes that channel 0 stores red (R), channel 1 green (G), and channel 2 blue (B) color values. The visual band difference value is computed with the following equation:

```
VBD = (2*G - R - B) / (2*G + R + B)                -1 <= VBD <= +1
```

In the Point display category of TerraScan **Settings**, a threshold value is defined for each vegetation index computation method. If the difference value of a point is smaller than the threshold, the point is displayed with intensity coloring (non-vegetation). If the difference value of a point is larger than the threshold, the point is displayed with a green coloring (= vegetation). The higher the vegetation index, the brighter the green.

In addition to the implemented methods, the list may include any User vegetation indexes defined in the corresponding category of TerraScan **Settings**.

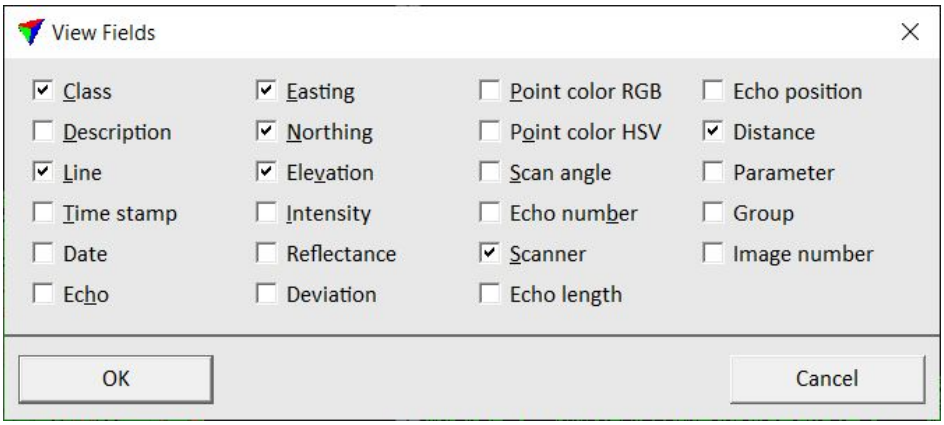
Fields

Fields command lets you select which attributes are displayed in the **TerraScan** window if point are loaded in TerraScan. The list of loaded points is only visible if the window is displayed as Medium dialog, Large dialog, or Wide dialog.

To select visible fields:

1. Select **Fields** command from the **View** pulldown menu.

   This opens the **View Fields** dialog:



2. Select fields you want to see in the list of loaded points and click OK.

FIELD NAME	DESCRIPTION
Class	Number of the point class. Mandatory attribute in TerraScan, must be unique.
Description	Description or name of the point class. Defined in the Point classes dialog.

FIELD NAME	DESCRIPTION
Line	Line number. May refer to the flight line, drive path or scan number in ALS, MLS or static TLS point clouds.
Time stamp	Point of time when a laser point was captured. Expressed in seconds.
Date	Date of data capture. This is derived from the time stamp of a point, if the time stamp format is GPS standard time or GPS time.
Echo	Return signal type. TerraScan distinguishes four return types: Only, First (of many), Intermediate, Last (of many).
Easting | Northing | Elevation	XYZ coordinates of points. Mandatory attribute in TerraScan.
Intensity	Reflectance value or strength of the return signal. Represents the reflectivity of the surface that was scanned.
Reflectance	*Riegl Extra Bytes* attribute: Reflectance.
Deviation	*Riegl Extra Bytes* attribute: Pulse shape deviation.
Point color RGB	Color of a point expressed in channels of the Red Green Blue color model. Uses 24-bit color values ranging from 0 to 65535. Color can be extracted from images or assigned using different methods.
Point color HSV	Color of a point expressed in values of the Hue Saturation Value color model.
Scan angle	Angle of a scan signal off from vertical.
Echo number	Number of return out of all returns from a scanner signal.
Scanner	Number of a scanner. Essential for differentiating data of different scanners in multiple-scanner systems. Assigned when a point cloud is loaded or imported into TerraScan.
Echo length	Relative length of a return signal compared to a typical return from a hard surface. Can be derived from waveform information stored for a point cloud.
Echo position	Difference in position of a peak of a return signal compared to a typical return from a hard

FIELD NAME	DESCRIPTION
	surface. Can be derived from waveform information stored for a point cloud.
Distance	Distance value of a point. Different computation methods.
Parameter	Attribute stored by processing software, *Riegl Extra Bytes* attribute: Amplitude.
Group	Group number assigned to a point.
Image number	Image number assigned to a point when color values are extracted from raw images.

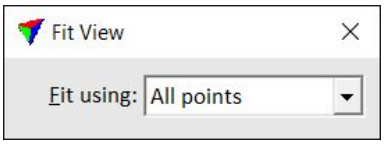
Fit view

Fit view command fits a CAD view window to an area covered by points loaded in TerraScan. The command fits the XY range as well as the display depth of a view to the points. It can fit to the extend of all loaded point or of visible point only.

To fit a view to the extend of loaded points:

1. Select **Fit view** command from the **View** pulldown menu.

   The **Fit View** dialog opens:



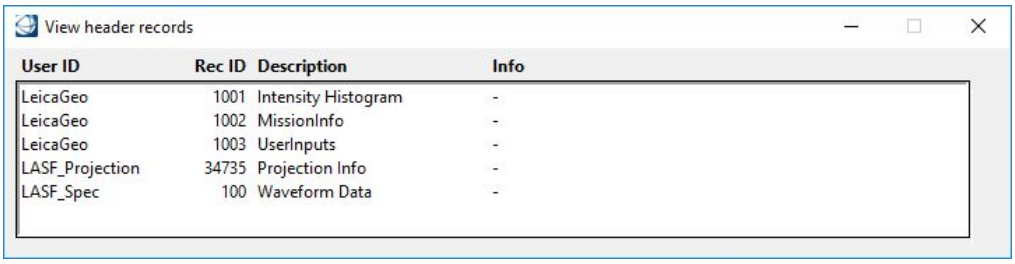
2. Select a method in the **Fit using** field: **All points** or **Visible points**.

3. Place a data click inside the view which you want to fit.

   This fits the view to the area covered by points and redraws the view.

Header records

Header records command opens a window that shows ***Variable Length Record Header*** information of LAS files. The window lists **User ID**, **Record ID**, **Description** and, if available, additional information for each ***Variable Length Record*** stored in an LAS file. The window is empty if no ***Variable Length Records*** are available.



User ID	Rec ID	Description	Info
LeicaGeo	1001	Intensity Histogram	-
LeicaGeo	1002	MissionInfo	-
LeicaGeo	1003	UserInputs	-
LASF_Projection	34735	Projection Info	-
LASF_Spec	100	Waveform Data	-

More information about the LAS format definition can be found on the webpages of the ASPRS.
External link *Terrasolid is not responsable for the content of webpages of other organizations.*

Working with Projects

Not UAV

A project definition in TerraScan helps to organize the work with a huge point cloud and to automate processing tasks. Basically, it is a method of dividing the whole point cloud data set into smaller, better manageable parts. These smaller geographical regions or blocks should be sized so that the laser data referenced by one block fits into the computer's RAM. There must be some space left in RAM for processing routines. See Memory usage of loaded points for more information.

TerraScan UAV does not include any project functionality.

A project definition is saved in a TerraScan project file with the extension *.PRJ. It is an ASCII file including:

- **header** - project settings.
- **block names** - laser file name and extension as link between the project and the referenced binary files storing the points.
- **block boundaries** - coordinates of the vertices of each block boundary.

Typical steps for creating a project are:

1. Use Load Airborne Points tool or Read points command to load a subset of points (every 10th, 50th, 100th,...) from all input files that belong to a project. This shows the geographical coverage and location of laser points without providing the full point density.
2. Create block boundaries. There are several tools and methods that support the creation of block boundaries:
   - Measure Point Density tool - estimate the amount of points inside a certain area by using a rectangle as sample area.
   - Any CAD tool for placing closed elements - place boundary shapes for the blocks. Each block should enclose an area with a manageable number of points. You should use snapping tools to avoid gaps and overlap between neighbouring block boundaries.
   - Design Block Boundaries tool - create shapes from a closed line work and get an approximate number how many points are inside each shape.
   - Create along centerline command - create blocks along a linear element.
   - Create along tower string command - create blocks along a tower string element for a powerline corridor.
   - Create grid block boundaries automatically during the import of point files. See Import points into project command.

The result of block boundary creation is always a set of shape elements which include the project area for processing. Each shape should enclose a part of the project area with a number of points that easily fit into the computer's memory.

3. Close the points loaded in step 1 using Close points command.
4. Select Define Project tool. This opens the TerraScan Project window.
5. Define a new project using New project command.
6. Select all block boundary shapes created in step 3.
7. Add blocks to the project using Add by boundaries command.
8. Save the project definition using Save project as command.

9. Import points from all input files into the project using Import points into project or Import directory commands.

Once the points are imported into the project, you can process the data of the referenced files in batch mode. You can run macros on the project which may include several processing steps. Macros can run in TerraScan but most of the macro actions can also be performed in TerraSlave. See Chapter Macros for detailed information.

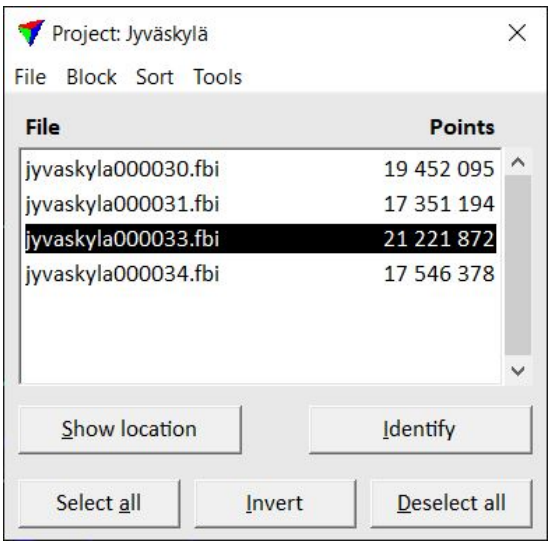
Before you process project data in batch mode, you would test the settings of macro actions based on loaded points in order to find the optimal parameter values for the project area. It is also recommended to test a macro on several blocks before you run it on the whole project.

Besides running macros on a project, there are several processing tasks that can be performed on project level. The corresponding commands are included in the pulldown menus of the TerraScan **Project** window and described in the following sections.

TerraScan Project window

The **Project** window contains all menu commands for creating and modifying project definitions, managing block definitions, and for running processing steps on project level.

Select the Define Project tool to open the **Project** window:



In the file list of the project window, all blocks are listed that belong to the project with the amount of points in the referenced laser binary file. If File locking is active for a project, or if a block binary file is set to be read only, the list also includes information about file locking.

To select a block, click on the name in the list. Press < Ctrl> to select several blocks.

To show the location of a block, select a line in the **Project** window. Click on the **Show location** button and move the mouse pointer into a view. This highlights the boundary of the selected block.

To identify a block, click on the **Identify** button and place a data click inside a block boundary in a view. This selects the corresponding line in the **Project** window. Several blocks can be identified if <Ctrl> is pressed while selecting block locations in the view.

You can use the **Select all** and **Deselect all** buttons in order to select and deselect all blocks. The **Invert** button selects all blocks that are previously not selected and deselects previously selected blocks.

Block pulldown menu

Commands from the **Block** pulldown menu are used to create and edit block boundaries.

TO	USE COMMAND
Add block definitions to a project	Add by boundaries
Add block definitions deduced from block binary files	Add using files
Edit a block definition	Edit definition
Delete a block definition	Delete definition
Merge selected blocks into one block	Merge blocks
Lock selected block files	Lock selected
Release the lock of a block file	Release lock
Draw block boundaries into the CAD file	Draw boundaries
Create block boundaries along a centerline	Create along centerline
Create block boundaries along a tower string	Create along tower string
Create block boundaries based on trajectory information only	Create along trajectories
Transform block boundaries	Transform boundaries

Add by boundaries

Not UAV

Add by boundaries command adds block boundaries to a project definition. This is usually the second step of the project creation after defining settings of the project itself.

Block boundaries can be created using CAD drawing tools for closed elements, or using one of the tools provided by TerraScan, such as Design Block Boundaries, Create along centerline, or Create along tower string.

The command lets you define names for the blocks. The same names are used for the block binary files which are created by importing points into the project. The names can contain a prefix and a unique, automatically derived number for each block. CAD text elements drawn inside the boundaries

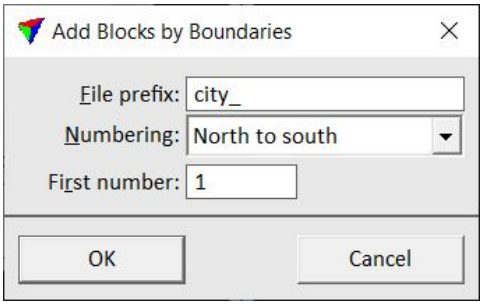
of each block can be used as names as well. Alternatively, a naming method can be defined in the Block naming formulas category of TerraScan **Settings**.

To add block boundaries to the project definition:

1. Select shape elements and (optional) text elements inside the shapes using the **Selection** tool.

2. Select **Add by boundaries** command from the **Block** pulldown menu.

   This opens the **Add Blocks by Boundaries** dialog:



3. Define settings and click OK.

   This adds the blocks to the project definition. Use Save project or Save project as commands in order to save the block boundaries in a project file.

SETTING	EFFECT
File prefix	Prefix for the names of project blocks and related block binary files.
Numbering	Method of assigning names or name parts to blocks: • **Selection order** - automatic numbering increases in the same order as boundary shapes have been selected. • **North to south** - automatic numbering increases geographically from north to south and secondarily west to east. • **South to north** - automatic numbering increases geographically from south to north and secondarily west to east. • **West to East** - automatic numbering increases geographically from west to east and secondarily south to north. • **East to West** - automatic numbering increases geographically from east to west and secondarily south to north. • **Trajectory order** - automatic numbering increases according to the order of data capturing. The time of data capturing is derived from the active trajectories in TerraScan.

SETTING	EFFECT
	• **Selected numbers** - the name is defined by a unique numerical text element that is placed inside each block boundary. • **Selected strings** - the name is defined by a text element that is placed inside each block boundary. The list may further include Block naming formulas that are defined in TerraScan **Settings**.
First number	Number of the first block to add. This is only considered if a method of automatic block numbering is selected in the **Numbering** list.

Add using files

Not UAV

Add using files command creates a list of block definitions from a collection of FastBinary, LAS or GeoTIFF files. It can be used to create a TerraScan project for files that have been delivered as tiled point cloud but without project definition.

The process deduces the bounding box for each file from the file header and uses that as block boundary. The block boundary may be a bit inaccurate if the position of the point cloud has been modified, for example by TerraMatch processing. However, this is not critical for further processing steps.

To create block definitions from files:

1. Create a project definition. Use the format of the block binary files you want to use for block creation.

2. Save the project file in the same folder where the block binary files are stored.

3. Select **Add using files** command from the **Block** pulldown menu.

   This opens the **Add files to project** dialog, a standard dialog for opening files.

4. Select all files you want to use for creating block definitions and click **Open**.

   This adds the block definitions to the project definition. Use Save project command in order to save the block definitions in a project file.

Create along centerline

Not UAV

Create along centerline command creates block boundaries along a centerline element. The block length is measured along the centerline element and each block has exactly the same width and length as defined. Only the last block along the centerline element may be shorter.

Optionally, the command creates text elements that are placed inside each block boundary. The resulting block boundaries and text elements can then be used to define blocks for a project with Add by boundaries command.

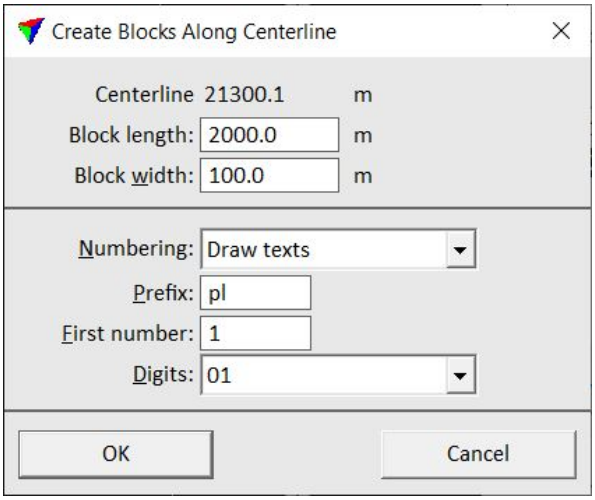
To create blocks along a centerline element:

1. Create a centerline element using CAD or TerraScan tools.

2. Select the centerline element.

3. Select **Create along centerline** command from the **Block** pulldown menu.

   This opens the **Create Blocks Along Centerline** dialog:



4. Define settings and click OK.

   This draws the block boundaries as shapes and, if defined, the text elements into the CAD file. The elements are drawn on the active level using the active symbology settings of the CAD platform.

SETTING	EFFECT
Centerline	The length of the selected centerline element is displayed.
Block length	Length of the blocks measured along the centerline element.
Block width	Width of the blocks.

SETTING	EFFECT
Numbering	Defines labeling of the blocks: **None** or **Draw texts**.
Prefix	Prefix for the text element drawn as label for each block. This is followed by an automatically increasing number.
First number	Number of the first block.
Digits	Defines the amount of digits for block numbering.

Create along tower string

Not UAV

Create along tower string command creates block boundaries along a tower string element. Any linear element can serve as a tower string for this command. The block length is measured along the tower string but a block boundary is drawn at the location of the vertex that is closest within the given block length. The last block ends at the last vertex of the tower string element. The width of the blocks is constant.

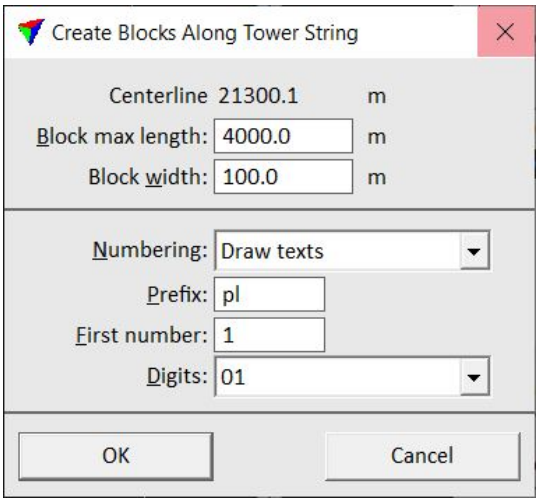
Optionally, the command creates text elements that are placed inside each block boundary. The resulting block boundaries and text elements can then be used to define blocks for a project with Add by boundaries command.

The command is most useful for creating blocks for a powerline project. See Chapter Powerlines for more information.

To create blocks along a tower string:

1. Draw a tower string using TerraScan Place Tower String tool or CAD drawing tools.
2. Select the tower string element.
3. Select **Create along tower string** command from the **Block** pulldown menu.

   This opens the **Create Blocks Along Tower String** dialog:



4. Define settings and click OK.

   This draws the block boundaries as shapes and, if defined, the text elements into the CAD file. The elements are drawn on the active level using the active symbology settings of the CAD platform.

SETTING	EFFECT
Centerline	The length of the selected tower string is displayed.
Block max length	Maximum length of a block.
Block width	Width of the blocks.
Numbering	Defines labeling of the blocks: **None** or **Draw texts**.
Prefix	Prefix for text string drawn as label for each block. This is followed by an automatically increasing number.
First number	Number of the first block.
Digits	Defines the amount of digits for block numbering.

Create along trajectories

Not UAV

Create along trajectories command creates block boundaries along trajectories. The trajectories must be active in TerraScan. The block length is measured along the trajectories. The block width is calculated from each trajectory line. If trajectories are closer to each other than half of the given block width, the width is computed from the outermost trajectories. The command is suited for MLS corridor projects.

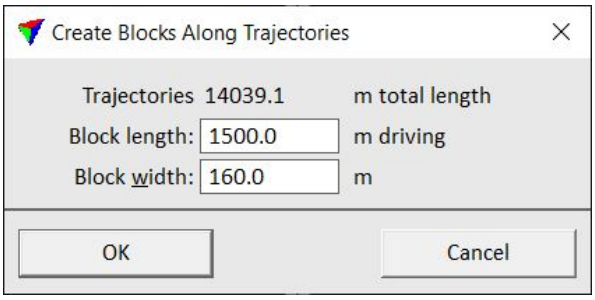
The resulting block boundaries can then be used to define blocks for a project with Add by boundaries command.

To create blocks along a centerline element:

1. Load trajectories into TerraScan.

2. Select **Create along trajectories** command from the **Block** pulldown menu.

   This opens the **Create Blocks Along Trajectories** dialog:



4. Define settings and click OK.

   This draws the block boundaries as shapes into the CAD file. The elements are drawn on the active level using the active symbology settings of the CAD platform.

SETTING	EFFECT
Trajectories	The length of the trajectories is displayed.
Block length	Length of the blocks measured along the trajectories.
Block width	Width of the blocks.

Delete definition

Not UAV

Delete definition command deletes one or more block definitions from a project definition.

To delete a block definition:

1. Select one or more blocks in the **Project** window.

2. Select **Delete definition** command from the **Block** pulldown menu.

   This deletes the selected blocks. If more than one block is selected, a message appears that asks you to confirm the removal.

3. Use Save project or Save project as commands in order to save the modification in a project file.

Draw boundaries

Not UAV

Draw boundaries command draws the boundaries and (optional) names of block definitions into the CAD file. This can be used, for example, to draw boundaries if points have been imported into the project without defining block boundaries beforehand or after blocks have been transformed.

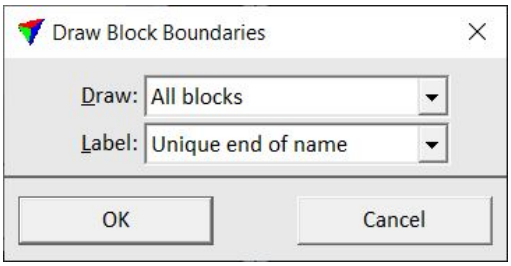
The command can also be used to draw the outer boundary of a project area into the CAD file.

To draw block boundaries into the CAD file:

1. (Optional) Select block definitions in the **Project** window to be drawn into the CAD file.

2. Select **Draw boundaries** command from the **Block** pulldown menu.

   This opens the **Draw Block Boundaries** dialog:



3. Select settings and click OK.

   The boundaries and, if selected, block labels are drawn as shape and text elements into the CAD file. The elements are drawn on the active level using the active symbology settings of the CAD platform.

SETTING	EFFECT
Draw	Boundaries to be drawn: **All blocks**, **Selected blocks**, or **Project outer boundaries**.
Label	Defines the way of labeling the blocks: • **None** - no text elements are created. • **Block number** - the complete block number is drawn. • **Full file name** - the complete file name without extension is drawn. • **Unique end of name** - only the last unique number or text string of the block names is drawn. This works only if more than one block is drawn into the CAD file. This is only active is **Draw** is set to **All blocks** or **Selected**.

Edit definition

Not UAV

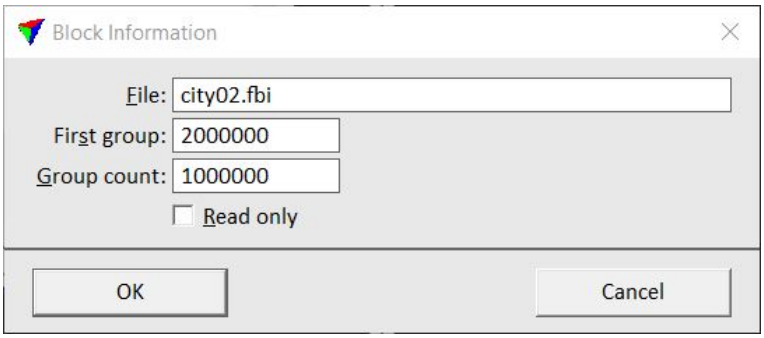
Edit definition command lets you edit a block name and lock a block binary file to be read only. If several blocks are selected in the **Project** window, the command lets you edit the source folder, prefix, and file name extension of the selected blocks.

To edit a single block definition:

1. Select a block definition in the **Project** window.

2. Select **Edit definition** command from the **Block** pulldown menu.

   This opens the **Block Information** dialog:



3. Define settings and click OK.

   This applies the modification to the block definition.

4. Use Save project or Save project as commands in order to save modification in a project file.

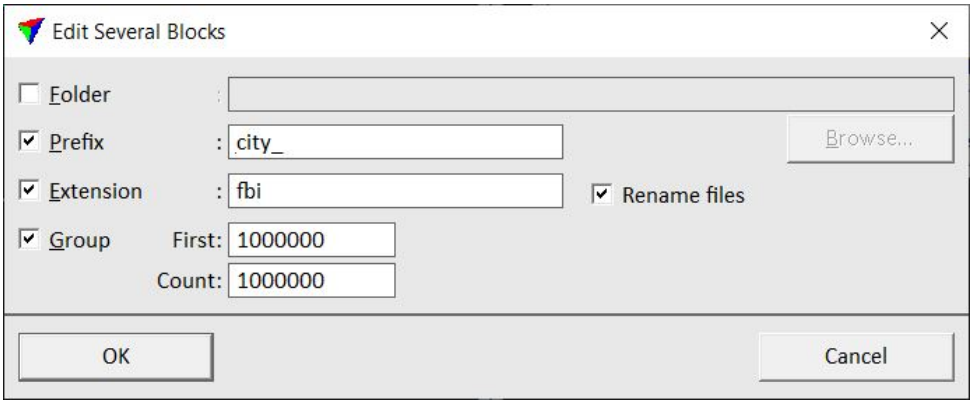
SETTING	EFFECT
File	Block name and file extension.
First group	Lowest possible group number in the block. Effects the Assign groups command or corresponding macro step.
Group count	Amount of group numbers for the block. Effects the Assign groups command or corresponding macro step.
Read only	If on, the block file can be opened only for reading and permanent modifications are not allowed.

To edit several block definitions:

1. Select several block definitions in the **Project** window.

2. Select **Edit definition** command from the **Block** pulldown menu.

This opens the **Edit Several Blocks** dialog:



3. Switch on the attributes that you want to modify.

4. Define new attribute values and click OK.

   This applies the modification to the block definitions.

5. Use Save project or Save project as commands in order to save modification in a project file.

SETTING	EFFECT
Folder	Path to the source folder of the block binary files. Use the **Browse** button in order to open a standard dialog for selecting a folder.
Prefix	Text string that replaces the current prefix in the beginning the block names. All non-numeric characters in the beginning of a block name are considered the prefix.
Extension	File extension that replaces the current extension in the end of the block names.
Group	The **First** field defines the lowest group number for the project. The **Count** field defines the amount of group numbers for each block. Effects the Assign groups command or corresponding macro step.

If the block file name in the project definition is changed after points have been imported into the project, the link between the project definition and the block binary file is lost unless the block binary file is renamed accordingly.

Lock selected

Not UAV

Lock selected command locks the selected block binary files and thus, disables processing of the files on any other computer than the locking computer. See File locking for more information about the file locking concept of TerraScan.

The command is only available if **Require file locking** is active in the **Project Information** dialog. See New project for a description of the dialog settings.

To lock selected block binary files:

1. Select one or more block names in the **Project** window.
2. Select **Lock selected** command from the **Block** pulldown menu.

   This locks the selected block. The name of the locking computer and the time when the lock was established are displayed in the **Project** window.

Merge blocks

Not UAV

Merge blocks command merges selected project blocks. It combines the block definitions and the binary files into one definition/file. The name of the block with the lowest number is used for the merged block.

The command is only available if at least two block definitions are selected in the **Projects** window. The process works only if there are block binary files available.

To merge blocks interactively:

1. Select two or more block definitions in the **Project** window.
2. Select **Merge blocks** command from the **Block** pulldown menu.

   An **Alert** dialog asks for confirmation of the selection.
3. Click OK.

   This creates a new block definition and binary file. The original binary files are not kept. An information dialog shows the number of merged blocks after the process is finished.
4. Use Save project or Save project as commands in order to save the modification in a project file.

Blocks can be merged automatically by using the Merge small blocks from the **Tools** pulldown menu. However, the result of the automatic process may not be optimal and can be refined by the manual command.

Release lock

Not UAV

Release lock command releases the lock for block binary files. The lock can be released only on the same workstation that locked the file or if the file is already locked for more than 24 hours. See File locking for more information about the file locking concept of TerraScan.

The command is only available if **Require file locking** is active in the **Project Information** dialog. See New project for a description of the dialog settings.

To release a lock for a block binary file:

1. Select one or more locked blocks in the **Project** window.
2. Select **Release lock** command from the **Block** pulldown menu.

   This releases the locking of the selected block. A message appears that informs about the success of the release.

Transform boundaries

Not UAV

Transform boundaries command transforms the block boundaries. This effects the coordinates of the block boundary vertices stored in the project definition file. The command can be used, for example, to update a project if the block binary files have been transformed into a new projection system.

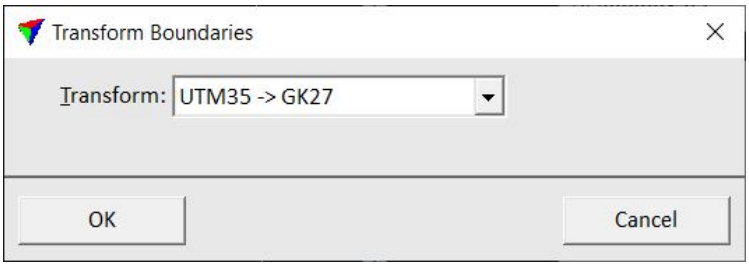
The command can use the following transformation definitions:

- **Dxyz** - text file storing difference values for easting, northing, and elevation.
- A specific transformation defined in Coordinate transformations / Transformations category of TerraScan **Settings**.

To transform block boundaries:

1. Select **Transform boundaries** command from the **Block** pulldown menu.

   This opens the **Transform Boundaries** dialog:



2. Select a transformation in the **Transform** list.

3. If **Dxyz** is selected in the **Transform** list, define the text file that stores the transformation values in the **File** field.

4. Click OK.

   This transforms the block boundaries. An information dialog shows the number of transformed block boundaries. Use Save project or Save project as commands in order to save the modification in a project file.

You may use Draw boundaries command to draw the transformed boundaries into the CAD file.

File pulldown menu

Commands from the **File** pulldown menu are used to create a project, edit project information and to import points into a project.

TO	USE COMMAND
Create a new project definition	New project
Open an existing project definition	Open project
Save changes to an existing project definition	Save project
Save a new project definition	Save project as
Edit project information and settings	Edit project information
Import laser files into the project	Import points into project
Import all laser files from a directory	Import directory

Edit project information

Not UAV

Edit project information command lets you edit the settings of a loaded project definition.

To edit a project:

1. Select **Edit project information** command from the **File** pulldown menu.

   This opens the **Project Information** dialog which is described for the New project command.

2. Edit the settings for the project definition and click OK.

3. Use Save project or Save project as commands in order to save the modifications of the project definition in a project file.

The **Default block values** in the **Project information** dialog do not effect existing blocks. Any changes of these values effect only new blocks that are added to the project. You can use the Edit definition command to change settings for existing blocks.

Import directory

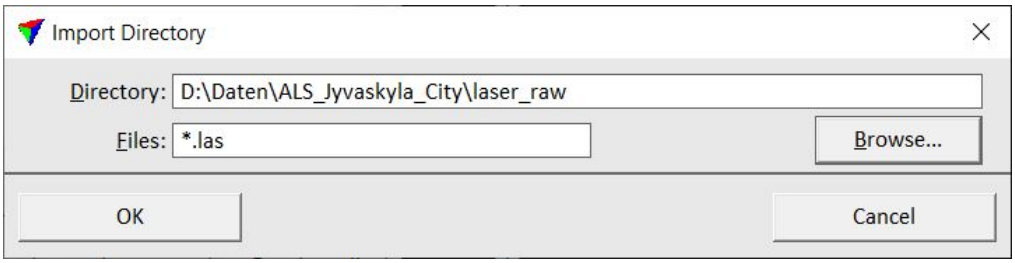
Not UAV

Import directory command imports point files into the project. All files of the same format in a directory are imported. The import process itself works in the same way as described for the Import points into project command.

To import all files in a directory into a project:

1. Select **Import directory** command from the **File** pulldown menu.

   This opens the **Import Directory** dialog:



2. Define settings and click OK.

   This opens the **Import Points into Project** dialog. Follow the steps of Import points into project procedure in order to import the files.

SETTING	EFFECT
Directory	Folder from which to import files. Click on the **Browse** button in order to select a folder in the **Browse for Folder** dialog, a standard dialog for selecting folders.
Files	Defines the files that are imported. You can use the * character as placeholder for any number of characters in a file name or extension. For example, ***.las** imports all LAS files from a folder, ***.*** imports all files from a folder.

Import points into project

Not UAV

Import points into project command imports point files into a project. The process reads the points from the input files and stores them into new files according to the block definitions. As a result, there is a new point file for each block of the project.

The file names and the format are derived from the block names which contain the name itself and the extension depending to the selected storage format. Thus, a file that is created in the import process is linked to a project by its name and extension.

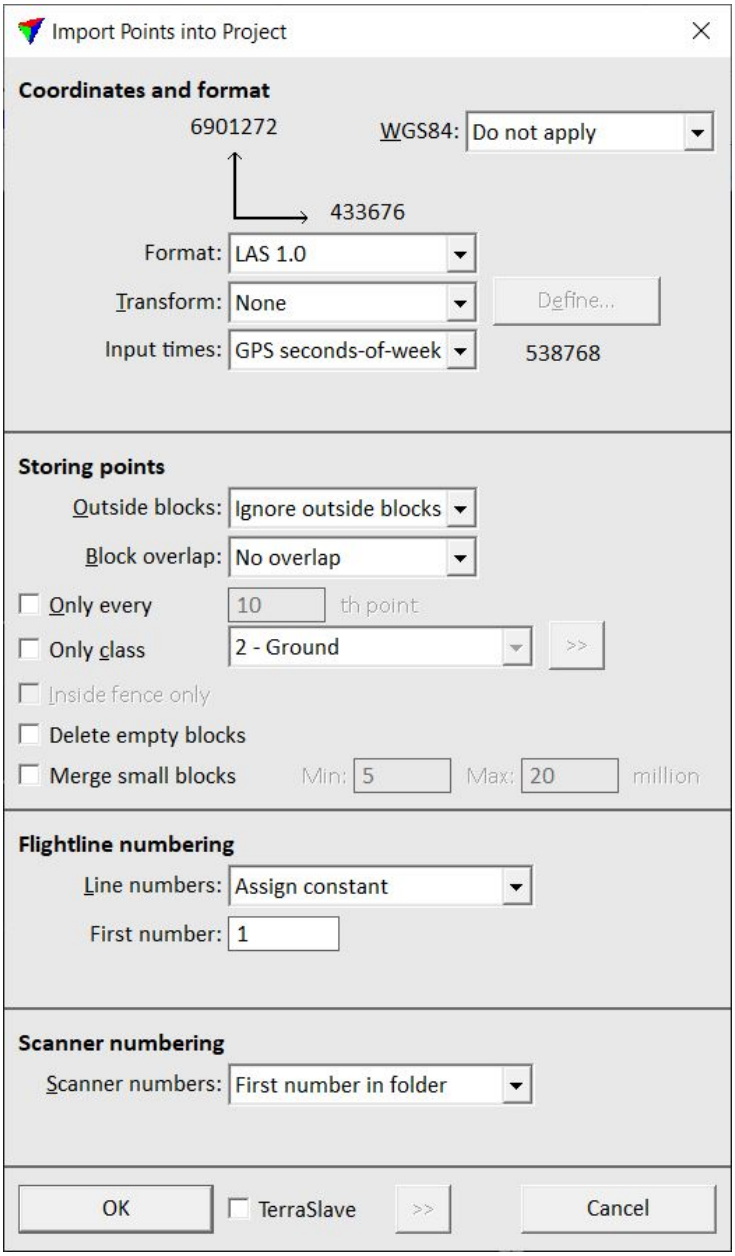
To import points into a project:

1. Select **Import points into project** command from the **File** pulldown menu.

   This opens the **Import points into project** dialog, a standard dialog for open files.

2. Select input files and click **Open**.

   This opens the **Import Points into Project** dialog:



3. Define settings and click OK.

   This starts the import of the input files. A progress bar shows the progress of the process.

After all files are imported, TerraScan shows a report. The report lists the imported files, the amounts of imported and ignored points for each file, and the overall amounts of imported and ignored points. It can be saved as a text file or sent to a printer by using commands from the **File** pulldown menu of the report window. If the import is done by TerraSlave, the report file is stored in the TerraSlave reports folder, such as C:\TERRA64\TSLAVE\REPORTS.

SETTING	EFFECT
Coordinates	The coordinate axes show the coordinate values of the first point in the laser file. This helps to decide if the points are in the correct coordinate system or if an coordinate transformation has to be applied.
WGS84	Transformation from WGS84 coordinates to another projection system applied during the import. The list contains projection systems that are active in Coordinate transformations / Built-in projection systems, Coordinate transformations / US State Planes, and Coordinate transformations / User projection systems categories of TerraScan **Settings**.
Define	Opens the **Transformation** dialog which lets you define a transformation. See Coordinate transformations / Transformations category for more information. This is only active if **Transform** is set to **Define now**.
Format	Format of the input files. This is automatically recognized by the software. For ASCII files, there might be more than one option.
Transform	Transformation applied to the points during the import process. The list contains transformation that are defined in Coordinate transformations / Transformations category of TerraScan **Settings**. Select **Define now** in order to define a new transformation.
Input times	GPS time format of the time stamps in the input files.
Survey date	Date when the data was captured. The format is day/month/year (dd/mm/yyyy). This is only active if **Input time** is set to **GPS seconds-of-week** and the time stamp format for the project is set to **GPS standard time** or **GPS time**.
Outside blocks	Defines how points outside the block boundaries are handled: • **Ignore outside blocks** - points outside boundaries are ignored.

SETTING	EFFECT
	• **Ignore outside selected** - points outside selected block boundaries are ignored. This is only active if blocks are selected in the list of blocks in the Project window. • **Create grid blocks** - the software creates new blocks for the points outside boundaries. The size and names of the new blocks are defined by the setting in the **Project information** dialog.
Block overlap	Defines how points in overlapping block areas are handled: • **No overlap** - points in the overlapping area are loaded only in the first of the overlapping blocks. The area is empty in all other overlapping blocks. • **Duplicate points** - points in the overlapping area are loaded into all blocks that overlap each other.
Only every	If on, only every **n** th point of the input files is imported where **n** is the given value.
Inside fence only	If on, only points that are located inside a MicroStation fence or selected shape element are imported.
Delete empty blocks	If on, blocks without points located inside the block area are deleted from the project definition.
Merge small blocks	If on, blocks with a small amount of points inside the block area are merged with neighboring blocks. The **Min** and **Max** values define the minimum and maximum amount of points per block. Blocks are only merged if the amount of points is smaller than the given minimum and if the merged block does not exceed the given maximum.
Default	Point class that is assigned to all imported points if no class attribute is stored in the input files. This is only active if **Format** is set to any text file format that does not include the class attribute.
Line numbers	Defines, how line numbers are assigned to the points during the import process: • **Use from file** - line numbers from source files are used. • **Assign constant** - the number given in the **First number** field is assigned to all points.

SETTING	EFFECT
	• **First number in name** - the first numerical sequence in a file name is used as line number. • **Last number in name** - the last numerical sequence in a file name is used as line number. • **First number in folder** - the first numerical sequence in the name of the folder containing the input files is used as line number. • **Last number in folder** - the last numerical sequence in the name of the folder containing the input files is used as line number. • **Deduce using time** - numbers are assigned based on trajectories loaded into TerraScan. The same process can be performed for by the Deduce using time command or the corresponding macro action. • **Increase by xy jump** - the line numbers increase from the given **First number** if the xy distance is bigger than the value given in the **By distance** field. • **Increase by time jump** - the line numbers increase from the given **First number** if a jump in time stamps occurs. This requires that trajectory information is available in TerraScan. • **Increase by file** - the line numbers increase from the given **First number** for each separate file. • **Increase by file name** - the line numbers increase from the given **First number** for each file with another file name. Files with the same name get the same number. • **Increase by directory** - the line numbers increase from the given **First number** for each file stored in another source folder. Files from the same source folder get the same number. The availability of options depends on the number of input files and the attributes stored for points in the input files.
Scanner number	Defines, how scanner numbers are assigned to the points during the loading process: • **Use from file** - scanner numbers from source files are used. • **Assign constant** - the number given in the **First number** field is assigned to all points.

SETTING	EFFECT
	• **Increase by file** - the scanner numbers increase from the given **First number** for each separate input file. • **First number in name** - the first numerical sequence in a file name is used as scanner number. • **Last number in name** - the last numerical sequence in a file name is used as scanner number. • **First number in folder** - the first numerical sequence in the name of the folder containing the input files is used as scanner number. • **Last number in folder** - the last numerical sequence in the name of the folder containing the input files is used as scanner number. • **From line number** - the line number is used as scanner number. The availability of options depends on the number of input files and the attributes stored for points in the input files.
TerraSlave	If on, the import process starts in TerraSlave. TerraScan can be used for other tasks while the import runs in TerraSlave.
>>	Opens the **TerraSlave Task Settings** dialog. You can select or deselect computers that will participate in the import task. See TerraSlave User Guide for more information.

If you import new points into a block that already contains points, the additional points are added to the existing block binary file. Therefore, if the import of raw data needs to be repeated, you must delete the existing block binary files before starting the next import process.

New project

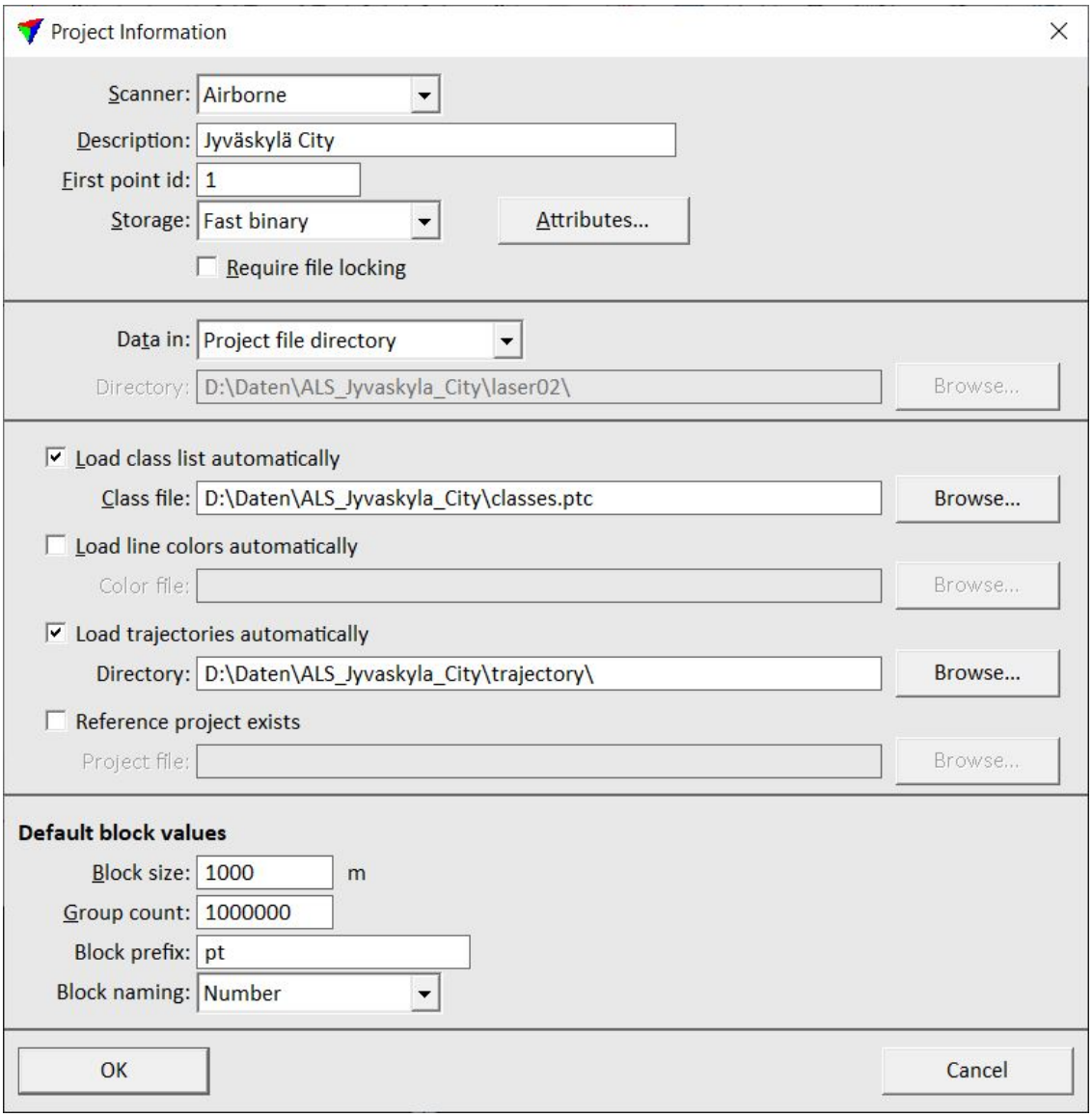
Not UAV

New project command creates a new project definition. The complete project definition includes some descriptive information and a list of block boundaries. For the definition of block boundaries see Add by boundaries.

To create a project definition:

1. Select **New project** command from the **File** pulldown menu.

   This opens the **Project Information** dialog:



2. Define settings and click OK.

SETTING	EFFECT
Scanner	Scanner type: **Airborne**, **Mobile**, **Ground based**, or **Mixed**.
Description	Descriptive text for the project.
First point id	Start ID number for the laser data file.
Storage	Block binary file format: **FastBinary**, **Scan binary 8 bit**, **Scan binary 16 bit**, **LAS 1.x**, **LAZ 1.x**, **GeoTIFF**. For more information see Choosing a project storage format.
Attributes	Opens the Attributes to store dialog. Only active attributes are stored in project block laser point files that belong to the project.

SETTING	EFFECT
Require file locking	If on, a project block file is marked as locked when a user opens it for modification with the Open block command. For more information see File locking.
Data in	Defines how the directory for the referenced laser data files is determined: • **Project file directory** - laser files are stored in the same directory as the project file. This is independent of the absolute path of the data and is therefore good in a network environment or when moving the data set from one computer to another. • **Separate directory** - laser files are stored at the location given in the **Directory** field.
Load classes automatically	If on, the defined **Class file** is automatically loaded with the project.
Load line colors automatically	If on, the defined **Color file** for line colors is automatically loaded with the project.
Load trajectories automatically	If on, trajectories from the given **Directory** are loaded automatically with the project.
Reference project exists	If on, the given **Project file** is defined as reference project to be used in corresponding tools, such as Compare with reference command.
Default	Size of rectangular, automatically created blocks. These blocks are created if points are imported from areas which are outside the pre-defined block boundaries.
Group count	Amount of group numbers available for one block in the project. Determines also the first group number of the first block (= lowest group number in a project). Effects the Assign groups command or corresponding macro step.
Block prefix	Prefix of block names that is used for automatically created block boundaries.
Block naming	Block name definition for automatically created block boundaries. The list contains **Number** as automatic numbering option and any user defined Block naming formulas defined in TerraScan **Settings**.

Choosing a project storage format

TerraScan supports multiple file formats from which you need to select one to be used for the block binary files. The best choice depends on a few factors:

- **FastBinary** is the best format for processing data in TerraScan.
- If your project has more than 255 lines, you should choose **FastBinary** or **Scan binary 16 bit** instead of **Scan binary 8 bit**. If your project has fewer than 255 lines, **Scan binary 8 bit** is a more compact format than **Scan binary 16 bit**.
- If your project is static ground-based, you must choose **FastBinary**, **Scan binary 8 bit**, or **Scan binary 16 bit**.
- If you need to transfer the laser files to other applications, the best choice is **LAS 1.x** which is an open industry standard format and usually used for laser data exchange. The version of the LAS format mainly depends on the attributes you want to store for each point.
- If your raw data is **LAS 1.4** and you need to deliver in **LAS 1.4**, use **LAS 1.4**. Otherwise, it's recommended to use **FastBinary** for processing and export into **LAS 1.4** for data delivery.
- If you want to store the data in compressed files, use an **LAZ 1.x** format. This reduces storage space requirements significantly and thus, is a good option for data delivery and archiving.
- You can store **Riegl Extra Bytes**, such as **Pulse width** and **Amplitude** attributes, in FastBinary, LAS 1.2+, or LAZ 1.2+ formats. The formats store the Riegl **Pulse width** as **Echo length** attribute and the Riegl **Amplitude** as **Parameter** attribute.
- You can store **Riegl Extra Bytes**, such as **Reflectance**, **Amplitude** and **Pulse shape deviation** attributes, in FastBinary, LAS 1.4, or LAZ 1.4 formats. The formats store the Riegl **Reflectance** as **Reflectance** attribute, the Riegl **Amplitude** as **Parameter** attribute and the Riegl **Pulse shape deviation** as **Deviation** attribute.

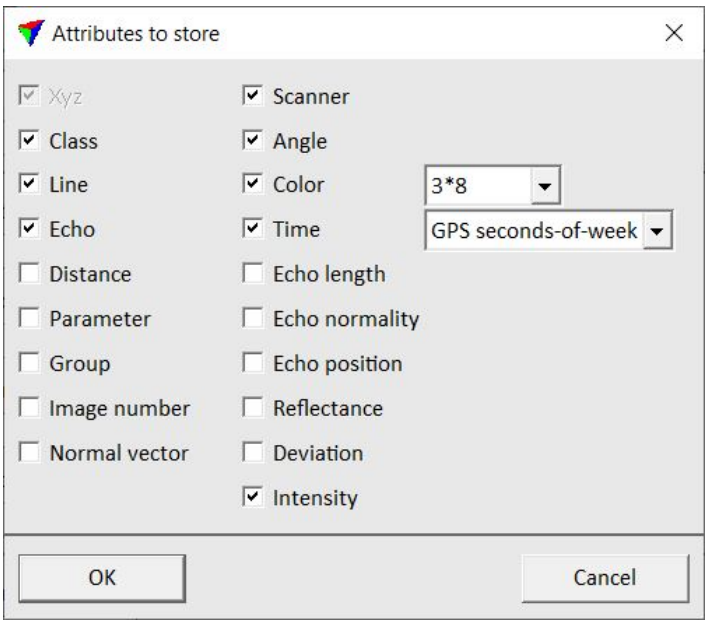
Attributes to store

The dialog defines what attributes are stored for each laser point that belongs to the project. Only active attributes are stored. It depends on the selected storage format which attributes are available.

When a new project is created, you normally active attributes that are stored in the raw laser files and which you want to keep in the block binary files. You may also activate additional attributes if you already know that you need them later. However, the selection of active attributes can be modified later by using the Edit project information command.

In addition to the attributes, the dialog defines the format of time stamps for the block binary files. If the selected format differs from the time stamp format of the raw laser files, the time stamps are converted into the format defined in the project definition. The conversion is done during the import of the raw laser files into the project.

If color values are stored for the block binary files, another selection is required for the amount of color channels. TerraScan can extract up to 10 color channels for each point. The maximum amount of color channels can only be stored in the TerraScan FastBinary format. LAS 1.4 format and later can store up to 4 color channels, LAS 1.2 format and later up to 3. If more than 3 channels are extracted, color values are stored as 16-bit values.



File locking

TerraScan supports a simple way of file locking for block binary files. The file locking should prevent two people from modifying the same data at the same time in a work group environment.

The file locking is active if **Require file locking** is switched on in the project definition. This is only needed if you have multiple people working on the same data set in a network.

TerraScan implements file locking in a simple manner. It does not lock the laser file itself. It creates a temporary file which shows that the laser file is undergoing modification. The temporary file has the same name and storage location as the block binary file and has the extension .LCK. This temporary file contains the name of the computer which has opened the laser file for modification. The creation time of the temporary file shows when the laser file was opened.

This relaxed locking method does not completely prevent modification of a laser file that has been locked. Only most of the TerraScan tools refuse to modify the data. Other applications do probably not recognize that the laser file is locked by TerraScan.

Example case

When file locking is active, the project storage directory might contain the following files:

BLOCK BINARY FILE	TEMPORARY LOCK FILE
h:\otaniemi\ota000317.fbi	h:\otaniemi\ota000317.lck
h:\otaniemi\ota000318.fbi	
h:\otaniemi\ota000319.fbi	h:\otaniemi\ota000319.lck
h:\otaniemi\ota000320.fbi	

The above files would indicate that blocks 000317 and 000319 are modified and locked.

Operations supporting file locking

The following actions lock a block binary file:

- Open block command if **Open for** is set to **Modification**.
- Read points command when opening a single block binary file.
- Load Airborne Points tool when opening a single block binary file.
- Load Ground Points tool when opening a single block binary file.
- Executing Adjust to geoid from the **Project** window.
- Executing Run macro from the **Project** window with a **Save points** setting on.

The same actions also check whether a laser file is locked or not.

Releasing a lock

The **Project** window has a Release lock command in the **Block** pulldown menu for releasing a locked block. The command releases a lock only if a file has been locked on the same workstation or if the locking was done more than 24 hours ago.

As a last precaution, the lock can be released by deleting the .LCK file.

Open project

Not UAV

Open project command opens an existing project definition.

To open a project:

1. Select **Open project** command from the **File** pulldown menu.

   This opens the **Open project** dialog, a standard dialog for selecting files.

2. Select a project file and click **Open**.

   This loads the selected project into TerraScan.

Save project

Not UAV

Save project command saves changes of the project definition to an existing project file. This can be used after changing project settings or block definitions.

To save changes to a project:

1. Select **Save project** command from the **File** pulldown menu.

This saves the project.

Save project as

Not UAV

Save project as command saves a project definition by creating a new project file. The project file is an ASCII file with the extension .PRJ. This can be used after creating a new project definition.

To save a project into a new project file:

1. Select **Save project as** command from the **File** pulldown menu.

   This opens the **Save project** dialog, a standard dialog for saving files.

2. Select a storage location and type a name for the project file

3. Click **Save**.

   This saves the project file.

Sort pulldown menu

Commands from **Sort** pulldown menu in **Project** window are used to display the list of blocks in a specific order. The order can be stored in the project file.

COMMAND	EFFECT
By name	Sort alphabetically ascending by block name.
By number	Sort ascending by last number in the block name.
By point count	Sort ascending by amount of points in the block binary files.
North to south	Sort by geographical location, north to south and secondarily west to east.
South to north	Sort by geographical location, south to north and secondarily west to east.
West to east	Sort by geographical location, west to east and secondarily south to north.
East to west	Sort by geographical location, east to west and secondarily south to north.

Tools pulldown menu

Commands from the **Tools** pulldown menu in the **Project** window are used to perform different actions on block binary files or based on block binary files on project level.

TO	USE COMMAND
Run a macro on a project	Run macro
Adjust the elevation of block files to a geoid model	Adjust to geoid
Adjust XYZ coordinates of block files	Adjust xyz
Convert project block binary files into another format	Convert storage format
Check the z accuracy of block files	Output control report
View statistics about points in block files	Show statistics
Check the coverage of block files	Check coverage
Validate block boundaries	Validate blocks
Merge small blocks automatically into one block	Merge small blocks
Copy points into block files from a reference project	Copy from reference
Assign color values from images to laser points in block files	Extract color from images
Assign echo properties to laser points	Extract echo properties
Export laser data from block files into lattice files	Export lattice models
Export laser data from block files into raster images	Export raster images
Export a 3D point cloud from ortho images	Export 3D ortho
Output collections from block files	Output collections
Draw shape elements for lines	Draw line boundaries

Adjust to geoid

Not UAV

Adjust to geoid command adjusts the elevation values of the block binary files to a local elevation model. The geoid model can be defined by a text file, a TerraModeler surface, or a selected linear chain.

The theory of geoid adjustment and the use of the command in TerraScan are explained in detail in section Geoid adjustment.

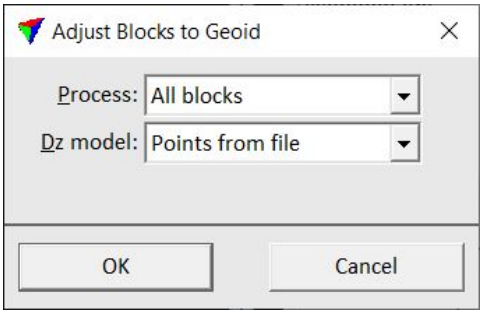
The command performs the same action on block binary files as the Adjust to geoid command on loaded points.

To run elevation adjustment on project blocks:

1. (Optional) Select block definitions in the **Project** window to be adjusted.

2. Choose **Adjust to geoid** command from the **Tools** pulldown menu.

   This opens the **Adjust Blocks to Geoid** dialog:



3. Define settings and click OK.

   This starts the adjustment process. The process effects the block binary files directly on the hard disc. It might be advisable to create a backup copy of the block binary files before starting the process.

SETTING	EFFECT
Process	Blocks to adjust: **All blocks** or **Selected blocks**.
Dz model	Input model for geoid corrections: **Points from file**, **Selected linear chain**, a specific surface model that is active in TerraModeler.
Extend	Distance from a selected linear element by which the linear chain is extended for elevation value corrections. This is only active if **Dz model** is set to **Selected linear chain**.

Adjust xyz

Not UAV

Adjust xyz command applies a varying XYZ correction to the block binary files. The correction model is defined by a text file containing rows with five fields: X Y dX dY dZ where X and Y define a fixed coordinate location and dX dY dZ define the change of XYZ coordinates at this fixed XY location.

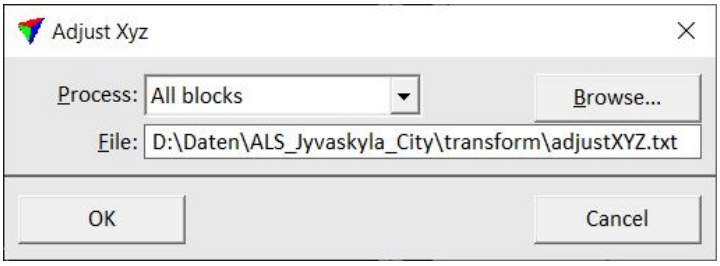
The command performs the same action on block binary files as the Transform loaded points command with setting **Transform = Dxyz** on loaded points.

To adjust block binary file coordinates to a varying XYZ correction model:

1. (Optional) Select block definitions in the **Project** window to be adjusted.

2. Choose **Adjust xyz** command from the **Tools** pulldown menu.

This opens the **Adjust Xyz** dialog:



3. Define settings and click OK.

   This starts the adjustment process. The process effects the block binary files directly on the hard disc. It might be advisable to create a backup copy of the block binary files before starting the process.

SETTING	EFFECT
Process	Blocks to adjust: **All blocks** or **Selected blocks**.
File	Text file that stores the correction values.

Check coverage

Not UAV

Check coverage command finds holes and areas of low point density in the project binary files. The area to be covered can be either defined by one or more selected polygons or by all block boundaries of the project. The application calculates the point density within sample areas and decides to which coverage level an area belongs.

The point coverage is defined in four levels: **Covered**, **Almost covered**, **Almost hole** and **Hole**. User settings define the point densities which are interpreted as being holes or as being fully covered. The command can create different output products in order to illustrate the point coverage. The options include:

- the creation of a density raster image showing the point coverage in a TIFF file. The raster images get the same name as the original block binary files.
- the creation of points in a uniform grid structure that fill the holes and areas of low point density. Three different classes can be used to separate the points according the coverage level. The elevation of the artificial points is set to be equal to the number of points found in the sampling area. The points are saved in TerraScan binary files which get the same name as the original block binary files. A project file with the name DENSITY.PRJ is created that references the binary files with the artificial point.
- the output of the density calculation results in a report window. You can save a text file from the report window.

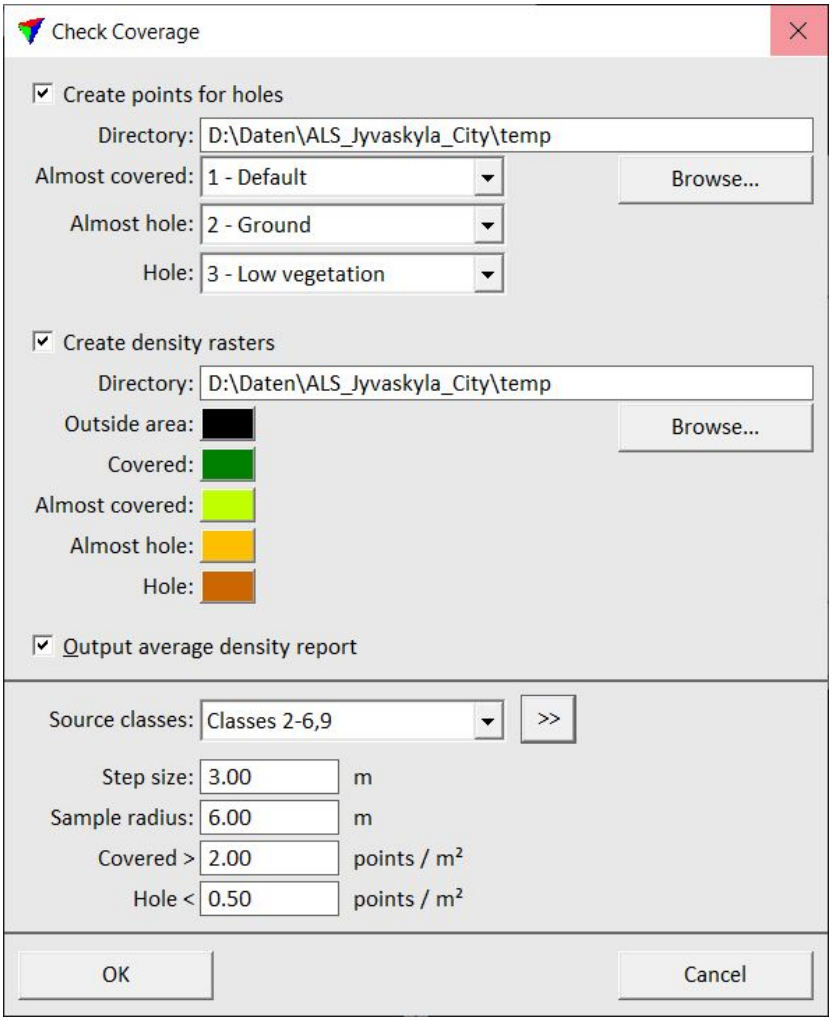
To check the laser point coverage in a project area:

1. (Optional) Draw and select one or more polygons around areas for which you want to check the coverage.

2. Select **Check coverage** command from the **Tools** pulldown menu.

3. If no polygons are selected, the application informs you that it uses all block boundaries for the check. Click OK.

   This opens the **Check Coverage** dialog:



4. Define settings and click OK.

   This starts the calculation of the point densities and the coverage. The software creates the files according to the settings in the given directories. An information window shows the progress of the process.

SETTING	EFFECT
Create points for holes	If on, points are created that fill holes, almost hole and almost covered areas.
Directory	Directory for storing the TerraScan binary files and the project file for the artificial points. This is only active if **Create points for holes** is switched on.

SETTING	EFFECT
Almost covered	Class for artificial points in almost covered areas. This is only active if **Create points for holes** is switched on.
Almost hole	Class for artificial points in almost hole areas. This is only active if **Create points for holes** is switched on.
Hole	Class for artificial points in holes. This is only active if **Create points for holes** is switched on.
Create density raster	If on, density raster images in TIFF format are created.
Directory	Directory for storing the density raster images. This is only active if **Create density raster** is switched on.
Outside area	Color in density raster images for areas outside the covered area. This is only active if **Create density raster** is switched on.
Covered	Color in density raster images for covered areas. This is only active if **Create density raster** is switched on.
Almost covered	Color in density raster images for almost covered areas. This is only active if **Create density raster** is switched on.
Almost hole	Color in density raster images for almost hole areas. This is only active if **Create density raster** is switched on.
Hole	Color in density raster images for holes. This is only active if **Create density raster** is switched on.
Output average density report	If on, a report window is displayed showing the amount of points per measurement area and the point density.
Source classes	Point class(es) that are included in the coverage calculation.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Source classes** field.
Step size	Defines the steps at which the point density is analyzed. Determines also the pixel size of the density raster images and the distance

SETTING	EFFECT
	between the artificial points that are created to fill holes and areas of low density.
Sample radius	Radius of the sample area from which to calculate the local point density.
Covered >	Point density required for an area being considered as covered.
Hole <	Point density for an area that will be considered as hole.

Since the file names of the files created by the process are fixed, existing files are overwritten without warning if the command is performed a second time on a project with the same directory settings.

Convert storage format

Not UAV

Convert storage format command converts project block binary files into another binary format. It does not keep the original binary files. At the same time, it modifies and saves the project definition according to the new binary file format.

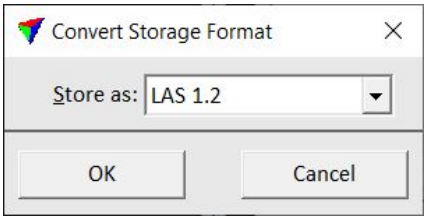
Before you convert block binary files into a new format, make sure that you do not loose any important attributes stored in the original binary files. For instance, if converting from **FastBinary** format to **LAS**, specific attributes, such as normal vector, group number, or even the color (LAS 1.0 or 1.1), are no longer available.

It is recommended to create a backup copy of the original project and block binary files before running the **Convert storage format** command.

To convert the storage format of project binary files:

1. Select **Convert storage format** command from the **Tools** pulldown menu.

   This opens the **Convert Storage Format** dialog:



2. Select a new format from the **Store as** list. The list contains all supported binary formats for project blocks. See Supported file formats for more information.

3. Click OK.

This starts the conversion process which effects the original block binary files and project file.

Copy from reference

Not UAV

Copy from reference command copies attributes of laser points from a reference project to the laser points of the active project. The command requires the definition of a reference project in the project information settings of the active project. See Edit project information for more information about defining a reference project.

There has to be a possibility to uniquely identify a point by its attributes in the active and reference project binary files in order to ensure that the correct attributes are copied for a point.

For LAS files, the combination of time stamp + echo information is unique for each laser point if all the data was captured in one GPS week or if GPS standard time stamps are used. Otherwise, the combination of line + time stamp + echo information defines a laser point unambiguously.

For TerraScan Binary and FastBinary files, the combination of time stamp + echo information is not unique for each laser point because of the resolution with which time stamps are stored in this format. Therefore, additional attributes that are the same in the active and reference project binary files have to be selected to identify laser points unambiguously.

An example case for using this command could be:

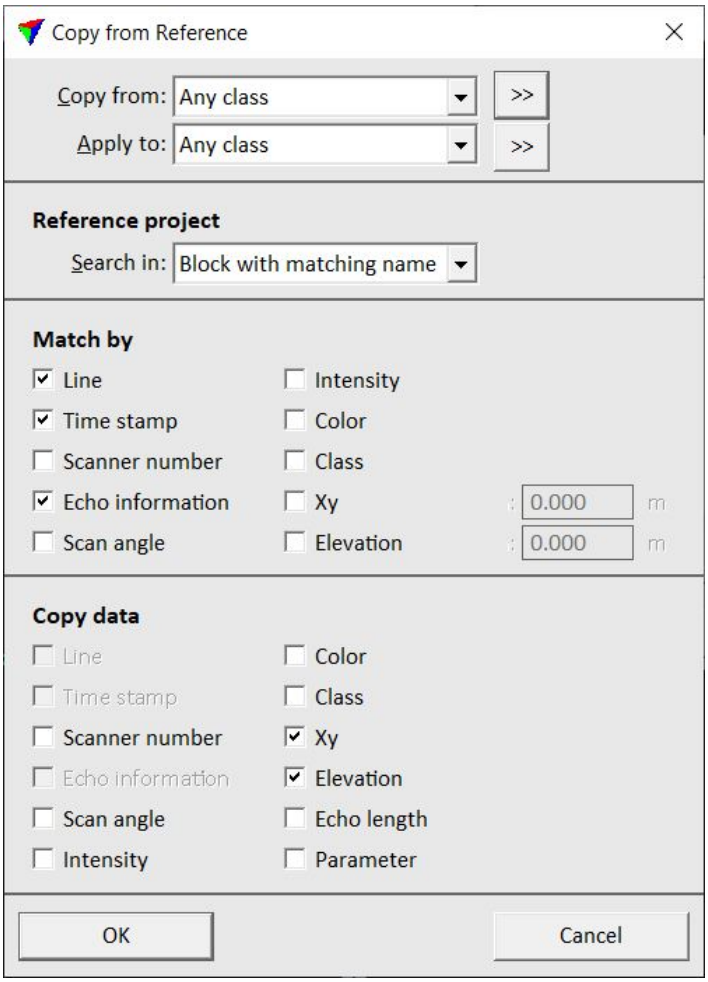
- Laser data has been imported into a project in folder *laser1*.
- Heading, Roll, Pitch (HRP) misalignment has been solved and applied to the laser points that are now stored in folder *laser2*.
- Automatic and manual classification steps have been performed.
- After the classification it has been realized that the HRP correction was wrong but the classification is good and took a lot effort.
- As a solution, the project in folder *laser1* is defined as reference project in the information dialog of the project in folder *laser2*.
- The attributes xy and z are copied from laser data in *laser1* to the data in *laser2* using the **Copy from reference** command.

This restores the coordinate values from the status before the HRP correction but preserves the classification that was done after the HRP correction.

To copy attributes from a reference project:

1. Select **Copy from reference** command from the **Tools** pulldown menu.

   This opens the **Copy from Reference** dialog:



2. Define settings and click OK.

   This copies the selected attributes to the points of the active project. An information window shows the progress of the process.

   After the process finished, a report is displayed that shows the number of points that have been changed in each block binary file. The report can be saved as a text file or printed directly using commands from the **File** pulldown menu in the **Copy parameters** window.

SETTING	EFFECT
Copy from	Point class(es) from which the attributes are copied.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Copy from** field.
Apply to	Point class(es) to which the attributes are copied.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can

SETTING	EFFECT
	select multiple source classes from the list that are then used in the **Apply to** field.
Search in	Method how corresponding blocks are searched in the reference project: • **Block with matching name** - the block names in the reference project are supposed to be the same as in the active project. • **Blocks close in xy** - the block boundaries of the reference project are supposed to be at the approximately same location as the block boundaries of the active project. The software finds matching points in the active and reference project blocks also if they are not exactly at the same XY location.
Match by	If on, the attribute is used to find matching points in the reference and active projects.
Copy data	If on, the attribute is copied from the laser points in the reference project to the corresponding laser points in the active project.

Draw line boundaries

Not UAV

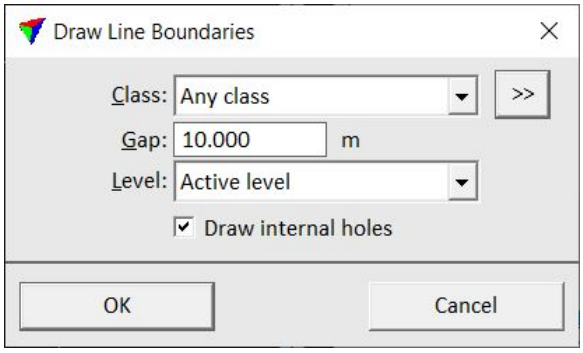
Draw line boundaries command draws shape elements for areas that are covered by single lines. The lines are identified by the line number attribute stored for laser points. In addition, the class attribute can be used to further limit the area enclosed by the shape.

The shape elements may be useful to ensure that there are no gaps in the data set or to test if data from specific lines can be removed without causing a gap. The process can draw the shapes on different CAD file levels for each line. This supports the check of line removal without the need to delete the elements.

To draw line boundaries:

1. Select **Draw line boundaries** command from the **Tools** pulldown menu.

   This opens the **Draw Line Boundaries** dialog:



2. Define settings and click OK.

   The software checks all blocks of the project and draws the line boundaries into the CAD file. The color of the shape elements is determined by the line number, line style and weight by the active symbology settings of the CAD platform. After drawing the shapes on separate levels, a MicroStation design file is compressed automatically.

SETTING	EFFECT
Class	Point class(es) that are included in the shape computation.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Class** field.
Gap	A new shape element is drawn if there is a gap larger than the given size between points of the same line.
Level	CAD file level(s) on which the shape elements are drawn: • **Active level** - shapes for all lines are drawn on the active level. • **Separate levels** - shapes for each line are drawn on a separate level. The levels for the shapes are created automatically with the name <LineX> where X is the line number.
Draw internal holes	If on, areas within lines without points larger than the given **Gap** size are enclosed by boundaries. They are defined as holes within the shape element for the line.

Export 3D ortho

Not UAV

Export 3D ortho command generates a point cloud from orthophotos. The process combines orthophoto pixel location and color, and laser point coordinates in order to create a high-density

colored point cloud. The resulting point cloud contains one point for each orthophoto pixel with the following attributes:

- **XY coordinates** - computed from the center of orthophoto pixels or used from the original laser points.
- **Z coordinate** - computed from a TIN generated from laser data or used from the original laser points.
- **Class** - defined by the source class in the laser data.
- **RGB color values** - determined by the pixel color, the laser point color, or a fixed color.

The command requires TerraScan and TerraPhoto running on the same computer. The orthophotos must be attached as TerraPhoto raster references in order to create the 3D point cloud. In addition, the laser data in the block binary files should be classified in order to distinguish point elevations on the ground, vegetation, building roofs, etc. This allows you to define different rules for the point cloud generation depending on the object types.

The process can also include vector elements in the point cloud computation, such as 3D building models or other 3D shapes.

The process creates a new TerraScan project file and block binary files for each orthophoto inside the area covered by blocks of the original laser project. The names of the blocks and binary files are determined by the names of the orthophotos. The point density and the amount of points per block binary file of the generated point cloud is determined by the pixel resolution of the orthophotos but also by the rules for the point cloud creation.

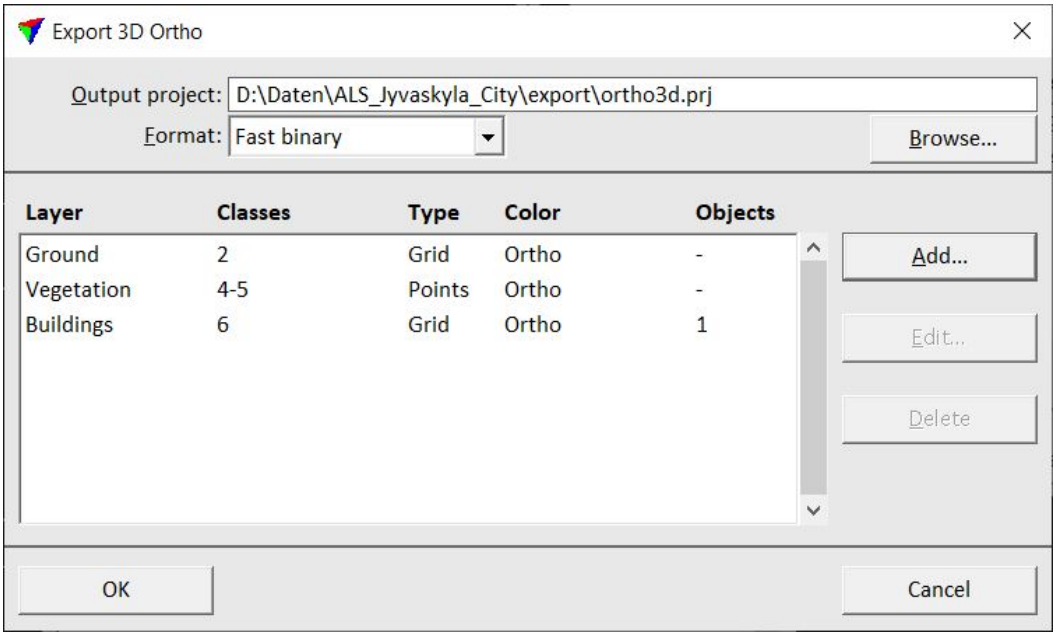
To export a 3D ortho point cloud:

1. Attach orthophotos by using the Manage Raster References tool of TerraPhoto.

2. Open the TerraScan project that references the classified laser binary files.

3. Select **Export 3D ortho** command from the **Tools** pulldown menu.

   This opens the **Export 3D Ortho** dialog:

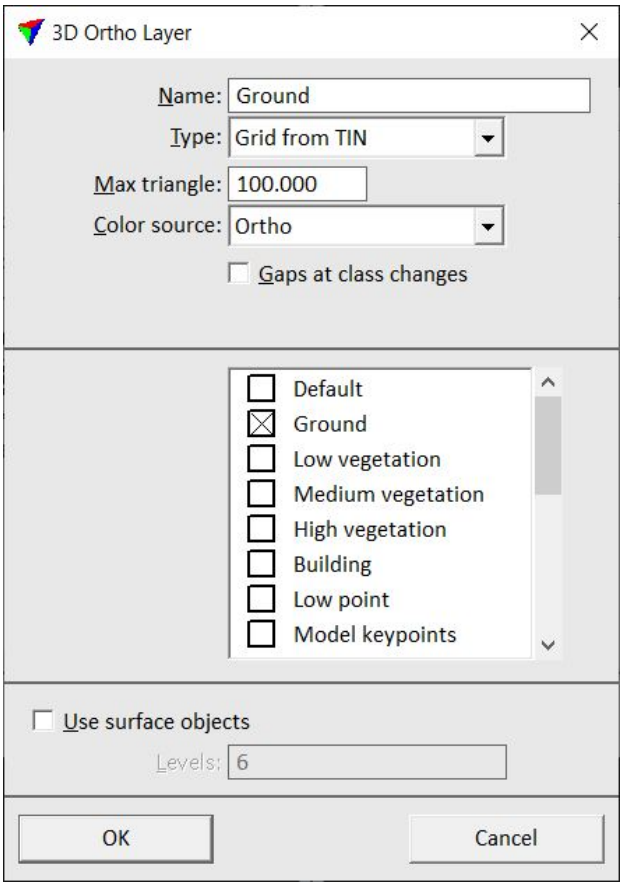


4. Define a storage directory and name for the **Output project**. The block binary files are stored in the same folder as the project file. Click on the **Browse** button in order to define the output project in a standard dialog.

5. Select a file format for the block binary files: **FastBinary** or **LAS 1.2**.

6. Click on the **Add** button in order to define rules for the export of different layers, such as ground, vegetation, or buildings.

   This opens the **3D Ortho Layer** dialog:



7. Define settings and click OK.

   You can **Edit** and **Delete** layers by using the corresponding buttons in the **Export 3D ortho** dialog.

8. Click OK in the **Export 3D Ortho** dialog.

   This starts the point cloud generation process. A progress window displays the progress of the process.

SETTING	EFFECT
Name	Descriptive name of the layer.
Type	Defines how points are extracted for the export: • **Grid from TIN** - a grid is extracted from a TIN created from the laser points. The XY location

SETTING	EFFECT
	of a grid point is determined by the pixel center, the elevation by the TIN. • **Points directly** - the original laser points are exported.
Max triangle	Maximum length of a triangle edge in the TIN. Determines how big gaps are filled with points generated by aerial triangulation. This is only active if **Type** is set to **Grid from TIN**.
Color source	Defines the source for extracting RGB values for the points: • **Ortho** - each point gets the color of the closest pixel in the orthophotos. • **Point color** - each point gets the color assigned to laser points. • **Fixed color** - each point gets a fixed color value.
Gaps at class changes	If on, a gap is enforced at boundaries between different classes in the source laser data. This avoids that point are generated by aerial triangulation between different point classes.
R G B	RGB color values of a fixed color assigned to points. This is only active if **Color source** is set to **Fixed color**.
List of classes	Select the source class(es) for this layer. The list contains the active classes in TerraScan.
User surface objects	If on, shape elements on the given **Levels** are used for determining the elevation of exported points inside a shape area.

The TerraScan project file for the exported point cloud is created with the **Color** attribute inactive. Therefore, you have to switch the attribute on and save the project before you can see the points displayed by color correctly. See Edit project information for instructions how to edit a project.

Export lattice models

Not UAV

Export lattice models command creates grid files with uniform distances between points from one or more selected laser point classes. For each grid point, the lattice model file stores XY coordinates and one of the following value types:

- elevation values
- point count/density values
- distance values computed for points with the Compute distance command or corresponding macro action

- analytical values, such as average intensity, horizontal or vertical distance, surface roughness

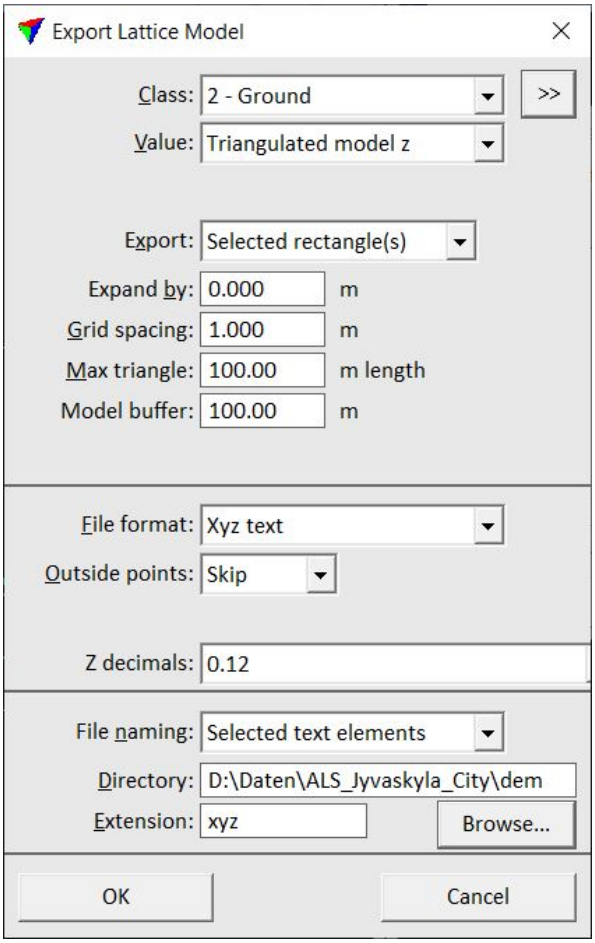
There are several formats supported to store lattice models as raster or text files. The command requires the selection of at least one polygon that defines the lattice model boundary. If several polygons are selected, the software creates a separate lattice model file for each polygon. Text elements placed inside the polygon(s) can be used as file names for the lattice model files. A polygonal area is always expanded to a rectangular area of a lattice model. The cells outside the polygon are filled with a defined "outside" value or can be skipped from output in XYZ text files.

To export a lattice model:

1. Draw rectangle(s) around area(s) from which you want to create lattice model(s). (Optional) Place text elements inside the rectangles. Select rectangle(s) and (optional) text(s).

2. Select **Export lattice model** command from the **Tools** pulldown menu.

   This opens the **Export Lattice Model** dialog:



3. Define settings and click OK.

   This starts the lattice model file creation.

   If **File naming** is set to **Enter name for each**, the **Export lattice model** dialog opens, a standard dialog for saving files.

4. Define a location and name for the lattice file and click **Save**.

Repeat step 4 for each lattice model.

SETTING	EFFECT
Class	Source class(es) for lattice model creation.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Class** field.
Value	Value stored for each grid point of a lattice model: • **Triangulated model z** - elevation value calculated from a TIN model of the points in the source class(es). • **Highest hit z** - elevation value determined by the highest point in the source class(es). • **Average hit z** - elevation value calculated as average of points in the source class(es) falling inside the grid cell. • **Lowest hit z** - elevation value determined by the lowest point in the source class(es). • **Closest hit z** - elevation value determined by the point closest to the grid cell center in the source class(es). • **Point count** - amount of points falling inside the cell. • **Point density** - amount of points per squared master unit. • **Average intensity** - average intensity value of points in the source class(es) falling inside the grid cell. • **Distance to point** - average horizontal distance between a grid point and the two closest points in the source class(es). • **Dz from ground** - vertical distance between a grid point and a surface model created from the Ground class(es). • **Surface roughness** - difference of a grid point from a plane fitted to the closest points in the source class(es). Represents the local elevation variation of points in the source class(es). • **Biggest distance** - biggest distance value inside the grid cell. Requires that a distance value is computed for each point using the command for loaded points or corresponding macro action. • **Average distance** - average distance value of points inside the grid cell. Requires that a distance value is computed for each point

	using the command for loaded points or corresponding macro action. • **Smallest distance** - smallest distance value inside the grid cell. Requires that a distance value is computed for each point using the command for loaded points or corresponding macro action.
Ground	Source class(es) for creating a surface model. This is only active if **Value** is set to **Dz from ground**.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Ground** field.
Export	Area to be exported into lattice model file(s): • **Whole area** - not available. • **Selected rectangle(s)** - the area inside selected polygon(s) is exported into separate files.
Expand by	Distance by which a selected polygon is expanded for the model export. The area covered by the expanded polygon is included in a lattice model file.
Grid spacing	Defines the distance between grid points and thus, the resolution of the lattice model.
Max triangle	Maximum length of a triangle edge for TIN creation. Effects how big gaps are filled in the lattice model by interpolating grid point values from the TIN. This is only active if **Value** is set to **Triangulated model z** or **Distance to point**.
Model buffer	Width of a buffer area around the actual model area that is considered for calculating grid point values. This is only active if **Value** is set to **Triangulated model z**, **Distance to point**, **Dz from ground**, or **Surface roughness**.
Fill gaps up to	Defines the size of gaps that are filled in the lattice model by deriving grid point values from closest points in the source class(es). This is only active if **Value** is set to **Highest|Average|Lowest hit z**, **Average intensity**, or **Surface roughness**.
Conserve memory	If on, the software first determines how many points will be loaded and thus, how much memory needs to be allocated for the exact number of points. This slows down the process but it is less likely to run out of memory.

File format	Format of the lattice model file: **ArcInfo**, **GeoTIFF**, **Intergraph GRD**, **Raw**, **Surfer ASCII** or **binary**, **Xyz text**.
Z unit	Unit of grid point values. Relevant for formats storing elevations as integers. This is only active if **File format** is set to **GeoTIFF**, **Intergraph GRD**, or **Raw**.
Outside points	Defines how the software handles grid cells that are not covered by points in the source class(es): **Skip** or **Output**. This is only active if **File format** is set to **Xyz text**.
Outside Z	Defines the value for grid cells that are inside the rectangular lattice model area but not covered by points in the source class(es). This is only active if **File format** is set to **ArcInfo**, **GeoTIFF**, **Surfer ASCII** and **binary**, **Xyz text**.
Z decimals	Determines the number of decimals stored for the grid point value. This is only active if **File format** is set to **ArcInfo**, **Surfer ASCII**, or **Xyz text**.
Create TFW files	If on, the software creates external georeference files for GeoTIFFs. This is only active if **File format** is set to **GeoTIFF**.
File naming	Defines how lattice model files are named: • **Enter name for each** - a name for each lattice model file has to be defined manually. • **Selected text elements** - selected text elements inside the lattice model area are used as file names.
Directory	Directory for storing lattice model files. Click on the **Browse** button in order to select a folder in the **Browse For Folder** dialog. This is only active if **File naming** is set to Selected **text elements**.
Extension	Extension of lattice model files. There may be a specific extension required for a certain file format, such as .GRD for ArcInfo or .TIF for GeoTIFF. This is only active if **File naming** is set to **Selected text elements**.

Lattice models can be produced by using

- Export lattice models command for loaded points.
- Export lattice macro action.
- Produce lattice models command in TerraModeler

Export raster images

Not UAV

Export raster image command generates raster images where pixel values are derived from laser point attributes. The source data is points referenced by a TerraScan project.

The raster image can be created as Windows bitmap (.BMP) or GeoTIFF (.TIF). The color of a pixel is determined using laser points whose coordinate values fall inside the pixel. The coloring attribute can be chosen as:

- **Elevation** - laser point elevation.
- **Elevation difference** - elevation difference between laser points of two different classes.
- **Intensity hits** - laser point intensity.
- **Intensity footprint** - average intensity of laser points within a footprint area overlapping the pixel.
- **Point color** - color values stored for laser points.
- **Point class** - laser point class.
- **Road intensity** - intensity of laser points computed by directional sampling along road alignment vectors.
- **Slope** - slope gradient based on normal vector computation.

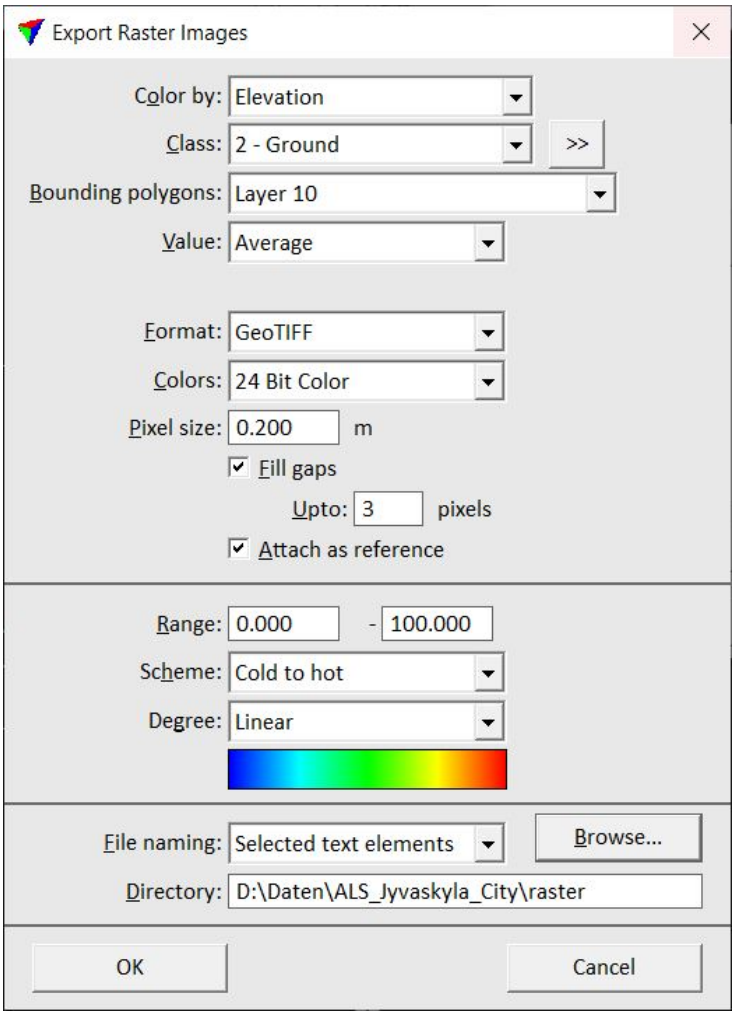
The boundary of each raster file must be defined as a rectangle drawn into the CAD file. The rectangle(s) must be selected. Text strings placed inside the rectangles can be used as file names for the output files.

To create colored raster images:

1. Select rectangles as raster file boundaries and (optional) text strings inside the boundaries.

2. Select **Export raster images** command from the **Tools** pulldown menu.

   This opens the **Export Raster Images** dialog:



3. Define settings. You may define your own coloring scheme by using the **Define** button.

4. Click OK.

   This starts the raster file generation.

   If **File naming** is set to **Enter name for each**, the **Export raster image** dialog opens, a standard dialog for saving files.

5. Define a location and name for the raster file and click Save.

   Repeat step 5 for each lattice model.

   The software creates a raster file for each selected rectangle.

SETTING	EFFECT
Color by	Coloring attribute. See description above.
Class	Point class(es) to use for creating the raster file. If **Color by** is set to **Elevation difference**, two classes have to be selected.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can

SETTING	EFFECT
	select multiple source classes from the list that are then used in the **Class** field.
Bounding polygons	Defines the area filled with values in the output raster files. Pixels outside the polygons get black as a background value.
Alignments	Level on which the road alignment vectors are located. This is only active if **Color by** is set to **Road intensity**.
Value	Determines the value for each pixel: • **Lowest** - smallest value of points inside the pixel area. • **Average** - average value of points inside the pixel area. • **Highest** - highest value of points inside the pixel area. This is only active if **Color by** is set to **Elevation**, **Elevation difference**, or **Intensity hits**.
Sampling	Computation of raster values based on directional sampling: • **Road intensity** - distance along the road alignment element(s). • **Slope** - radius around a point.
Format	Format of the output file: **Windows BMP**, **GeoTIFF**, or **GeoTIFF + TFW**.
Colors	Color depth of the output file for elevation or intensity coloring: • **24 Bit Color** - true color image. • **256 Colors** - 256 color image. • **Grey Scale** - 8 bit gray scale image.
Pixel size	Size of each pixel in the output file.
Fill gaps	If on, gaps up to the given number of pixels are filled in places where there are no laser points inside a pixel area.
Attach as reference	If on, the output image is attached as a raster reference. This requires TerraPhoto to run.
Range	Defines the value range that is covered by the color scheme for **Elevation** and **Intensity** coloring. Should be set to the general elevation or intensity range covered in the laser data to ensure that all values are represented by the complete color scheme.

SETTING	EFFECT
Unit	Defines the unit for **Slope** coloring: **Degree** or **Percentage**.
Scheme	Type of coloring scheme for elevation or intensity coloring: • **Cold to hot** - varies from blue for low pixel values via cyan, green, and yellow to red for high pixel values. This is a common coloring scheme for elevation coloring. • **Hot to cold** - varies from red for low pixel values via cyan, green, and yellow to blue for high pixel values. • **Selected colors** - a user-defined coloring scheme can be created by clicking on the **Define** button. The dialog for color scheme definition depends on the selection of the color depth in the **Colors** field, 24-Bit Color or 256 Colors. • **Black to white** - varies from black for low pixel values to white for high pixel values. This is only active if **Colors** is set to **Grey scale**. This is the common coloring scheme for intensity coloring. • **White to black** - varies from white for low pixel values to black for high pixel values. This is only active if **Colors** is set to **Grey scale**.
Degree	Determines how the color changes in color schemes are computed. **Warm** and **Hot** move a coloring scheme towards the red-yellow color range, **Cool** and **Cold** towards the blue-cyan color range. For gray scale images, **Light** moves the gray scale towards white and light gray, **Dark** towards black and dark gray. **Linear** defines a linear distribution of colors.
File naming	Defines how raster files are named: • **Enter name for each** - a name for each file has to be defined manually. • **Selected text elements** - selected text elements inside the raster image boundary are used as file names.
Directory	Directory for storing raster image files. Click on the **Browse** button in order to select a folder in the **Browse For Folder** dialog. This is only active if **File naming** is set to Selected **text elements**.

Raster images can be produced based on points loaded in TerraScan by using the Export raster image command.

Extract color from images

Not UAV

Extract color from images command extracts RGB color values from raster images and assigns the color values to laser points in block binary files. The color sources can be orthophotos attached as TerraPhoto raster references or raw images in an active TerraPhoto image list. In addition, a color point file can be used to balance colors of the raw images before the color values are assigned to the laser points. The command requires TerraPhoto or TerraPhoto Lite running on the same computer.

TerraScan can extract up to 10 color channels for each point. The maximum amount of color channels can only be stored in the TerraScan FastBinary format. LAS 1.4 format and later can store up to 4 color channels, LAS 1.2 format and later and TerraScan Binary files up to 3.

The color for a laser point is derived by resampling the color values of all the pixels inside a circular area around the point. There are different methods of color value extraction from raw images which are either suitable for airborne or mobile data sets.

The process can involve the computation and storage of image numbers. The number of the image used for extracting the color can be stored for each laser point. This requires the storage of points in TerraScan FastBinary format. The image number stored as laser point attribute is required for advanced coloring methods for mobile point clouds.

The command performs the same action on block binary files as the Extract color from images command on loaded points.

To extract color from attached orthophotos or raw images:

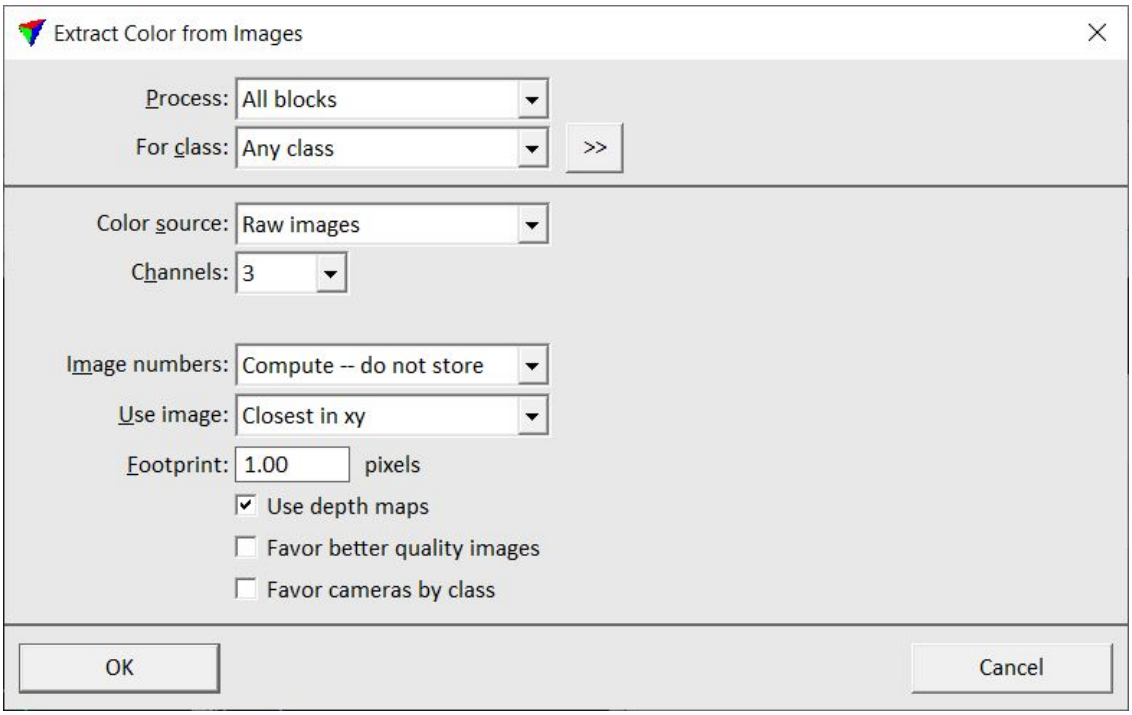
1. Attach reference images in TerraPhoto's Manage Raster References tool.

   OR

1. Load an image list into TerraPhoto.

2. Select **Extract color from images** command from the **Tools** pulldown menu.

   This opens the **Extract Color from Images** dialog:



3. Define settings and click OK.

   This derives color values for the laser points from the defined source images.

SETTING	EFFECT
Process	Blocks to process: **All blocks** or **Selected blocks**.
For class	Laser point class(es) for which colors are extracted.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **For class** field.
Color source	Source files for color extraction: • **Ortho images** - colors are extracted from attached TerraPhoto raster references. • **Raw images** - colors are extracted from raw images in an active image list in TerraPhoto. • **Raw images & color points** - colors are extracted from raw images and from a color point file.
Channels	Amount of color channels to be extracted.
Cpt file	Location and name of a color point file. This is only active if **Color source** is set to **Raw images & color points**.

SETTING	EFFECT
Image numbers	Method how the software handles the computation and storage of image numbers from raw images: • **Compute -- do not store** - image numbers are computed but not stored for laser points. • **Compute and store** - image numbers are computed and each laser point stores the number of the image from which it gets the color. • **Use stored** - stored image numbers of laser points are used for color extraction.
Use image	Method how the software determines the raw image for extracting a color for a laser point: • **Closest in 3d** - closest camera XYZ position. Optimized for airborne data sets. • **Closest in xy** - closest camera XY position. Optimized for airborne data sets. • **Closest in time** - closest time stamp. Optimized for airborne data sets. • **Mobile -- closest in time** - closest time stamp. Optimized for mobile data sets. • **Mobile -- ground surface** - best ground surface visibility. Optimized for mobile data sets. • **Mobile -- closest in 3d** - closest camera XYZ position. Optimized for mobile data sets.
Footprint	Radius of a circular area around each laser point within which pixel color values are resampled. Given in meters for a method optimized for airborne data sets and in pixels for mobile data sets. This is the only active if **Color source** is set to **Ortho images**.
Max distance	Maximum distance between a raw image and a laser point. Images outside that distance are not considered for color extraction. This is only active if **Use image** is set to any method optimized for mobile data sets.
Max time diff	Maximum time difference between a raw image and a laser point. Images outside that difference are not considered for color extraction. This is only active if **Use image** is set to **Mobile -- closest in time**.
Use depth maps	If on, depth maps files are included in the color extraction process. See **TerraPhoto User**

SETTING	EFFECT
	Guide for more information about depth maps.
Favor better quality images	If on, the quality attribute stored for raw images in an image list is considered in the color extraction process. See **TerraPhoto User Guide** for more information about the quality attribute.
Favor cameras by class	If on, the settings in the TerraPhoto mission file related to favoring cameras for coloring points are considered in the color extraction process. See **TerraPhoto User Guide** for more information about the mission definition.

Extract echo properties

Not UAV

Extract echo properties command extracts information from waveform data and assigns it as attributes to the laser points. The command requires that waveform data and a scanner waveform profile are available. The processing steps for preparing the extraction of waveform-related information are described in detail in Chapter Waveform Processing.

The command can extract the following attributes:

- **Echo length** - relative length (millimeter) of a return signal compared to a typical return from a hard surface.
- **Echo normality** - difference in shape of a return signal compared to a typical return from a hard surface.
- **Echo position** - difference in position of a peak of a return signal compared to a typical return from a hard surface.

The echo length can be used for the visualization of points and for classifying points. For instance, a classification By echo length prior to ground classification can improve the result of the Ground routine especially in areas of low vegetation.

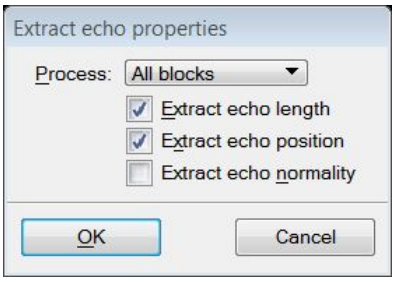
The echo properties can be stored in TerraScan FastBinary files. The command can be used to extract echo properties for all blocks or a selection of blocks defined in a project.

The command performs the same action on block binary files as the Extract echo properties command on loaded points.

To extract echo properties:

1. Select **Extract echo properties** command from the **Tools** pulldown menu.

   This opens the **Extract Echo Properties** dialog:



2. Select which blocks to process: **All blocks** or **Selected blocks**.

3. Select what properties you want to extract by switching the corresponding options on.

4. Click OK.

   This starts the extraction process. It assigns the extracted attributes to all laser points of the processed block binary files for which waveform information is available. Depending on the amount of points, the process may take some time.

Merge small blocks

Not UAV

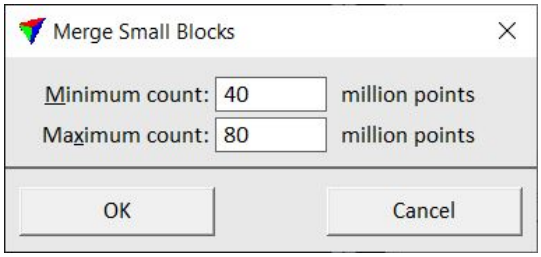
Merge blocks command merges small project blocks with neighbour blocks. It combines the block definitions and the binary files into one definition/file. This may be useful, for example, if automatically created grid block boundaries have been used for importing points. At the boundaries of the project areas, there may be blocks with low amounts of points which can be merged automatically into bigger blocks.

The process works only if there are block binary files available.

To merge blocks automatically:

1. Select **Merge small blocks** command from the **Tools** pulldown menu.

   This opens the **Merge Small Blocks** dialog:



2. Define a number of points in the **Minimum count** and **Maximum count** fields. This determines the point count limits of the merged blocks. The process stops if no block can be merged with a neighbouring block without exceeding the maximum count limit.

3. Click OK.

   This creates new block definitions and binary files. The original binary files are not kept. An information dialog shows the number of merged blocks after the process is finished.

4. Use Save project or Save project as commands in order to save the modification in a project file.

Blocks can be merged interactively by using the Merge blocks from the **Block** pulldown menu. This may be useful if the automatic merging process does not lead to a satisfying result.

Output collections

Not UAV

Collection shapes can be used to produce output files where logical groups of points are put together. You can perform the grouping by placing collections shapes around the objects using the Place Collection Shape tool. The output process starts from the **Project** window and automatically gathers all necessary points from the block binary files.

The output files can be created in two ways:

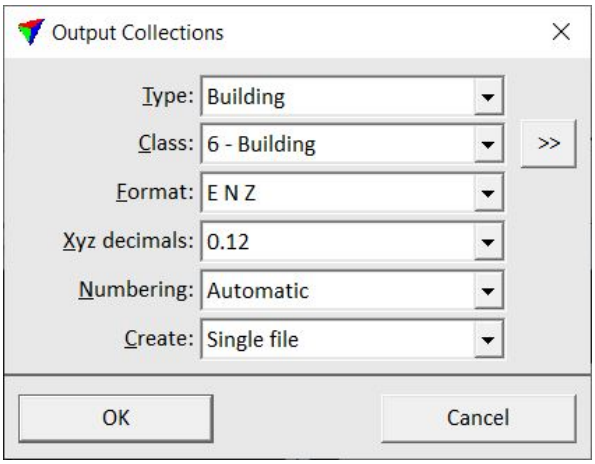
- The points inside each collection shapes are written into a separate output file. The name of the output files are determined by an optional prefix and either the number of the collection shape or an automatically increasing number.
- All points inside collection shapes are written into one output file.

The output file format should be a user-defined text file format which includes a **Collection** field so that points belonging to different collections can be distinguished from each other. See File formats / User point formats category of TerraScan **Settings** for more information. However, any text file format or TerraScan binary format is a valid output format for point collections.

To output collection shapes:

1. Select **Output collections** command from the **Tools** pulldown menu.

   This opens the **Output Collections** dialog:



2. Define settings and click OK.

3. If **Create** is set to **Single file**, the software opens the **Collection output file** dialog, a standard dialog for saving files. Define a location, name, and extension for the output file and click **Save**.

   This starts the output process.

SETTING	EFFECT
Type	Collection shape type to output. The list contains all collection shape types that are defined in Collection shapes category of TerraScan **Settings**.
Class	Point class(es) to output.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Class** field.
Format	Output file format.
Xyz decimals	Number of decimals used for coordinate values in the output files. This is only active if a text file format is chosen as output format.
Numbering	Defines the number of a point collection and the numbering method for naming separate output files: • **Automatic** - automatically increasing numbers starting from 1. • **From texts** - text elements drawn inside collection shapes.
Create	Defines the output: **Single file** or **One for each shape**.
Directory	Directory for storing output files. This is only active if **Create** is set to **One for each shape**.
Prefix	Defines the first part of the output file names which is followed by a number. This is only active if **Create** is set to **One for each shape**.
Extension	File name extension. Allowed extensions are .TXT or .ASC for text files and .BIN for TerraScan Binary files. This is only active if **Create** is set to **One for each shape**.

Output control report

Not UAV

Output control report command creates a report of elevation differences between laser points and ground control points or a surface model. This can be used, for example, to check the elevation accuracy of a laser data set and to calculate a correction value for improving the elevation accuracy of the laser points.

The ground control points have to be stored in a space-delimited text file in which each row has three or four fields: (optionally) identifier, easting, northing and elevation. The identifier field is

usually a number but it may include non-numeric characters as well. The surface model has to be an active model in TerraModeler.

The output report shows statistical information about the elevation difference between points and surface model, such as points used for computing the values, average difference and magnitude, standard deviation, and RMS value. It does not show elevation difference values for single point locations.

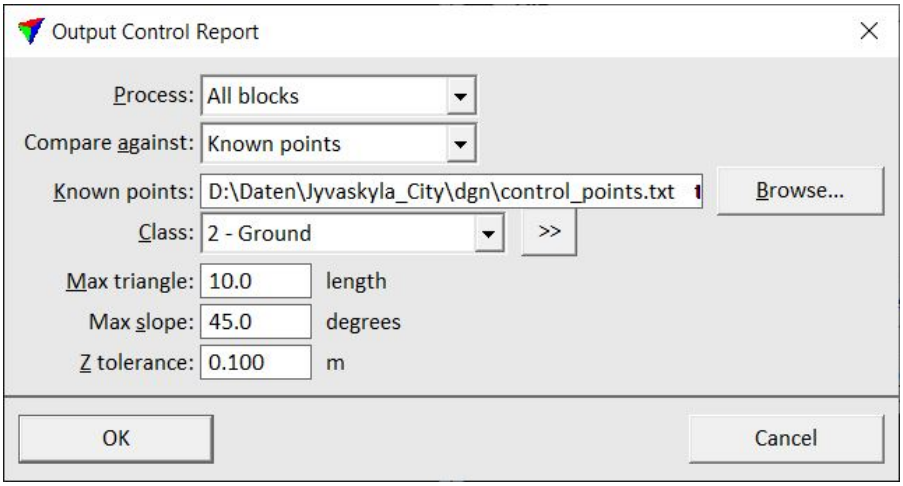
The command performs the same action on block binary files as the Output control report command on loaded points.

To create a control report:

1. (Optional) Select block definitions in the **Project** window to be adjusted.

2. Select **Output control report** command from the **Tools** pulldown menu.

   This opens the **Output Control Report** dialog:



3. Define settings and click OK.

   This calculates the elevation differences and opens the **Control report** window. The content of the report depends on the source file used for the comparison, ground control points in a text file or a surface model. The results and the further usage of a comparison with ground control points is described in detail in Section Systematic elevation correction.

SETTING	EFFECT
Process	Blocks to include in the process: **All blocks** or **Selected blocks**.
Compare against	Source file for elevation value comparison: • **Known points** - text file that contains coordinates of ground control points. • **<surface model>** - name of a specific surface model active in TerraModeler.
Known points	Location and name of the file that stores the coordinates of the ground control points. This

SETTING	EFFECT
	is only active if **Compare against** is set to **Known points**.
Class	Point class(es) used for the elevation value comparison.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Class** field.
Max triangle	Maximum length of a triangle edge. The software creates a triangle from the closest 3 laser points around a ground control point location. If the triangle edge length exceeds the given value, the control point is not used for the report.
Max slope	Maximum terrain slope for which an elevation difference is computed.
Z tolerance	Normal elevation variation (noise level) of laser points. This value is used only when computing the terrain slope so that small triangles do not exceed the Max slope value.

Run macro

Not UAV

The main benefit of defining a project is the ability to perform batch processing on blocks. **Run macro** command lets you run a TerraScan macro on project level. This requires that you first define a macro which includes all the processing steps to perform. Then, you can run the macro on all or selected project blocks.

A macro can run on project level either by using TerraScan or by using TerraSlave. Using TerraSlave has the advantage that TerraScan and the CAD platform are not blocked when a macro is processed.

For detailed information about macros, see Chapter Macros.

Show statistics

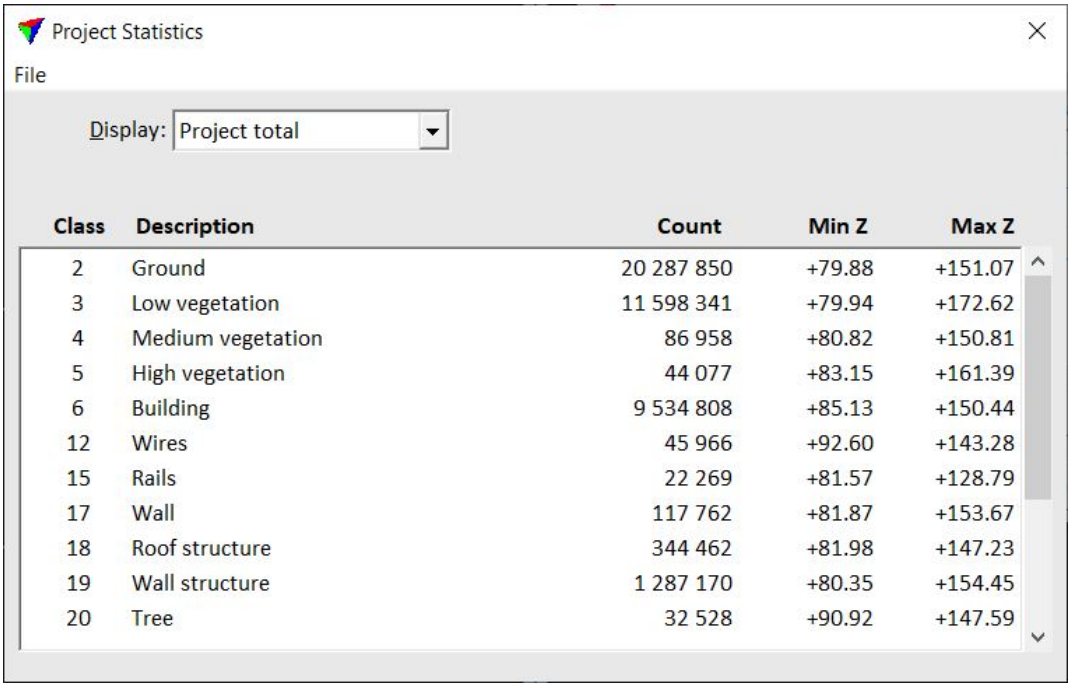
Not UAV

Show statistics command calculates simple statistics for the block binary files. The output dialog includes information about classes, point count as well as minimum, maximum, and median elevation values. Besides for the whole project, these values are also calculated for each block and line. The statistics can be saved into a text file.

To calculate statistics for a project:

1. Select **Show statistics** command from **Tools** pulldown menu.

The calculation process starts and an information window is displayed that shows the progress of the process. After finishing the calculation, the **Project statistics** dialog opens:



Class	Description	Count	Min Z	Max Z
2	Ground	20 287 850	+79.88	+151.07
3	Low vegetation	11 598 341	+79.94	+172.62
4	Medium vegetation	86 958	+80.82	+150.81
5	High vegetation	44 077	+83.15	+161.39
6	Building	9 534 808	+85.13	+150.44
12	Wires	45 966	+92.60	+143.28
15	Rails	22 269	+81.57	+128.79
17	Wall	117 762	+81.87	+153.67
18	Roof structure	344 462	+81.98	+147.23
19	Wall structure	1 287 170	+80.35	+154.45
20	Tree	32 528	+90.92	+147.59

Check the statistics for the project, different blocks, or lines by selecting the corresponding settings in the **Display** and **Block** or **Line** lists.

The statistical values for all classes occurring in the block binary files are listed in the lower part of the dialog.

You can save the statistics into a text file using **Save as** command from the **File** menu in the **Project statistics** dialog.

The text file stores the point count for each class and the elevation values if **Display** is set to **Project total**, and the point count for each class per block or line if **Display** is set to **By block** or **By line**.

SETTING	EFFECT
Display	Content of statistics display: • **Project total** - values for the whole project are displayed. • **By block** - values are shown for the selected block. • **By line** - values are shown for the selected line.
Block	Name of the block for which statistical values are shown. This is only active if **Display** is set to **By block**.

SETTING	EFFECT
Line	Number of the line for which statistical values are shown. This is only active if **Display** is set to **By line**.

Validate blocks

Not UAV

Validate blocks command checks block definitions of a project regarding duplicated block names, small area blocks, and overlap between block boundaries. This helps to analyze automatically created block boundaries and names before the block binary files are created.

To validate blocks:

1. Select **Validate blocks** command from the **Tools** pulldown menu.

This opens the **Block validity check** window that displays the results of the validation in a report.

The report can be saved into a text file or sent to a printer using **Save as text** or **Print** commands from the **File** pulldown menu.

Manage Trajectories

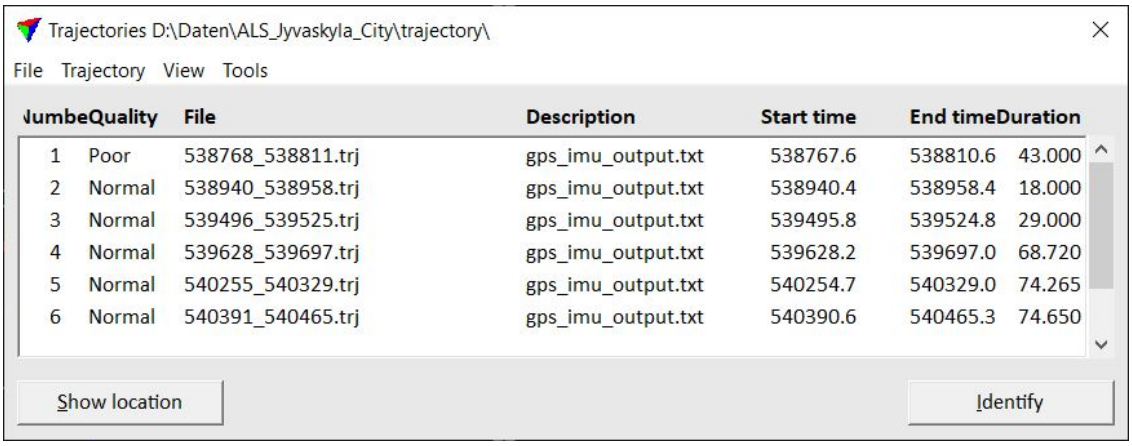
Trajectories are required for many processing steps in TerraScan and TerraMatch. They provide positional and, usually for moving systems, attitude information of the scanner system for each point of time during the data collection.

Normally, the raw trajectory is produced by so-called post-processing software that combines the input of GPS and INS sensors. The raw trajectory may be provided in a binary or ASCII file format. TerraScan is able to import common binary formats of post-processing software as well as a number of ASCII formats. Additional text file input formats for trajectories can be defined in File formats / User trajectory formats category of TerraScan **Settings**. All imported trajectories are converted into the TerraScan trajectory binary format (*.TRJ). See Trajectory file formats for more information.

All commands related to trajectories is combined in the TerraScan **Trajectories** window which is opened by the Manage Trajectories tool.

TerraScan Trajectories window

The **Trajectories** window contains pulldown menu commands for importing, modifying, and managing trajectory information.



Number	Quality	File	Description	Start time	End time	Duration
1	Poor	538768_538811.trj	gps_imu_output.txt	538767.6	538810.6	43.000
2	Normal	538940_538958.trj	gps_imu_output.txt	538940.4	538958.4	18.000
3	Normal	539496_539525.trj	gps_imu_output.txt	539495.8	539524.8	29.000
4	Normal	539628_539697.trj	gps_imu_output.txt	539628.2	539697.0	68.720
5	Normal	540255_540329.trj	gps_imu_output.txt	540254.7	540329.0	74.265
6	Normal	540391_540465.trj	gps_imu_output.txt	540390.6	540465.3	74.650

The list in the window shows all TerraScan trajectory files that are stored in the active trajectory folder. The active directory is shown in the title bar of the window.

To select a trajectory, click on the line in the list. Press <Ctrl> to select several trajectories.

To show the location of a trajectory, select a line in the list. Click on the **Show location** button and move the mouse pointer into a view. This displays the selected trajectory. With a data click inside the view you can center the selected trajectory in the view.

To identify a trajectory, click on the **Identify** button and place a data click close to a trajectory in a view. This selects the corresponding line in the **Trajectories** window.

File pulldown menu

Commands from the **File** pulldown menu in the **Trajectories** window are used to import trajectory information into TerraScan and to export trajectory information into text files.

TO	USE COMMAND
Set the active trajectory folder	Set directory
Import trajectory files	Import files
Import trajectory files from a folder and its sub-folders	Import directory
Import positions of tripod-mounted scanners	Import scanner positions
Deduce trajectories from point clouds of tripod-mounted scanners	Deduce scanner positions
Import separate text files from GPS and INS sensors	Merge from GPS and INS
Import accuracy files for trajectories	Import accuracy files
Export trajectory information into text files	Output positions

Deduce scanner positions

Deduce scanner positions command creates trajectory information for tripod mounted scanner data where:

- each scan has its own line number in the point cloud. The line information is used to separate one scanner position from another.
- scanner appears as a circular void area without points. The software assumes that the scanner is located in the center of the void area.

The routine computes the average XY position of points in one line and then searches for the void area. It creates a trajectory file with one position. The process may fail if a scanner position is close to a wall or if the void area is not circular.

You can use this method, if no text file for scanner positions is available for import.

The deduced trajectory allows processes for the point clouds that rely on trajectory information, such as cut overlap, classify by range, compute normal vectors towards the direction of scanner, etc..

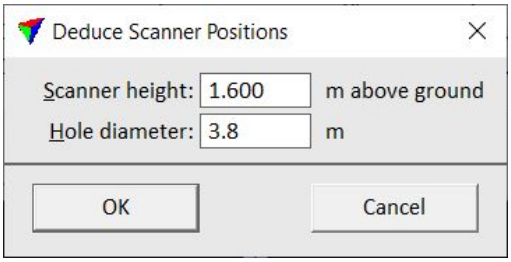
To deduce scanner positions:

1. Set a directory for storing trajectories.

2. Load the point cloud produced by the scanner. You may load point clouds from several scanner positions.

3. Select **Deduce scanner positions** from the **File** pulldown menu.

The **Deduce Scanner Positions** dialog opens:



4. Define settings and click OK.

   This writes the scanner positions as TerraScan trajectory binary file(s) into the active trajectory directory. The name of a file is determined by the line number.

SETTING	EFFECT
Scanner height	Height of the scanner above the ground.
Hole diameter	Diameter of the circular void area around the scanner position.

Import accuracy files

Import accuracy files command imports an output file from post-processing software that contains accuracy estimates for each trajectory position. The file includes the RMS values for XYZ positions as well as for heading, roll, and pitch angles. It is connected to the trajectory file by the time stamps.

TerraScan can import the following accuracy file formats:

- Applanix SMRMSG_*.OUT
- Leica IPAS .SOL
- Riegl .POQ

The RMS values are stored in the binary trajectory files. TerraScan stores only four RMS values for each trajectory position: x/y, z, heading, roll/pitch.

The information from the accuracy files is used for strip matching computations in TerraMatch and for drawing trajectories into the CAD file.

To import an accuracy file:

1. Import the trajectory file(s) as described in Import files or Import directory.

2. Select **Import accuracy files** command from the **File** pulldown menu.

   This opens the **Import accuracy files** dialog, a standard dialog for opening files.

3. Open the accuracy file delivered by the post-processing software.

This reads the file and connects the RMS values to the trajectory positions. The values are saved automatically to the binary trajectory files in the active trajectory directory. A dialog informs about the number of positions for which RMS values are available.

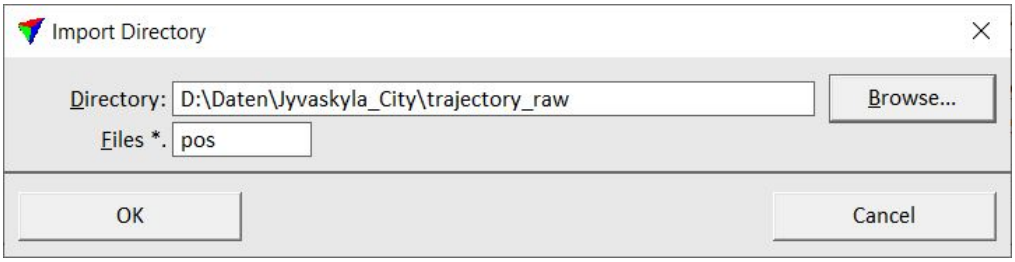
Import directory

Import directory command imports trajectory files into TerraScan. All files of the same format in a directory are imported. The import process itself works in the same way as described for the Import files command.

To import all trajectory files in a directory:

1. Select **Import directory** command from the **File** pulldown menu.

   This opens the **Import Directory** dialog:



2. Define settings and click OK.

   This opens the **Import trajectories** dialog. Follow the steps of Import files procedure in order to import the files.

SETTING	EFFECT
Directory	Folder from which to import files. Click on the **Browse** button in order to select a folder in the **Browse for Folder** dialog.
Files	Defines the extension of files that are imported. You can use the * character as placeholder for any file extension or type a specific extension.

Import files

Import files command is used to import raw trajectories into TerraScan. During the import, trajectory information is converted into TerraScan trajectory binary files (*.TRJ).

The input files must contain at least time-stamped position and, for most processing tasks, attitude information. The input files can be:

- text files in one of the implemented ASCII formats, see Supported file formats
- binary files from Applanix or Riegl software, see Supported file formats
- text files in a user-defined file format, see File formats / User trajectory formats

During the import, the software assigns some attributes to the trajectories and can apply coordinate transformations and/or a time stamp format conversion. Most of the settings defined in the import process can be changed later for the converted trajectory files by using the Edit information command or commands from the Tools pulldown menu.

If external accuracy files are available for Riegl POF (*.POQ) and Applanix SBET.OUT (SMRMSG_*.OUT) trajectory formats, the command reads the information automatically from the files matching the trajectory files. In this case, Import accuracy files is no longer necessary in a separate step.

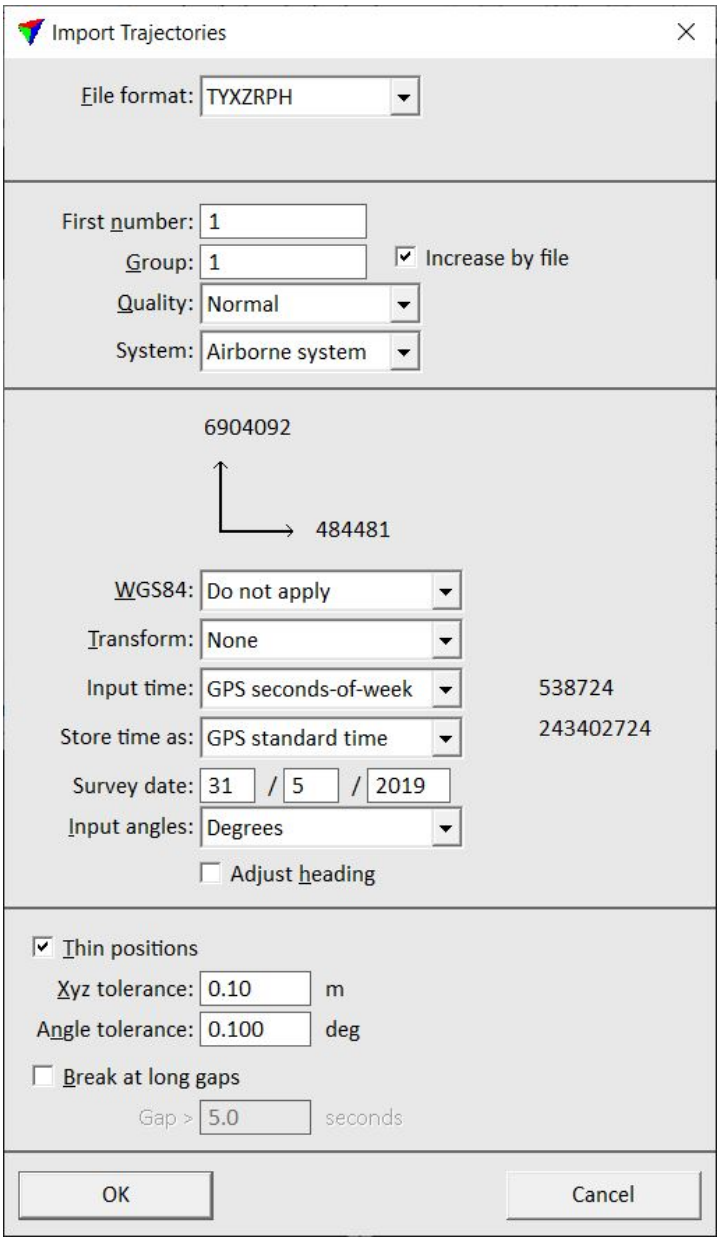
To import a raw trajectory:

1. Select **Import trajectories** command from the **File** pulldown menu.

   This opens the **Import trajectories** dialog, a standard dialog for opening files.

2. Select the raw trajectory file(s) and click **Open**.

   The **Import Trajectories** dialog opens:



3. Define settings and click OK.

   This imports the trajectory file(s) and stores them as TerraScan trajectory binary file(s) into the active trajectory directory. The name of a file is determined by the seconds values of the first and last position in a trajectory file separated by an underline character.

SETTING	EFFECT
File format	File format of the input file(s). This is usually recognized automatically for implemented input formats.
Attitude format	Format of the INS file. This is only active if Merge from GPS and INS command is used to import trajectory information.

SETTING	EFFECT
First number	Number assigned to the first trajectory file. If more than one file is imported, the files are numbered incrementally.
Group	Group number assigned to trajectory file(s). Groups may indicate, for example, different flight sessions and can be used by TerraMatch processes.
Increase by file	If on, the group number is automatically increased for each imported trajectory file.
Quality	Quality attribute assigned to trajectory file(s). Quality may indicate, for example, the accuracy of trajectories and can be used for TerraMatch and TerraScan processes.
System	Scanner system used for data collection. This may add lever arm corrections to trajectory positions and thus, effect the computation of the scanner location at the moment of measuring a laser point.
WGS84	Transformation from WGS84 coordinates to another projection system applied during the import. The list contains projection systems that are active in Coordinate transformations / Built-in projection systems, Coordinate transformations / US State Planes, and Coordinate transformations / User projection systems categories of TerraScan **Settings**.
Transform	Transformation applied during the import. The list contains transformations defined in Coordinate transformations / Transformations category of TerraScan **Settings**.
Input time	Format of time stamps in the raw trajectory file(s): GPS seconds-of-week, GPS standard time, Unix time, or GPS time.
Store time as	Format of time stamps in the converted files: **GPS seconds-of-week**, **GPS standard time**, or **GPS time**. If the format is different from the **Input time** format, time stamps are converted.
Survey data	Date when the trajectory data was captured. The format is day/month/year (dd/mm/yyyy). This is required for the conversion of time stamps from **GPS seconds-of-week** to GPS **standard time** and is only active if **Input time** and **Store time** as are set accordingly.

SETTING	EFFECT
Input angles	Format of angle values in the raw trajectory file(s): **Degrees**, **Radians**, or **TopEye radians**. This is usually set automatically for implemented input formats.
Adjust heading	If on, the software applies a meridian convergence correction to heading values. The correction is based on the projection system set for **WGS84** or the coordinate transformation set for **Transform**.
Thin positions	If on, intermediate positions are skipped as long as the trajectory accuracy stays within the given tolerances.
Xyz tolerance	Maximum allowed locational change of the trajectory caused by thinning. This is only active if **Thin positions** is switched on.
Angle tolerance	Maximum allowed angular change of the trajectory caused by thinning. This is only active if **Thin positions** is switched on.
Break at long gaps	If on, the software splits the trajectory if there is a gap between consecutive trajectory positions that is longer than the given **Gap** value. The gap value is given in seconds.

TerraScan and TerraMatch do not need highly accurate trajectory information. It is beneficial to remove unnecessary positions with **Thin positions** setting when importing a raw trajectory. This reduces the amount of memory consumed by trajectory information and speeds up processes.

Import scanner positions

Import scanner positions command is used to import positions of tripod-mounted scanners into TerraScan. The scanner positions must be stored in space-delimited text files of the format:

- number easting northing elevation

The number of a scanner position must be a numerical value, no alphanumerical or other characters are allowed. During the import, the software assigns some attributes to the scanner positions and can apply coordinate transformations. Most of the settings defined in the import process can be changed later for the converted trajectory files by using the Edit information command or commands from the Tools pulldown menu.

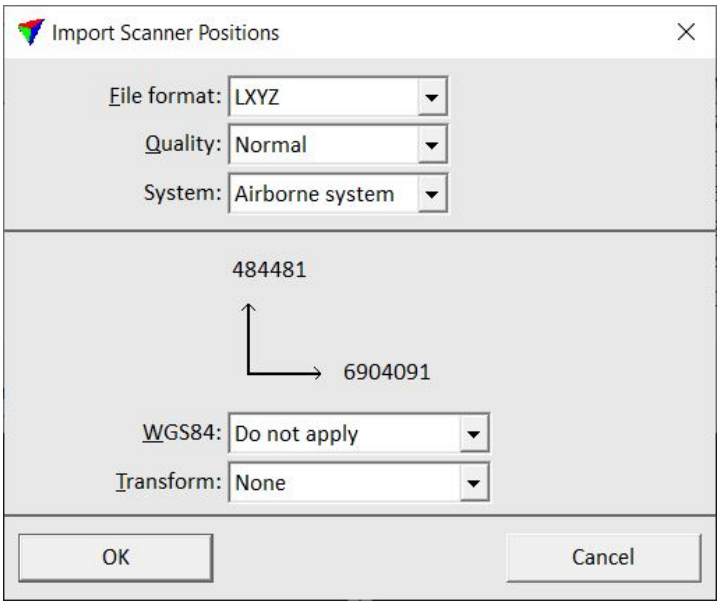
To import scanner positions from a text file:

1. Select **Import scanner positions** command from the **File** pulldown menu.

   This opens the **Import scanner positions** dialog, a standard dialog for selecting files.

2. Select the text file and click **Done**.

   The **Import Scanner Positions** dialog opens:



3. Define settings and click OK.

   This imports the text file and stores the scanner positions as TerraScan trajectory binary file(s) into the active trajectory directory. The name of a file is determined by the position number.

SETTING	EFFECT
File format	File format of the input file. This is usually recognized automatically for implemented input formats.
Quality	Quality attribute assigned to trajectory file(s). Quality may indicate, for example, the accuracy of trajectories and can be used for TerraMatch and TerraScan processes.
System	Scanner system used for data collection. This may add lever arm corrections to trajectory positions and thus, effect the computation of the scanner location at the moment of measuring a laser point.
WGS84	Transformation from WGS84 coordinates to another projection system applied during the import. The list contains projection systems that are active in Coordinate transformations / Built-in projection systems, Coordinate transformations / US State Planes, and Coordinate transformations / User projection systems categories of TerraScan **Settings**.

SETTING	EFFECT
Transform	Transformation applied during the import. The list contains transformations defined in Coordinate transformations / Transformations category of TerraScan **Settings**.

Merge from GPS and INS

Merge from GPS and INS command creates a trajectory binary file for TerraScan from separate GPS and INS files. The GPS file contains time stamps and coordinates for the trajectory positions, while the INS file includes time stamps and attitude angle values for the same trajectory positions. The software combines the two input files using the time stamps.

The GPS and INS files are usually text files. The format of the files can be defined in File formats / User trajectory formats of TerraScan **Settings**.

To create a trajectory from GPS and INS files:

1. Select **Merge from GPS and INS** command from the **File** pulldown menu.

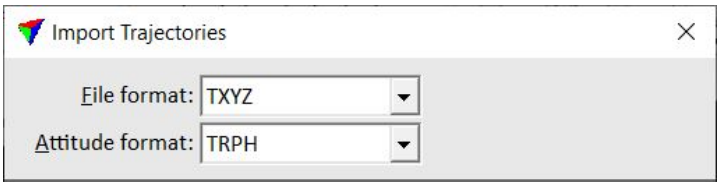
   This opens the **GPS positions files** dialog, a standard dialog for opening files.

2. Open the file that contains the positional information.

   This opens the **INS attitude files** dialog, a standard dialog for opening files.

3. Open the file that contains the attitude information.

   The **Import Trajectories** dialog opens.



4. Check the **File format** and **Attitude format**. The software tries to determine the format of the GPS and INS files automatically.

   If necessary, select the correct format from the selection lists.

   For all other settings of the dialog, see Import files for a description.

5. Define settings and click OK.

   The software combines the two input files and creates the binary trajectory file in the active trajectory directory.

Output positions

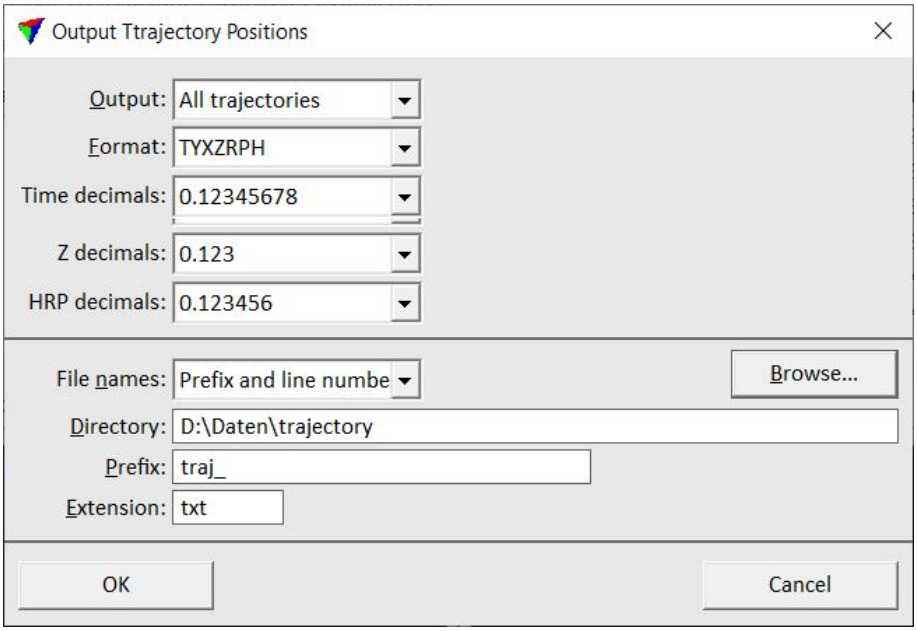
Output positions command saves trajectory positions into text files. It creates a separate text file for each trajectory file. The format of the output files can be defined in File formats / User trajectory formats of TerraScan **Settings**. There are also two implemented output formats. The software writes one line for each trajectory position into an output file.

To create text files for trajectory positions:

1. (Optional) Select the trajectories in the list of the **Trajectories** window for which you want to save positions into text files.

2. Select **Output positions** command from the **File** pulldown menu.

   This opens the **Output Trajectory Positions** dialog:



3. Define settings and click OK.

   This writes the trajectory positions into text files.

SETTING	EFFECT
Output	Trajectories for which text files are created: **All trajectories** or **Selected only**.
Format	Text file format, defines which attributes are stored in the columns of the text file. The list contains two implemented formats: • **TYXZRPH** - time northing easting elevation roll pitch heading • **TXYZ** - time easting northing elevation and any formats defined for output in File formats / User trajectory formats.

SETTING	EFFECT
Time decimals	Amount of decimals written for the time stamps.
Z decimals	Amount of decimals written for elevation values.
HRP decimals	Amount of decimals written for attitude values.
File names	Method of naming the output files: • **Prefix and line number** - name contains a prefix and the trajectory number. • **Same as trj file** - name of the trajectory binary file is used.
Directory	Folder into which the output files are written. Click on the **Browse** button in order to select a folder in the **Browse for Folder** dialog.
Prefix	Text string added in the beginning of an output file name. This is only active if **File names** is set to **Prefix and line number**.
Extension	File name extension.

Set directory

Set directory command is used to define the active trajectory directory. The software writes trajectory files into this folder during the import process. It loads TerraScan trajectory files from a folder if it is set as active directory and files do already exist. Usually, this is the first command you use when you start working with trajectories.

It is good practice to reserve a folder in your project directory structure for storing trajectories imported into TerraScan. In some cases, it might be advisable to save a new copy of TerraScan trajectories. Then, you would have multiple trajectory directories in a project and change the active directory whenever needed in order to access the correct set of trajectory files.

To set the active trajectory directory:

1. Select **Set directory** command from the **File** pulldown menu.

   This opens the standard dialog for selecting a folder.

2. Select a folder and click OK.

   This sets the active directory to the given folder. TerraScan scans the directory. If there are TerraScan trajectory files in the folder, it reads the header information from each file into memory and displays them in the list.

Tools pulldown menu

Commands in the **Tools** pulldown menu are used to manipulate trajectories and to create macros automatically based on trajectory information.

TO	USE
Split a trajectory manually	Split
Split trajectories automatically at turnarounds	Cut turnarounds
Split trajectories and keep only parts inside a polygon	Delete outside polygons
Split trajectories at gaps in laser data	Split at laser gaps
Link trajectories to waveform files	Link to waveform files
Apply new numbers to trajectories	Renumber trajectories
Remove unnecessary trajectory positions	Thin positions
Transform trajectory coordinates	Transform
Add lever arms to trajectory positions	Add lever arm
Adjust trajectory elevations to a geoid model	Adjust to geoid
Convert trajectory angle values	Convert angles
Convert trajectory time stamps	Convert time stamps
Create macros automatically based on trajectory information	Create macro / For stops and turns
	Create macro / For poor accuracy
	Create macro / For repeated passes
Correct trajectory drift during a stop	Smoothen stops
Draw trajectories into the CAD file	Draw into design

Add lever arm

Add lever arm command applies a lever arm to trajectories. A lever arm is expressed by the X,Y, and Z components of a vector between the original trajectory position and the lever arm-corrected position.

The direction of the three vector components relative to the trajectory or system movement direction is as follows:

- **X** - positive values to the right, negative to the left.
- **Y** - positive values forward, negative backward.
- **Z** - positive values up, negative down.

A lever arm should be applied, if the trajectory has been computed for the IMU and the point cloud has been generated without considering the lever arm values. Then, the lever arm vector describes the distance between the IMU and the scanner. However, this is commonly done by post-processing software for one scanner systems. For multiple scanner systems, the Scanner systems definition in TerraScan **Settings** defines the lever arms of the different scanners which can be applied in the import process of trajectories. See Import files command for more information.

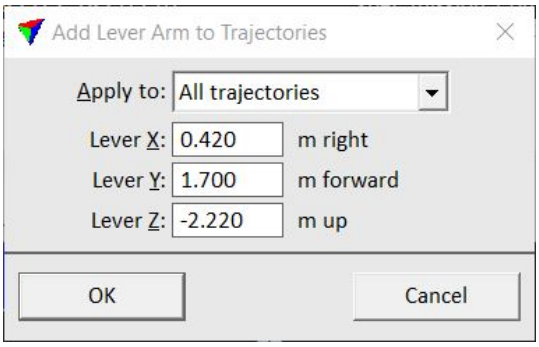
An application example for applying lever arms to active trajectories is to project the trajectory of a MMS survey to the center of a rail track. In this case, the lever arm vector describes the distance between the IMU (trajectory location) and the center between the wheels of the vehicle carrying the system along the tracks.

To add a lever arm to trajectories:

1. (Optional) Select the trajectory file(s) you want to modify in the **Trajectories** window.

2. Select **Add lever arm** command from the **Tools** pulldown menu.

   This opens the **Add Lever Arm to Trajectories** dialog:



3. Define settings and click OK.

   This modifies the trajectory positions and updates the trajectory files in the active trajectory directory. An information dialog shows the number of the trajectories effected by the process.

SETTING	EFFECT
Apply to	Trajectories effected by the process: **All trajectories** or **Selected only**.
Lever X	X component of the lever arm vector.
Lever Y	Y component of the lever arm vector.
Lever Z	Z component of the lever arm vector.

Adjust to geoid

Adjust to geoid command applies an elevation correction to trajectory files. The command is used, for example, to transform the WGS84-based ellipsoidal elevation values of a raw trajectory file to a local height model. The input model for geoid adjustment must be provided in one of the following formats:

- **Points from file** - text file containing space-delimited X Y dZ- points.
- **TerraModeler surface** - triangulated surface model created from X Y dZ - points. The surface model in TerraModeler has the advantage that you can visualize the shape of the adjustment model.
- **Selected linear chain** - linear element of which the vertices represent the X Y dZ - points.

XY are the easting and northing coordinates of the geoid model points, dZ is the elevation difference between ellipsoidal and local heights at the location of each geoid model point. Intermediate adjustment values of the model are derived by aerial (text file or surface model as input) or linear (linear element as input) interpolation between the known geoid model points.

You can find more detailed information about elevation adjustment in Section Geoid adjustment.

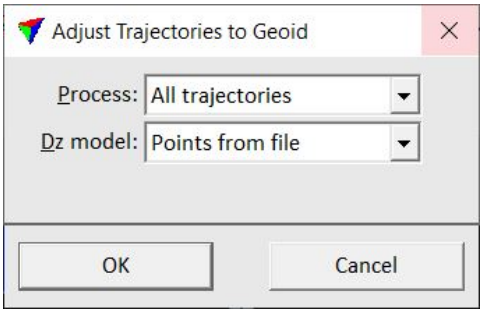
To adjust trajectories to a geoid model:

1. (Optional) Load a geoid model into TerraModeler.

2. (Optional) Select trajectory file(s) to adjust.

3. Select **Adjust to geoid** command from the **Tools** pulldown menu.

   This opens the **Adjust Trajectories to Geoid** dialog:



4. Define settings and click OK.

   If **Points from file** is selected as the **Dz model**, the **Geoid dz file** dialog opens, a standard dialog for opening files.

5. Define the text file that contains the geoid point coordinates and elevation differences and click **Open**.

   This applies the elevation adjustment to all or selected trajectories. The modification is saved to the trajectory binary files in the active trajectory directory. An information dialog shows the minimum and maximum values of the adjustment.

SETTING	EFFECT
Process	Trajectories to adjust: • **All trajectories** - all trajectories in the list. • **Selected only** - selected trajectories only.
Dz model	Source file that provides the geoid correction model: • **Points from file** - text file. • **Selected linear chain** - linear element selected in the CAD file. • <**surface**> - name of the geoid model surface loaded in TerraModeler.

SETTING	EFFECT
Extend	Distance of a linear extension. This is only active if **Dz model** is set to **Selected linear chain**.

Convert angles

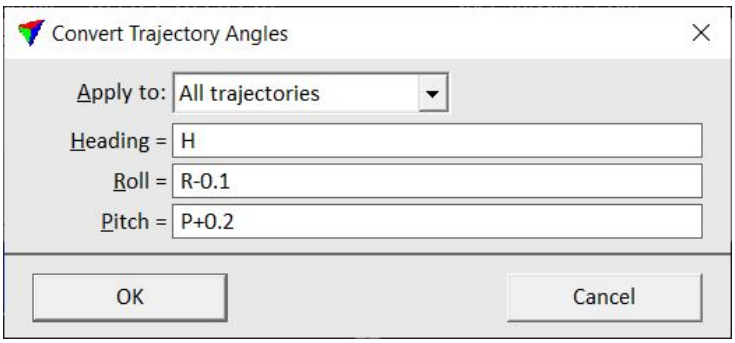
Convert angles command lets you apply a mathematical equation to the attitude angles heading, pitch, and roll of each trajectory position. The current angle value can be accessed by using constants H (heading), R (roll) and P (pitch). Thus, the command can also be used to exchange angle values.

To convert angles of trajectory positions:

1. (Optional) Select trajectory file(s) for which to manipulate angles.

2. Select **Convert angles** command from the **Tools** pulldown menu.

   This opens the **Convert Trajectory Angles** dialog:



3. Define equations and click OK.

   This computes the new values for the orientation angles. The modification is saved to the trajectory binary files in the active trajectory directory. An information dialog shows the number of effected trajectories.

SETTING	EFFECT
Apply to	Trajectories for which the computation of new angles is applied: • **All trajectories** - all trajectories in the list. • **Selected only** - selected trajectories only.
Heading	Equation for modifying the heading angle.
Roll	Equation for modifying the roll angle.
Pitch	Equation for modifying the pitch angle.

Convert time stamps

Convert time stamps command can be used to convert the format of time stamps. Supported conversions are:

INPUT FORMAT	CONVERTED FORMAT
GPS seconds-of-week	GPS standard time
	GPS time
GPS standard time	GPS seconds-of-week
	GPS time
GPS time	GPS seconds-of-week
	GPS standard time
GPS seconds-of-day	GPS seconds-of-week
	GPS standard time
	GPS time
Unix time	GPS seconds-of-week
	GPS standard time
	GPS time
UTC seconds-of-day	GPS seconds-of-week
	GPS standard time
	GPS time

The conversion is necessary, for example, if data collected in several weeks is processed together in one project. Then, GPS seconds-of-week time stamps result in repeated values and GPS standard time must be used in order to provide unique time stamps for each trajectory position. This is a requirement for many processes that rely on trajectory information. Some post-processing software generates data with Unix or UTC seconds-of-day time stamps. They must be converted into another GPS time format as well.

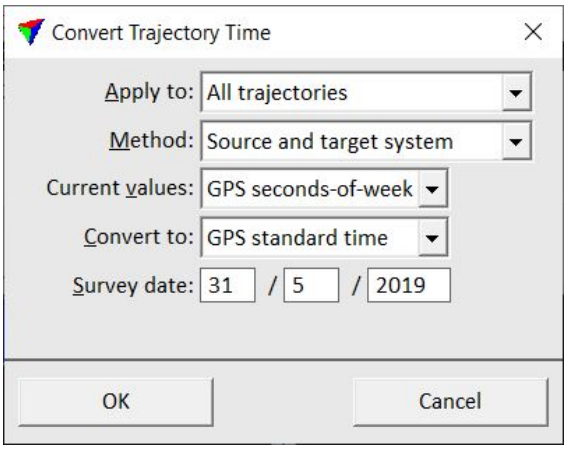
It is essential that time stamps of trajectories and point cloud data are stored in the same GPS time format.

To convert time stamps:

1. (Optional) Select trajectory file(s) for which to manipulate time stamps.

2. Select **Convert time stamps** command from the **Tools** pulldown menu.

   This opens the **Convert Trajectory Time** dialog:



3. Define settings and click OK.

   This converts the trajectory time stamps to the new format. The modification is saved to the trajectory binary files in the active trajectory directory. An information dialog shows the number of effected trajectories.

SETTING	EFFECT
Apply to	Trajectories for which the conversion of time stamps is applied: • **All trajectories** - all trajectories in the list. • **Selected only** - selected trajectories only.
Method	Defines the computation method of modifying time stamps: • **Source and target system** - conversion from one time format to another. • **Multiply and add constant** - constant value by which time stamps are multiplied or that is added to current time stamps.
Current values	Original time stamp format of the trajectory positions. This is only active if **Method** is set to **Source and target system**.
Convert to	Target time stamp format. This is only active if **Method** is set to **Source and target system**.
Survey date	Date when the trajectory data was captured. The format is day/month/year (dd/mm/yyyy). This is only active for the conversion from **GPS seconds-of-week** or **UTC seconds-of-day** to **GPS standard time** or **GPS time**.
Multiply by	Factor by which current time stamps are multiplied. This is only active if **Method** is set to **Multiply and add constant**.
Add	Value to add to GPS time stamps when converted from UTC seconds-of-day time stamps. Refers to the leap-seconds that by which GPS time is ahead

SETTING	EFFECT
	of UTC time. This is only active for the conversion from **UTC seconds-of-day** to any GPS time format. Value to add to current time stamps if **Method** is set to **Multiply and add constant**.

Create macro / For poor accuracy

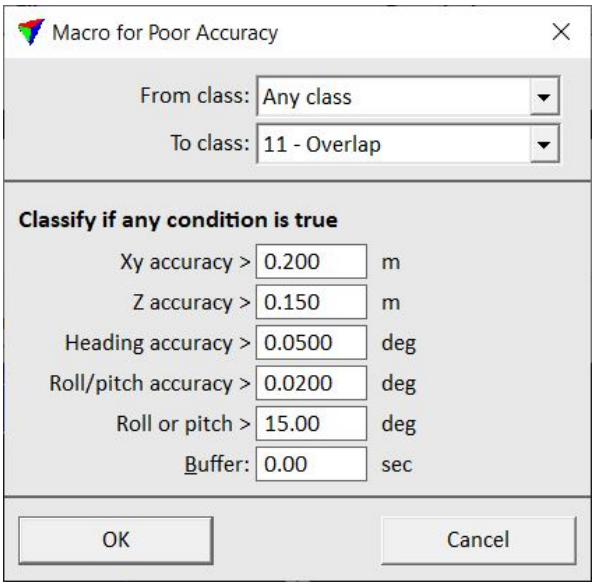
Create macro / For poor accuracy command creates a TerraScan macro automatically. The macro is used to classify points that were collected from locations of bad trajectory accuracy.

Especially in MLS data sets, there might be places of poor trajectory accuracy caused, for example, by the lack of GPS signals. The command can be used to identify such locations based on trajectory position attributes. The process uses accuracy values that are assigned to trajectory positions. See Import accuracy files for more information. If the search finds poor accuracy locations, the resulting macro contains steps that classify points based on time intervals.

To create a macro for poor accuracy:

1. Select **Create macro / For poor accuracy** command from the **Tools** pulldown menu.

   This opens the **Macro for Poor Accuracy** dialog:



2. Define settings and click OK.

   The software computes poor accuracy time ranges from the trajectory positions and creates a TerraScan macro. An information dialog shows the number of added time intervals.

   The macro can be saved and applied to the laser points. See Chapter Macros for more information about macros in TerraScan.

SETTING	EFFECT
From class	Point class(es) from which to classify points. The list contains the active point classes in TerraScan.
To class	Target class for points collected during poor trajectory accuracy. The list contains the active point classes in TerraScan.
Xy accuracy	If the xy accuracy value is bigger than the given value, the trajectory position is added to the macro.
Z accuracy	If the z accuracy value is bigger than the given value, the trajectory position is added to the macro.
Heading accuracy	If the heading accuracy value is bigger than the given value, the trajectory position is added to the poor accuracy macro.
Roll/pitch accuracy	If the roll/pitch accuracy value is bigger than the given value, the trajectory position is added to the macro.
Roll or pitch	If the value of roll or pitch angle is bigger than the given value, the trajectory position is added to the macro.
Buffer	Number of seconds that is added to each poor accuracy time interval. The seconds are added at the beginning and in the end of a time interval.

Create macro / For repeated passes

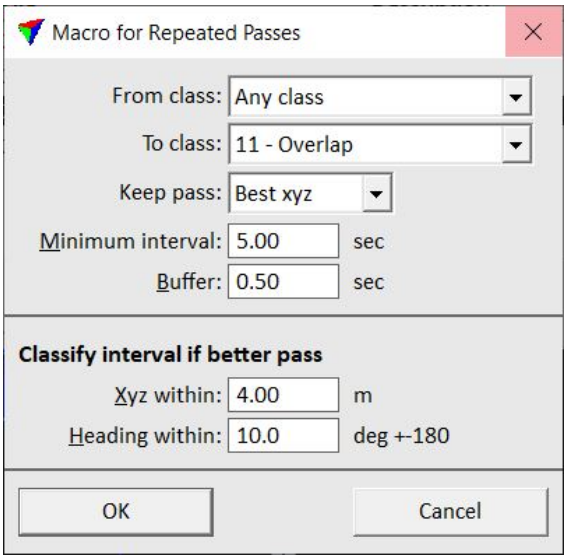
Create macro / For repeated passes command creates a TerraScan macro automatically. The macro is used to classify points collected from the same location in several strips.

Especially in MLS data sets of roads inside urban areas, there might be places where data was collected several times during a survey. The command can be used to identify such locations based on trajectory position attributes. The process can make use of accuracy values that are assigned to trajectory positions. See Import accuracy files for more information. If the search finds repeated pass locations, the resulting macro contains steps that classify points based on time intervals.

To create a macro for repeated passes:

1. Select **Create macro / For repeated passes** command from the **Tools** pulldown menu.

   This opens the **Macro for Repeated Passes** dialog:



2. Define settings and click OK.

   The software computes time ranges of repeated passes from the trajectory positions and creates a TerraScan macro. An information dialog shows the number of added time intervals.

   The macro can be saved and applied to the laser points. See Chapter Macros for more information about macros in TerraScan.

SETTING	EFFECT
From class	Point class(es) from which to classify points. The list contains the active point classes in TerraScan.
To class	Target class for points collected in repeated passes. The list contains the active point classes in TerraScan.
Keep pass	Defines which data is kept in the original class: • **First** - data of the first pass. • **Last** - data of the last pass. • **Best xyz** - data of any pass with the best positional accuracy. • **Best hrp** - data of any pass with the best attitude accuracy.
Minimum interval	Minimum time interval within which the software searches for repeated pass points.
Buffer	Number of seconds that is added to each repeated pass time interval. The seconds are added at the beginning and in the end of a time interval.
Xyz within	Defines the maximum distance between repeated passes. If passes are less than the given distance apart from each other, they are considered repeated passes.

SETTING	EFFECT
Heading within	Defines the maximum angular difference between repeated passes. If the heading angle between passes is smaller than the given value, they are considered as repeated passes.

Create macro / For stops and turns

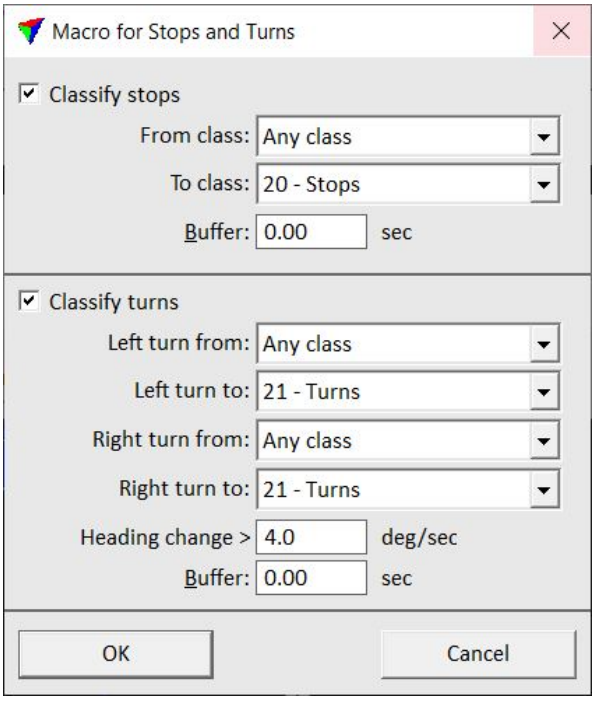
Create macro / For stops and turns command creates a TerraScan macro automatically. The macro is used to classify points of an MLS data set that were collected during stops or in sharp turns.

Since stops and turns cause a slowing-down of the vehicles speed, the scanner collects significantly more data than at normal operating speed. The command can be used to identify locations of stops and turns based on trajectory position attributes. If the search for stop and turns finds such locations, the resulting macro contains steps that classify points based on time intervals.

To create a macro for stops and turns:

1. Select **Create macro / For stops and turns** command from the **Tools** pulldown menu.

   This opens the **Macro for Stops and Turns** dialog:



2. Define settings and click OK.

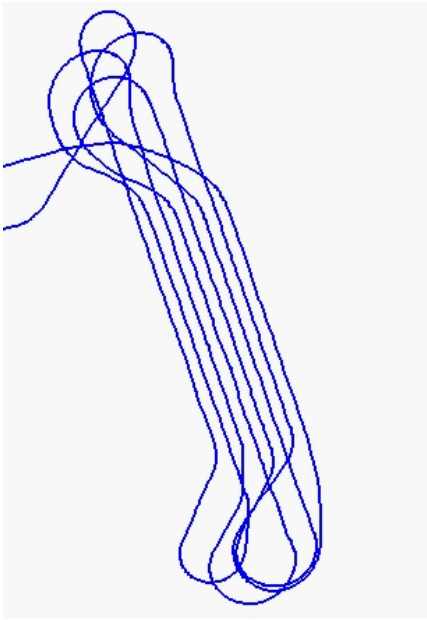
   The software computes stops and turns from the trajectory positions and creates a TerraScan macro. An information dialog shows the number of added time intervals.

   The macro can be saved and applied to the laser points. See Chapter Macros for more information about macros in TerraScan.

SETTING	EFFECT
Classify stops	If on, the software searches for stops.
From class	Point class(es) from which to classify points. The list contains the active point classes in TerraScan.
To class	Target class for points collected during stops. The list contains the active point classes in TerraScan.
Buffer	Number of seconds that is added to each stop time interval. The seconds are added at the beginning and in the end of a stop.
Classify turns	If on, the software searches for turns.
Left turn from	Point class(es) from which to classify points collected during turns to the left. The list contains the active point classes in TerraScan.
Left turn to	Target class for points collected during a turns to the left. The list contains the active point classes in TerraScan.
Right turn from	Point class(es) from which to classify points collected during turns to the right. The list contains the active point classes in TerraScan.
Left turn to	Target class for points collected during a turns to the right. The list contains the active point classes in TerraScan.
Heading change	Minimum change in heading angle that defines a turn. As long as the heading angle between consecutive trajectory positions changes more than the given degree value per second, the change is considered a turn and the respective time stamps are added to the macro step.
Buffer	Number of seconds that is added to each turn time interval. The seconds are added at the beginning and in the end of a turn.

Cut turnarounds

Cut turnarounds command splits a trajectory into several trajectories that do not overlap themselves anymore. It does not remove any parts of the original trajectory. The following figure illustrates the method:



Original trajectory

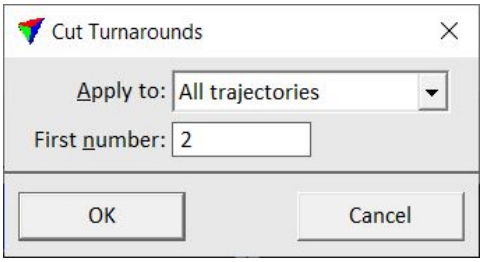
Resulting trajectories

To cut trajectories at turnarounds:

1. (Optional) Select the trajectory file(s) you want to cut in the **Trajectories** window.

2. Select **Cut turnarounds** command from the **Tools** pulldown menu.

   This opens the **Cut Turnarounds** dialog:



3. Define settings and click OK.

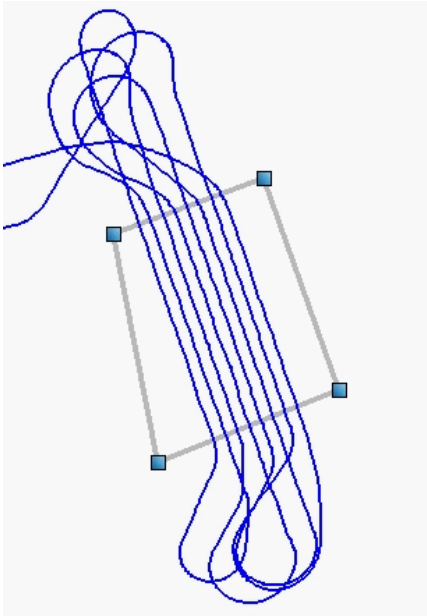
   This splits trajectories whenever there is a close to 180 degree turn. The application deletes the old trajectory file(s) and creates new files in the active trajectory directory.

SETTING	EFFECT
Apply to	Trajectories effected by the process: **All trajectories** or **Selected only**.
First number	Number of the first additional trajectory that is created by the process.

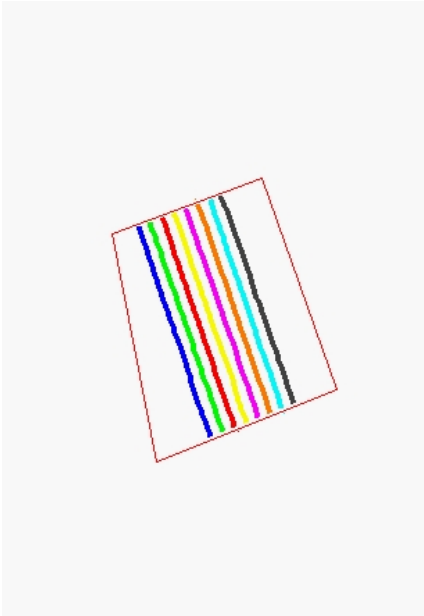
Delete by polygons

Delete by polygons command cuts trajectories at the boundary of a shape element. It keeps only trajectory lines inside or outside the shape and deletes all other parts.

This is often the easiest way to split trajectories of aerial projects. The figure below illustrates the method:



Original trajectory



Resulting trajectories using Delete = outside polygons.

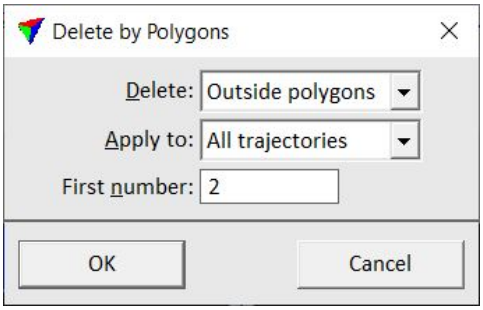
To delete trajectories by polygons:

1. Use CAD tools to draw polygon(s) around areas where you want to keep or delete trajectory information. Select the polygon(s).

2. (Optional) Select the trajectory file(s) you want to cut in the **Trajectories** window.

3. Select **Delete by polygons** command from the **Tools** pulldown menu.

   This opens the **Delete by Polygons** dialog:



4. Define settings and click OK.

This deletes all trajectory parts inside or outside the selected polygon(s). The application deletes the old trajectory file(s) and creates new files in the active trajectory directory. An information dialog shows the result of the process.

SETTING	EFFECT
Delete	Determines which part of a trajectory is deleted: **Inside polygons** or **Outside polygons**.
Apply to	Trajectories effected by the process: **All trajectories** or **Selected only**.
First number	Number of the first additional trajectory that is created by the process.

Draw into design

Draw into design command draws the trajectories as line elements into the CAD file. The line elements are drawn on the active level using the active line width and line style settings of the CAD file. The color(s) of the line elements are defined by the command's settings.

The command can use accuracy values that are assigned to trajectory positions. See Import accuracy files for more information.

The line elements are drawn by placing a vertex for each trajectory position. The lines can by simplified by removing positions within a given tolerance.

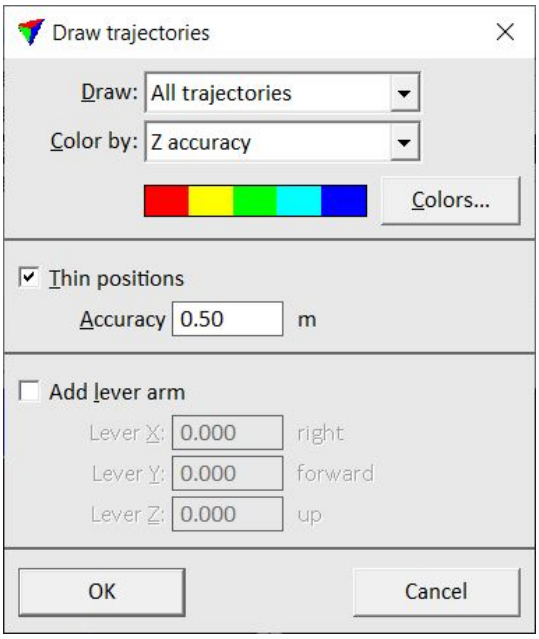
You can apply lever arms to trajectories when drawing them into the CAD file. This is useful, for example, if a centerline or other line elements along rails are derived from the trajectories of an MLS survey. The lever arm values are only applied to the line elements drawn into the CAD file but do not effect the original trajectory files.

To draw trajectory lines into the CAD file:

1. (Optional) Select trajectory file(s) to draw.

2. Select **Draw into design** command from the **Tools** pulldown menu.

   This opens the **Draw trajectories** dialog:

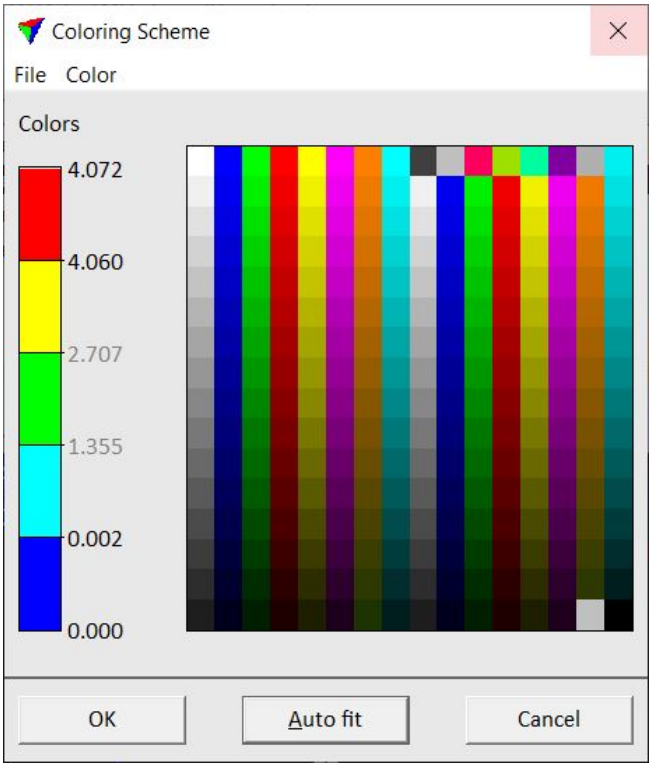


3. Define settings.

SETTING	EFFECT
Draw	Trajectories that are drawn: • **All trajectories** - all trajectories in the list. • **Selected only** - selected trajectories only.
Color by	Determines how the color is chosen for drawing a trajectory line: • **Active color** - the active color of the CAD file is used. • **Trajectory number** - the color whose number in the active color table of the CAD file corresponds to the trajectory number is used. • **Xy accuracy** - x/y accuracy values are applied to a color scheme. • **Z accuracy** - z accuracy values are applied to a color scheme. • **H accuracy** - heading accuracy values are applied to a color scheme. • **Rp accuracy** - roll/pitch accuracy values are applied to a color scheme.
Colors	Button to open the coloring scheme for accuracy-based coloring methods.
Thin positions	If on, intermediate trajectory positions are skipped when the line is drawn as long as the line accuracy stays within the given positional **Accuracy** tolerance.

4. If the trajectory is drawn with an accuracy-based coloring option, click on the **Colors** button.

This opens the **Coloring Scheme** dialog, the TerraScan dialog for 256 Colors:



5. (Optional) Define your own coloring scheme for drawing trajectories.

6. Click on the **Auto fit** button in order to fit the colors to RMS value ranges.

7. Click OK to the **Coloring scheme** dialog.

8. Click OK to the **Draw trajectories** dialog.

   This draws the line element(s) into the CAD file.

You can undo the drawing of trajectories by using the **Undo** command of the CAD platform.

Link to waveform files

Link to waveform files command links waveform files to trajectories. This is required for any processing tasks based on waveform data. See Chapter Waveform Processing for more information. You can check if a waveform file is linked to a trajectory in the **Trajectory information** dialog that is opened by the Edit information command.

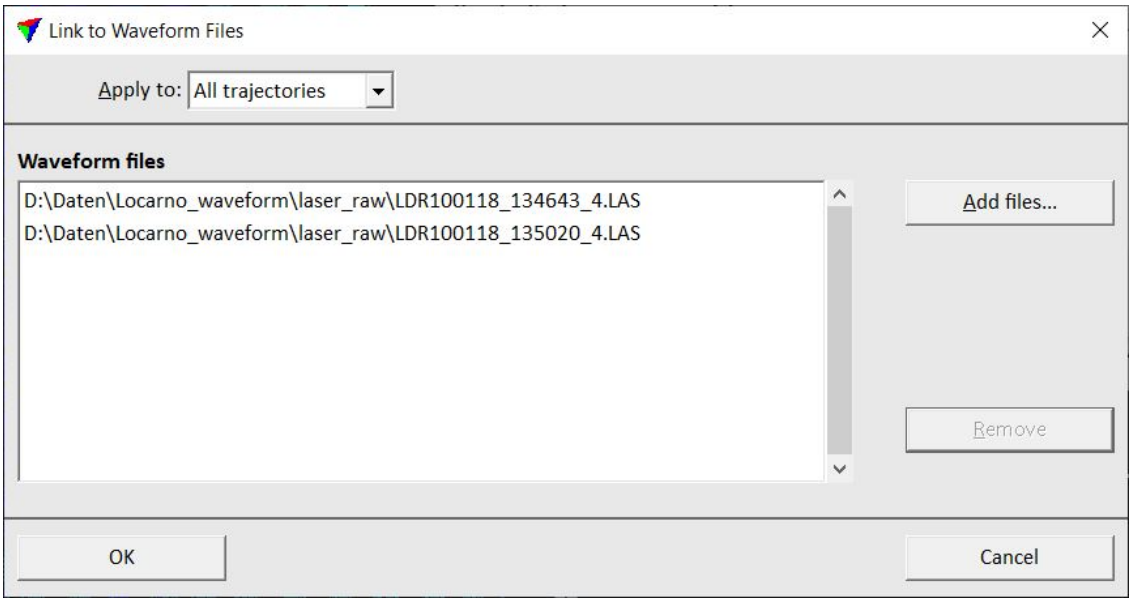
In order to link a waveform file to a trajectory, select the LAS file even if the waveform data is stored in WDP files. If waveform files are linked with the **Link to waveform files** command, the software creates index files for waveform files. Index files can speed up further processing steps that require waveform information. However, they are not necessarily required.

To link trajectories to waveform files:

1. (Optional) Select the trajectory file(s) you want to link in the **Trajectories** window.

2. Select **Link to waveform files** command from the **Tools** pulldown menu.

   This opens the **Link to Waveform Files** dialog:



3. Click on the **Add files** button.

   This opens the **Waveform files** dialog, a standard dialog for opening files.

4. Select the waveform file(s) and click **Open**.

   This adds the file(s) to the list of waveform files in the **Link to waveform files** dialog.

5. Click OK.

   This links trajectories to the waveform files and updates the trajectory files in the active trajectory directory. An information dialog shows the amount of trajectories that are effected by the process.

SETTING	EFFECT
Apply to	Trajectories linked to waveform files: **All trajectories** or **Selected only**.
Add files	Opens a dialog for selecting waveform files.
Remove	Removes a selected waveform file from the list.

Renumber trajectories

Renumber trajectories command applies a new numbering to trajectories. It assigns increasing numbers to the trajectories according to their order in the **Trajectories** window.

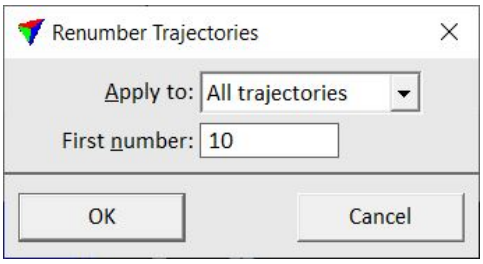
Renumbering can be useful, for example after a new sorting has been applied to the active trajectories using the Sort command.

To renumber trajectories:

1. (Optional) Select the trajectory file(s) you want to sort in the **Trajectories** window.

2. Select **Renumber trajectories** command from the **Tools** pulldown menu.

   This opens the **Renumber Trajectories** dialog:



3. Define settings and click OK.

   This assigns new numbers to the trajectories and updates the trajectory files in the active trajectory directory. An information dialog shows the number of the trajectories effected by the process.

SETTING	EFFECT
Apply to	Trajectories effected by the process: **All trajectories** or **Selected only**.
First number	Number of the first trajectory in the list.

Smoothen stops

Smoothen stops command detects stops that occurred during a survey drive and creates a correction for trajectory drift at stop locations. The process looks into trajectory information and finds stop intervals. Then, it computes a smooth curve through each stop interval. The correction is the difference between the smooth curve and the actual trajectory positions.

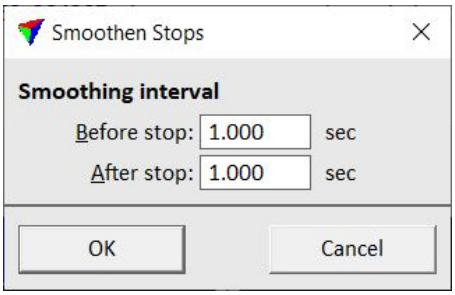
The corrections are stored in a .TMS file and can be applied by using the Apply correction macro action or the Apply corrections tool of TerraMatch.

At the moment, the software stores only the elevation correction in the correction file. The correction of a horizontal drift is difficult to compute and may not provide a valid improvement for the data.

To create a corrections file for trajectory drift at stop locations:

1. Select **Smoothen stops** command from the **Tools** pulldown menu.

   This opens the **Smoothen Stops** dialog:

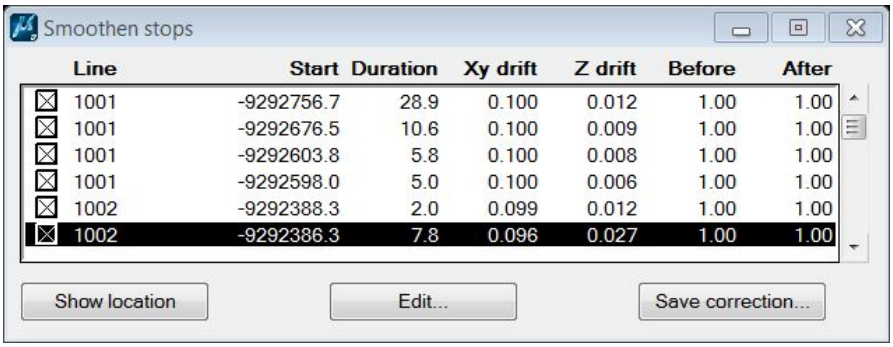


2. Define settings and click OK.

SETTING	EFFECT
Before stop	Number of seconds before a stop that is included in the time interval for computing the corrections.
After stop	Number of seconds after a stop that is included in the time interval for computing the corrections.

The process starts. If the software does not find any stops in the active trajectories, an information dialog is displayed.

If stops are found, the corrections are computed and another **Smoothen stops** dialog opens:

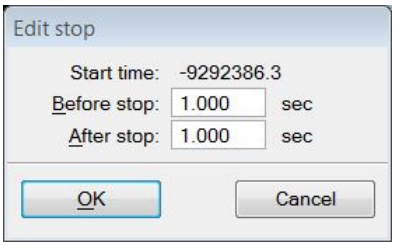


The dialog shows for each stop the trajectory line number, the start time, duration in seconds, XY and Z drift values, and the number of seconds before and after the stop time that are included in the correction computation.

You can switch on/off the correction for stop locations by toggling the cross on or off in the field before the line number. If the cross is off, the correction is not stored in the correction file.

To show the location of a stop interval, select a line in the **Smoothen stops** dialog. Click on the **Show location** button and move the mouse pointer into a view. This highlights the selected interval location with a cross. Place a data click inside the view in order to center the location in the view.

3. (Optional) Click on the **Edit** button to open the **Edit stop** dialog:



You can define new values for the time interval that is used to compute correction values.

4. Click on the **Save corrections** button.

   This opens the **Correction file** window, a standard dialog for saving files.

5. Define a storage location and name for the corrections file and click **Save**.

Split

Split command can be used to split a trajectory manually into smaller parts. The command lets you define the location for splitting the trajectory with a data click. This can be used if the automatic methods for splitting trajectories do not apply or lead to the requested result.

To split a trajectory:

1. Select **Split** command from the **Tools** pulldown menu.

   If the mouse pointer is moved inside a CAD file view, the closest trajectory is highlighted.

2. Identify the trajectory to split with a data click.

   A red cross shows dynamically the split location.

3. Define the position at which to split the trajectory with a data click.

   This splits the trajectory at the given position. The application deletes the old trajectory file and creates two new files in the active trajectory directory.

There are also automatic ways to split a trajectory. See Cut turnarounds, Delete by polygons, and Split at laser gaps commands for more information.

Split at laser gaps

Split at laser gaps command cuts trajectories if there is a gap in laser data. It removes part of trajectories where there is no laser data available.

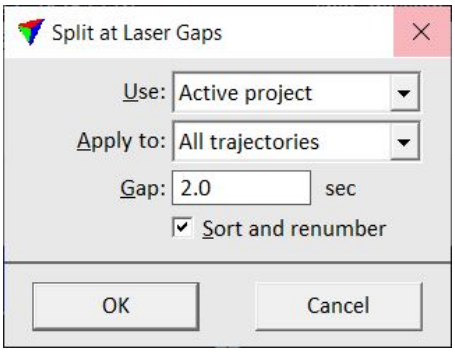
The command requires points or a project loaded into TerraScan.

To split trajectories at laser gaps:

1. (Optional) Select the trajectory file(s) you want to cut in the **Trajectories** window.

2. Select **Split at laser gaps** command from the **Tools** pulldown menu.

   This opens the **Split at Laser Gaps** dialog:



3. Define settings and click OK.

   This deletes all trajectory parts outside the area covered by laser data. The application deletes the old trajectory file(s) and creates new files in the active trajectory directory. An information dialog shows the result of the process.

SETTING	EFFECT
Use	Source data used for detecting gaps in a point cloud: **Active project** or **Loaded points**. An option is only available if either a project or a point cloud is loaded into TerraScan.
Apply to	Trajectories effected by the process: **All trajectories** or **Selected only**.
Gap	Minimum time interval that defines a gap in the laser data. If there is data missing for a longer time, the trajectory is split.
Sort and renumber	If on, the trajectories are sorted by time and renumbered after splitting.

Thin positions

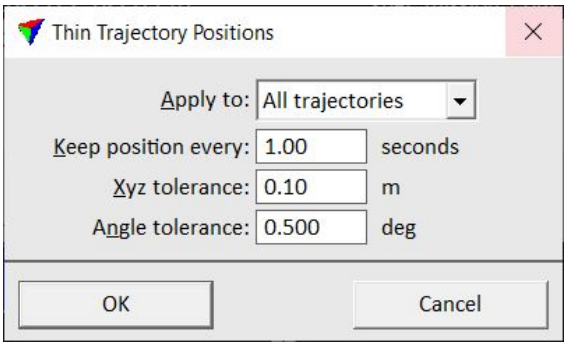
Thin positions command removes positions from trajectories. The process removes positions as long as the trajectory line stays within given accuracy tolerance values.

To thin positions of trajectories:

1. (Optional) Select the trajectory file(s) you want to thin in the **Trajectories** window.

2. Select **Thin positions** command from the **Tools** pulldown menu.

   This opens the **Thin Trajectory Positions** dialog:



3. Define settings and click OK.

   This removes unnecessary trajectory positions and updates the trajectory files in the active trajectory directory. An information dialog shows the number of the trajectories effected by the process.

SETTING	EFFECT
Apply to	Trajectories effected by the process: **All trajectories** or **Selected only**.
Keep position every	Time interval between two trajectory positions to keep.
Xyz tolerance	Maximum allowed locational change of the trajectory caused by thinning.
Angle tolerance	Maximum allowed angular change of the trajectory caused by thinning.

Thinning can also be applied to a trajectory in the import process. See Import files command for more information.

Transform

Transform command applies a transformation to the coordinates of a trajectory. The transformation can be, for example, a change of the projections system or any other transformation defined in Coordinate transformations / Transformations category of TerraScan **Settings**.

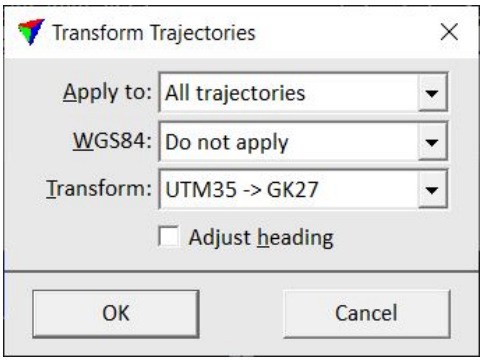
You can find more detailed information about transformations in Chapter Coordinate Transformations.

To transform a trajectory:

1. (Optional) Select trajectory file(s) to transform.

2. Select **Transform** command from the **Tools** pulldown menu.

   This opens the **Transform Trajectories** dialog:



3. Define settings and click OK.

   The coordinates of the trajectory are changed. The modification is saved to the trajectory binary files in the active trajectory directory. An information dialog shows the number of effected trajectories.

SETTING	EFFECT
Apply to	Trajectories to transform: • **All trajectories** - all trajectories in the list. • **Selected only** - selected trajectories only.
WGS84	Target projection system for applying a transformation from WGS84 coordinates to the given projection system. You can choose from any of the built-in or user-defined projections systems which are set as active in Coordinate transformations / Built-in projection systems, Coordinate transformations / US State Planes, or Coordinate transformations / User projection systems categories of TerraScan **Settings**.
Transform	User-defined transformation to apply. The list includes transformations that is defined in Coordinate transformations / Transformations of TerraScan **Settings**.
Adjust heading	If on, the software applies a meridian convergence correction to heading values. The correction is based on the projection system set for **WGS84** or the coordinate transformation set for **Transform**.

Trajectory pulldown menu

Commands from the **Trajectory** pulldown menu are used to modify information of a trajectory, to assign a new trajectory number, to delete trajectory files, and to view the positions of a trajectory.

TO	USE COMMAND
Modify trajectory information	Edit information

TO	USE COMMAND
Assign a number by identifying a laser point	Assign number
Set accuracy estimate values for trajectory positions	Set accuracy
Delete selected trajectories	Delete
View positions of a selected trajectory	View positions

The commands of the pulldown menu are only available if at least one trajectory is selected in the **Trajectories** window.

Assign number

Assign number command lets you modify the trajectory number based on laser points loaded in TerraScan. It assigns the line number of the laser point closest to a data click as trajectory number to the selected trajectory.

The command applies the number only to one trajectory at a time. If several trajectories are selected, only the first one is effected by the number assignment.

To assign a trajectory number from laser points:

1. Load laser data into TerraScan.
2. Select a trajectory in the list of the **Trajectories** window.
3. Select **Assign number** command from the **Trajectory** pulldown menu.
4. Place a data click inside a view.

   This assigns the line number of the point closest to the data click location as number to the trajectory.

Delete

Delete command deletes one or more selected trajectory files. The entries for the files are removed from the list and the binary files are deleted from the hard disc.

To delete trajectories:

1. Select the trajectory file(s) in the list of the **Trajectories** window.
2. Select **Delete** command from the **Trajectory** pulldown menu.

   A dialog asks to confirm the removal of the file(s).
3. Click **Yes** in order to delete the selected file(s).

A dialog informs about the deletion process.

Edit information

Edit information command opens a dialog that contains basic information and attributes stored for a selected trajectory. The attributes can be modified. Modifications are immediately stored in the binary trajectory file.

In addition, up to two video files and a waveform file can be linked to a trajectory. The video file settings are not actively used by TerraScan but required for the compatibility of trajectories with TerraPhoto.

The **Waveform** field may contain a link to a single waveform file, for example, if one LAS or WDP file contains data of one line (trajectory). Alternatively, a file name with wild-cards can be used to establish a link between one trajectory and several waveform files. Example: "Laser_20160308_*.las" or "Laser_20160308_FL??_???.las" can be used to link waveform files with names "Laser_20160308_FL20_123.las" and "Laser_20160308_FL21_124.las" to one trajectory.

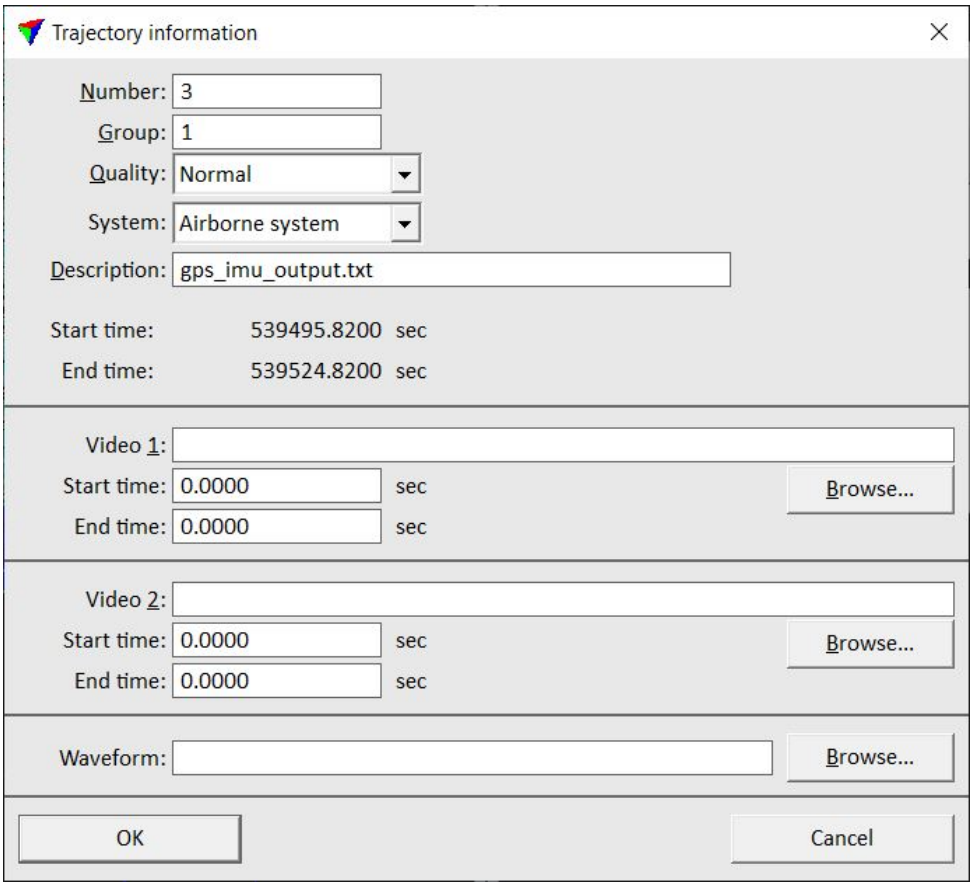
In order to link a waveform file to a trajectory, select the LAS file even if the waveform data is stored in WDP files. Only one file name in the field is allowed. If several waveform files need to be linked, select one file and then, replace the varying part(s) of the file name with wild-cards. If waveform files are linked in the Trajectory information dialog, no index files for waveform files are created.

To modify trajectory information:

1. Select a trajectory in the list of the **Trajectories** window.

2. Select **Edit information** command from the **Trajectory** pulldown menu.

   This opens the **Trajectory information** dialog:



3. Define settings and click OK.

   This modifies the information in the header of the corresponding .TRJ file.

SETTING	EFFECT
Number	Number of the trajectory.
Group	Group number of the trajectory. Group numbers may indicate, for example, different flight sessions and are used for TerraMatch processes.
Quality	Quality attribute of the trajectory. Quality may indicate, for example, the accuracy of trajectories and can be used for TerraMatch and TerraScan processes.
System	Scanner system used for data collection. This determines lever arm corrections that are added to trajectory positions and thus, effects the computation of the scanner location at the moment of measuring a laser point.
Description	Text that describes the trajectory. By default, the name of the raw trajectory file is used as descriptive text.

SETTING	EFFECT
Video 1	Primary video file linked to the trajectory. This is not actively used by TerraScan but required for the compatibility of trajectories with TerraPhoto.
Start time	GPS time stamp of the start position of Video 1.
End time	GPS time stamp of the end position of Video 1.
Video 2	Secondary video file linked to the trajectory. This is not actively used by TerraScan but required for the compatibility of trajectories with TerraPhoto.
Start time	GPS time stamp of the start position of Video 2.
End time	GPS time stamp of the end position of Video 2.
Waveform	Waveform data file linked to the trajectory. See Link to waveform files command and Chapter Waveform Processing for more information.

If you select several trajectories in the **Trajectories** window, the **Edit information** command opens the **Edit several trajectories** dialog. This dialog allows you to modify only settings which may apply for several trajectories, such as **Group**, **Quality**, and **System** settings.

Set accuracy

Set accuracy command lets you define accuracy estimates for trajectory positions. The selected estimate value is applied to all positions of one or more selected trajectories. The command can be used to set reasonable accuracy estimates for trajectories, if no system-based accuracy files are available. Alternatively, a minimum accuracy estimate value can be enforced. The accuracy estimates are treated like RMS values. A list of values is provided in the command's dialog from which you can select the best estimate.

If accuracy estimates are available from the post-processing software, the files can be imported by using the Import accuracy files command.

The accuracy estimate values are stored in the binary trajectory files. TerraScan stores only four values for each trajectory position: x/y, z, heading, roll/pitch.

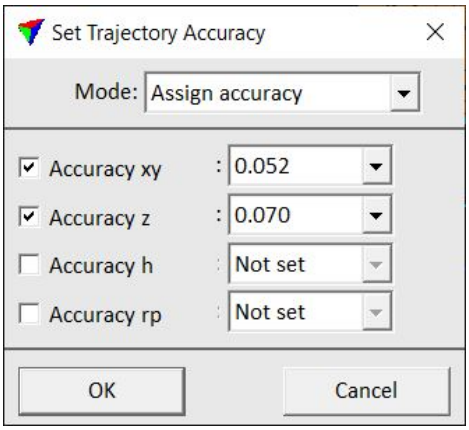
The information from the accuracy files is used for strip matching computations in TerraMatch and for drawing trajectories into the CAD file.

To set accuracy values for trajectory positions:

1. Select one or more trajectories in the list of the **Trajectories** window.

2. Select **Set accuracy** command from the **Trajectory** pulldown menu.

   This opens the **Set Trajectory Accuracy** dialog:



3. Switch on accuracy estimates that you want to define and select a value from the list.

4. Click OK.

   This assigns the values to the trajectory positions and saves the .TRJ file(s). An information dialog shows the number of effected trajectories.

SETTING	EFFECT
Mode	Determines whether a fixed accuracy estimate is set or a minimum value is enforced: • **Assign accuracy** - sets a fixed accuracy estimate value for all trajectory positions. • **Enforce minimum** - sets an accuracy estimate value for a trajectory position if the original value is smaller than the new given value.
Accuracy xy	If on, accuracy estimates for xy positions are set.
Accuracy z	If on, accuracy estimates for z positions are set.
Accuracy h	If on, accuracy estimates for heading attitude are set.
Accuracy rp	If on, accuracy estimates for roll and pitch attitude are set.

View positions

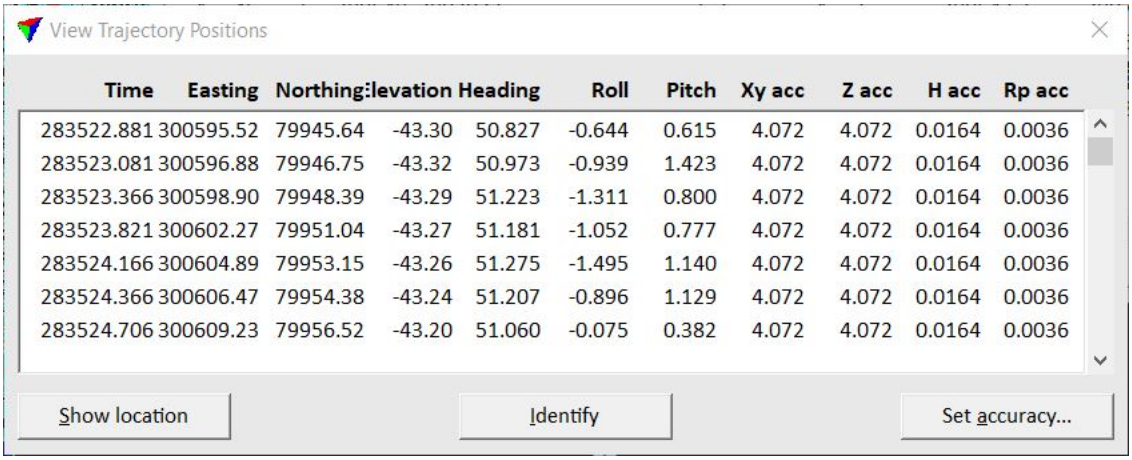
View positions command can be used to display the single positions of a trajectory file. The command opens a window that shows the list of positions and for each position the attributes stored in the trajectory file. This may include the time stamp, coordinate values, heading, roll, and pitch values, as well as RMS values.

To view trajectory positions:

1. Select a trajectory file in the list of the **Trajectories** window.

2. Select **View positions** command from the **Trajectory** pulldown menu.

This opens the **View Trajectory Positions** dialog which contains the list of trajectory positions:



Time	Easting	Northing	Elevation	Heading	Roll	Pitch	Xy acc	Z acc	H acc	Rp acc
283522.881	300595.52	79945.64	-43.30	50.827	-0.644	0.615	4.072	4.072	0.0164	0.0036
283523.081	300596.88	79946.75	-43.32	50.973	-0.939	1.423	4.072	4.072	0.0164	0.0036
283523.366	300598.90	79948.39	-43.29	51.223	-1.311	0.800	4.072	4.072	0.0164	0.0036
283523.821	300602.27	79951.04	-43.27	51.181	-1.052	0.777	4.072	4.072	0.0164	0.0036
283524.166	300604.89	79953.15	-43.26	51.275	-1.495	1.140	4.072	4.072	0.0164	0.0036
283524.366	300606.47	79954.38	-43.24	51.207	-0.896	1.129	4.072	4.072	0.0164	0.0036
283524.706	300609.23	79956.52	-43.20	51.060	-0.075	0.382	4.072	4.072	0.0164	0.0036

To show the location of a trajectory position, select a line in the list of positions. Click on the **Show location** button and move the mouse pointer into a view. This highlights the selected position with a cross. Place a data click inside a view in order to center the display at the selected position.

To identify a position, click on the **Identify** button and place a data click close to a trajectory in a view. This selects the line of the position closest to the data click in the View trajectory positions dialog.

You can set the accuracy estimates (RMS values) for one or more selected trajectory positions by using the **Set accuracy** button. The options for defining an accuracy estimate are the same as for the Set accuracy command.

View pulldown menu

Commands of the **View** pulldown menu are used to sort trajectory files in the list and to select attribute fields for being displayed in the window.

TO	USE COMMAND
Sort trajectories according to specific criteria	Sort
Select fields to be displayed in the window	Fields

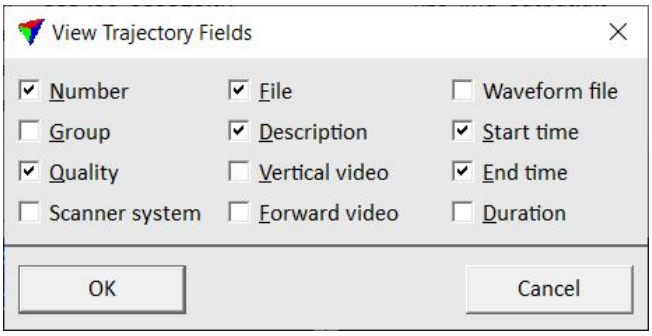
Fields

Fields command lets you select which attributes are displayed for each trajectory in the **Trajectories** window.

To select visible fields:

1. Select **Fields** command from the **View** pulldown menu.

   This opens the **View Trajectory Fields** dialog:



2. Select fields and click OK.

FIELD	DESCRIPTION
Number	Trajectory number.
Group	Group number of the trajectory.
Quality	Attribute that indicates the quality of the trajectory.
Scanner system	Scanner system assigned to the trajectory.
File	Name of the trajectory binary file on the hard disk.
Description	Description of the trajectory given in the **Trajectory information** dialog.
Vertical video	Name of the video file defined as Video 1 in the **Trajectory information** dialog. See Edit information command.
Forward video	Name of the video file defined as Video 2 in the **Trajectory information** dialog. See Edit information command.
Waveform file	Path and name of a waveform file linked to the trajectory.
Start time	Time stamp at the start of the trajectory.
End time	Time stamp at the end of the trajectory.
Duration	Length of the trajectory in seconds.

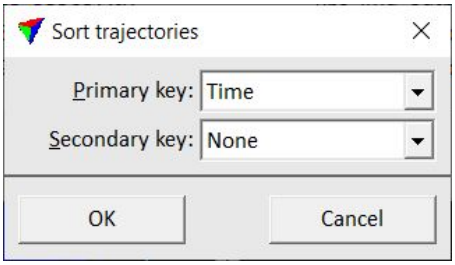
Sort

Sort command defines the display order of trajectory files in the list. The trajectories can be sorted by up to two attributes.

To sort trajectory files:

1. Select **Sort** command from the **View** pulldown menu.

This opens the **Sort trajectories** dialog:



2. Select a **Primary key** and **Secondary key** for sorting.

3. Click OK.

The display order of the trajectory files in the list is changed according to the settings.

SETTING	EFFECT
Primary key	Attribute used first for sorting the trajectories: • **Number** - increasing trajectory numbers. • **Group** - increasing group numbers. • **Time** - increasing time stamps.
Secondary key	Attribute used second for sorting the images: • See **Primary key** attributes. • **None** - no secondary key is used for sorting.

Fit Geometry Components

Preliminary draft

Note Lite, Not UAV, Not Spatix

This is an introduction to the Fit Geometry Components tool in TerraScan. The tool can be use to find a road geometry by fitting its components to the points found in a MicroStation line string. The application can generate horizontal and vertical road geometries. Normally, the horizontal geometry is solved first. The work on the vertical geometry only starts after the horizontal geometry has been finalized. A normal overall workflow in this case is:

1. Create a filtered survey vector.

2. Find preliminary horizontal geometry.

3. Modify the horizontal geometry.

4. Find preliminary vertical geometry.

5. Modify the vertical geometry.

6. Export your work.

However, if the horizontal geometry is not required, it is possible to start working on the vertical geometry once the filtered survey vector has been created. In this case the workflow is:

1. Create a filtered survey vector.

2. Find preliminary vertical geometry.

3. Modify the vertical geometry.

4. Export your work.

The modifications to the geometry components are stored in the CAD file. The saved components can be used to continue working on the geometry at a later point of time. The application also supports saving the geometry components into a text file in a proprietary file format. The vertical and horizontal geometries are saved into separate files. The application can then import geometries from files that are in its proprietary file format. Finally, the geometry components can be exported into selected exchange formats, such as simplified LandXML among others.

Content of this Chapter

This Chapter describes the recommended steps for producing the horizontal and vertical geometries. The first step in the generation of a road geometry is the filtering of the input survey vector which is described in Section Survey / Filter survey vector.

The generation of a horizontal geometry and the options for working with its components are discussed in Sections Horizontal / Create geometry and Modifying the horizontal geometry.

The generation of a vertical geometry and commands for working with its components are introduced in Sections Vertical / Create geometry / From horizontal components, Create geometry / From line string, and Modifying the vertical geometry. The commands for modifying vertical geometry components appear identical to the ones used for horizontal geometry. The main difference is:

- the horizontal geometry is compared to the filtered survey vector.
- the vertical geometry is generated using the elevation values of the filtered survey vector and the station values calculated using the horizontal geometry.

Section Activate an existing geometry shows how to continue with previously created geometries.

Section Tools for component modification provides detailed description of the commands for modifying the geometry components. Finally, all other commands included in the component fitting application are introduced in Section Other commands.

User settings

You can modify the colors and levels which are used by Fit Geometry Components in Component fitting / Colors category and Component fitting / Levels category of TerraScan **Settings**.

The initial values for the levels are:

FEATURE	LEVEL
Horizontal geometry	23
Vertical geometry	24
3D representation of the geometry	25
Vertical geometry frame	20
Filtered survey data	21
Internal vertical line string	22

The levels will be automatically created if they do not exist in the CAD file.

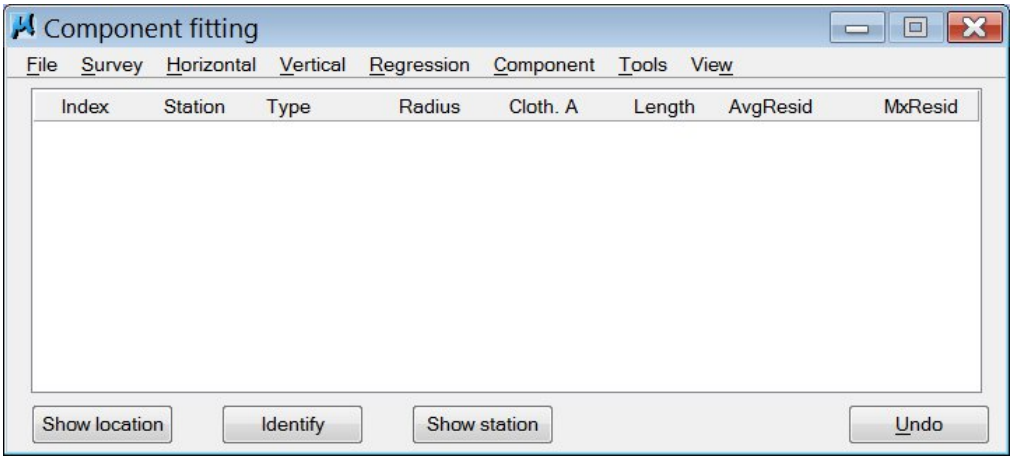
You can adjust settings for line weights and styles in Component fitting / Weights and styles category. The layout of the profile used to present the vertical geometry component fitting results is defined in Component fitting / Profile category.

If you want the application to save the results of component fitting automatically, you can enable this option in Component fitting / Operation category of TerraScan **Settings**.

Starting the Fit Geometry Components application

The application can be started from TerraScan Fit Geometry Components tool which is part of the Road toolbox.

This opens the **Component fitting** main window:



The main window contains pulldown menus with commands, buttons, and the list of geometry components. The list is displayed as soon as a geometry has been created. The menu commands are activated when their use becomes possible during the geometry generation progress. All commands can also be activated by key-in commands and most of them are specified in section Tools for component modification.

The following list provides an overview of the functionality included in the pulldown menus:

1. File

   Remove non-essential but helpful elements created during the work with geometry components from the CAD file.

   Export the geometry into a text file.

2. Survey

   Create a filtered version of the survey vector.

   View some basic properties of a CAD file line string.

   Calculate and display curvature information for the filtered line string.

3. Horizontal

   Create a preliminary horizontal geometry.

   Activate a horizontal geometry from the elements in the CAD file.

   Read horizontal geometry from a text file. Only the application's own file format is supported.

   Save the horizontal geometry into a text file.

4. Vertical

   Create a preliminary vertical geometry.

   Activate a vertical geometry from the elements of the CAD file.

   Read vertical geometry from a text file. Only the application's own file format is supported.

   Save the vertical geometry as a text file.

5. Regression

   Automatic modification of single or all components so that they follows the filtered line string more closely.

   Refitting the lengths of the segments.

6. Component

   Tools for modifying the circular arc radius of curvature, joining of components, and changing the type of a component. Note, that these tools make a modification considering only the continuity of the model, the continuity of its tangent and curvature, if applicable.

   Finding components using different criteria, for example the shortest component.

7. Tools

   Modifying some set of components of the current geometry automatically.

   Visualizing the residual or error of the fit.

   Modifying the weight of different points of the survey vector.

8. View

   Making the vertical or the horizontal geometry active.

See Component fitting main window for more information about the content of the main window after a geometry has been created.

Survey / Filter survey vector

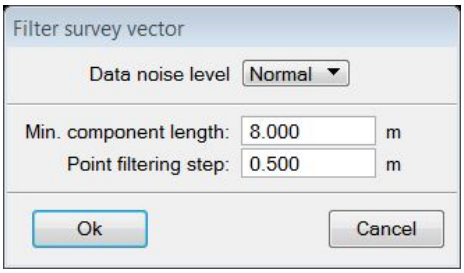
This is the first step in both of the workflows presented in the Chapter's introduction. Filter survey vector command starts from the selected MicroStation line string element. The filtering process modifies the line string in order to make it better suited for the task of creating geometry components. The goal is to get a line string that does not have vertices too close to each other or too far apart. In ideal case, this step would not be needed. However, since the application does not make any assumption on the source of the survey vector, this command is used to make sure that the input line string is reasonable.

To create a filtered survey vector

1. Select a line string element using MicroStation **Selection** tool.

2. Select **Filter survey vector** command from the **Survey** pulldown menu.

   This opens the **Filter survey vector** dialog:



3. Define settings and click OK.

SETTING	EFFECT
Data noise level	Controls, in part, how many points the application automatically adds to the line string using linear interpolation. • Typically, this parameter is **Normal**. • However, if there are gaps in the line string or the data is noisy, use **High**. • Value **Low** can be used if the component length is large compared to the average point distance and data is smooth.
Min. component length	Determines, together with **Data noise level**, how many points are added to the survey line string in the filtering process. The goal is to have approximately six points for each minimum component length. Typically, the value should be at maximum 0.5 * the known minimum component length. Larger values can cause the application to miss components which results in a poor-quality geometry.
Point filtering step	Defines the approximate minimum distance between the vertices in the filtered line string. For a line string with a large vertex density, the parameter can be used to filter out vertices and speed up the calculations. Typically, the value should be much smaller than the minimum component length, for example one sixth of the minimum component length.

The filtered line string should not be modified since any changes to its position or shape is directly translated to the geometry components.

Horizontal / Create geometry

Create geometry command generates a preliminary horizontal geometry.

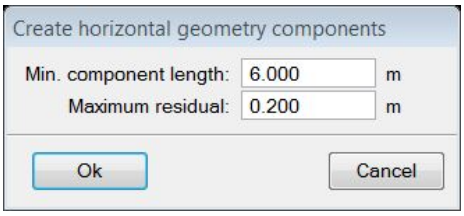
The preliminary geometry is a combination of the geometry components arcs and lines. You can include transition curves using the Tools / Change curved set mode command. Almost all commands for modifying a geometry work on curved component sets with and without transition curves. The transition curve type that is currently implemented is clothoid.

To generate a preliminary horizontal geometry:

1. Select the filtered survey vector element using MicroStation **Selection** tool.

2. Select **Create geometry** command from the **Horizontal** pulldown menu.

   This opens the **Create horizontal geometry components** dialog:



3. Define settings and click OK.

   This starts the automatic process for generating a preliminary geometry.

SETTING	EFFECT
Min. component length	The application attempts to generate a geometry that has components longer than this value. Usually, the components are much longer and this value only comes into effect in extreme cases. Typically, the value should be at maximum 0.5 * the known minimum component length. Larger values can cause the application to miss components and producing poor-quality geometry.
Maximum residual	Determines the goal for the level of agreement between the generated geometry and the vertices of the line string. In other words, the application adds components or continues to try to converge the components until this level of agreement is reached. Typically, the value is comparable to the noise level of the data. In test cases, average residuals were less than 0.1 m, maximum residuals about 0.2 m. A value that is too small can cause the preliminary geometry generation to be slow or fail.

Road fit: results

The **Road fit: results** dialog is opened after the preliminary horizontal geometry components have been created:

The dialog shows a general overview of the geometry. It lists the total number of the components in the geometry as well as the number of different types of components (line, circular arc, clothoid). The dialog also reports the differences between the geometry and the filtered line string by giving

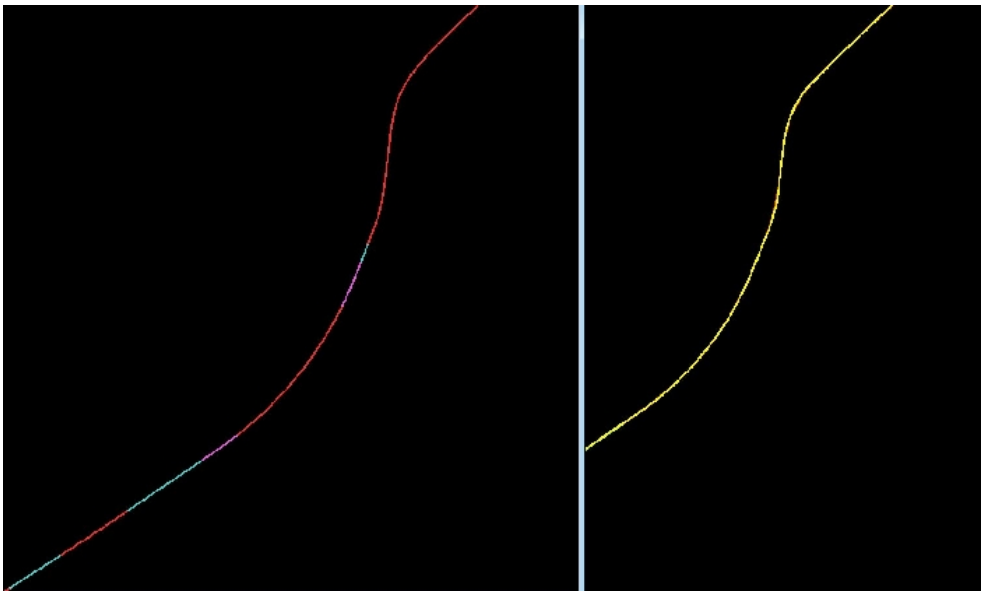
- the largest distance from the geometry to a point in the line string (**Worst point**)
- the largest average residual that a single component has (**Worst comp**.)
- the average residual (**Average residual**).

The two **Show** buttons can be used to pan the view to the component with the worst point-wise residual or the worst average residual. The values of the dialog are automatically updated as the geometry is modified. Checking these values is a fast way to detect problems in the geometry.

Current geometry

The horizontal geometry is stored in the CAD file (on level 23 as the default). The different components of the geometry are color-coded, the default colors for lines is light blue, for circular arcs red and for clothoids purple. The colors for components and the level for the geometry can be defined in TerraScan **Settings**, see also User settings.

An example of a segment of a road with different types of components can be seen in the illustration below.

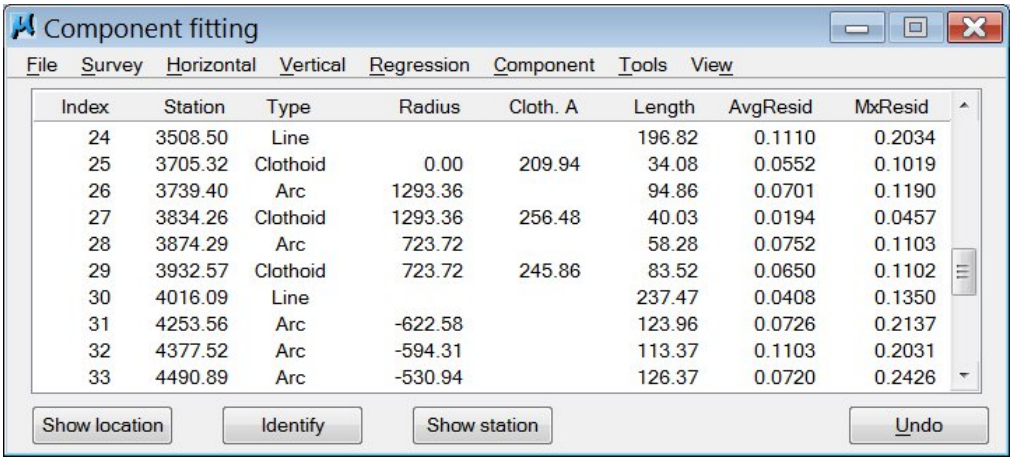


The elements written in the CAD file are standard MicroStation elements grouped together into a cell. For the component fitting application, they are mostly the visual representation of the geometry but include information about the components. This means that the components can be regenerated from the elements of the CAD file.

The geometry components in the CAD file are automatically updated if they are modified with the commands of the component fitting application. Modifications of the elements drawn for components in the CAD file are not recommended but they do not effect the components that are stored internally by the component fitting application. If components are redrawn by the component fitting application, they are always redrawn at their position that is stored internally.

Component fitting main window

After the preliminary geometry generation, the geometry components are listed in the Component fitting main window. The only way to select a component for modification is by selecting it in the list of this window.



Index	Station	Type	Radius	Cloth. A	Length	AvgResid	MxResid
24	3508.50	Line			196.82	0.1110	0.2034
25	3705.32	Clothoid	0.00	209.94	34.08	0.0552	0.1019
26	3739.40	Arc	1293.36		94.86	0.0701	0.1190
27	3834.26	Clothoid	1293.36	256.48	40.03	0.0194	0.0457
28	3874.29	Arc	723.72		58.28	0.0752	0.1103
29	3932.57	Clothoid	723.72	245.86	83.52	0.0650	0.1102
30	4016.09	Line			237.47	0.0408	0.1350
31	4253.56	Arc	-622.58		123.96	0.0726	0.2137
32	4377.52	Arc	-594.31		113.37	0.1103	0.2031
33	4490.89	Arc	-530.94		126.37	0.0720	0.2426

To show the location of a geometry component, select a line in the **Component fitting** window. Click on the **Show location** button and move the mouse pointer into a view. This highlights the selected

component. Place a data click inside the view in order to center the component in the view. If horizontal and vertical geometries are present, you can select in the **Show location** dialog which one to use for panning a view.

To identify a geometry component, click on the **Identify** button and place a data click close to a component in a view. This selects the corresponding line in the **Component fitting** window. Only components of the currently active geometry (vertical or horizontal) can be selected.

To display the station and the signed residual between the geometry and the line string for the currently active (horizontal or vertical) geometry, click on the **Show station** button and move the mouse pointer into a view. Positive residuals of a horizontal geometry mean that the geometry component is on the left of the line string. Positive residuals of a vertical geometry mean that the geometry component is below the line string.

Use the **Undo** button in order to undo a modification. You can undo up to 20 modifications.

VALUE	DESCRIPTION
Index	Index of the component. Numeration starts at 0.
Station	Starting station of the road component. The value indicates the distance from the first vertex of the filtered line string to the first vertex of the component measured along the filtered line string.
Type	Type of the component: **Line**, **Arc**, or **Clothoid**. Arc means a circular arc.
Radius	Starting radius of curvature of the component. The sign of the radius indicates the direction of the turn (counter-clockwise or clockwise). In the case of circular arcs, the starting radius is always finite. However, 0.00 is displayed for circular arcs if the value for the radius is larger than the number of digits available in the field.
Cloth. A	Value of the clothoid parameter.
Length	Length of the component. It is possible to end up with 0.0 length components when components are modified. You should try to join the zero-length components with other components.
AvgResid	Average distance between the component and vertices of the line string that are closer to this component than any other. Value 0.00 means that none of the vertices are closer to this component than any other component. You should try join such components with other components.

VALUE	DESCRIPTION
MxResid	Maximum distance from the component to a vertex of the line string that is closer to this component than any other. Value 0.00 means that none of the points are closer to this component than any other component.

Modifying the horizontal geometry

There are two alternative goals for modifying the horizontal geometry:

- obtaining a continuous curvature geometry. This means that the geometry includes transition curves between a line and a circular arc as well as between circular arcs with different radii. At the moment, the only supported transition curve type is a clothoid.
- obtaining continuous tangent geometry. This means that the geometry contains lines and arcs but no transition curves.

The application fully supports geometry where a set of curved components with continuous curvature is connected with a line to another set of components with only continuous tangent.

The changes in the geometry can be followed by viewing the changes in the Component fitting main window, the Road fit: results dialog, and in the graphical display of the geometry components in MicroStation views. Additionally, you can display exaggerated residual vectors by using the Tools / Residual display command.

The workflow for producing a horizontal geometry given below aims at obtaining a continuous curvature geometry that represents the line string. The modification steps for going from a preliminary geometry to the final geometry may include in some order the following steps:

Basic workflow

1. Change circular arcs to lines.

   Tools / Remove arcs with large radius

   Component / Change selected

2. Break-up curved sets with a change in the sign of curvature.

   Tools / Break S-curve set

3. Join circular arcs with similar radii.

   Tools / Join arcs with similar radii

   Component / Join selected

4. Add transition curves, change the mode of the curved set from continuous tangent to continuous curvature (or back).

   Tools / Change curved set mode

Done repeatedly during the basic workflow

1. Refit the line components.

   Regression / Refit selected lines

   Regression / Refit all lines

2. Refit individual circular arcs or join several components into an arc.

   Regression / Refit selected components as arc

3. Refit the lengths of all components.

   Regression / Refit component lengths

4. Find an alternative set of components for a segment of the geometry

   Tools / Fit alternative components

Finalizing

In the very last part of the geometry modification process, the radii of the circular arcs can be changed to desired values.

- Tools / Fix to even radius
- Component / Modify selected
- Tools / Radius table

Vertical / Create geometry / From horizontal components

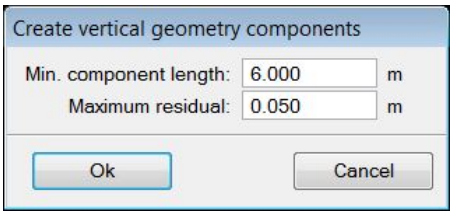
Create geometry / From horizontal components command generates a preliminary vertical geometry. It uses the filtered survey vector element and horizontal geometry components as basis for computing the vertical geometry components.

To generate vertical geometry from horizontal components:

1. Select the filtered survey vector element using MicroStation Selection tool.

2. Select **Create geometry / From horizontal components** command from the **Vertical** pulldown menu.

   This opens the **Create vertical geometry components** dialog:



3. Define settings and click OK.

4. Move the mouse pointer into a MicroStation view. The horizontal geometry components closest to the mouse pointer are highlighted.

5. Place a data click inside the view in order to select the highlighted components for which to create vertical geometry.

   This starts the automatic process for generating a preliminary vertical geometry.

6. Place the frame for displaying the profile of the vertical geometry components by a data click inside a MicroStation view.

   The elements drawn in the profile frame have station and elevation values scaled as specified in the Component fitting / Profile category of TerraScan **Settings**. The frame is drawn on the level defined in the Component fitting / Profile category of TerraScan **Settings**.

SETTING	EFFECT
Min. component length	The application attempts to generate a geometry that has components longer than this value. Usually, the components are much longer and this value seldom comes into effect.
Maximum residual	Determines the goal for the level of agreement between the generated geometry and the vertices of the line string. In other words, the application adds components or continues to try to converge the components until this level of agreement is reached. Typically, the maximum residual that can be achieved for the vertical geometry is smaller than for the horizontal geometry.

Create geometry / From line string

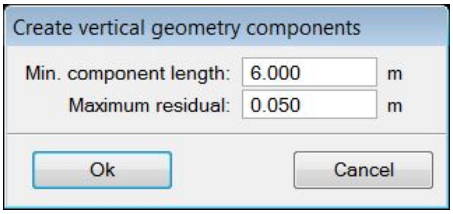
Create geometry / From line string command generates a preliminary vertical geometry. It uses only the filtered survey vector element as basis for computing the vertical geometry components.

To generate vertical geometry directly from the filtered survey line string:

1. Select the filtered survey vector element using MicroStation **Selection** tool.

2. Select **Create geometry / From line string** command from the **Vertical** pulldown menu.

   This opens the **Create vertical geometry components** dialog:



3. Define settings and click OK.

This starts the automatic process for generating the preliminary geometry.

4. Place the frame for displaying the profile of the vertical geometry components by a data click inside a MicroStation view.

   The elements drawn in the profile frame have station and elevation values scaled as specified in the Component fitting / Profile category of TerraScan **Settings**. The frame is drawn on the level defined in the Component fitting / Levels category of TerraScan **Settings**.

SETTING	EFFECT
Min. component length	The application attempts to generate a geometry that has components longer than this value. Usually, the components are much longer and this value only comes into effect in extreme cases.
Maximum residual	Determines the goal for the level of agreement between the generated geometry and the vertices of the line string. In other words, the application adds components or continues to try to converge the components until this level of agreement is reached. Typically, the maximum residual that can be achieved for the vertical geometry is smaller than for the horizontal geometry.

Modifying the vertical geometry

The vertical geometry can be modified with the same tools as the horizontal geometry. However, the assumption is that the final result does not include transition curves, such as clothoids.

The modification steps for going from a preliminary vertical geometry to the final geometry may include in some order the steps listed in the next three Sections.

Basic workflow

1. Change circular arcs to lines.

   Tools / Remove arcs with large radius

   Component / Change selected

2. Break-up curved sets with a change in the sign of curvature.

   Tools / Break S-curve set

3. Join circular arcs with similar radii.

   Tools / Join arcs with similar radii

   Component / Join selected

Done repeatedly during the basic workflow

1. Refit the line components.

   Regression / Refit selected lines

   Regression / Refit all lines

2. Refit individual circular arcs.

   Regression / Refit selected components as arc

3. Refit the lengths of all components.

   Regression / Refit component lengths

4. Find an alternative set of components for a segment of the geometry

   Tools / Fit alternative components

Finalizing

In the very last part of the geometry modification process, the radii of the circular arcs can be changed to desired values.

- Tools / Fix to even radius
- Component / Modify selected
- Tools / Radius table

Activate an existing geometry

You can activate previously generated geometries, for example if your work with the Component fitting application has been interrupted. After activating a geometry, the geometry components are displayed again in the **Component fitting** main window and can be modified.

Only one geometry, horizontal or vertical, can be active at a time. Use the View / Horizontal components or View / Vertical components commands in order to activate the other geometry for modification.

To continue the work on previously generated geometries:

1. Select the filtered survey vector in the CAD file by using the MicroStation **Selection** tool.

2. Select **Activate geometry** command from the **Horizontal** pulldown menu in order to activate a horizontal geometry.

   OR

3. Select **Activate geometry** command from the **Vertical** pulldown menu in order to activate a vertical geometry.

4. Identify the geometry components drawn in the CAD file with a data click inside a view.

This lists the geometry components in the **Component fitting** main window. You can continue with modifications of the components as described in Sections Modifying the horizontal geometry and Modifying the vertical geometry. If the vertical geometry is associated with a horizontal geometry, the horizontal geometry should still be present in the CAD file and is automatically available for activation.

Tools for component modification

This Section describes commands of the **Component fitting** main window that can be used to modify horizontal and vertical geometry components. Many commands require the selection of one or more components in the main window's list of components. These commands may be inactive if there is no selection of components made beforehand.

Component / Change selected

The operation of **Change selected** command depends on the type of the selected components.

Key-in: road change components

Change into a line

The application tries to change the components into a line if

- the selection is a circular arc or a set of circular arcs that belong to a set of curved components with continuous tangent.
- the selection includes a line.

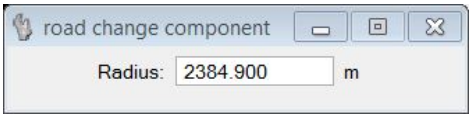
In this case, the change is attempted automatically and you have only the option of accepting or rejecting it.

Change into a circular arc

The application tries to change the components into a circular arc if

- the selection is part of a set of curved components with continuous curvature

In this case, the command opens the **Road change component** dialog:



The **Radius** value shown in the dialog is an estimate made by the application. The sign of the radius is also determined automatically. You may set the new radius of the circular arc to a specific value by typing the value into the **Radius** field. Changing curved components into a line can be attempted by setting the radius to 0.00 in the **Road change component** dialog.

The change is attempted and if a solution is found, a preview of the modified component(s) is shown in the MicroStation view. If you accept the modification by a data click, the component list is updated.

Small radius values are more likely to produce an acceptable solution.

Component / Insert line between arcs

Insert line between arcs command can be used to insert a line, if possible, between two circular arcs. It only considers the geometry components and disregards the original line string. This means, that the command may create components that do not fit well to the line string. However, it can be used to force the insertion of a line component in cases where other commands fail.

The insertion of a line is only possible between two connected arcs. This means that the command does not work if the arcs are connected by a clothoid.

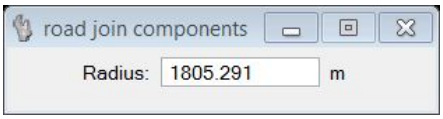
Component / Join selected

The operation of **Join selected** command depends on the type of selected components.

Key-in: road join segments

Join circular arcs

If the selected components include circular arcs, the elements are joined into a single circular arc. In this case, the command opens the **Road join components** dialog:



The **Radius** value shown in the dialog is an estimate made by the application. The sign of the radius is determined automatically. You may set the new radius of the circular arc to a specific value by typing the value into the **Radius** field.

The change is attempted and if a solution is found, a preview of the modified component(s) is shown in the MicroStation view. If you accept the modification by a data click, the component list is updated.

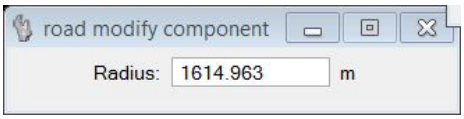
Small radius values are more likely to produce an acceptable solution.

Join clothoids

If the selection only includes clothoids, there is no option to set the radius and the dialog does not appear. You have only the option of accepting or rejecting the result.

Component / Modify selected

Modify selected command is used to modify the radius of a single selected circular arc component. It opens the **Road modify component** dialog:



The **Radius** value shown in the dialog is an estimate made by the application. The sign of the radius is also determined automatically. You may set the new radius of the circular arc to a specific value by typing the value into the **Radius** field.

If a continuous solution is found, a preview of the modified component is shown in the MicroStation view. If you accept the modification by a data click, the arc radius is changed and the component list is updated.

Key-in: road modify component

Regression / Refit all lines

Refit all lines command refits all line components to the vertices of the line string. Often, this is a quick way to make a significant improvements in the accuracy.

The command is most effective for lines connecting two curved sets with only continuous tangent. It modifies the radius of the neighbouring circular arcs. This means that this command should not be used after radius values have been set to specific values manually.

In the case the residuals in the components are large, the command may reduce the neighbours of the line to zero length components. These components should then be removed by using the Component / Join selected command.

The command does not have a dialog.

Key-in: road fit all lines

Regression / Refit component lengths

Refit component lengths command improves the fit between the geometry components and the vertices of the line string by adjusting the lengths of the components.

The local convergence of the process stops if

- the length of one of the components approaches zero.
- one of the components does not have any vertices associated with it.

In these cases you should try to remove the components by joining them to other components.

Occasionally, an improvement in the accuracy for one part of the geometry can result in a worse accuracy for other parts. You should always check the overall changes after using this command.

The command does not have a dialog.

Key-in: road refit lengths

Regression / Refit selected components as arc

Refit selected components as arc command can be used to refit the radius of a single circular arc or to join selected components into a circular arc with optimal radius. The process automatically searches for a circular arc radius that locally improves the fitting between a geometry component and the vertices of the line string.

The automatic process does not always find the solution that minimizes the residuals. In these cases, you may use Component / Modify selected or Component / Change selected commands in order to set the radius of the circular arc manually.

The command does not have a dialog.

Key-in: road refit as arc

Regression / Refit selected lines

Refit selected lines command can be used to refit selected line components to the vertices of the line string.

The command is most effective for lines connecting two curved segments with only continuous tangent. It modifies the radius of the neighbouring circular arcs. This means that the command should not be used after radius values have been set to specific values.

In the case the residuals of the components are large, the command often reduces the neighbours of the line to zero length components. These components should then be removed by using the Component / Join selected command.

The command does not have a dialog.

Key-in: road fit selected lines

Tools / Break S-curve set

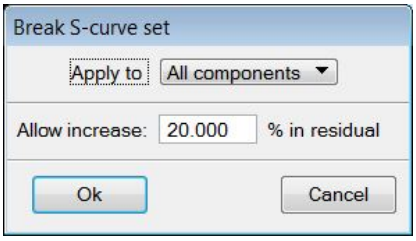
Break S-curve set command is used for breaking-up curved sets with a change of curvature (circular arcs with opposite signs for radius values). The process may change one or two of the circular arcs into a line or add a line between two arcs.

To break-up S-curve sets automatically:

1. (Optional) Select circular arc components in the main window of the application.

2. Select the **Break S-curve set** command from the **Tools** pulldown menu.

   This opens the **Break S-curve set** dialog:



3. Define settings and click OK.

   This starts the process.

SETTING	EFFECT
Apply to	Defines to which components the process is applied: **All components** or Selected **components**.
Allow increase	Determines the maximum allowed increase in residual values caused by breaking-up S-curved sets.

Key-in: road break s-curve

Tools / Change curved set mode

Change curved set mode command lets you change the mode of a curved set from continuous tangent to continuous curvature or vice versa. The application checks curved sets in the selected components and tries to find a best fit by

- using *clothoid - circular arc - clothoid* combinations if the goal is a continuous curvature set.
- using a set of *circular arcs* if the goal is a continuous tangent set.

The command can cause relatively large increases in the residuals for the components created for the selected curved set.

The command does not have a dialog.

Key-in: road change local mode

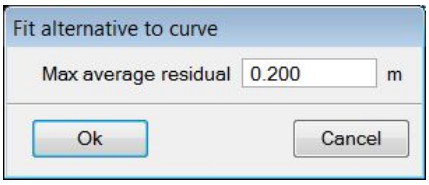
Tools / Fit alternative components

Fit alternative components command can be used to create an alternative set of components for a curved set or a line.

The operation of the command depends on the type of the selected components in the main window. The selection must only include either curved components or a line but not both, curved components and lines or several lines.

A curved set

If the selection includes components of a curved set (circular arcs, clothoids), the command opens the **Fit alternative to curve** dialog:



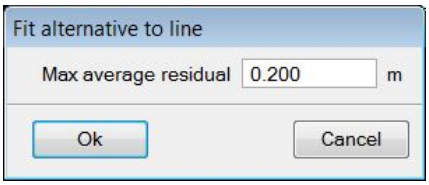
The **Max average residual** value determines the maximum allowed average residual of the alternative components.

The application tries to find an alternative set of components for the curve set that includes the selected component(s). The analysis of the maximum average residual is done for the curved set and the line components before and after it. If a solution is found that fulfills the accuracy requirement, the list of components in the main window is updated.

The lines before and after the curved set can dominate in the accuracy check. This means that it is unrealistic to try to find alternative components for a curved set that are more accurate than the lines before and after it.

A line

If the selection includes a line component, the command opens the **Fit alternative to line** dialog:



The **Max average residual** value determines the maximum allowed average residual of the alternative components.

The application tries to find an alternative set of components for the selected line. The result normally includes both, lines and arcs. The analysis of the maximum average residual is done for the line and the arc components before and after it. If a solution is found that fulfills the accuracy requirement, the list of components in the main window is updated.

The command works only for lines that are connected to arcs. Lines connected to clothoids are not yet supported.

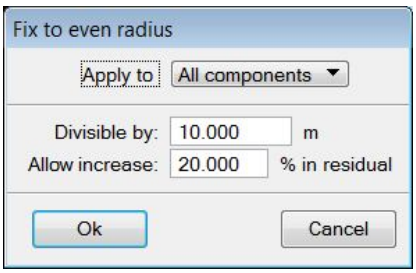
Tools / Fix to even radius

Fix to even radius command is used in the last stage of the geometry production workflow. It quickly changes the radii of all or the selected circular arcs to the closest even meter, five meter, or any other radius value.

To fix radii to even values automatically:

1. (Optional) Select circular arc components in the main window of the application.
2. Select the **Fix to even radius** command from the **Tools** pulldown menu.

   This opens the **Fix to even radius** dialog:



3. Define settings and click OK.

   This starts the process.

SETTING	EFFECT
Apply to	Defines to which components the process is applied: **All components** or **Selected components**.
Divisible by	Defines the common factor for all radii.
Allow increase	Determines the maximum allowed increase in residual values caused by fixing radius values of arcs.

Key-in: road arcs even

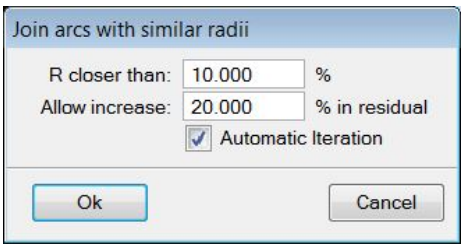
Tools / Join arcs with similar radii

Join arcs with similar radii command joins pairwise neighbouring circular arcs with similar radii into a single circular arc. It is applied to all circular arc components.

To join arcs with similar radii automatically:

1. Select the **Join arcs with similar radii** command from the **Tools** pulldown menu.

   This opens the **Join arcs with similar radii** dialog:



2. Define settings and click OK.

   This starts the process.

SETTING	EFFECT
R closer than	Defines the maximum relative difference in radius values that is allowed between of two arcs to be joined.
Allow increase	Determines the maximum allowed increase in residual values caused by joining two arcs.
Automatic iteration	Usually, the command needs to run more than once in order to join all components with similar radii. If this option is on, the iteration is continued automatically until no more arcs can be changed.

Key-in: road arcs join

Tools / Radius table

Sub-commands of the **Radius table** command can be used to apply radius values specified in a text file to circular arc components of the active geometry. The text file must contain only the radius values separated by space or line break.

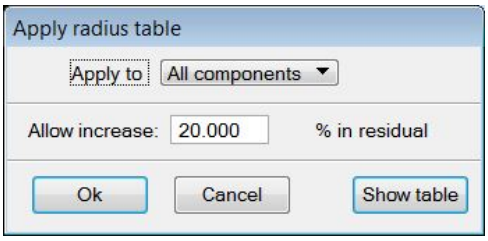
To apply radius values from a text file:

1. Select **Radius table / Read radius table...** command from the **Tools** pulldown menu.

2. (Optional) Select circular arc components in the main window of the application.

3. Select **Radius table / Apply radius table** command from the **Tools** pulldown menu.

   This opens the **Apply radius table** dialog:



4. Define settings and click OK.

   This starts the process.

SETTING	EFFECT
Apply to	Defines to which components the process is applied: **All components** or **Selected components**.
Allow increase	Determines the maximum allowed increase in residual values caused by applying radius values of arcs.
Show table	Opens a preview of the radius values read from the text file.

Tools / Remove arcs with large radius

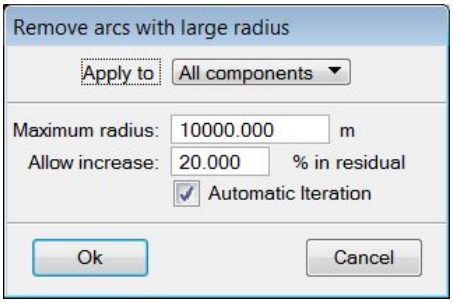
Remove arcs with large radius command offers a fast way to change circular arcs with large radii to lines.

To remove arcs with large radii automatically:

1. (Optional) Select circular arc components in the main window of the application.

2. Select the **Remove arcs with large radius** command from the **Tools** pulldown menu.

   This opens the **Remove arcs with large radius** dialog:



3. Define settings and click OK.

This starts the process.

SETTING	EFFECT
Apply to	Defines to which components the process is applied: **All components** or **Selected components**.
Maximum radius	Maximum allowed radius of arcs. If the radius of an arc is larger, the application tries to convert the arc into a line.
Allow increase	Determines the maximum allowed increase in residual values caused by converting arcs into lines.
Automatic iteration	Usually, the command needs to run more than once in order to get rid of large radius components. If this option is on, the iteration is continued automatically until no more arcs can be changed. However, this can cause the residuals to accumulate.

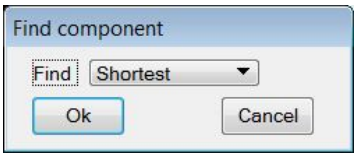
Key-in: road arcs max

Component / Find component

Find component command can be used to find the component that

- is shortest.
- is longest.
- has the largest radius of curvature.
- has the smallest radius of curvature.
- does not have any vertices of the lines string associated with.

The command automatically selects the first component that matches the selected criteria in the main window.



File / Close all

Close all command can be used to remove all unnecessary results generated by the component fitting application from the CAD file.

I/O-commands

Input/Output commands are used to import geometry components from text files, save or export geometry components into text files, or to read in a line string that can serve as starting element for the component fitting process.

TOOL	DESCRIPTION
Horizontal/Open geometry	Imports a horizontal geometry from a text file in the application's proprietary file format. ***Key-in: road read horizontal components***
Vertical/Open geometry	Imports a vertical geometry from a text file in the application's proprietary file format. ***Key-in: road read vertical components***
Horizontal/Save geometry as	Exports a horizontal geometry into a text file in the application's proprietary file format. ***Key-in: road save horizontal components***
Vertical/Save geometry as	Exports a vertical geometry into a text file in the application's proprietary file format. ***Key-in: road save vertical components***
File/Export to LandXML	Exports geometries into a LandXML file. ***Key-in: road write landxml***
File/Export to Tekla 11/12	Exports geometries to Tekla 11/12 format ***Key-in: road write tekla1112***

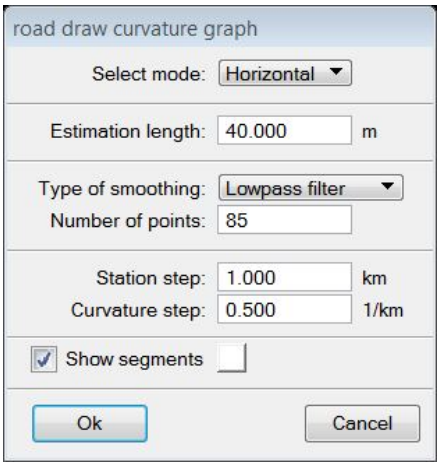
Survey / Draw curvature graph

Draw curvature graph command can be used to display the curvature information of a filtered line string and the geometry components in a graph.

To draw a curvature graph:

1. Select a line string element using the MicroStation **Selection** tool.
2. Select **Draw curvature graph** command from the **Survey** pulldown menu.

   This opens the **Draw curvature graph** dialog:



3. Define settings and click OK.

4. Define the location of the graph drawing with a data click in a MicroStation view.

   This draws the graph as MicroStation cell element into the CAD file. The graph is drawn on the active level.

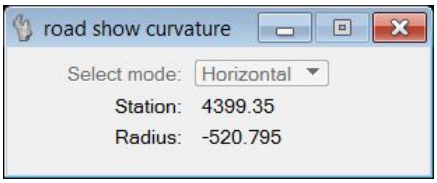
SETTING	EFFECT
Select mode	Defines for which geometry the graph is drawn: **Horizontal** or **Vertical**.
Estimation length	Specifies the length of a line string segment used for estimating the curvature. Higher values reduce the oscillations in curvature values.
Type of smoothing	Determines the smoothing mode: **None**, **Moving average**, **Moving median**, or **Lowpass filter**.
Number of points	Number of points used for smoothing.
Station step	Determines the horizontal scale markings in the graph.
Curvature step	Determines the vertical scale markings in the graph.
Show segments	If on, components of an existing geometry are drawn in the graph using the selected color.

The application uses the same graph drawing if the command is performed for a second time during the same work session. The connection gets lost if the **Component fitting** main window is closed.

Survey / Show curvature

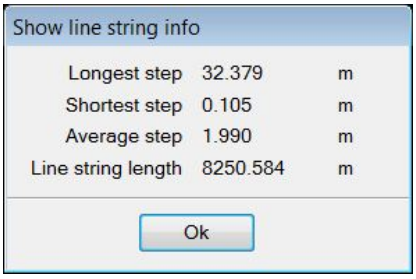
Show curvature command can be used to display the curvature along the line string. The command requires that the Survey / Draw curvature graph command has been performed before and that the graph is drawn in the CAD file.

The curvature is displayed dynamically for the station closest to the mouse pointer and calculated using the parameters specified in the dialog of the Survey / Draw curvature graph command. The station value can be selected either from the line string or from the curvature graph.



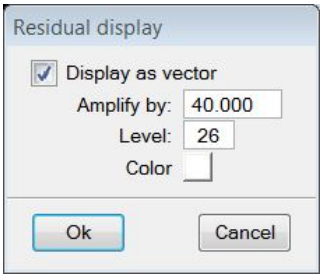
Survey / Show line string info

Show line string info command opens a dialog that shows general information about vertices of a selected line string. This includes the longest, shortest, and average distances between consecutive vertices, and the length of the line string.



Tools / Residual display

Residual display command shows an exaggerated residual vector for the active geometry. The settings for drawing residual vector into the CAD file are defined in the **Residual display** dialog:



SETTING	EFFECT
Display as vector	If on, the residual vector is displayed in the CAD file. Switch this off if you want to remove the display of the residual vector.
Amplify by	Scaling factor of the residual vector.
Level	Level on which the residual vector is drawn. Refers to a MicroStation level number.
Color	Color of the residual vector. Uses the active color table of MicroStation.

You can also remove the display of a residual vector by using File / Close all command.

Tools / Set point weights

Set point weights command can be used to specify weights for the vertices of the line string. The weights are used in the component fitting process. You can define separate weights for the horizontal and vertical geometries.

The default weight or a vertex is 1.0. If the components should follow some vertices more closely, set the weight of these vertices to a value larger than 1.0. To decrease the weight of the vertices for the fitting process, set the weight to a value smaller than 1.0. Weight values must be larger than 0.0.

The weights are drawn as text elements into the CAD file. Only manually set weight values are drawn, the default weight of 1.0 is not drawn.

To set the weight of a vertex:

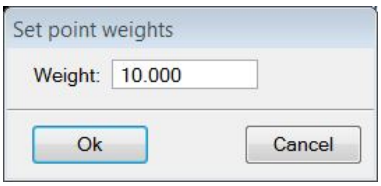
1. Select the filtered survey string using MicroStation **Selection** tool.

2. Select **Set point weights** command from the **Tools** pulldown menu.

3. Move the mouse pointer close to the line string. The vertex closest to the mouse pointer is highlighted by a small circle.

4. Select a vertex with a data click. You can select several vertices by pressing the <**Ctrl**> key while selecting a vertex.

   This opens the **Set point weights** dialog:



5. Define the weight value for the selected vertex (vertices) in the **Weight** field.

6. Click OK.

   The weight values are written into the CAD file next to the filtered survey vector. They are drawn as MicroStation text elements on the level defined in Component fitting / Levels category of TerraScan **Settings**. Weights of vertices for a vertical geometry are also displayed in the profile window.

View / Horizontal components

Horizontal command activates the horizontal geometry for modification. It also changes the content of the main window to display the horizontal geometry components.

View / Vertical components

Vertical command activates the vertical geometry for modification. It also changes the content of the main window to display the vertical geometry components.

Coordinate Transformations

TerraScan can apply a coordinate transformation to point clouds at different steps of the processing workflow, for example, when loading or importing points, working with the points in RAM, processing points in batch mode in a TerraScan project or with a macro step, or writing points to output files. Coordinate transformations may also be applied to trajectories.

TerraScan divides coordinate transformations into several categories:

- Projection system transformations - used to transform coordinates from one coordinate system to another.
- User-defined transformations - coordinate transformations which can be defined by a number of parameters or equations.
- Geoid adjustment - used to transform elevation values from one height model to another.
- Systematic elevation correction - applies a single correction value to elevation values.

Projection system transformations

The transformation of coordinates from one coordinate system to another is a common task. Usually, the coordinates of raw laser data or trajectories are given in WGS84 or some UTM projections system values. For data processing and/or delivery, it is often necessary to transform these coordinates into another (national) projection system.

Coordinates in WGS84 system can be provided as longitude, latitude, ellipsoidal elevation values or geocentric XYZ values. TerraScan automatically recognizes the coordinate value format when it reads the points or trajectories.

The transformation into the destination coordinate system is usually done when point cloud data or trajectories are imported into TerraScan, for example, at the beginning of the processing workflow, or when data is prepared for delivery.

A projection system has to be activated in Coordinate transformations / Built-in projection systems or Coordinate transformations / US State Planes categories of TerraScan **Settings**. Only active projection systems are available for transformations. If your local projection system is not implemented in TerraScan, you can def e it in Coordinate transformations / User projection systems category of TerraScan **Settings**.

After activating the projection system(s), you can define a transformation of type **Projection change** in Coordinate transformations / Transformations category of TerraScan **Settings**. See also Projection change transformation for more information.

User-defined transformations

There are different types of transformations that can be used to manipulate the coordinate values of point cloud data and trajectories in TerraScan. The implemented transformation types are:

- Linear transformation
- Known points transformation

- Xy multiply transformation
- 3D translate & rotate transformation
- 3D Affine transformation

You can define the values for the transformation parameters in Coordinate transformations / Transformations category of TerraScan **Settings**.

Geoid adjustment

The elevation values of raw laser data and trajectories are often provided as ellipsoidal height values. Usually, these values need to be transformed into orthometric values of a local height system.

For larger areas, the adjustment from ellipsoidal to orthometric height values can not be defined as one mathematical formula. Therefore, the elevation adjustment model needs to be defined by using local points for which the elevation difference between the height systems is known.

In TerraScan, the elevation adjustment can be performed for loaded points or for project blocks in batch mode. See Adjust to geoid command for loaded points, Adjust to geoid command for project blocks, and Adjust to geoid command for trajectories for more information.

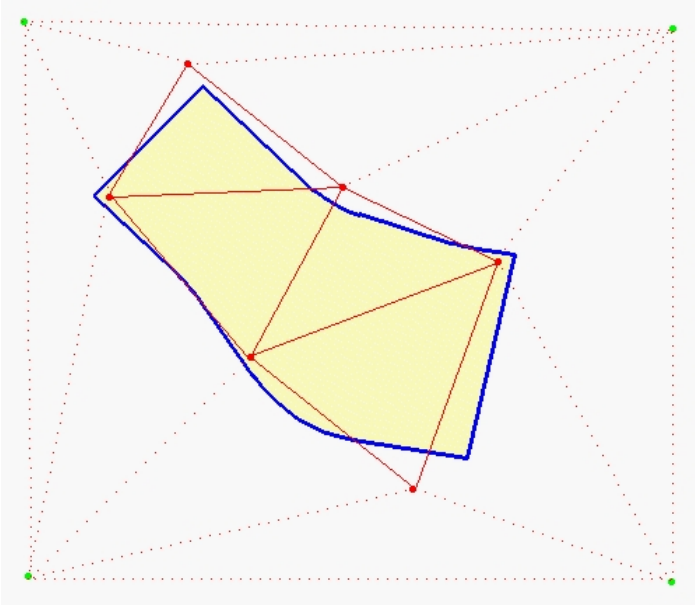
Elevation adjustment model

The input model for geoid adjustment must be provided in one of the following formats:

- **Points from file** - text file containing space-delimited X Y dZ- points.
- **TerraModeler surface** - triangulated surface model created from X Y dZ - points. The surface model in TerraModeler has the advantage that you can visualize the shape of the adjustment model.
- **Selected linear chain** - linear element of which the vertices represent the X Y dZ - points.

XY are the easting and northing coordinates of the geoid model points, dZ is the elevation difference between ellipsoidal and local heights at the location of each geoid model point. Intermediate adjustment values of the model are derived by aerial (text file or surface model as input) or linear (linear element as input) interpolation between the known geoid model points.

The figure below illustrates the aerial interpolation method. The yellow shape represents a project area covered by laser data, the red points symbolize known X Y dZ - points and the green points interpolated X Y dZ - points. The red (dotted) lines show the triangulated model.



The six known points in the illustration above do not create a model that completely encloses the laser data area. If the model does not provide any additional information, TerraScan automatically adds four corner points (green points in the illustration) to expand the elevation adjustment model. Each added corner point has the same dz value as the closest known point.

It is recommended to use an adjustment model that contains the complete project area and thus, provides more accurate elevation information for project boundaries.

Systematic elevation correction

A systematic elevation correction needs to be applied if the point cloud data are systematically shifted in elevation. The systematic shift can be detected by comparing the point cloud with ground control points (GCPs).

TerraScan can do the comparison automatically. The GCPs must be provided in a text file which stores an identifier, X, Y, and Z coordinates in space-delimited fields, one line for each control point. The identifier is normally a number but it may include non-numeric characters as well.

In the point cloud, at least the points on the ground around the GCP locations should be classified into a separate class. In practice, the check of a systematic elevation shift is often done after the ground points have been classified in the point cloud.

In TerraScan, the check of a systematic elevation shift can be performed for loaded points or for project blocks in batch mode. See Output control report command for loaded points and Output control report command for project blocks for more information.

TerraScan performs the following steps:

1. Read the GCPs from a text file.

2. Scan through project blocks and load laser points from a given class within a given search radius around each GCP.

OR

3. Scan through loaded points.

4. Create a small triangulated surface model (TIN) from the laser points within a given search radius around each GCP.

5. Compute a laser data elevation for each GCP XY location from the TIN. This effectively interpolates an elevation value from three laser points which are closest to a GCP.

6. Output a Control point report that lists all GCPs, the laser elevation value, and the difference between laser elevation and GCP elevation.

The report of the elevation comparison shows, among other things, an **Average dz** value which represents the average elevation shift of the point cloud from the GCP elevations. This value gives some indication about the elevation accuracy of the point cloud. Furthermore, the value can be used in a Linear transformation in order to improve the elevation match between point cloud data and GCPs.

To apply a systematic elevation correction to a point cloud, proceed as follows:

1. Create a control report using either Output control report command for loaded points or Output control report command for project blocks.

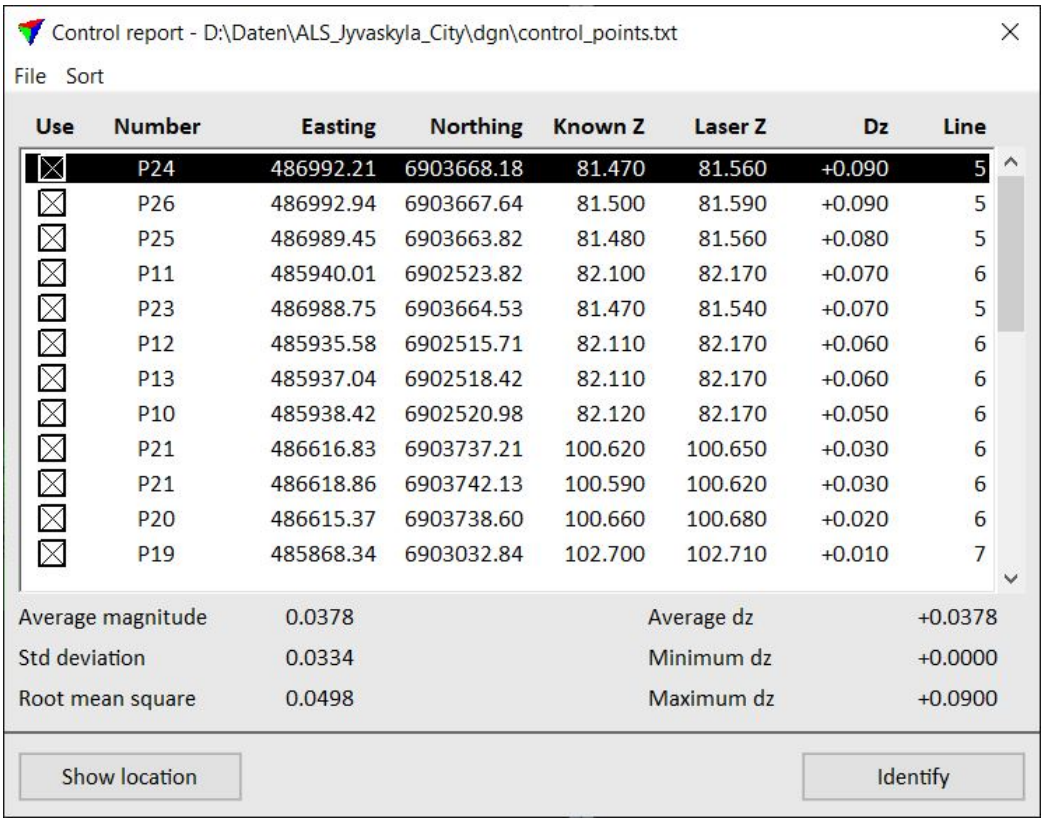
2. Define a Linear transformation in TerraScan Settings. Use the given Average dz value from the report with the inverse sign as Add constant Z value in the transformation definition.

3. Apply the transformation using Transform loaded points command for loaded points or the Transform points action of a macro for project blocks or multiple files.

You can type the dz value directly into the **Transform loaded points** dialog if you want to apply the elevation adjustment to loaded points only. In this case, you do not need to define a transformation in TerraScan **Settings**.

Control point report

The control point report is shown in the **Control report** window:



The window contains the list of all GCPs in the input text file. For each point, the following information is shown:

- **Use** - determines whether a GCP is used in the comparison or not. Switch control points on or off by clicking on the square.
- **Number** - identifier of the GCP.
- **Easting** - easting coordinate of the GCP.
- **Northing** - northing coordinate of the GCP.
- **Known Z** - elevation coordinate of the GCP.
- **Laser Z** - elevation value derived from the laser points at the GCP's XY location.
- **Dz** - difference between Known Z and Laser Z. If the value exceeds a limit defined in the Control report settings, the value is displayed in red. If a control point is outside the area covered with point cloud data, the Dz value is shown as "*outside*".
- **Intensity** - weighted intensity value of the three closest laser points at the GCP's XY location. A closer laser point influences the value more than a more distant laser point. This is displayed if the option in the Control report settings is switched on.
- **Line** - line number assigned to the laser points at the GCP's XY location. This is displayed if the option in the Control report settings is switched on.

Below the GCP list, some statistical information computed from the elevation difference values is provided. This includes average magnitude, standard deviation, and root mean square. Additionally, the average, minimum, and maximum value of elevation differences is displayed.

If a line in the list is selected, the CAD file views defined in the Control report settings are centered at the location of the corresponding GCP.

To show the location of a GCP, select a line in the list. Click on the **Show location** button and move the mouse pointer into a view. This highlights the selected GCP with a square.

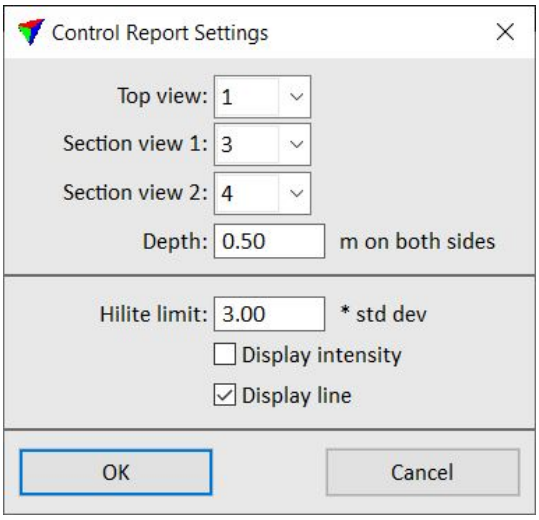
To identify a GCP, click on the **Identify** button and place a data click close to a GCP in a view. This selects the corresponding line in the list.

The GCPs in the list can be sorted in different ways using the commands from the **Sort** pulldown menu.

The report can be saved into a text file or sent to a printer using **Save as text** or **Print** commands from the **File** pulldown menu.

Control report settings

The display settings for the control point report can be changed using **Settings** command from the **File** pulldown menu. This opens the **Control Report Settings** dialog:

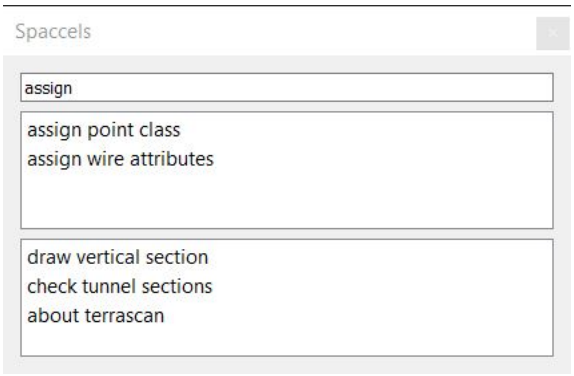
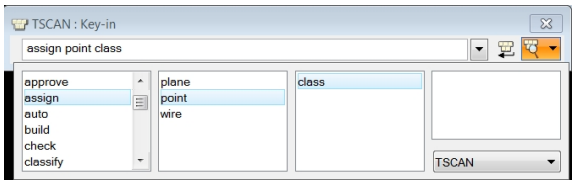


SETTING	EFFECT
Top view	Top view that is updated if a GCP is selected.
Section view 1	First section view that is updated if a GCP is selected. The section is drawn in east-west direction.
Section view 2	Second section view that is updated if a GCP is selected. The section is drawn in north-east direction.
Depth	Depth of a section in the section views. The actual depth shown in a section view is the given value * 2.
Hilite limit	Determines the limit for displaying elevation difference values in red in the report.

SETTING	EFFECT
Display intensity	If on, the average intensity value of the laser points at the GCP location is displayed in the report.
Display line	If on, the line number of the laser points at the GCP location is displayed in the report.

Key-in commands/Spaccels

Key-in commands (MicroStation) or **Spaccels** (**Sp**atix **accel**erate**s**) are a way to speed up the call of tools and menu commands. The CAD platforms offer command lines where you can type and execute the commands. In addition, commands can be assigned to keys (function keys in MicroStation). This speeds up some manual tasks significantly as you can call tools by pressing a key instead of mouse clicks. Especially tools with optional parameters in their call commands are well suited for speeding up manual work with keys.

SPATIX	MICROSTATION
Tools in TerraScan can be started by entering a spaccel in the Spatix **Spaccels** window. The window contains a command line and two lists that help to find the correct syntax of a command.	Most of the tools in TerraScan can be started by entering a key-in command in the MicroStation **Key-in** line. The **Browse Key-in** option of the **Key-in** line can be used to find out the syntax of a key-in command.
	
The upper list in the **Spaccels** window lists all available spaccels. This includes commands for calling tools of Spatix and any loaded IxApp, such as Terra applications. If you know approximately the beginning of the command syntax, start typing the first word. The list of spaccels is reduced to those that start with the typed letters. This helps to find the correct command syntax. You can select a spaccel from the list with a double-click. This writes the spaccel in the command line on the top of the window. Press <Enter> in order to execute the command. This starts the corresponding tool. The lower list in the **Spaccels** window lists spaccels that have been executed. To repeat a command, you may select it from this list with a double-click and press <Enter>.	If you select TSCAN in the list at the lower right corner of the **Browse Key-in** dialog, the selection of commands is limited to TerraScan commands only. There are four list fields that show available commands. Select the first word of a command in the left list. This adds the word to the command line and displays matching second words in the second-left list field. Select the second word of a command. This adds the word to the command line and displays any matching third words in the next list field. Continue until a command is complete. If you know approximately, how a command looks like, you may start typing the command in the **Key-in** line. The software automatically completes words of the command, so you just type the first letter(s) and then, confirm the suggested word with <Space>. Press <Enter> in order to execute a key-in command. This starts the corresponding tool or performs another action called by the command.

SPATIX	MICROSTATION
In Spatix, you can assign commands to any key or combination of keys. Key assignments are defined in the **Shortcuts** window. The window lists all tools/function calls of the software and lets you define a key (combination) for selected ones. In addition, **Spaccels** for commands with optional parameters can be defined and assigned to a key (combination) as well.	In MicroStation, you can assign commands to function keys. This is done in the **Function keys** category of **User Settings**. You first need to select the function key and then, type the correct command in the command line. Set the command with <Enter>.

This Chapter lists a selection of commands and their optional parameters. Some of them you may consider assigning to keys. For each command, a link to the corresponding tool or menu command is given. Use this link to jump to a more detailed description of the tool/command.

The syntax of commands/spaccels is the same in all CAD platforms. Also optional parameters for function calls are defined in the same way.

KEY-IN COMMAND	KEY-IN COMMAND	KEY-IN COMMAND
Add point to ground	Move backward	Scan delete outside fence
Assign point class	Move forward	Scan fit view
Classify above line	Open block	Scan move sun
Classify below line	Open scan project	Scan run macro
Classify close to line	Scan app main	Scan undo display
Classify fence	Scan app mainwin	Travel step backward
Classify inside shape	Scan create surface	Travel step forward
Classify using brush	Scan display	View tower next
Classify view	Scan display close	View tower previous
Fix elevation	Scan display dialog	
Mouse point adjustment	Scan delete inside fence	

Add Point To Ground

Add Point To Ground key-in command adds a point to the ground class and re-iterates its closest environment. You identify the laser point to add after starting the command with a data click.

Syntax:

add point to ground

Corresponding menu command: Add point to ground

Assign Point Class

Assign Point Class key-in command assigns a point class to a single laser point or to all points belonging to the same group. You identify the laser point with a data click after starting the command.

Syntax:

assign point class from=2/to=6/classify=single/select=highest/within=2.0

Parameters:

PARAMETER	EFFECT
from=n	Source class. Use **from=999** or **from=any** to use any visible points.
to=n	Target class.
classify=x	Points to classify: **single** or **group**.
select=x	Point to classify within the search radius: **closest** to mouse pointer, **highest**, or **lowest**.
within=n	Search radius for selecting the **highest** or **lowest** point.

Corresponding tool: Assign Point Class

Classify Above Line

Classify Above Line key-in command classifies points above a given line in a section view. You specify the line with two data clicks after starting the command.

Syntax:

classify above line [From] To

Parameters:

PARAMETER	EFFECT
from	Source class(es). Use **999** or no value at all to classify any visible points. Separate several source classes by a comma or minus, for example **class=1,3-5**.
to	Target class.

Corresponding tool: Classify Above Line

Classify Below Line

Classify Below Line key-in command classifies points below a given line in a section view. You specify the line with two data clicks after starting the command.

Syntax:

classify below line [from] to

Parameters:

PARAMETER	EFFECT
from	Source class(es). Use **999** or no value at all to classify any visible points. Separate several source classes by a comma or minus, for example **class=1,3-5**.
to	Target class.

Corresponding tool: Classify Below Line

Classify Close To Line

Classify Close To Line key-in command classifies points above, below or close to a given line in a section view. You define the line with two data clicks inside a section view.

Syntax:

classify close to line abovefrom=any/aboveto=3/abovetol=0.1

Parameters:

PARAMETER	EFFECT
abovefrom=n	Source class above line. Use **999** or **any** to classify any visible points.
aboveto=n	Target class above line.
abovetol=x	Tolerance above line.
closefrom=n	Source class close to line. Use **999** or **any** to classify any visible points.
closeto=n	Target class close to line.
belowfrom=n	Source class below line. Use **999** or **any** to classify any visible points.
belowto=n	Target class below line.
belowtol=x	Tolerance below line.

Corresponding tool: Classify Close To Line

Classify Fence

Classify Fence key-in command classifies points inside an existing fence. It also starts the Place Fence tool so that you can immediately place a new or the next fence.

Syntax:

classify fence [from] to

Parameters:

PARAMETER	EFFECT
from	Source class(es). Use **999** or no value at all to classify any visible points. Separate several source classes by a comma or minus, for example **class=1,3-5**.
to	Target class.

Corresponding tool: Classify Fence

Classify Inside Shapes

Classify Inside Shapes key-in command classifies points inside selected shapes. Source class, destination class and expand distance may be set as parameters if the key-in command.

The command requires that at least one shape element is selected in the CAD file. It classifies the points inside selected shapes immediately, no manual action is required.

Syntax:

classify inside shapes from=5/to=6/expand=1.0

Parameters:

PARAMETER	EFFECT
from=n	Source class(es). Use **from=999** or **from=any** to classify from any class.
to=n	Target class.
expand=n	Distance by which to expand the shapes. Positive values expand and negative values shrink the shapes.

Corresponding macro action: By polygons

Classify Using Brush

Classify Using Brush key-in command classifies points inside a circular or rectangular brush. You define the points to classify placing a data click and moving the brush over the points in a top or section view. Stop the tool by placing another data click.

Syntax:

classify using brush from=2/to=6/size=15/shape=circle

Parameters:

PARAMETER	EFFECT
from=n	Source class(es). Use **from=999** or **from=any** to classify any visible points. Separate several source classes by a comma or minus, for example **class=1,3-5**.
to=n	Target class.
size=n	Brush size. Given in pixels on screen.
shape=n	Shape of the brush: **circle** or **rectangle**.

Corresponding tool: Classify Using Brush

Classify View

Classify View key-in command classifies points which are visible inside a view. You select the view with a data click after starting the command.

Syntax:

classify view [from] to

Parameters:

PARAMETER	EFFECT
from	Source class(es). Use **999** or no value at all to classify any visible points.
to	Target class.

There is no corresponding tool, menu command, or macro action.

Fix Elevation

Fix Elevation key-in command changes the elevations of laser points inside given polygons. Each laser point inside one polygon is fixed to the same elevation value. The polygons used for this action can be defined by a MicroStation fence, selection or a specific CAD file level.

The command manipulates the points inside polygons immediately, no manual action is required.

Syntax:

fix elevation class=2,8/percentile=5/level=10/color=7

Possible parameters:

PARAMETER	EFFECT
class=n	Classes to modify. Use **class=999** or **class=all** to modify all points. Separate several classes by a comma or minus, for example **class=1,3-5**.
polygon=n	Type of polygons: **fence**, **selected**, or **level**.
percentile=n	If given, elevation values are computed from the points inside the polygon and n specifies the percentile (**0=minimum**, **50=median**, **100=maximum**) of points.
elevation=n	If given, elevation values of all points are set to the fixed value, for example **elevation= 0.0**.
level=n	CAD file level to search for polygons. If given, it forces **polygon=level**.
color=n	If given, filters polygons by color in addition to level. Uses CAD file color numbers.
weight=n	If given, filters polygons by line weight in addition to level. Uses CAD file line weight numbers.
style=n	If given, filters polygons by line style in addition to level. Uses CAD file line style numbers.

Corresponding tool: Fix elevation

Mouse Point Adjustment

Mouse Point Adjustment key-in command activates the Mouse Point Adjustment tool. As long as the tool is active, all data clicks are adjusted to the elevation and/or horizontal location of points loaded into TerraScan.

Syntax:

mouse point adjustment AdjZ=1/AdjXy=0/Point=Closest/Class=2/Within=0.5

Possible parameters:

PARAMETER	EFFECT
AdjZ=n	If given, the elevation of data clicks (vertices of elements) is adjusted. **n** defines a constant value that is added to the point cloud elevation when vertices are placed. If **n=0**, the parameter is deactivated.
AdjXy=n	If **n=1**, the parameter is activated and the xy location of data clicks (vertices) is adjusted. If **n=0**, the parameter is deactivated.
Point	Points or surface model from which element vertex coordinates are derived: **Closest**, **Highest**, **Average**, **Percentile**, **Lowest**, **TIN model**.
class=n	Point class to adjust to. **n** defines the class number.
Within=n	**n** defines the radius of the search area around the mouse pointer location.

Corresponding tool: Mouse Point Adjustment

Move Backward

Move Backward key-in command activates the Move Section tool and moves a section view backward. The key-in command must include a view number (1-8). The section is moved backward by the full depth or half of the depth of the section view, depending on the active setting in the Move Section tool dialog.

Syntax:

move backward [stayactive] ViewNumber

Parameters:

PARAMETER	EFFECT
stayactive	If given, the Move Section tool stays active until a reset click is placed. The section extend is displayed in a top view. A reset click deactivates the Move Section tool and re-activates the tool used before starting the key-in command. If not given, the Move Section tool does not stay active and the section extend is not displayed. The tool used before starting the key-in command stays active.

PARAMETER	EFFECT
ViewNumber	View in which the action is applied. View numbers can range from 1-8.

Corresponding tool: Move Section

Move Forward

Move Forward key-in command activates the Move Section tool and moves a section view forward. The key-in command must include a view number (1-8). The section is moved forward by the full depth or half of the depth of the section view, depending on the active setting in the Move Section tool dialog.

Syntax:

move forward [stayactive] ViewNumber

Parameters:

PARAMETER	EFFECT
stayactive	If given, the Move Section tool stays active until a reset click is placed. The section extend is displayed in a top view. A reset click deactivates the Move Section tool and activates the tool used before starting the key-in command. If not given, the Move Section tool does not stay active and the section extend is not displayed. The tool used before starting the key-in command stays active.
ViewNumber	View in which the action is applied. View numbers can range from 1-8.

Corresponding tool: Move Section

Open Block

Open Block key-in command opens a project block for viewing or modification.

Syntax:

open block [BlockFile [options]]

Parameters:

PARAMETER	EFFECT
BlockFile	Block binary file to open. Can be specified by the file name or the unique block number.
OPTIONS	
neighbours=n	Distance to load points from neighbouring blocks.
fit=n	View(s) to fit where n is between -1 and 8 (**-1=all**, **0=none**, **1-8=view**).
lock=n	If n is 0, opens block for viewing only. If n is not zero, opens block for modification.

Corresponding menu command: Open block

Open Scan Project

Open Scan Project key-in command opens a project in TerraScan.

Syntax:

open scan project ProjectFile

Parameters:

PARAMETER	EFFECT
ProjectFile	Path and name of the TerraScan project.

Corresponding command: Open project in the TerraScan **Project** window

Scan App Main

Scan App Main opens the **TerraScan** toolbox. By default, the toolbox is opened when TerraScan is loaded. The command can be used to re-open the toolbox after it was accidentally closed.

Syntax:

scan app main

Scan App Mainwin

Scan App Mainwin opens the **TerraScan** window. By default, the window is opened when TerraScan is loaded. The command can be used to re-open the window after it was accidentally closed.

Syntax:

scan app mainwin

Scan Create Surface

Scan Create Surface key-in command creates a TerraModeler surface model. The command starts TerraModeler if it is not yet running. It creates the surface model, keeps the model in the memory and stores the model file on the hard disk. The storage location and name of the model file on the hard disk are the same as of the CAD file.

Syntax:

scan create surface class name

Parameters:

PARAMETER	EFFECT
class	Point class from which the surface model is created.
name	Name of the surface model.

Corresponding menu command: Create surface model

Scan Display

Scan Display key-in command changes the view display mode for laser points.

Syntax:

scan display off=all/on=2/color=class

Parameters:

PARAMETER	EFFECT
view=n	View to which the display settings are applied. View numbers can range from 1-8, for example **view=3**. If this parameter is given, it must be the first one and causes an immediate display change to the specified view. If not given, you select a view with a data click.
on=n	Class(es) to switch on. Examples: **on=2** or **on=1,5-8** or **on=all**.
off=n	Class(es) to switch off. Examples: **off=7** or **off=3-11** or **off=all**.
color=n	Coloring mode, n can be **class**, **echo**, **elevation**, **line**, **intensity**, **distance**, **color**, **lineint**, **echolen**,

PARAMETER	EFFECT
	scanner, **dimension**, **group**, **shading**, or u. Example: **color=class**.
weight=n	Weight (point size) where n is line weight value 0-7 or -1 for class-based weight. Examples: **weight=1** or **weight=-1**.
lineon=n	Lines to switch on. Examples: **lineon=1-3** or **lineon=all**.
lineoff=n	Lines to switch off. Examples: **lineoff=1-3** or **lineoff=all**.
sparse=n	Sparse display mode. Examples: **sparse=on** or **sparse=off**.
depth=	Depth display mode. Examples: **depth=on** or **depth=off**.

Corresponding settings can be done in the Display mode dialog.

Scan Display Close

Scan Display Close key-in command closes the Display mode dialog.

Syntax:

scan display close

Scan Display Dialog

Scan Display Dialog key-in command opens the **Display mode** dialog.

Syntax:

scan display dialog

Corresponding menu command: Display mode

Scan Delete Inside Fence

Scan Delete Inside Fence key-in command deletes points inside a fence or selected polygon. You accept the action with a data click inside the view where the fence is drawn or the polygon is selected.

The command requires that a fence is drawn or a polygon is selected before it is started.

Syntax:

scan delete inside fence

Corresponding menu command: Delete / **Inside fence**

Scan Delete Outside Fence

Scan Delete Outside Fence key-in command deletes points outside fence or selected polygon. You accept the action with a data click inside the view where the fence is drawn or the polygon is selected.

The command requires that a fence is drawn or a polygon is selected before it is started.

Syntax:

scan delete outside fence

Corresponding menu command: Delete / **Outside fence**

Scan Fit View

Scan Fit View key-in command fits a view to display the area covered by loaded points. You can specify the view to fit as an optional parameter or by selecting it with a data click.

Syntax:

Scan Fit View [View]

PARAMETER	EFFECT
View	View to which the action is applied. View numbers can range from 1-8.

Corresponding menu command: Fit view

Scan Move Sun

Scan Move Sun key-in command moves the sun direction (azimuth) in shaded views.

Syntax:

Scan Move Sun Degree

PARAMETER	EFFECT
Degree	Value to add to the current sun direction, angle by which the sun direction is moved.

Corresponding dialog setting: Display mode dialog, **Azimuth** setting for **Color by Shading**.

Scan Run Macro

Scan Run Macro key-in command executes a macro.

Syntax is:

Scan Run Macro Macrofile

PARAMETER	EFFECT
Macrofile	File name of the macro to execute. If the file name does not include a directory path, the value of TSCAN_MACRODIR environment variable is used as the directory to search from. See Configuration Variables for MicroStation for more information.

Corresponding menu command: Run macro

Scan Undo Display

Scan Undo Display key-in command makes an undo for the last display mode change.

Syntax:

scan undo display

There is a corresponding **Undo display** button in the Display mode dialog.

Travel Step Backward

Travel Step Backward key-in command moves Travel Path windows by one section backward. This works only, if the Travel Path tool has been used to set up the **Travel Player** of TerraScan.

Syntax:

travel step backward [Count]

PARAMETER	EFFECT
Count	Number of steps to move.

There is a corresponding button in the TerraScan **Travel Player** which is opened by the Travel Path tool.

Travel Step Forward

Travel Step Forward key-in command moves Travel Path windows by one section forward. This works only, if the Travel Path tool has been used to set up the **Travel Player** of TerraScan.

Syntax:

travel step forward [Count]

PARAMETER	EFFECT
Count	Number of steps to move.

There is a corresponding button in the TerraScan **Travel Player** which is opened by the Travel Path tool.

View Tower Next

View Tower Next key-in command moves to the next tower location in the Tower Spans list. This works only, if the View Tower Spans tool has been used to set up the the display of tower locations in TerraScan.

Syntax:

view tower next

The same action is performed by selecting a new line in the Tower Spans dialog.

View Tower Previous

View Tower Previous key-in command moves to the previous tower location in the Tower Spans list. This works only, if the View Tower Spans tool has been used to set up the the display of tower locations in TerraScan.

Syntax:

view tower previous

The same action is performed by selecting a new line in the Tower Spans dialog.

Color schemes

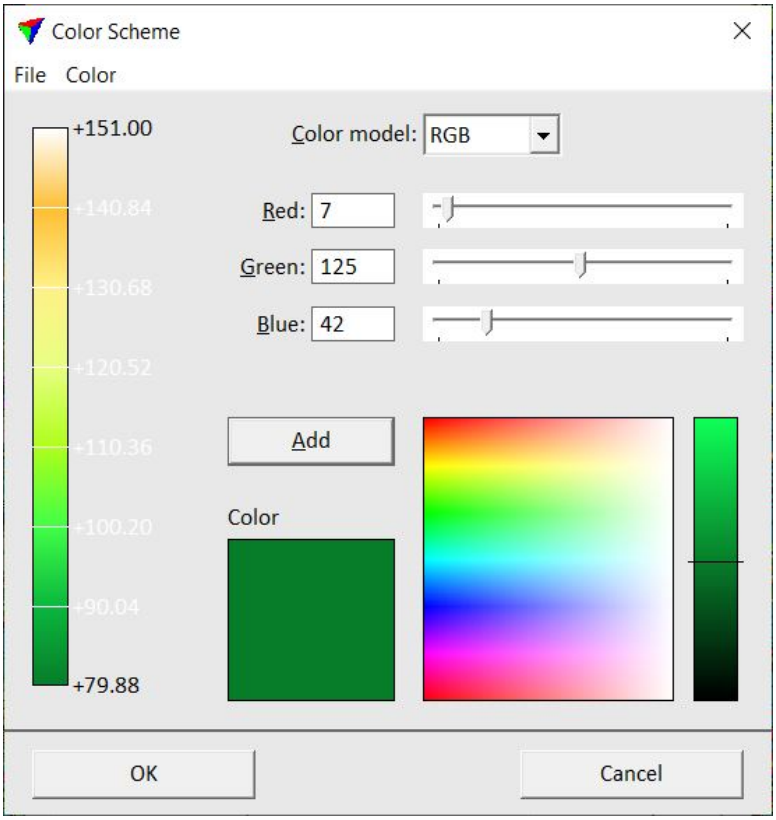
TerraScan uses different color depths and models for displaying or exporting colored data. Besides predefined color schemes, many display and export functions provide the option to define user-specific color schemes. The TerraScan dialogs for defining these schemes differ depending on the color depth:

- 24-Bit Color schemes - allows the definition of more than 16 mio colors. This is suited for seamless color gradients.
- 8-Bit Color schemes - allows the definition of 256 colors. This is suited for the differentiation of several value ranges.
- **Grey scale** - common black to white gradient. This is suited for displaying intensity values but sometimes also for other values. Grey-scale schemes are defined in the same way as other color schemes.

For defining single color values, TerraScan uses the standard Windows dialog for selecting a color.

24-Bit Color scheme definition

24-Bit **Color Scheme** dialog:



Select a **Color model**: **RGB** or **HSV**.

Add colors to the color scheme shown on the left side of the dialog. Colors can be selected by typing values in the **Red**, **Green**, **Blue** or **Hue**, **Saturation**, **Value** fields, by moving the sliders next to the fields, by clicking inside the color box, or by clicking on the color bar right of the color box. The selected color is shown in the **Color** field left of the color box. Click the **Add** button to add the color to the color scheme. Colors are always added to the lower end of the color scheme which means that the color added first is for the highest elevation or intensity value.

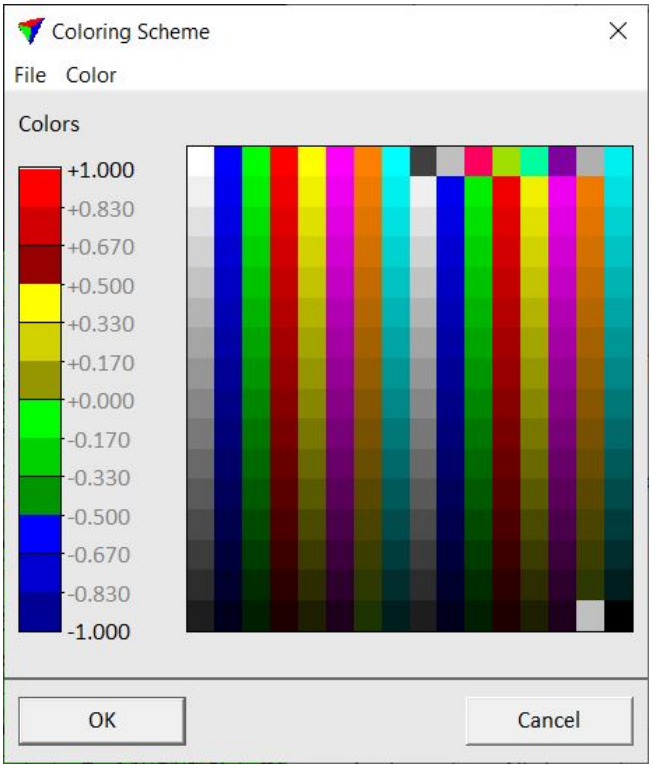
The boundary values between colors can be changed by clicking on the value next to the color scheme. This opens the **Color value** dialog. Switch **Fixed** on and define a value.

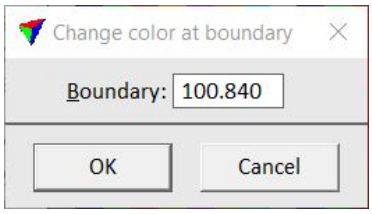
To change colors of a color scheme, select **Remove all** or **Remove last** command from the **Color** pulldown menu. This removes all colors or the color added last from the color scheme.

A color scheme can be saved into a text file with the extension *.SMC using the **Save As** command from the **File** pulldown menu. The file stores the RGB or HSV values for each color of the scheme in rows. A previously saved color scheme file can be loaded with the Open command from the File pulldown menu. Color values can be also changed by editing the color scheme file in a text editor.

8-Bit Color scheme definition

8-bit **Coloring Scheme** dialog:





To change a color scheme, select **Remove all** command from the **Color** pulldown menu. This removes the active color scheme. Add a new color from the color table with a data click on the color field. New colors are always added at the lower end of the color scheme which means that the first selected color is for the highest elevation range. Select **Remove last** from the **Color** pulldown menu in order to remove the color added last to the coloring scheme.

The elevation values defining the boundaries between two colors can be changed by clicking on the color field in the color scheme or on the number. The **Change color at boundary** dialog opens where a fixed elevation value can be set. If **Auto fit** is applied, any fixed elevation values get lost.

A color scheme can be saved into a text file with the ending .CLR. The text file lists the numbers of the selected colors. To save a color scheme, select **Save as** command from the **File** pulldown menu. To load a color scheme, select **Open** from the **File** pulldown menu.

Classification Routines

The Routine sub-menu in the Classify pulldown menu of the **TerraScan** window offers a number of classification routines that effect or rely on points.

The Classify sub-menu in the Group pulldown menu of the **TerraScan** window contains classification routines that effect or rely on groups of points.

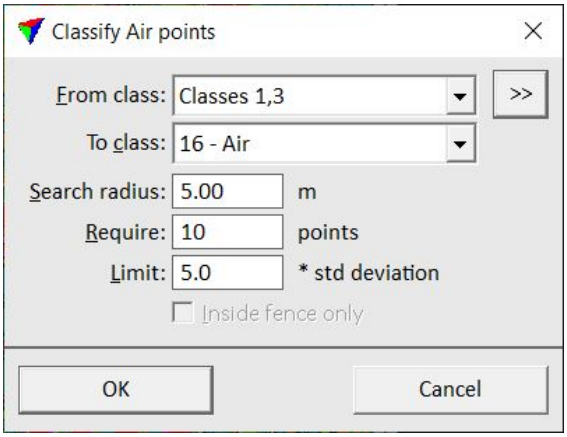
In addition, there are some classification routines which can be used in macros only. This Chapter describes the basic logic used in the classification routines.

Air points

Not Lite

Air points routine classifies points which are clearly higher than the median elevation of surrounding points. It can be used to classify noise up in the air.

For each point, the routine finds all neighbouring points within a given search radius. It computes the median elevation of the points and the standard deviation of the elevation values. A point is classified as air point if it is more than the standard deviation multiplied by a given factor above the median elevation. The comparison using the standard deviation results in the routine being less likely to classify points in places where there is more elevation variation.



SETTING	EFFECT
From classes	Source class(es). Select several classes by pressing the **Ctrl-key** while selecting a class from the list.
To class	Target class.
Search radius	2D search radius around a point. For points within this radius the median elevation and standard deviation are computed. Normally a value between 2.0 - 10.0 m.

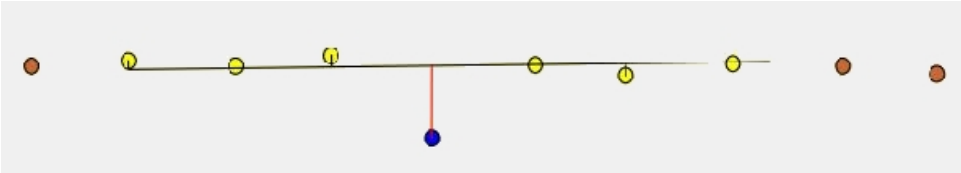
SETTING	EFFECT
Require	Minimum amount of points required within the **Search radius**. A point is not classified if there are not enough points within the search radius.
Limit	Factor for multiplication with the standard deviation of elevation values within the search radius. A point is classified, if it is more than **Limit * std deviation** above the median surface of the points in the search radius.
Inside fence only	If on, only points inside a fence or selected polygon(s) are classified.

Below surface

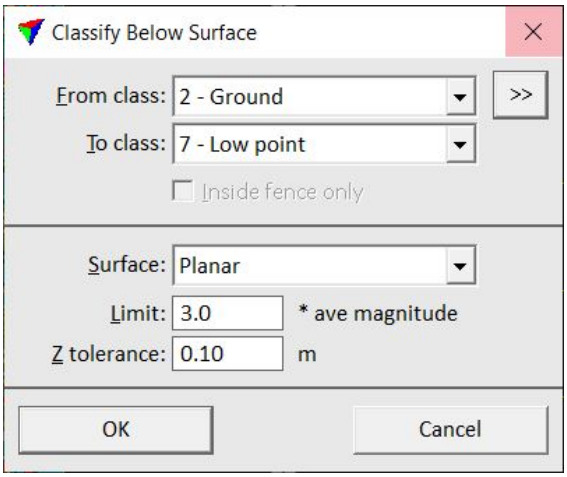
Not Lite

Below surface routine classifies points which are lower than neighbouring points in the source class. This routine can be run, for example, after ground classification to locate points which are a bit below the true ground surface.

For each point, the routine finds up to 25 closest neighbour points in the source class. It fits a planar or curved plane to the neighbouring points and computes the average magnitude of the elevation differences between the points and the plane. If the point is more than the average magnitude multiplied by a given factor below the plane, it is classified. If the point is above the plane or less than a given tolerance value below the plane, it is not classified.



Below point classification



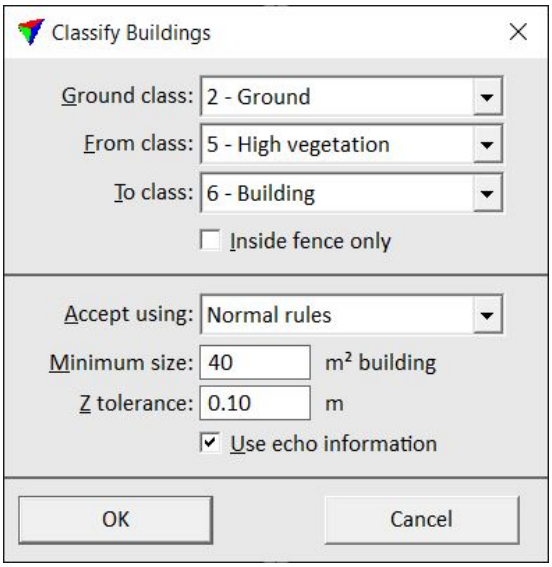
SETTING	EFFECT
From class	Source class(es) from which points are classified.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **From class** field.
To class	Target class.
Inside fence only	If on, points inside a fence or selected polygon(s) are classified.
Surface	Shape of the surface that is used as reference for classifying points: **Planar** or **Curved**.
Limit	Factor for multiplication with the average magnitude of the elevation differences of neighbouring points. A point is classified, if it is more than **Limit * avg magnitude** below the plane fitted through the neighbour points.
Z tolerance	Maximum allowed elevation variation of a point below the fitted plane. A point within this tolerance distance is not classified.

Buildings

Not Lite

Buildings routine classifies points on building roofs which form a planar surface. The routine requires that ground points have been before. It is also advisable to classify points above the ground into a separate class, so that this class contains points in an elevation range above ground where building roofs are included.

The routine starts from empty areas in the ground class and tries to find points on planar surfaces above these areas.

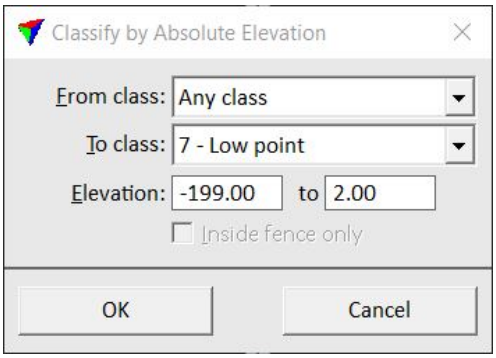


SETTING	EFFECT
Ground class	Point class that contains points on the ground.
From class	Source class(es).
To class	Target class.
Inside fence only	If on, points inside a fence or selected polygon(s) are classified.
Accept using	Defines how easily a group of points is accepted as building roof points. The more strict the rules, the closer the points must follow a plane surface.
Minimum size	Smallest size of a building footprint. Points are only classified if the footprint area of the whole building is larger than the given value.
Z tolerance	Minimum elevation difference of a point from the plane fitted through the points on a roof. The value is related to the noise level of the points on roof planes.
Use echo information	If on, the echo type attribute of laser points is considered in the classification process. This can support the classification because points on roofs mostly belong to the echo type 'only echo' whereas vegetation usually contains lot of 'first of many' and 'intermediate' echoes.

By absolute elevation

Not Lite

By absolute elevation routine classifies points within a given elevation range. It uses the elevation values stored for laser points in order to decide which points are classified. This can be used to classify error points high up in the air or clearly below the ground level.



SETTING	EFFECT
From class	Source class(es).
To class	Target class.
Elevation	Elevation range within which points are classified. Given in absolute elevation values.
Inside fence only	If on, points inside a fence or selected polygon(s) are classified.

By angle

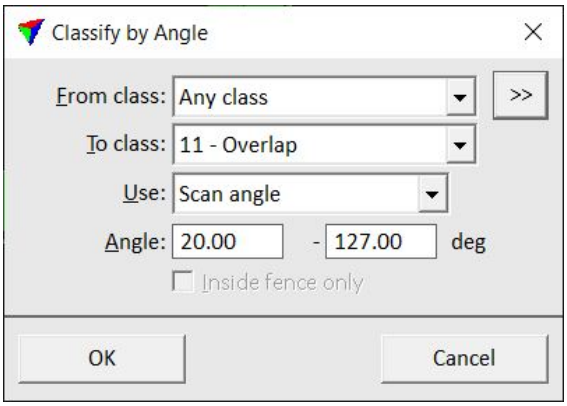
Not Lite

By angle routine classifies points according to the scan angle, the angle from vertical, or the angle from the line edge.

The **Scan angle** can be stored as attributes for laser points in TerraScan FastBinary and LAS files. If points from other file formats are used, or if the attribute is not stored, a scan angle is computed based on the trajectories and their values can range from -128 and +127 degree.

Angles from vertical are computed based on trajectories and can range from 0 to 90 degree.

Angles from edge uses the angle attribute stored for laser points and thus, requires that points are stored in TerraScan FastBinary or LAS files. The option can be used to classify points along scan line edges at a high angular accuracy.



SETTING	EFFECT
From class	Source class(es).
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **From class** field.
To class	Target class.
Use	Type of angle used for classification: **Scan angle**, **Angle from vertical**, or **Angle from edge**.
Angle	Range of angle values. A point is classified if its angle value falls within the given range.
Inside fence only	If on, points inside a fence or selected polygon(s) are classified.

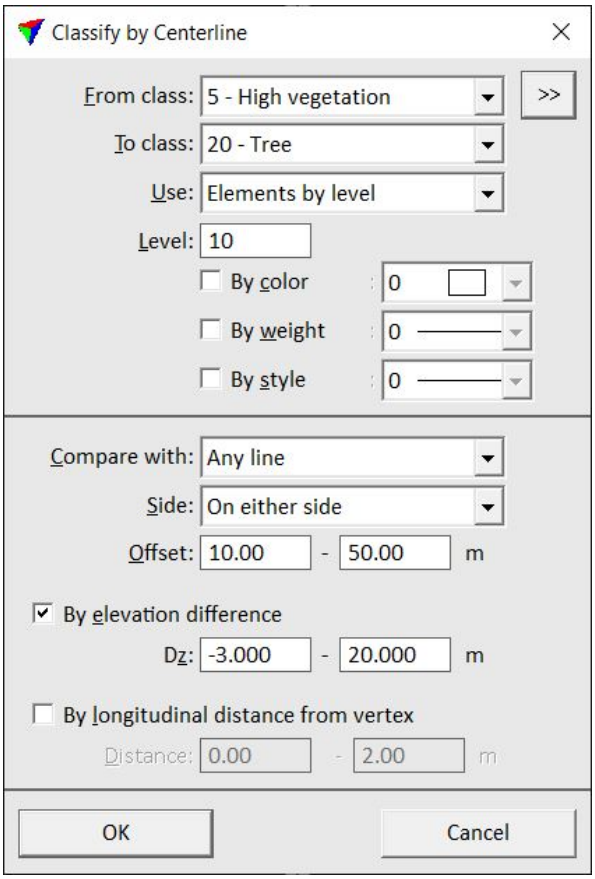
By centerline

Not Lite

By centerline routine classifies points based on their distance to a linear element.

The linear elements that are used by this routine can be lines, polylines (Spatix) or line strings (MicroStation), arcs, polygons (Spatix) or shapes (MicroStation), big elements (Spatix) or complex elements (MicroStation) consisting valid simple element types. The classification is performed by using either selected elements or elements on a specified CAD file level.

If multiple elements are used, each laser point is classified according to the offset distance to either the closest linear element or to any of the linear elements. In addition to the 2D distance, the elevation distance can be included in the classification process. Further, the classification can be limited to a certain longitudinal distance from vertices, which is useful, for example, to classify points along a line string but only in a certain area around vertices.

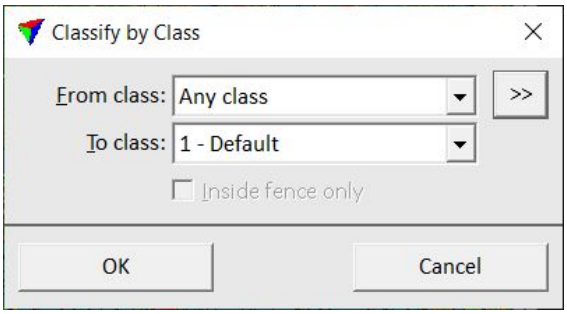


SETTING	EFFECT
From class	Source class(es).
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **From class** field.
To class	Target class.
Use	Defines what elements are used: • **Selected linear elements** - any selected elements in the CAD file. This requires the selection of elements before starting the routine. • **Elements by level** - any elements that are located on a given CAD file level.
Level	Number of the CAD file level where elements are located that are used for the classification. This is only active if **Use** is set to **Elements by level**.
By color	If on, elements on the given **Level** are further filtered by the selected color. Click on the color field in order to select the color. Uses

SETTING	EFFECT
	the active color table of MicroStation. This is only active if **Use** is set to **Elements by level**.
By weight	If on, elements on the given **Level** are further filtered by the selected weight. Click on the list of line weights in order to select the weight. Uses the line weights of MicroStation. This is only active if **Use** is set to **Elements by level**.
By style	If on, elements on the given **Level** are further filtered by the selected style. Click on the list of line styles in order to select the style. Uses the line styles of MicroStation. This is only active if **Use** is set to **Elements by level**.
Compare with	Method of comparing points to linear elements: • **Any line** - points are classified if they are within the distance of any linear element. • **Closest line** - points are classified if they are within the distance of the closest linear element.
Side	Side on which to classify points: **On left side**, **On either side**, or **On right side**. The side is relative to the digitization direction of the linear element.
Offset	Minimum and maximum 2D distance. Points within the offset range are classified.
By elevation difference	If on, only points within the given elevation distance range from the linear element are classified. Define the elevation offset in the **Dz** fields.
By longitudinal distance from vertex	If on, only points within the given longitudinal distance range from the closest element vertex are classified. The longitudinal distance is measured in both directions from a vertex along the linear element. Define the distance in the **Distance** fields.

By class

By class routine simply classifies all points from one or several given class(es) to another class.



SETTING	EFFECT
From class	Source class from which to classify points.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **From class** field.
To class	Target class to classify points into.
Inside fence only	If on, points inside a fence or selected polygon(s) are classified.

By color

Not Lite

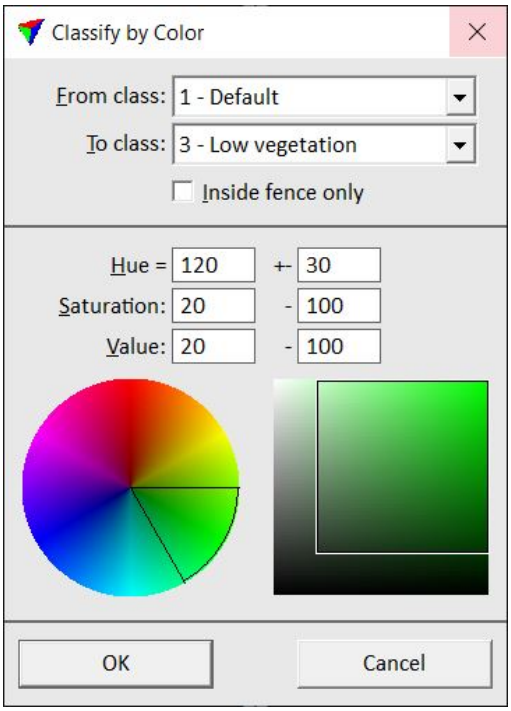
By color routine classifies points based on color values. RGB color values can be stored as attributes for laser points in TerraScan FastBinary files, Binary 8-bit/16-bit files, LAS 1.2+ files, and user-defined text file formats that include the color channels as attributes. The color values have to be assigned to the laser points before using this classification routine. You may, for example, extract colors from images using Extract color from images command.

The routine classifies points whose color attribute falls into a specified HSV color range. The HSV color model consists of three components:

- **Hue** - pure color value on a 360-degree color circle ranging from Red (0 deg) via Yellow (60 deg), Green (120 deg), Cyan (180 deg), Blue (240 deg), Magenta (300 deg) back to Red.
- **Saturation** - intensity or purity of the color. A smaller saturation sets the color closer to a gray shade.
- **Value** - lightness or darkness of the color. A smaller value sets the color closer to black.

In the routine's dialog, you define a range of values for each component. A point is classified, if its color value falls within the given ranges. The dialog contains a color circle which illustrates the selected **Hue** range. The circle can also be used to select another Hue value with a data click. Further, the dialog contains a color field that illustrates the selected **Saturation** (x-axis) and **Value** (y-axis) ranges.

Color values for laser points are stored as RGB values but the corresponding HSV values can be displayed in the **TerraScan** window by using the Fields command and switching on the **Point color HSV** attribute.



SETTING	EFFECT
From class	Source class(es).
To class	Target class.
Inside fence only	If on, points inside a fence or selected polygon(s) are classified.
Hue	Hue value and tolerance.
Saturation	Minimum and maximum value for saturation component.
Value	Minimum and maximum value for value component.

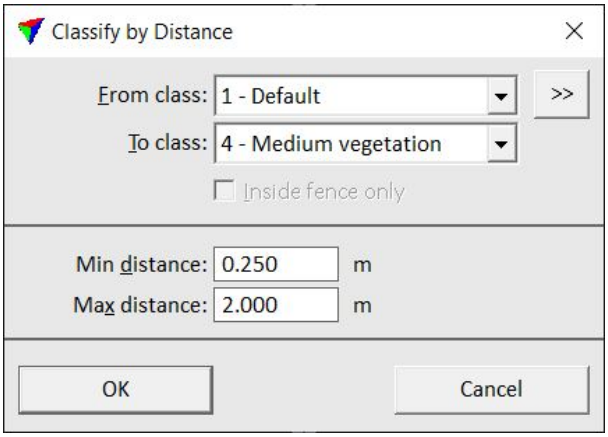
You can effectively classify by gray scale if you set **Hue** tolerance to +- 180. Then, all hue values fall within the range.

By distance

Not Lite

By distance routine classifies points which are within a given distance range. The routine requires distance values that have been computed by using the Compute distance command or macro action.

If distance from ground values have been computed, the **By distance** routine results in the same classification as the By height from ground routine.

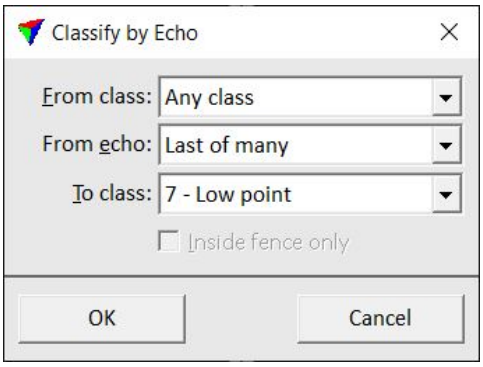


SETTING	EFFECT
From class	Source class(es).
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **From class** field.
To class	Target class.
Inside fence only	If on, points inside a fence or selected polygon(s) are classified.
Min distance	Minimum distance value.
Max distance	Maximum distance value.

By echo

Not Lite

By echo routine classifies points based on echo type or echo number. Echo type and echo number can be stored as attributes for each laser point. Echo number is stored only in TerraScan FastBinary files, LAS files, or user-defined text file formats that include the echo number attribute.

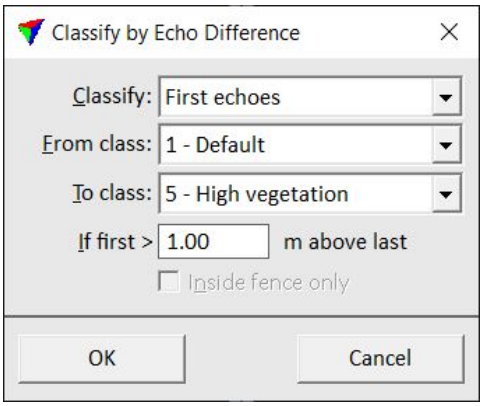


SETTING	EFFECT
From class	Source class(es).
From echo	Echo type from which to classify points: • **Only echo** - only echo from a return signal. • **First of many** - first echo from a return signal which produced at least two echoes. • **Intermediate** - intermediate echo(es) from a return signal which produced at least three echoes. • **Last of many** - last echo from a return signal which produced at least two echoes. • **Any first** - combination of **Only echo** and **First of many**. • **Any last** - combination of **Only echo** and **Last of many**. • **First... Seventh** - points with a specific echo number. This requires a file format that stores the echo number as attribute for each point.
To class	Target class to classify points into.
Inside fence only	If on, points inside a fence or selected polygon(s) are classified.

By echo difference

Not Lite

By echo difference routine classifies points based on elevation difference between first and last echoes. The echo type can be stored as attribute for each laser point. The classification effects only points with the echo types **first of many** or **last of many**.



SETTING	EFFECT
Classify	Echo type from which to classify points: **First echoes** or **Last echoes**.
From class	Source class(es).
To class	Target class.
If first | last >	Elevation difference between first and last echos. A point is classified, if the difference is larger than the given value.
Inside fence only	If on, points inside a fence or selected polygon(s) are classified.

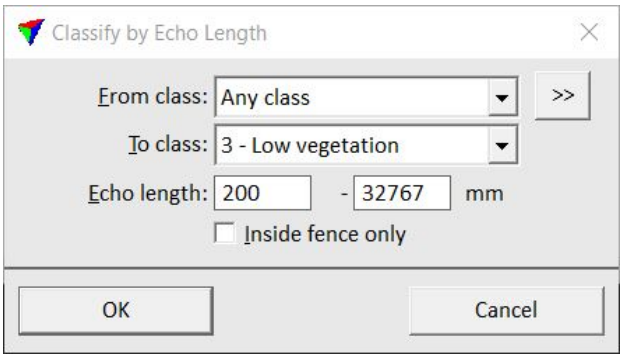
By echo length

Not Lite

By echo length routine classifies points based on the length of a return signal that resulted in a point. The echo length can be stored as attribute for each laser point in TerraScan FastBinary format. It describes the relative length of a return signal compared to a typical return from a hard surface, for example:

- -50 - the echo is 50 mm shorter than a typical hard surface return.
- +845 - the echo is 845 mm longer than a typical hard surface return.

The routine requires that the echo length has been extracted for the points from waveform information. See Chapter Waveform Processing for more information about waveform processing in TerraScan.



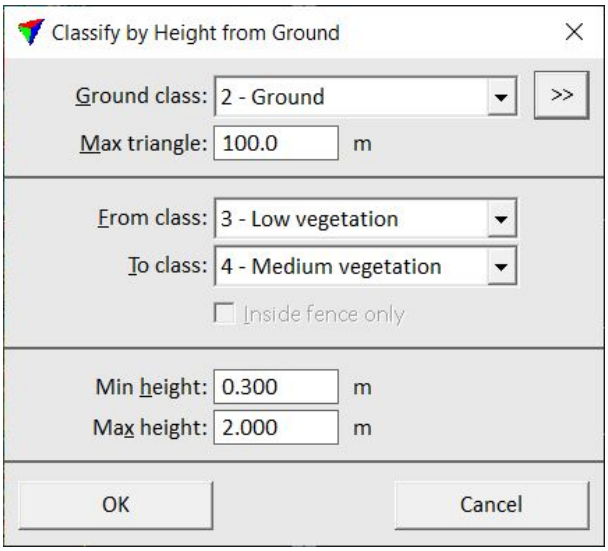
SETTING	EFFECT
From class	Source class(es).
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **From class** field.
To class	Target class.
Echo length	Range of echo length values. A point is classified, if its echo length value falls within the given range.
Inside fence only	If on, points inside a fence or selected polygon(s) are classified.

By height from ground

Not Lite

By height from ground routine classifies points which are within a given height range compared to a reference surface. The surface can be temporarily created from points in one or several classes or a surface active in TerraModeler can be used. Most often, the previously classified ground class is used as the surface for classifying points above the ground level.

You might use this routine for to classify points into different vegetation classes for preparing, for example, building classification, powerline processing, or tree detection. As a result, the highest vegetation class should include all hits on the target objects of interest (building roofs, wires and towers, or trees).



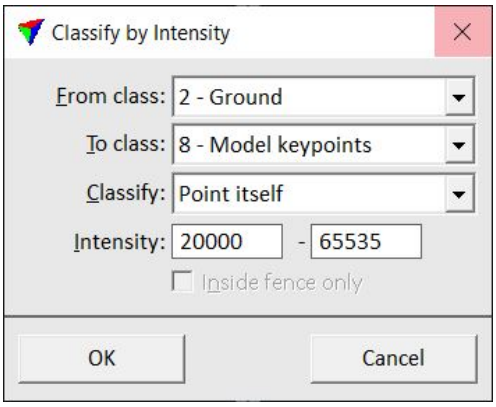
SETTING	EFFECT
Ground class	Point class(es) used for creating the reference surface or the name of a surface model that is active in TerraModeler.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Ground class** field.
Max triangle	Maximum length of a triangle edge in a temporary surface model.
From class	Source class(es).
To class	Target class.
Inside fence only	If on, points inside a fence or selected polygon(s) are classified.
Min height	Minimum height above the reference surface. If a negative value is given, points below the reference surface are classified.
Max height	Maximum height above the reference surface. The value must be bigger than the value of **Min height**. If a negative value is given, points below the reference surface are classified.

By intensity

Not Lite

By intensity routine classifies points within a given range of intensity values. The intensity value can be stored as attribute for each laser point. It describes the strength of the return signal and is effected by the type of surface material hit by the laser beam.

The routine can be used to quickly classify points that are possible hits on rails or on paint markings because the metal surface or the white paintings produce high intensity values while the surrounding area, such as dark gravel or asphalt often result in low intensity values.



SETTING	EFFECT
From class	Source class(es).
To class	Target class.
Classify	Determines which points are classified: • **Point itself** - points with an intensity value in the given range are classified. • **Later echoes** - points with last and intermediate echo types are classified if the intensity value of the point with first echo type of the same return signal is within the given range. Points with first and only echo types are not effected with this setting.
Intensity	Range of intensity values within which a point is classified. The routine classifies points with MinInt <= Intensity <= MaxInt. MinInt and MaxInt are given in absolute intensity values.
Inside fence only	If on, points inside a fence or selected polygon(s) are classified.

By normal vector

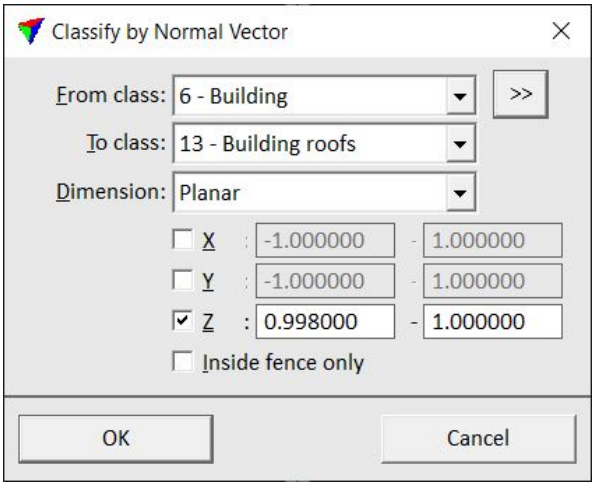
Not Lite

By normal vector routine classifies points based on normal vector values and/or dimension types.

Normal vector and dimension can be stored as attributes for points in TerraScan FastBinary files. They must be computed by using the Compute normal vectors command before starting the classification routine.

The normal vector is computed for points with planar dimension. It contains three components, X, Y, and Z. Points can be classified by using the dimension attribute only, or by specifying certain normal vector components and their values. The values can range from -1.0 to +1.0 for each component. For instance, the value +1.0 for the Z component means that the normal vector points upward from a completely horizontal surface.

The routine is useful, for example, to classify points on very flat parts of a road surface in a mobile data set or to classify points on roof planes facing to a certain direction in an airborne data set.



SETTING	EFFECT
From class	Source class(es).
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **From class** field.
To class	Target class.
Dimension	Dimension of points to be classified: **Not known**, **Linear**, **Planar**, or **Complex**. See Compute normal vectors command for a more detailed description of dimensions.

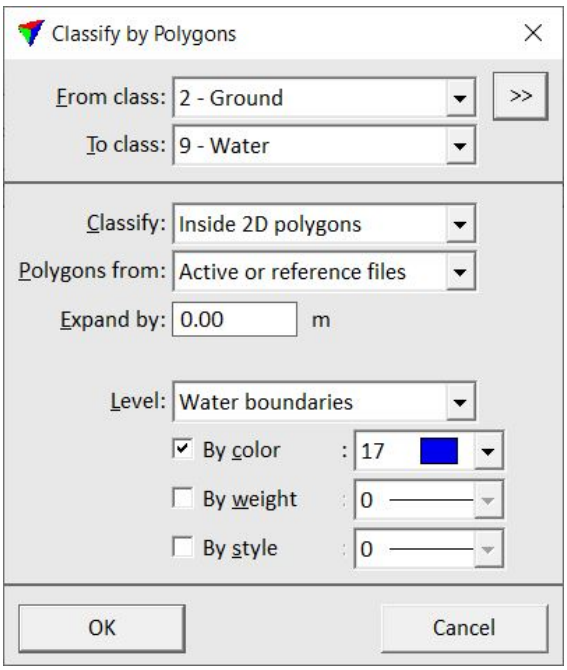
SETTING	EFFECT
X	If on, points are classified, if the X component of their normal vector falls within the given range of values.
Y	If on, points are classified, if the Y component of their normal vector falls within the given range of values.
Z	If on, points are classified, if the Z component of their normal vector falls within the given range of values.
Inside fence only	If on, points inside a fence or selected polygon(s) are classified.

By polygons

Not Lite

By polygons routine classifies points that are located inside or outside 2D shapes, or close to 3D shapes. The shapes must be drawn in the CAD file. They must be located on a specified level in the active CAD file or in a reference CAD file *(MicroStation only)* in order to be used in the classification routine. Optionally, the shapes can be filtered by color, weight, or style settings.

The routine is only available in macros. See Chapter Macros for more information about macro creation and macro actions.



SETTING	EFFECT

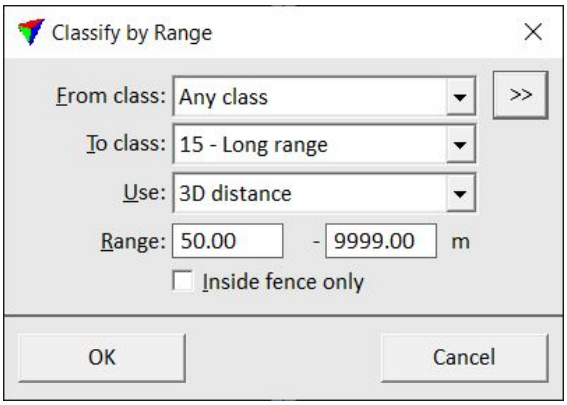
From class	Source class(es).
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **From class** field.
To class	Target class.
Classify	Defines the type of reference polygons: • **Inside 2D polygons** - points inside a polygon are classified. The elevation of the polygon is ignored. • **Outside 2D polygons** - points outside a polygon are classified. The elevation of the polygon is ignored. • **Close to 3D polygons** - points within the given **Offset** from a 3D polygon are classified.
Polygons from	Source where shapes are drawn: • **Active or reference files** - in the active CAD file or in an attached reference file. *(MicroStation only)* • **Active CAD file** - in the active CAD file. • **Reference file** - in an attached CAD file. *(MicroStation only)*
Expand by	Distance that is added to the boundary location of shapes. A positive value expands shapes, a negative value shrinks shapes.
Offset	Distance from a 3D polygon within which points are classified. A negative value refers to the inside, a positive value to the outside of a polygon, such as a building wall. This is only active if **Classify** is set to **Close to 3D polygons**.
Level	Number of the CAD file level where shapes are located that are used for the classification.
By color	If on, shapes on the given **Level** are further filtered by the selected color. Click on the color field in order to select the color. Uses the active color table of the CAD file.
By weight	If on, shapes on the given **Level** are further filtered by the selected weight. Click on the list of line weights in order to select the weight. Uses the line weights of the CAD file.
By style	If on, shapes on the given **Level** are further filtered by the selected style. Click on the list of line styles in order to select the style. Uses the line styles of the CAD file.

By range

Not Lite

By range routine classifies points based on the range. The range is defined as the distance of a point from the scanner. The distance can be measured as 3D, horizontal, or vertical distance.

This classification routine is primarily used with laser data from mobile scanners and can be used, for example, to classify points at long ranges or points that are clearly below the ground level.

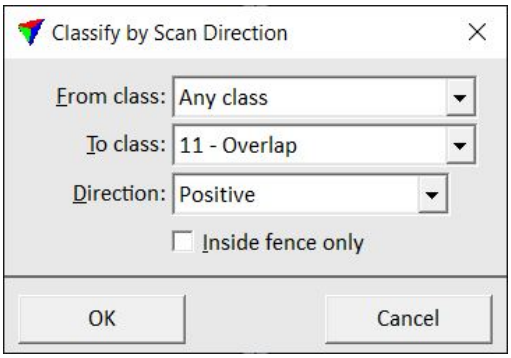


SETTING	EFFECT
From class	Source class(es).
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **From class** field.
To class	Target class.
Use	Defines how the range between a point and the scanner is measured: • **3D distance** - 3D distance. • **Xy distance** - horizontal distance. • **Offset distance** - left/right distance. • **Forward distance** - forward/backward distance. • **Dz** - vertical distance.
Range	Distance range. A point is classified, if it is within the given distance range from the scanner.
Inside fence only	If on, points inside a fence or selected polygon(s) are classified.

By scan direction

Not Lite

By scan direction routine classifies points in negative or positive scan direction, or edge points. This requires that laser points are stored in LAS files. The classification is based on the scan direction and edge of flight line bit fields present in LAS files.

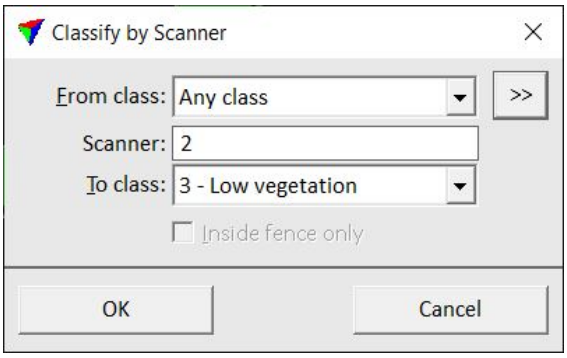


SETTING	EFFECT
From class	Source class(es).
To class	Target class.
Direction	Scan direction from which to classify points: **Negative**, **Positive**, or **Edge point**.
Inside fence only	If on, points inside a fence or selected polygon(s) are classified.

By scanner

Not Lite

By scanner routine classifies points according to the scanner number. The number of the scanner that recorded a point can be stored as attribute for laser points in TerraScan FastBinary and LAS files. This is primarily used for laser data from multiple-scanner systems.



SETTING	EFFECT
From class	Source class(es).
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **From class** field.
Scanner	Number(s) of the scanner(s) from which to classify points. Separate several scanner numbers by a comma or minus, for example 1-3,5. Use 0-255 for all scanners.
To class	Target class.
Inside fence only	If on, points inside a fence or selected polygon(s) are classified.

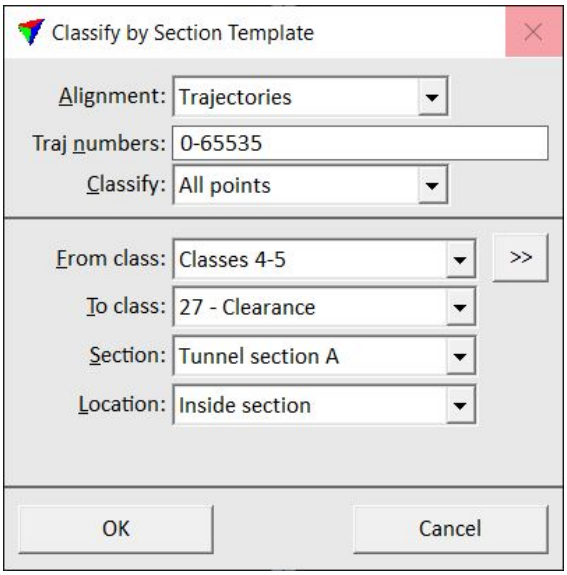
By section template

Not Lite

By section template routine classifies points based on the shape of a section. The points to classify are defined by a three dimensional alignment element and a section template.

This classification routine is useful for classifying points along roads, rails, in tunnels, or along other corridors in order to analyze clearance areas.

The section template must be defined in Section templates category of TerraScan **Settings** before using the classification routine. Then, it can be used to classify points which are inside the section, outside the section, or on and close to the section outline. Further, the routine requires a 3D alignment element which is used to move the section template along. A selected linear element can act as alignment element or alternatively, trajectories that are active in TerraScan.

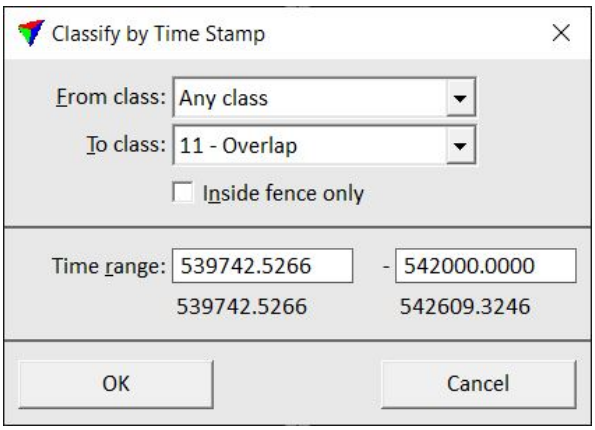


SETTING	EFFECT
Alignment	Elements that are used as alignment elements: • **Selected vectors** - any selected linear elements in the CAD file. The element(s) must be selected before starting the routine. • **Trajectories** - trajectories that are active in TerraScan.
Traj numbers	Number(s) of trajectories used as alignment elements. Separate several trajectory numbers by a comma or minus, for example 1,3-5. This is only active if **Alignment** is set to **Trajectories**.
Classify	Points to classify relative to trajectories: • **All points** - all points are compared to any trajectory line. • **Points from same line** - points are compared only with their associated trajectory line. This is only active if **Alignment** is set to **Trajectories**.
From class	Source class(es).
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **From class** field.
To class	Target class.
Section	Name of the section template. The list includes all section templates that are defined in Section templates category of TerraScan **Settings**.
Location	Points to classify relative to the section template: • **Inside section** - points inside the section. • **Outside section** - points outside the section. • **On section** - points on and close to the section.
Tolerance	Defines the maximum distance of a point from the section outline in order to be classified. This is only active if **Location** is set to **On section**.

By time stamp

Not Lite

By time stamp routine classifies points within a specified time range. This requires time stamps stored for each laser point.



SETTING	EFFECT
From class	Source class(es).
To class	Target class.
Inside fence only	If on, points inside a fence or selected polygon(s) are classified.
Time range	Range of time stamps. A point is classified, if its time stamp value falls within the given range. The numbers below the input fields show the minimum and maximum time stamps of loaded points.

By vegetation index

Not Lite

By vegetation index routine classifies points based on a vegetation index computation. The routine requires that color values have been extracted for the points. Color values for laser points can be extracted from raw images or raster attachments loaded into TerraPhoto, or assigned per class by setting values for red, green, and blue channels. See Extract color from images and Assign color to points for corresponding commands in TerraScan.

TerraScan can assign up to 10 color channels for each point. The maximum amount of color channels can only be stored in the TerraScan FastBinary format. LAS 1.4 format and later can store up to 4 color channels, LAS 1.2 format and later up to 3.

There are two implemented methods for computing the vegetation index.

Normalized difference

The method assumes that channel 0 stores red (R) and channel 3 near-infrared (NIR) color values. The normalized difference value is computed with the following equation:

```
ND = (NIR - R) / (NIR + R)                    -1 <= ND <= +1
```

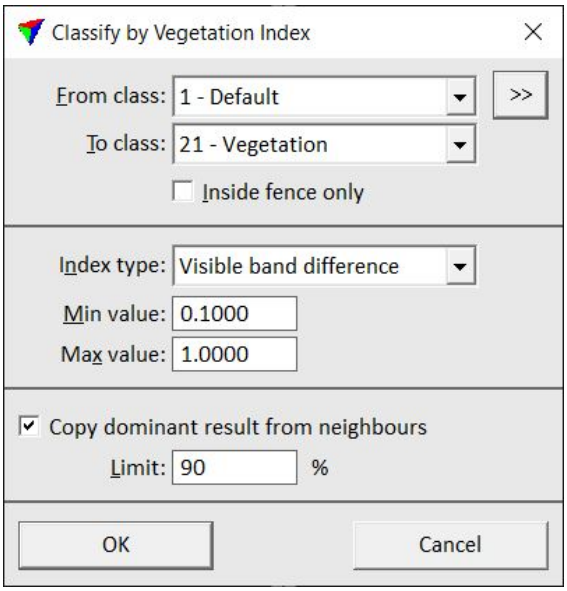
Visual band difference

The method assumes that channel 0 stores red (R), channel 1 green (G), and channel 2 blue (B) color values. The visual band difference value is computed with the following equation:

```
VBD = (2*G - R - B) / (2*G + R + B)                -1 <= VBD <= +1
```

The routine is well suited for separating points on vegetation from non-vegetation points. You can check how well the vegetation index computation separates vegetation from non-vegetation points by displaying points by vegetation index. The larger the difference value, the more likely a point is representing vegetation.

In addition to the implemented methods, there may be User vegetation indexes defined in the corresponding category of TerraScan **Settings**.



SETTING	EFFECT
From class	Source class(es).
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **From class** field.
To class	Target class.
Inside fence only	If on, points inside a fence or selected polygon(s) are classified.

SETTING	EFFECT
Index type	Method of vegetation index computation: **Normalized difference**, **Visual band difference**, or any User vegetation index defined in TerraScan **Settings**.
Min value	Minimum difference value to be classified.
Max value	Maximum difference value to be classified.
Copy dominant result from neighbours	If on, the software checks neighbouring points before classifying a point. If the percentage of neighbouring points given in the **Limit** field are classified, the point is classified as well. This avoids that single isolated points are classified as vegetation or non-vegetation.

Closeby points

Not Lite

Closeby points routine classifies points close to points from another class, another line, or another scanner. For each point, the software finds points within a given 2D or 3D search radius. It checks whether the points within the radius fulfill the given conditions, such as being in a specific class, from another line or scanner. If all defined conditions are true, the point is classified.

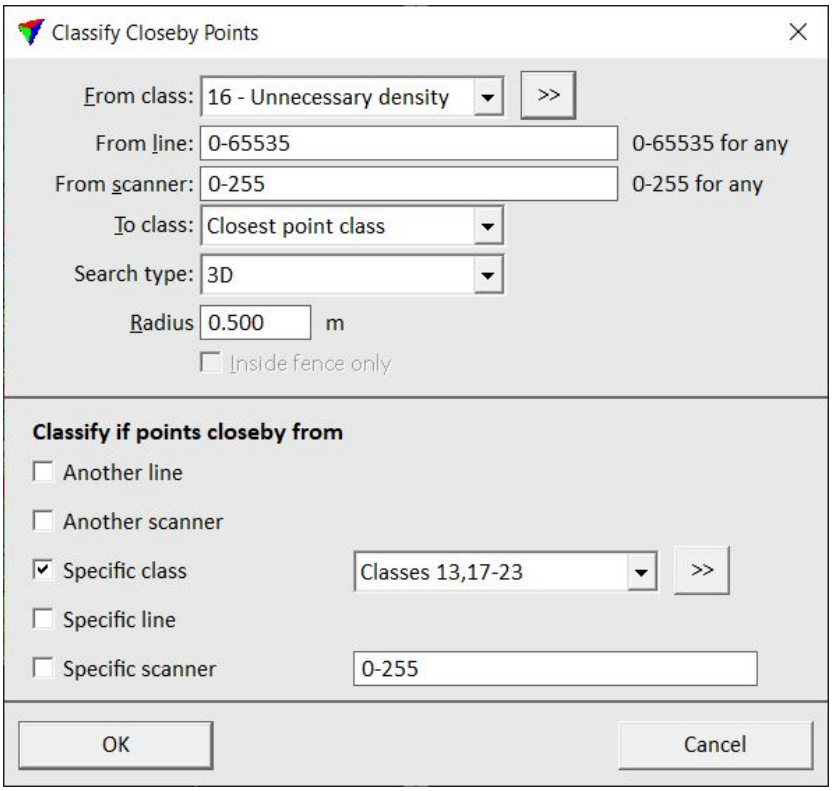
Example of a high-density point cloud:

- thin the point cloud, keep a central point and classify other points into a temporary point class
- apply classification routines to the thinned point cloud, classify points into any object-specific point classes
- apply the **Closeby points** routine in order to densify the object classes with points from the temporary point class

In the above example, the processing speed is increased by thinning a dense point cloud before running processing-intensive classification routines. The **Closeby points** routine is fast to apply for a large number of points.

Example for a multiple-scanner system data set:

- scanner 1 is better quality than scanner 2
- classify ground from each scanner into separate classes
- use ground points of scanner 2 only in places where there are no ground points of scanner 1.



SETTING	EFFECT
From class	Source class(es).
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **From class** field.
From line	Only points in the given line(s) are effected. Use a comma or minus to separate several line numbers, for example 2-5,10. 0-65535 refers to all lines.
From scanner	Only points captured by the given scanner(s) are effected. Use a comma or minus to separate several scanner numbers, for example 1-3,5. 0-255 refers to all scanners.
To class	Target class. Chose **Closest point class** if you want to deduce the class of a point in the source class(es) from the closest point in the **Specific class** selection.
Search type	Type of the search radius: • **2D** - a 2D search radius (horizontal dimension) is applied for finding closeby points.

SETTING	EFFECT
	• **2D above** - points are only classified if the reference points are above and within a 2D search radius. • **2D below** - points are only classified if the reference points are below and within a 2D search radius. • **3D** - a 3D search radius is applied for finding closeby points. • **3D above** - points are only classified if the reference points are above and within a 3D search radius. • **3D below** - points are only classified if the reference points are below and within a 3D search radius.
Radius	Size of the search radius around each point.
Inside fence only	If on, only points inside a fence or selected polygon(s) are classified.
Another line	If on, a point is classified if another point from any other line is within the given search radius.
Another scanner	If on, a point is classified if another points from any other scanner is within the given search radius.
Specific class	If on, a point is classified if another point from the given class(es) is within the given search radius.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Specific class** field.
Specific line	If on, a point is classified if another point from the given line(s) is within the search radius. Use a comma or minus to separate several line numbers, for example 2-5,10.
Specific scanner	If on, a point is classified if another point from the given scanner(s) is within the search radius. Use a comma or minus to separate several scanner numbers, for example 1-3,5.

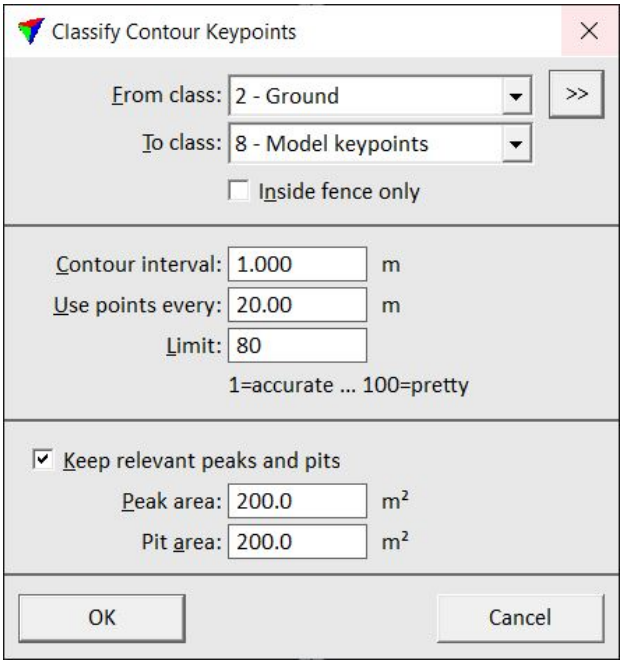
Contour keypoints

Contour keypoints routine classifies points for the production of contours. It works similar to the Model keypoints routine but produces a keypoint model which is suited for creating smooth, cartographic contour lines for map production.

If contours are derived from the original ground points, they are very accurate, detailed, and mathematically correct but less suited for being used for representation purposes. Small details in contour lines are caused by ground points close to contour line locations. Therefore, contour keypoints do not include points that are close to these locations.

The volumetric difference between a contour keypoint model and the model created from the original ground points is controlled by the **Limit** value for contour keypoints classification. The value determines how much the contours derived from the keypoints are smoothed. So higher the limit value, the smoother, nicer-looking the contours are but the less accurate the keypoint model is compared with the original ground model.

In addition to contour lines, the classification routine can also consider peaks and pits in the ground model. The values for **Peak area** and **Pit area** are used to define the minimum area that is enclosed by a contour line on top of hills or in depressions.



SETTING	EFFECT
From class	Source class(es).
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **From class** field.
To class	Target class.
Inside fence only	If on, points inside a fence or selected polygon(s) are classified.
Contour interval	Minimum interval planned for producing contours.

SETTING	EFFECT
Use points every	Minimum point density of contour keypoints. In areas of completely flat ground, there is a keypoint at approximately the given distance.
Limit	Determines the accuracy of the contour keypoint model and thus, the smoothness of the resulting contours.
Keep relevant peaks and pits	If on, areas on top of hills and in depressions are considered in the model.
Peak area	Minimum size of a peak area enclosed by a contour line.
Pit area	Minimum size of a pit area enclosed by a contour line.

Ground

Not Lite

Ground routine classifies ground points by creating a triangulated surface model iteratively. The routine is best suited for classifying ground in airborne laser data sets and in data sets where there is mainly natural terrain. For classifying ground in mobile data sets where the majority of ground is on hard surfaces, such as roads, use the Hard surface routine instead of the ground routine.

The routine is sensitive to low error points in the point cloud. Therefore, before running it, you should run one or more classification steps using the Low points routine.

The ground routine starts by selecting local low points that are confident hits on the ground. The initial point selection is controlled with the Max building size parameter. If the maximum building size is, for example, set to 60.0 m, the routine assumes that any 60 by 60 m area has at least one point on the ground level and that the lowest point is on the ground level.

Then, the routine builds a surface model (TIN) from the initial ground points. The triangles in this initial model are mostly below the ground level and only the vertices are touching the ground. In the following iterations, the routine molds the model upwards by adding more and more points. Each added point makes the model following the true ground surface more closely.

The iteration parameters of the routine determine how close a point must be to a triangle plane for being accepted as ground point and added to the model. **Iteration angle** is the maximum angle between a point, its projection on the triangle plane and the closest triangle vertex. This is the main parameter controlling how many points are classified into the ground class. The smaller the **Iteration angle**, the less eager the routine is to follow variation in the ground level, such as small undulations in terrain or points on low vegetation. Use a smaller angle value (close to 4.0) in flat terrain and a bigger value (close to 10.0) in mountainous terrain.

Iteration distance makes sure that the iteration does not make big jumps upward if triangles are large. This avoids ground points that are too high, for example within low vegetation or on low buildings.

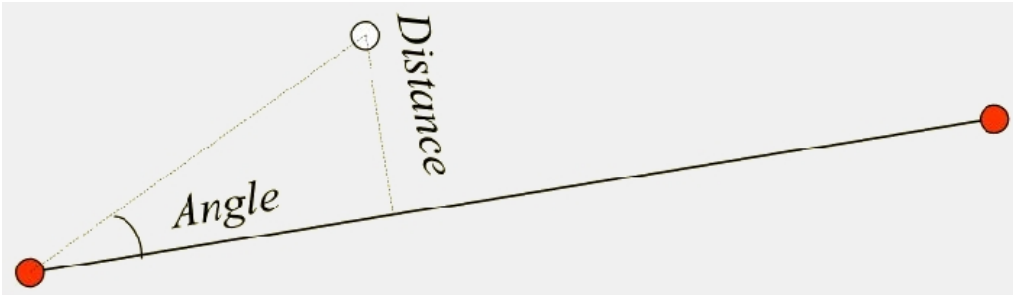
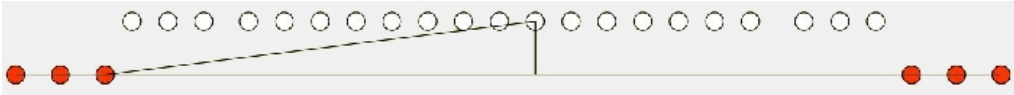
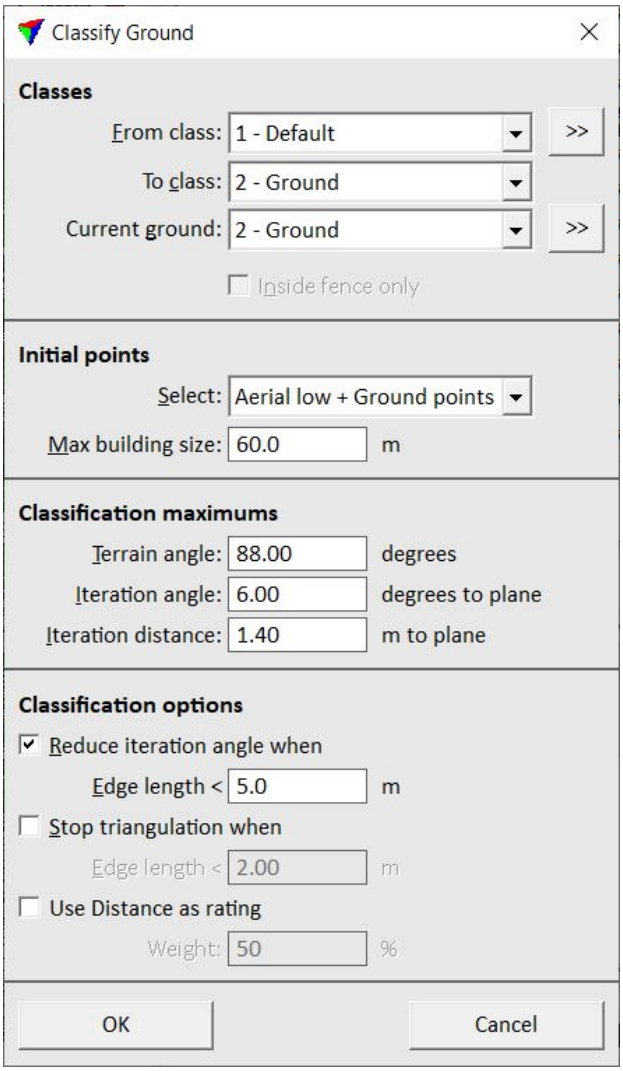


*Illustration of **Iteration angle** and **Iteration distance** parameters in the ground routine.*



*A smaller **Iteration distance** value avoids classification of ground points on low objects.*

The iteration angle can be reduced automatically, if the triangles become small. This reduces the eagerness to classify more ground points inside small triangles and thus, avoids unnecessary point density of the ground model. The iteration angle inside small triangles approaches zero if the longest triangle edge is shorter than a given **Edge length** value. Furthermore, the iteration can be stopped completely if triangle edges are shorter than a given limit.



SETTING	EFFECT
From class	Source class(es).
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **From class** field.
To class	Target class.
Current ground	Class(es) with already classified ground points. These points influence the classification of additional ground points but they are not effected themselves.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Current ground** field.

SETTING	EFFECT
Inside fence only	If on, points inside a fence or selected polygon(s) are classified.
Select	Selection of initial ground points. If starting a new ground classification, use **Aerial low + Ground points**. Use **Current ground points** when you want to continue the ground classification in a previously classified area.
Max building size	Defines the search area for initial ground points. The value should be close to the edge length of largest building in the project area.
Terrain angle	Steepest allowed slope in the ground surface. Use 88-90 degree if there are man-made objects in the project area. Use the estimated maximum terrain slope plus 10-15 degree if there is only natural terrain.
Iteration angle	Maximum angle between a point, its projection on the triangle plane and the closest triangle vertex.
Iteration distance	Maximum distance from a point to the triangle plane. Normally values between 0.5 and 1.5 m.
Reduce iteration angle when	If on, reduces the eagerness to add new points to ground inside a triangle if every edge of the triangle is shorter than **Edge length**. Avoids the addition of unnecessary point density to the ground model and reduces memory requirement.
Stop triangulation when	If on, stops processing points inside a triangle if every edge of the triangle is shorter than **Edge length**. Avoids the addition of unnecessary point density to the ground model and reduces memory requirement.
Use distance as rating	If on, the vegetation index stored as distance is used as probability factor for how likely a point is to be ground. This can improve the ground classification result for photogrammetric point clouds. Set the **Weight** higher, the more confident the vegetation index separates vegetated from non-vegetated areas.

After running the automatic ground routine, there might be the need to improve the result of ground classification manually. You may use half-automatic ways in order to do this task. One possibility is to run the ground routine on loaded points and inside a fence using the **Inside fence only** option of the routine. Alternatively, the Add point to ground command can be used to classify more points into the ground class within a limited area.

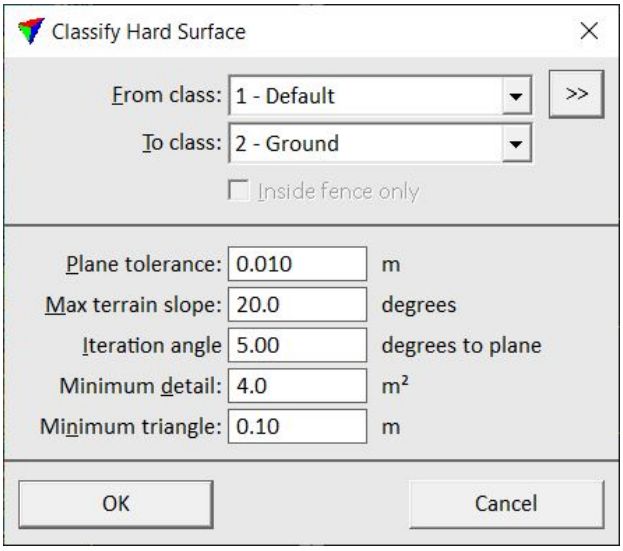
Hard surface

Not Lite

Hard surface routine classifies ground points by creating a triangulated surface model iteratively. The routine classifies dominant median surface points. It is best suited for classifying ground in mobile laser data sets and in data sets where there is mainly hard ground surface, such as paved roads or other areas. For classifying ground in airborne data sets where the majority of ground is in natural terrain, use the Ground routine instead of the hard surface routine. If you need to classify both, natural terrain and paved areas, you may need to draw polygons around natural/paved areas in order to apply on of the routines inside the polygons and the other routine for the rest of the data.

In contrast to the ground routine, the hard surface routine is not sensitive to low error points in the point cloud. Therefore, you do not need to run any low point classification before classifying ground with this routine. However, you may run other routines in order to limit the amount of points in the source class for the hard surface routine.

The hard surface routine is eager to classify points that form a local plane. The **Plane tolerance** given in the routine's settings determines how well the points must fit the plane. This is the main parameter controlling how many points are classified into the ground class.



SETTING	EFFECT
From class	Source class(es).
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source

SETTING	EFFECT
	classes from the list that are then used in the **From class** field.
To class	Target class.
Inside fence only	If on, points inside a fence or selected polygon(s) are classified.
Plane tolerance	Determines how well the points on a hard surface must fit a plane.
Max terrain slope	Maximum slope of the hard surface.
Iteration angle	Maximum angle between a point, its projection on the triangle plane and the closest triangle vertex.
Minimum detail	Minimum size of a continuous hard surface area.
Minimum triangle	Minimum edge length of a triangle.

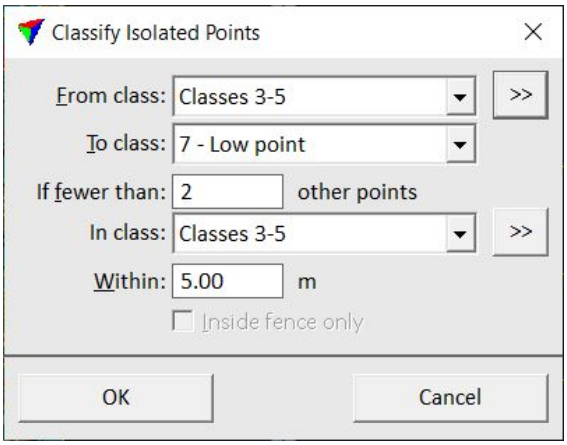
After running the automatic hard surface routine, there might be the need to improve the result of the classification manually. You may use half-automatic ways in order to do this task. One possibility is to run the hard surface routine on loaded points and inside a fence using the **Inside fence only** option of the routine.

Isolated points

Not Lite

Isolated points routine classifies points which have less neighbour points within a 3D search radius than defined in the routine's settings. This routine is useful for finding isolated points up in the air or below the ground.

In addition, it can be used to find points that do not have neighbour points in another class(es) within the given search radius. The same example provided for the Closeby points routine applies.



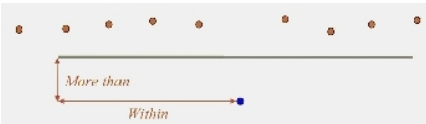
SETTING	EFFECT
From class	Source class(es).
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **From class** field.
To class	Target class.
If fewer than	A point is classified if there are less than the given number of neighbouring points within a 3D search radius. Normally a value between 1-5m.
In class	A point is classified if there are not enough neighbour points in the given class(es) within the search radius.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **In class** field.
Within	Size of the 3D search radius.
Inside fence only	If on, points inside a fence or selected polygon(s) are classified.

Low points

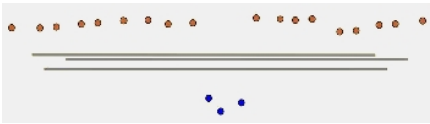
Not Lite

Low points routine classifies single points or groups of points which are lower than other points in the surrounding. It is often used to search for possible error points which are clearly below the ground.

The routine compares the elevation of each point or point group with any other point within a given 2D radius. If the point or point group is clearly lower than any other point, it is classified.



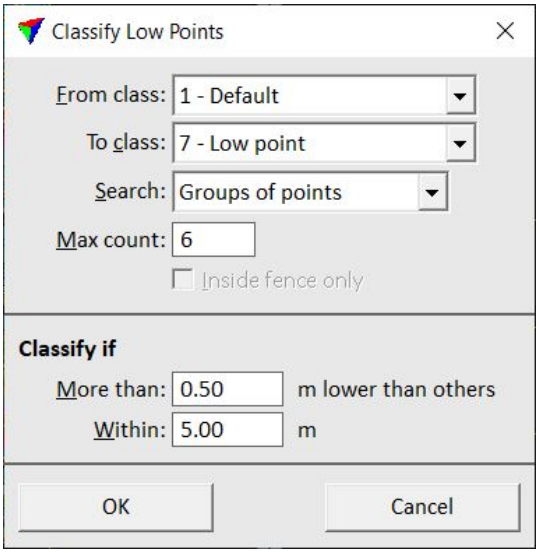
Single low point classification



Group of low points classification

The routine finds the lowest point or point group in a data set when it runs once. If there are low points on several elevation levels below the ground, it should be executed several times with different settings. Typically, the routine is included in a macro for ground classification in an airborne

laser data set and depending on the amount of error points below the ground, there are two or even more steps for low point classification.



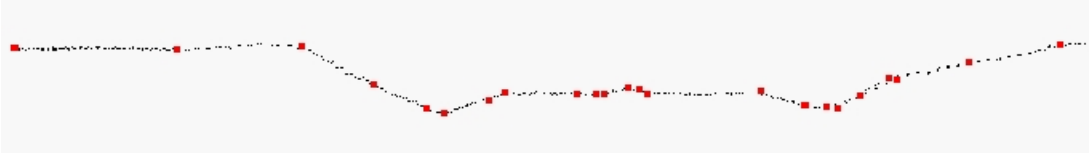
SETTING	EFFECT
From class	Source class(es).
To class	Target class.
Search	Defines the target points: **Single points** or **Groups of points**.
Max count	Maximum amount of points in a group of low points. This is only active if **Search** is set to **Groups of points**.
Inside fence only	If on, points inside a fence or selected polygon(s) are classified.
More than	Minimum height difference between a single point or a group of points and a point in the surrounding. Normally a value between 0.3 - 1.0 m.
Within	Size of the 2D search radius. Normally values between 2.0 - 8.0 m.

Model keypoints

Model keypoints routine classifies points which are needed to create a triangulated surface model of a given accuracy. This routine is normally used to create a thinned data set from previously classified as ground points. For **LAS 1.4** files, you can set the keypoint bit instead of classifying the points.

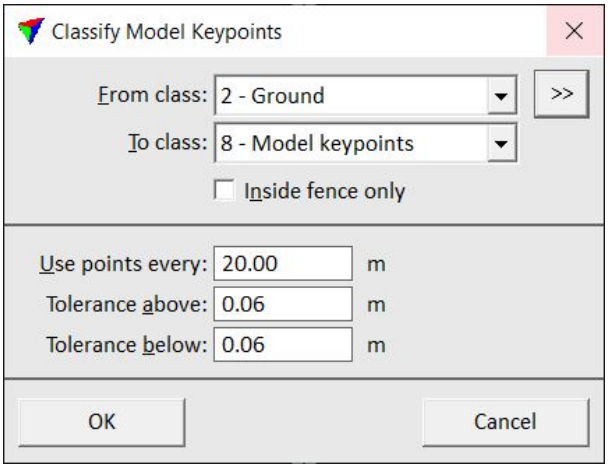
The routine tries to find a relatively small set of points (= keypoints) which create a triangulated model of the given accuracy. It starts by classifying all points from the source class, which is usually

the ground class, into the target class. Then, it iteratively removes all points from the target class that are not required in a surface model of the given accuracy.



Model keypoints (red points) as subset of ground points (black points).

You control the accuracy of the keypoint model with elevation tolerance settings **Above** and **Below** model. These settings determine the maximum allowed elevation difference between the keypoint model and a surface model created from the original ground points. The **Use points every** setting ensures a minimum point density in the final model. For example, if the distance between a point and its closest neighbour must not be less than 10 meter, you should set the **Use points every** setting to 10.0 m.



SETTING	EFFECT
From class	Source class(es). Contains usually ground points.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **From class** field.
To class	Target class. For **LAS 1.4** files, you can choose **Set keypoint bit** as alternative.
Inside fence only	If on, points inside a fence or selected polygon(s) are classified.
Use points every	Minimum point density of model keypoints. In areas of completely flat ground, there is a keypoint at approximately the given distance.
Tolerance above	Maximum allowed elevation difference upward between the triangulated keypoint

SETTING	EFFECT
	model and a model of the original source points.
Tolerance below	Maximum allowed elevation difference downward between the triangulated keypoint model and a model of the original source points.

Railroad

Not Lite

Railroad routine classifies points which match the elevation pattern of a railroad track.

The conditions for a point that matches the required pattern are:

- A point must have some points in the vicinity which are 0.05 - 0.35 m lower.
- A point can not have any points in the vicinity which would be 0.15 - 2.50 m higher.
- A point can not have any points in the vicinity which would be more than 0.35 m lower.

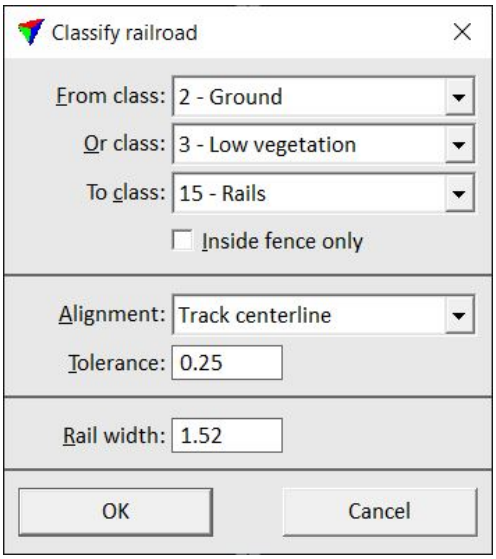
The routine tries to find points that match the elevation pattern and a given **Rail width ± Tolerance** setting. Further, it may be supported by an alignment element that can be created using the Place Railroad String tool in TerraScan or any other line string placement tool of the CAD platform.

The alignment element can be used in three different ways:

- **None** - No alignment is available. The routine tries to find points that match the elevation pattern and have another matching point at **Rail width ± Tolerance** distance in any direction.
- **Track centerline** - The alignment follows the centerline between two rails. The routine tries to find points that match the elevation pattern and are at **(0.5 * Rail width) ± Tolerance** distance from the alignment.
- **General direction** - The alignment defines the general direction of the railroad. The routine tries to find all points which match the elevation pattern and have another matching point at **Rail width ± Tolerance** distance. The line between the two points must be close to perpendicular to the alignment direction.

The alignment element(s) must be selected before starting the classification routine.

The routine is suited for ALS data sets with a comparatively low point density. For dense MLS data sets, there are better-suited tools for processing data of railroads in the Railroad toolbox, such as Find Rails tool.



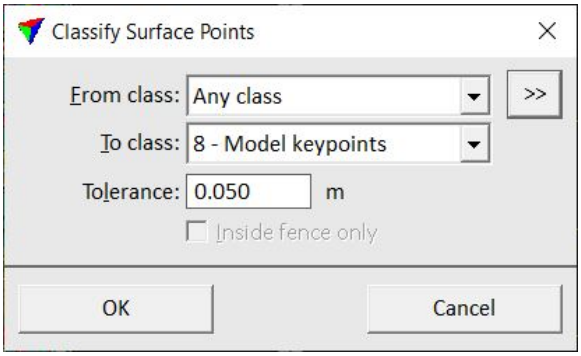
SETTING	EFFECT
From class	Source class(es).
Or class	Second source class.
To class	Target class.
Inside fence only	If on, points inside a fence or selected polygon(s) are classified.
Alignment	Usage of an alignment: **None**, **Track centerline**, or **General direction**.
Tolerance	Horizontal offset by which the alignment can differ from the true centerline locations. This is only active if **Alignment** is set to Track **centerline**.
Rail width	Track width, distance from rail center to rail center. The field is followed by another input field that is only active if **Alignment** is set to **None** or **General direction**. The field defines the horizontal offset by which the track width can differ.

Surface points

Not Lite

Surface points routine classifies points that fit to locally smooth surfaces. At classifies points on planar surfaces of any direction and also on rounded surface. The routine can be used as pre-classification for ground, building roof or wall classification in noisy point clouds, such as photogrammetric point clouds.

After classifying surface points, it might be useful to smooth the points using the **Xyz** method before running additional classification routines.

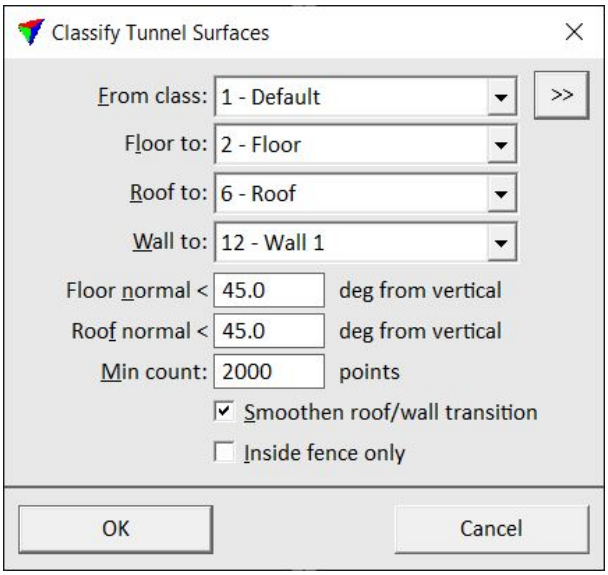


SETTING	EFFECT
From class	Source class(es).
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **From class** field.
To class	Target class.
Tolerance	Offset from the main surface up to which points are included in the classification. Given in CAD file units.
Inside fence only	If on, points inside a fence or selected polygon(s) are classified.

Tunnel surfaces

Not Lite

Tunnel surface routine classifies points collect inside tunnels. The routine classifies points on the tunnel roof, floor and walls. It requires the computation of normal vectors which can be done with the Compute normal vectors command or macro action. The normal vector direction should be turned towards the inside of the tunnel determined by either a trajectory or a vector element.



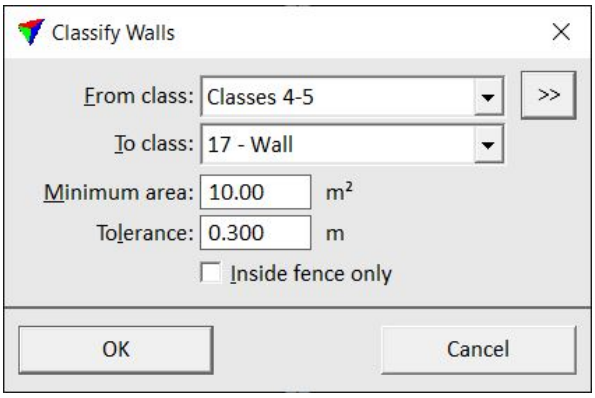
SETTING	EFFECT
From class	Source class(es).
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **From class** field.
Floor to	Target class for points on the tunnel floor.
Roof to	Target class for points on the tunnel roof.
Wall to	Target class for points on the tunnel walls.
Floor normal	Maximum angle off from vertical of the normal vector for points on the tunnel roof.
Roof normals	Maximum angle off from vertical of the normal vector for points on the tunnel floor.
Min count	Minimum amount of points per surface patch belonging to one target class. If a patch contains less points, it is added to the larger neighbouring class.
Smoothen roof/wall transitions	If on, the transition between roof and wall surface patches is smoothed.
Inside fence only	If on, points inside a fence or selected polygon(s) are classified.

Walls

Not Lite

Wall routine classifies points on wall-like surfaces in mobile data sets. The routine requires that normal vectors for the points in the source class(es) are available. Normal vectors can be stored as attributes for points in TerraScan FastBinary files. They must be computed by using the Compute normal vectors command.

The routine classifies points on planar surfaces that are nearly vertical (within 10 degree) and larger than a given minimum area. It classifies the points on the surface and also smaller details up to a given distance from the main surface.



SETTING	EFFECT
From class	Source class(es).
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **From class** field.
To class	Target class.
Minimum area	Minimum size of a wall surface. Given in squared CAD file units.
Tolerance	Offset from the main planar wall surface up to which points are included in the classification. Given in CAD file units.
Inside fence only	If on, points inside a fence or selected polygon(s) are classified.

Wire danger points

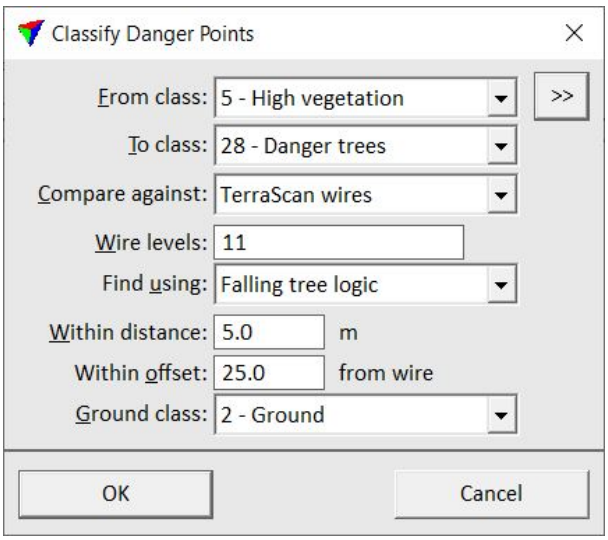
Not Lite

Wire danger points routine classifies points which are within a certain distance from reference elements. The reference elements can be wire elements produced by tools of the Vectorize Wires toolbox or any linear element drawn in the CAD file.

The routine can be used to classify points on potential danger objects along powerlines, rail tracks, roads or other corridor objects. There are three methods how potential danger points are defined:

- **Vertical distance to wire** - points within a vertical distance from a reference element are classified.
- **3D distance to wire** - points within a 3D radius around a reference element are classified.
- **Falling tree logic** - each point is considered as the tip of a tree with its trunk at the xy location of the point and the elevation of the base point on the ground. If the point "falls like a tree" and falls into a 3D buffer area around a reference element, it is classified.

The routine is only available in macros. See Chapter Macros for more information about macro creation and macro actions. However, the routine works similar as the classification part of the Find Danger Objects tool in the View Powerline toolbox.



SETTING	EFFECT
From class	Source class(es).
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **From class** field.
To class	Target class.
Compare against	Defines the reference elements: **TerraScan wires** or **Any linear elements**.

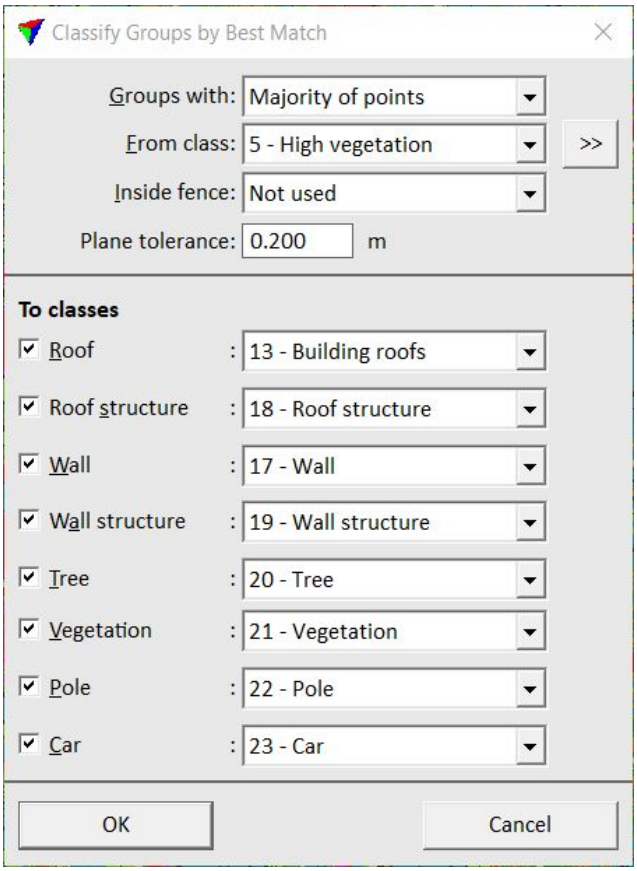
SETTING	EFFECT
Wire levels	CAD file level(s) on which the reference elements are placed. Separate several level numbers by a comma or minus, for example 1-10,13.
Find using	Method of danger point definition: **Vertical distance to wire**, 3D **distance to wire**, or **Falling tree logic**.
Within distance	Radius that defines a 3D buffer around a reference element. The buffer is added to the reference element for computing distances to points.
Within offset	Horizontal distance from a reference element that defines a corridor. Points within the given corridor are analyzed whether they are potential danger points or not.
Ground class	Point class with ground points. This is only active if **Find using** is set to **Falling tree logic**.

By best match

Not Lite

By best match routine classifies groups of points into several classes according to object types. For each group, the routine tests the probability that the group represents an object, such as a building roof, a wall, a tree, a pole, and so on. The group is classified as the object that gets the highest probability.

The routine requires that points have been assigned to groups using the Assign groups command or corresponding macro action.



SETTING	EFFECT
Groups with	Determines which groups are classified: • **One or more points** - groups with one or more points in the source class. • **Majority of points** - groups with the majority of points in the source class. • **All points** - groups with all points in the source class.
From class	Source class from which to classify points.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **From class** field.
Inside fence	Determines how a fence or selected polygon(s) effect the classification: • **Not used** - fence or selected polygons are ignored. • **One or more points** - groups are classified if one or more points are inside. • **Average xy** - groups are classified if the average xy point is inside. • **Majority of points** - groups are classified if the majority of points is inside.

SETTING	EFFECT
	• **All points** - groups are classified if all points are inside.
Plane tolerance	Distance how well points of a group must match a plane equation. This effects the classification of groups as roofs and walls. If a group is large enough and fulfills the planarity condition, it's classified as roof or wall. The smaller the tolerance value, the less likely a group is classified as roof or wall.
To classes	If an option is switched on, the routine checks the probability of groups fitting to the object type. If a group fits to an object type, the points of the group are classified into the given target class.

By centerline

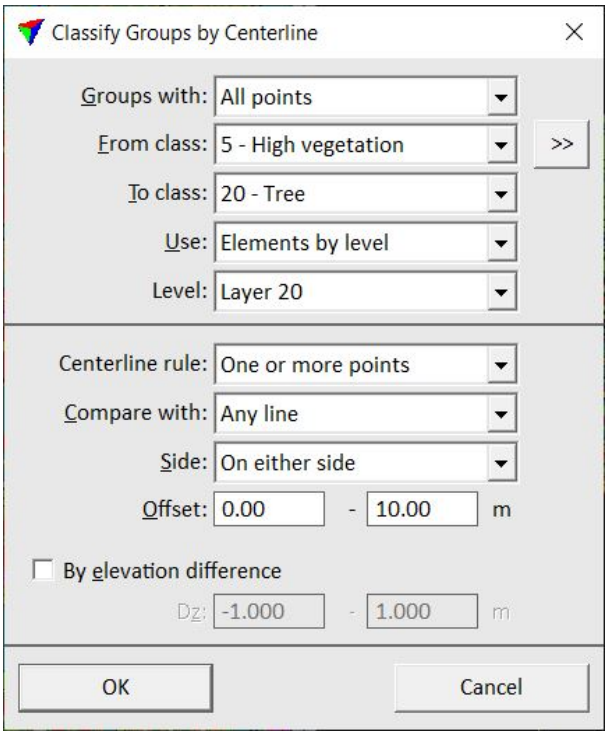
Not Lite

By centerline routine for groups classifies points into another class if all points, the majority of points, or just a single point of a group is within a given distance from a linear element.

The linear elements that are used by this routine can be lines, polylines (Spatix) or line strings (MicroStation), arcs, polygons (Spatix) or shapes (MicroStation), big elements (Spatix) or complex elements (MicroStation) consisting valid simple element types.. The classification is performed by using either selected elements or elements on a specified CAD file level.

If multiple elements are used, each laser point is classified according to the offset distance to either the closest linear element or to any of the linear elements. In addition to the horizontal distance, the elevation distance can be included in the classification process.

The routine requires that points have been assigned to groups using the Assign groups command or corresponding macro action.



SETTING	EFFECT
Groups with	Determines which groups are classified: • **One or more points** - groups with one or more points in the source class. • **Majority of points** - groups with the majority of points in the source class. • **All points** - groups with all points in the source class.
From class	Source class.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **From class** field.
To class	Target class.
Use	Defines what elements are used: • **Selected linear elements** - any selected elements in the CAD file. This requires the selection of elements before starting the routine. • **Elements by level** - any elements that are located on a given CAD file level.
Level	Name of the CAD file level where elements are located that are used for the classification. This is only active if **Use** is set to **Elements by level**.

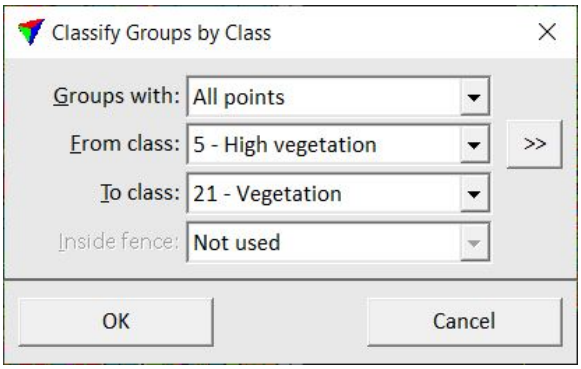
SETTING	EFFECT
Centerline rule	Determines the reference point(s) within a group: • **One or more points** - groups are classified if one or more points are within the given distance. • **Average xy** - groups are classified if the average xy point is within the distance. • **Majority of points** - groups are classified if the majority of points is within the distance. • **All points** - groups are classified if all points are within the distance.
Compare with	Determines which linear element effects the classification of a group: • **Any line** - groups are classified if they are within the distance of any linear element. • **Closest line** - groups are classified if they are within the distance of the closest linear element.
Side	Side on which to classify groups: **On left side**, **On either side**, or **On right side**. The side is relative to the digitization direction of the linear element.
Offset	Minimum and maximum horizontal distance. Groups within the offset range are classified.
By elevation difference	If on, only groups within the given elevation distance range from the linear element are classified. Define the elevation offset in the **Dz** fields.

By class

Not Lite

By class routine for groups classifies points into another class if all points, the majority of points, or just a single point of a group belongs to the source class.

The routine requires that points have been assigned to groups using the Assign groups command or corresponding macro action.



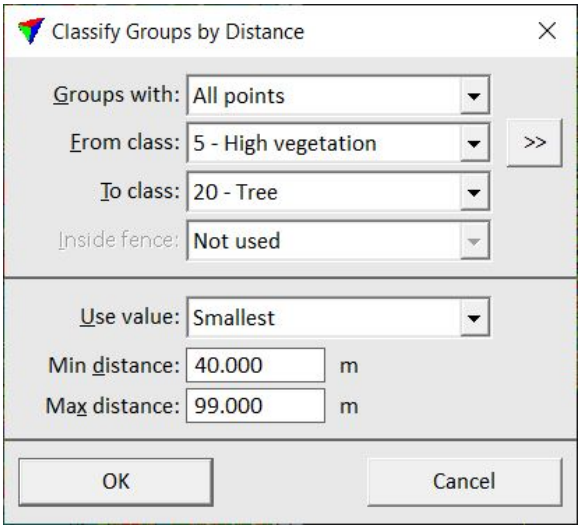
SETTING	EFFECT
Groups with	Determines which groups are classified: • **One or more points** - groups with one or more points in the source class. • **Majority of points** - groups with the majority of points in the source class. • **All points** - groups with all points in the source class.
From class	Source class from which to classify points.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **From class** field.
To class	Target class to classify points into.
Inside fence	Determines how a fence or selected polygon(s) effect the classification: • **Not used** - fence or selected polygons are ignored. • **One or more points** - groups are classified if one or more points are inside. • **Average xy** - groups are classified if the average xy point is inside. • **Majority of points** - groups are classified if the majority of points is inside. • **All points** - groups are classified if all points are inside.

By distance

Not Lite

By distance routine classifies groups of points which are within a given distance range.

The routine requires that points have been assigned to groups using the Assign groups command or corresponding macro action.



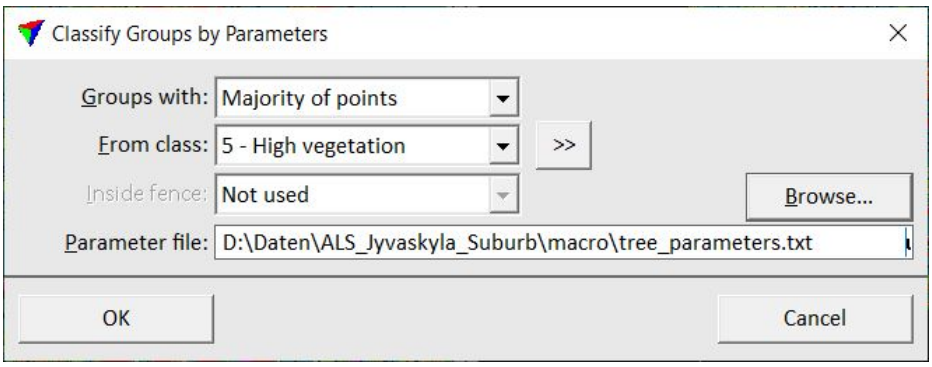
SETTING	EFFECT
Groups with	Determines which groups are classified: • **One or more points** - groups with one or more points in the source class. • **Majority of points** - groups with the majority of points in the source class. • **All points** - groups with all points in the source class.
From class	Source class from which to classify points.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **From class** field.
To class	Target class to classify points into.
Inside fence	Determines how a fence or selected polygon(s) effect the classification: • **Not used** - fence or selected polygons are ignored. • **One or more points** - groups are classified if one or more points are inside. • **Average xy** - groups are classified if the average xy point is inside. • **Majority of points** - groups are classified if the majority of points is inside. • **All points** - groups are classified if all points are inside.
Use value	Value that determines the distance of a group:

SETTING	EFFECT
	• **Biggest** - groups is classified if the biggest distance value is within the given distance range. • **Median** - groups is classified if the median distance value is within the given distance range. • **Average** - groups is classified if the average distance value is within the given distance range. • **Smallest** - groups is classified if the smallest distance value is within the given distance range.
Min distance	Minimum distance value.
Max distance	Maximum distance value.

By parameters

Not Lite

By parameters routine classifies groups of points according to parameters stored in a parameter file. The parameter file can be created by using the Test parameters command.



SETTING	EFFECT
Groups with	Determines which groups are classified: • **One or more points** - groups with one or more points in the source class. • **Majority of points** - groups with the majority of points in the source class. • **All points** - groups with all points in the source class.
From class	Source class from which to classify points.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can

SETTING	EFFECT
	select multiple source classes from the list that are then used in the **From class** field.
Inside fence	Determines how a fence or selected polygon(s) effect the classification: • **Not used** - fence or selected polygons are ignored. • **One or more points** - groups are classified if one or more points are inside. • **Average xy** - groups are classified if the average xy point is inside. • **Majority of points** - groups are classified if the majority of points is inside. • **All points** - groups are classified if all points are inside.
Parameter file	Text file that stores the parameters for classification. Use the **Browse** button in order to open a standard dialog for opening a file.

By vegetation index

Not Lite

By vegetation index routine classifies groups of points based on a vegetation index computation. The routine requires that color values have been extracted for the points. Color values for laser points can be extracted from raw images or raster attachments loaded into TerraPhoto, or assigned per class by setting values for red, green, and blue channels. See Extract color from images and Assign color to points for corresponding commands in TerraScan.

TerraScan can assign up to 10 color channels for each point. The maximum amount of color channels can only be stored in the TerraScan FastBinary format. LAS 1.4 format and later can store up to 4 color channels, LAS 1.2 format and later up to 3.

There are two methods for computing the vegetation index.

Normalized difference

The method assumes that channel 0 stores red (R) and channel 3 near-infrared (NIR) color values. The normalized difference value is computed with the following equation:

```
ND = (NIR - R) / (NIR + R)                    -1 <= ND <= +1
```

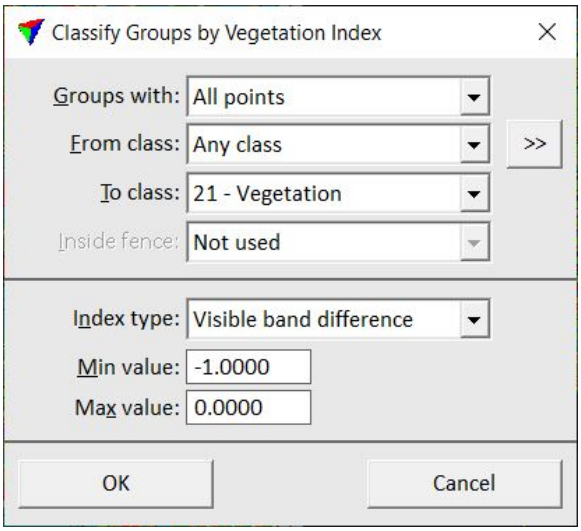
Visual band difference

The method assumes that channel 0 stores red (R), channel 1 green (G), and channel 2 blue (B) color values. The visual band difference value is computed with the following equation:

```
VBD = (2*G - R - B) / (2*G + R + B)                -1 <= VBD <= +1
```

The routine is well suited for separating points on vegetation from non-vegetation points. You can check how well the vegetation index computation separates vegetation from non-vegetation points by displaying points by vegetation index. The larger the difference value, the more likely a point is representing vegetation.

The routine further requires that points have been assigned to groups using the Assign groups command or corresponding macro action.



SETTING	EFFECT
Groups with	Determines which groups are classified: • **One or more points** - groups with one or more points in the source class. • **Majority of points** - groups with the majority of points in the source class. • **All points** - groups with all points in the source class.
From class	Source class(es).
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **From class** field.
To class	Target class.
Inside fence	Determines how a fence or selected polygon(s) effect the classification: • **Not used** - fence or selected polygons are ignored. • **One or more points** - groups are classified if one or more points are inside. • **Average xy** - groups are classified if the average xy point is inside. • **Majority of points** - groups are classified if the majority of points is inside.

SETTING	EFFECT
	• **All points** - groups are classified if all points are inside.
Index type	Method of vegetation index computation: **Normalized difference** or **Visual band difference**.
Min value	Minimum difference value to be classified.
Max value	Maximum difference value to be classified.

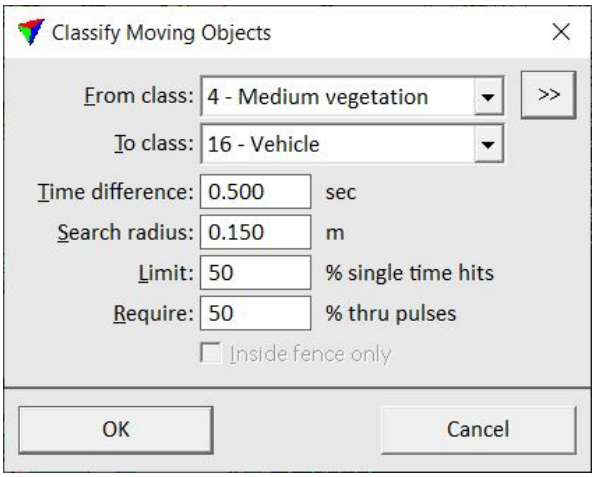
Moving objects

Not Lite

Moving objects routine classifies groups of points that are captured from moving objects. It can be used for mobile laser data sets. The routine requires that data from multiple lines or scanners is available.

The routine classifies groups of points that are captured at one moment of time and at another moment of time, when the area was captured by another scanner or in another line, there are no points at the same location but point from behind the moving object.

The routine requires that points have been assigned to groups using the Assign groups command or corresponding macro action.



SETTING	EFFECT
From class	Source class(es).
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **From class** field.

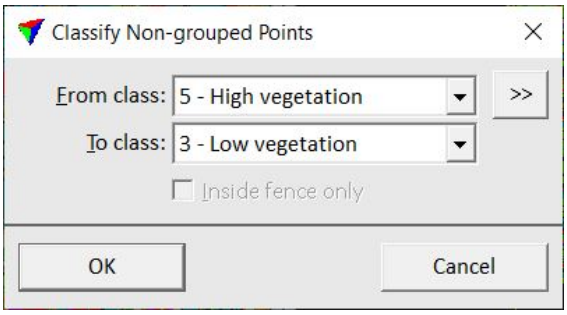
SETTING	EFFECT
To class	Target class.
Time difference	Minimum time difference between points on the moving object and points from behind the moving object.
Search radius	3D radius around each point within which only points of the same group can be located. Points on a moving object are considered to belong to the same group. A group is classified if there are no points from another group or without a group number falling within the given search radius.
Limit	A group is classified, if at least the given percentage of points are hits from the moving object captured at one point of time.
Require	
Inside fence only	If on, points inside a fence or selected polygon(s) are classified.

Non-grouped points

Not Lite

Non-grouped points routine classifies points that are not assigned to any group.

The routine requires that points have been assigned to groups using the Assign groups command or corresponding macro action.



SETTING	EFFECT
From class	Source class(es).
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **From class** field.

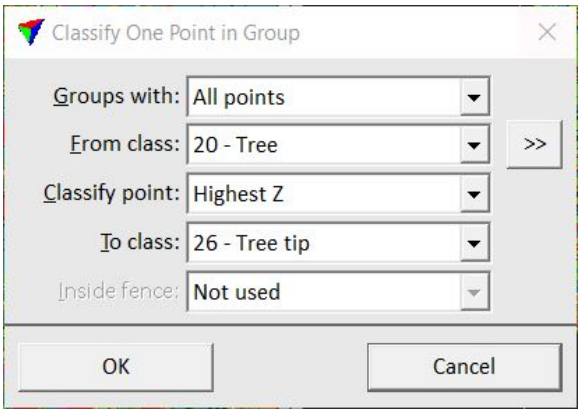
SETTING	EFFECT
To class	Target class.
Inside fence only	If on, points inside a fence or selected polygon(s) are classified.

One point in group

Not Lite

One point in group routine for groups classifies one point for each group that fullfills the classification condition. This may be the point with the highest or lowest value of point attributes, or the point closest to the 3D center of the group.

The routine requires that points have been assigned to groups using the Assign groups command or corresponding macro action.



SETTING	EFFECT
Groups with	Determines which groups are classified: • **One or more points** - groups with one or more points in the source class. • **Majority of points** - groups with the majority of points in the source class. • **All points** - groups with all points in the source class.
From class	Source class from which to classify points.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **From class** field.
Classify point	Classification condition: • **Highest Z** - highest elevation value.

SETTING	EFFECT
	• **Lowest Z** - lowest elevation value. • **Closest to 3D center** - closest to the 3D center of the group. • **Highest intensity** - highest intensity value. • **Lowest intensity** - lowest intensity value. • **Biggest distance** - biggest distance value. This requires that a distance value has been computed for the points in the source class. • **Smallest distance** - smallest distance value. This requires that a distance value has been computed for the points in the source class.
To class	Target class to classify points into.
Inside fence	Determines how a fence or selected polygon(s) effect the classification: • **Not used** - fence or selected polygons are ignored. • **One or more points** - groups are classified if one or more points are inside. • **Average xy** - groups are classified if the average xy point is inside. • **Majority of points** - groups are classified if the majority of points is inside. • **All points** - groups are classified if all points are inside.

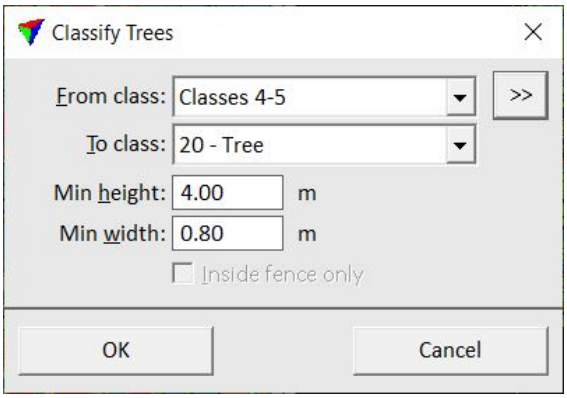
Trees

Not Lite

Trees routine classifies groups of points that are captured from tree crowns. It can be used for mobile laser data sets. The routine requires that normal vectors have been computed using the Compute normal vectors command or corresponding macro action. The normal vector attributes can be stored for points in TerraScan FastBinary files. The routine classifies irregular groups of points.

Further, it is recommended to classify points above the ground into a separate class, so that this class contains points in an elevation range above ground where tree crowns are included.

The routine requires that points have been assigned to groups using the Assign groups command or corresponding macro action.



SETTING	EFFECT
From class	Source class(es).
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **From class** field.
To class	Target class.
Min height	Minimum height of a point group above ground considered for classification.
Min width	Minimum width of a point group considered for classification.
Inside fence only	If on, points inside a fence or selected polygon(s) are classified.

Another way to classify trees is provided by the By best match routine for point groups.

Macros

Macros provide a method to automate processing steps. The best level of automation is reached when using macros together with a project definition. See Chapter Working with Projects for information about projects in TerraScan.

Macros consist of a number of processing steps which are executed one after the other. Processing steps can classify points, modify points, delete points, transform points, output points, update views, execute commands, or call functions from other terra applications. Macros may also contain comments.

Macros are stored as text files with the default extension .MAC.

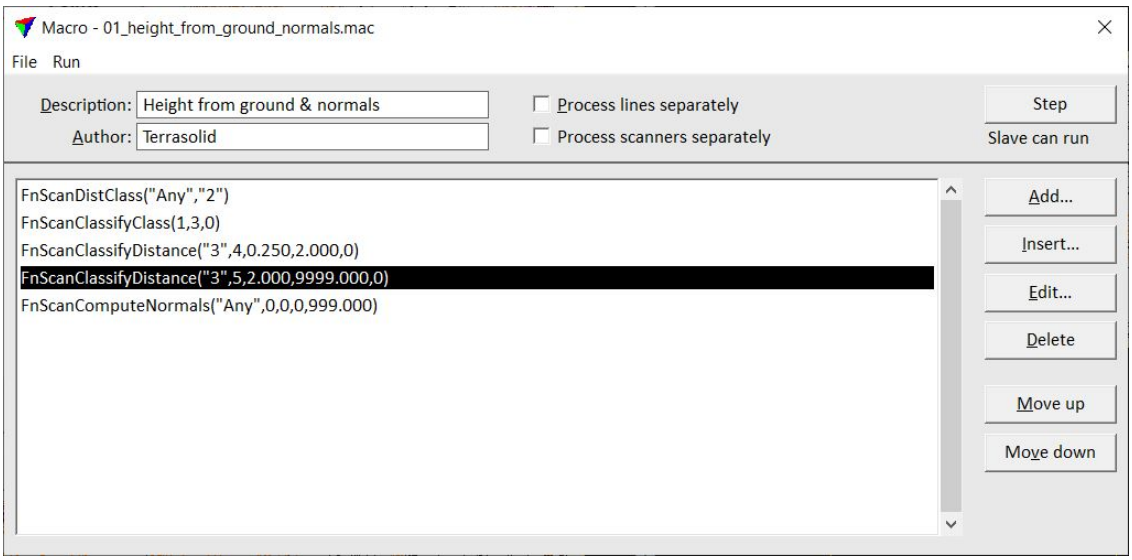
Create a macro

A macro can be created in TerraScan using the Macro command from the **Tools** pulldown menu in the **TerraScan** window. The **Macro** dialog lets you add, modify, delete, and arrange processing steps and comments. It also provides pulldown menu commands to open, save, and run macros. Each line in the macro dialog represents one comment line or one processing step that is performed on the point cloud when running a macro.

To create a macro:

1. Choose **Macro** command from the **Tools** pulldown menu in the **TerraScan** window.

   This opens the **Macro** dialog:



2. (Optional) Type a description of the macro in the **Description** field.

3. (Optional) Type your name in the **Author** field.

4. Click **Add** to add a new processing step to the macro.

This opens the **Macro step** dialog, a dialog for defining Macro actions and related settings.

5. Define settings for the macro step and click OK.

   You can continue to steps 4 and 5 until all the steps are complete.

6. Select **Save as** from the **File** pulldown menu to save the macro into a file.

SETTING / BUTTON / COMMAND	EFFECT
Description	Description of the macro.
Author	Name of the author.
Process lines separately	If on, the macro steps are performed for each line separately. Line refers to data of a single line in an ALS data set or a single drive path in an MLS data set. Lines are identified in the point cloud by the line attribute, a unique number for each point.
Process scanners separately	If on, the macro steps are performed for each scanner separately. Scanner refers to data of a single scanner in a multiple scanner system (usually MLS systems). Scanners are identified in the point cloud by the scanner attribute, a unique number for each point.
Step	By clicking this button, the selected macro step is executed on loaded points. With this button, you may check settings of macro actions step-by-step.
Slave can (not) run	The note indicates whether a macro can be processed in TerraSlave or not. See TerraSlave User Guide for more information.
Add	Add a new macro step.
Insert	Insert a new macro step before the selected macro step.
Edit	Modify settings for the selected macro step.
Delete	Delete the selected macro step.
Move up	Moves a selected macro step one line up in the macro.
Move down	Moves a selected macro step one line down in the macro.
File / New	Create a new macro. This removes any active macro from the macro dialog and starts a new macro definition. If another macro is still active, a dialog asks you to confirm whether you want to start a new macro or not.

SETTING / BUTTON / COMMAND	EFFECT
File / Open	Open an existing macro file. This opens a standard Windows dialog for opening files.
File / Save	Save changes to an existing macro file.
File / Save As	Save a macro into a new file. This opens a standard Windows dialog for saving files.
Run / On loaded points	Run the active macro on loaded points.
Run / On selected files	Run the active macro on selected files.

Run macros in TerraSlave

Not UAV

TerraSlave is a program for executing TerraScan macros. It is included in a full TerraScan version but it can also run with an own license on a computer without TerraScan and a CAD platform installed. It can be launched from TerraScan Run macro on blocks dialog.

The main benefits of using TerraSlave are:

- The CAD platform and TerraScan are immediately free for interactive work while the macro is running.
- It can use more memory and processor power than an application running on top of a CAD platform.
- It can queue several tasks for being executed one after the other without further interaction.
- It can utilize several computers in the network.
- Powerful servers can be used for batch processes.

Besides executing the macro steps, TerraSlave provides a simple task management system called TerraDispatcher. This includes the display of processed working segments and computers involved in the processing task, the ability to abort tasks, the ability to restart processing for a working segment and other options to interact with the process. The TerraSlave User Guide describes the application in detail.

There are some macro actions and processing options that do not work with TerraSlave. This includes:

- Start a task on computer 1 and start another task on computer 2 modifying the same data set.
- Run on another computer, if local paths are used. Network directories used in macros must be shared in order to enable TerraSlave to read and write data into a folder over the network.
- Run on multiple computers, if an output step writes into the same output file for all blocks (Output points or Output by line macro steps).
- Use **Inside fence only** setting in a macro step.
- Use Keyin command step.
- Use a step that requires functionality of other Terra applications, such as Apply correction and Create model.
- Use Write to design step.

The Macro dialog shows whether a macro can run in TerraSlave or not. See Create a macro for more information. The options for TerraSlave processing are stored in the macro file header:

- **SlaveCanRun=1** - Macro can be executed by TerraSlave.
- **AnotherComputerCanRun=1** - Macro can be executed by TerraSlave on another computer.
- **CanBeDistributed=1** - Macro can be executed by TerraSlave on multiple computers.

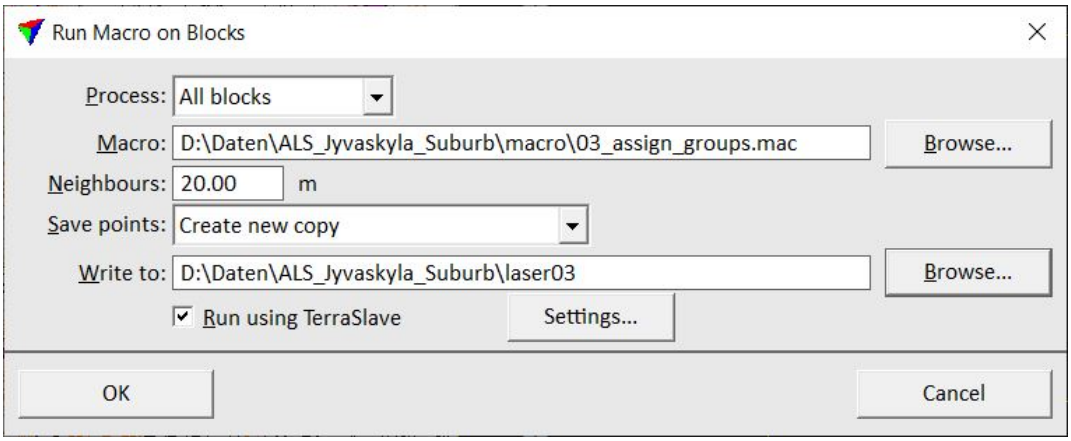
If one of these options is set to **0**, the option is not available due to the above listed restrictions for running macros in TerraSlave. Macros are stored as text files, so you can check the file header in any text editor.

To run a macro in TerraSlave:

1. (Optional) Select blocks in the TerraScan Project window if you want to execute the macro on selected blocks only.

2. Select **Run macro** command from the **Tools** pulldown menu in the TerraScan Project window.

   This opens the **Run Macro on Blocks** dialog:

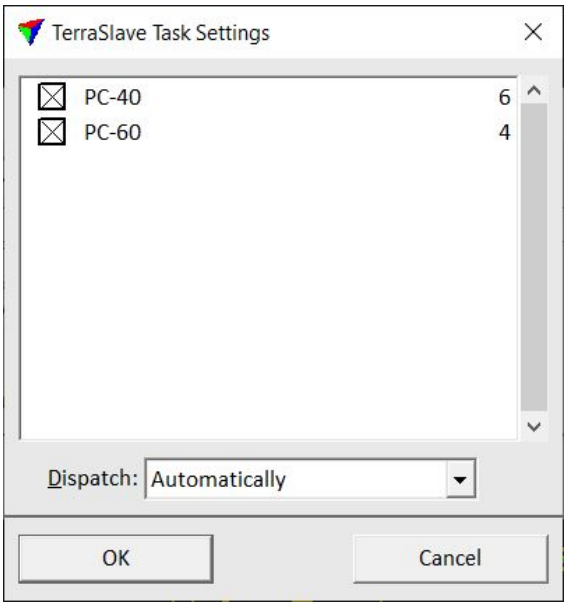


3. Define settings as described in Run a macro on a project.

4. Switch on **Run using TerraSlave**.

5. Click on **Settings** button.

   This opens the **TerraSlave Task Settings** dialog:



The list shows the computers set for TerraSlave processing in the Slave computers category of TerraScan **Settings**. By default, all computers are switched on. Switch off a computer that you want to exclude from processing the macro. You can also change the number of instances used for TerraSlave processing on a computer. Click on the number behind the computer name and type a new number in the text field. Press <Enter> to set the new number.

By default, TerraDispatcher will take care of dispatching the task to computers and instances automatically. Select **Manually** in the **Dispatch** list if you want to control this manually.

6. Select another option if you want to execute the macro on another computer.

7. Click OK.

8. Click OK to the **Run macro on blocks** dialog.

   This launches TerraDispatcher if it is not already running and starts processing the task in TerraSlave. If another process is already running, the task file is added to the queue and processed as soon as the other tasks are finished. See TerraSlave background processing workflow for a more detailed description of TerraSlave task management.

Run a macro on a project

Not UAV

Most often, macros are executed on a TerraScan project. See Working with Projects for detailed information about projects in TerraScan.

You can run a macro on all or selected blocks of a project. A macro can be performed by either using TerraScan or TerraSlave.

After a macro has been executed on a project in TerraScan, the **Macro execution** window appears. It shows a report that lists for each block binary file the amount of loaded points, the executed macro steps and a return value, as well as the amount of saved points, if applicable. The return value can be the amount of points that has been effected by the macro step or another value specifying the result

of the macro step. If a processing step failed, the line in the report appears in red color. A definition of the return values for many processing functions is given in Section Function Prototypes. The macro report can be saved as text file using the **Save as** command from the **File** pulldown menu or printed using the **Print** command from the **File** pulldown menu in the **Macro execution** dialog.

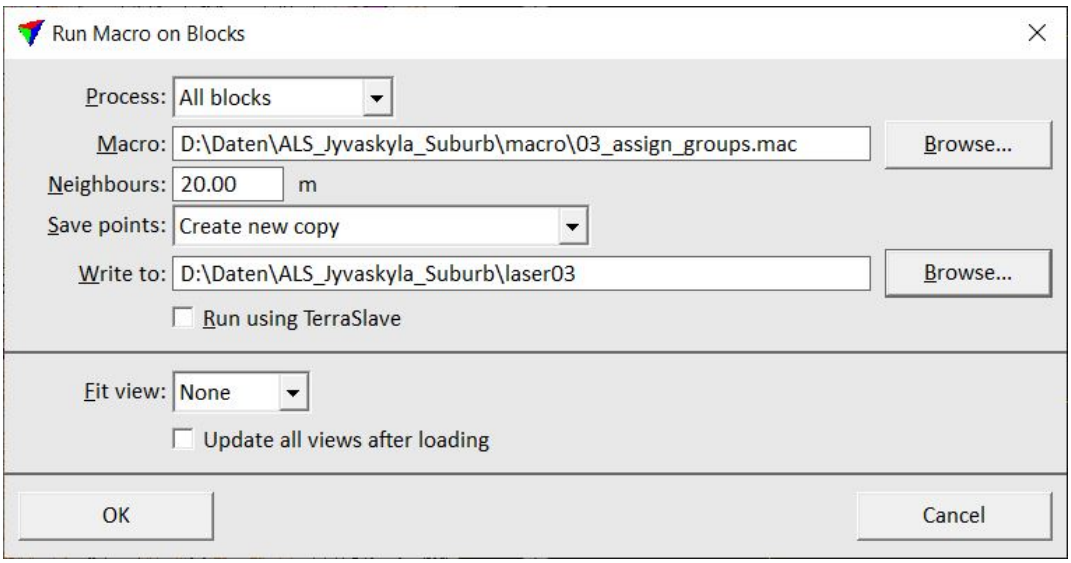
If a macro is executed in TerraSlave, the report of the macro execution is saved as text file in the TerraSlave installation folder, such as C:\TERRA64\TSLAVE\REPORTS. One report file is created for each block included in the macro process. The TerraSlave User Guide describes in detail the processing workflow of TerraSlave.

To execute a macro on blocks:

1. (Optional) Select blocks in the TerraScan Project window if you want to execute the macro on selected blocks only.

2. Select **Run macro** command from the **Tools** pulldown menu in the TerraScan Project window.

   This opens the **Run Macro on Blocks** dialog:



3. Define settings and click OK.

   This executes the macro on all or selected blocks. The application loops through the blocks, loads the corresponding block binary file, performs the macro step(s) and, if applicable, saves the modified points back to the block binary file or into another output file.

SETTING	EFFECT
Process	Blocks to process: **All blocks** or **Selected blocks**.
Macro	Macro file to execute.
Neighbours	Distance from the boundary of the active block for which the application loads points from neighbouring blocks. See Section Neighbour points for more information.
Save points	Method of saving the points after processing:

SETTING	EFFECT
	• **Do not save** - points are not saved to the original block binary files. Use this, if the macro includes an output step which writes the results into new files. • **Write over original** - original block binary files are overwritten. • **Temporary copy & replace original** - a copy is created for each processed block binary file. TerraScan moves the copies at the end of the macro process to replace the original block binary files. • **Create new copy** - a new file is created for each processed block binary file. Additionally, the project file is copied to the directory where the new binary files are stored. The original block binary files are not changed.
Temporary	Directory for storing the temporary copies of the block binary files. This is only active if **Save points** is set to **Temporary copy & replace original**.
Write to	Directory for storing the new block binary files. This is only active if **Save points** is set to **Create new copy**.
Run using TerraSlave	If on, the macro is performed by TerraSlave. See Section Run macros in TerraSlave for more information.
Fit view	Views to fit after loading a block binary file: **None**, **All**, or any CAD file view specified by its number.
Update all views after loading	If on, all views are updated after loading a block binary file.

Neighbour points

Neighbour points should be loaded in addition to the points of the active block, if the macro includes certain processing steps. For these steps, the application needs information from neighbouring areas in order to process the points along block boundaries correctly and to ensure smooth transitions between blocks. The steps include:

- all classification steps that include a search radius parameter or rely on points within a local environment around a point, such as Air points, Closeby points, Isolated points, Low points, Below surface
- Ground, Hard surface and Surface classification
- Model keypoints and Contour keypoints classification
- Buildings and Walls classification

- Assign groups, Fix border groups, Classification routines for groups, Split groups by class
- Compute distance, Compute normal vectors, Compute section parameters, Compute slope arrows
- Cut overlap
- Export lattice
- Find paint lines, Find poles, Find wires
- Fix elevation
- Scale intensity
- Search tie lines
- Smoothen points, Thin points
- Vectorize buildings

The distance for loading neighbour points measured from the block boundary outwards depends on the processing step. If only a local environment around a point is required, a small area of neighbour points may be enough. For ground, keypoints, or building classifications, the maximum edge length of buildings in the project area can be used as estimation for the neighbour points area. For other processing steps, the parameters of the step may also provide some good estimation, such as lengths, distances, radius values, etc..

If the distance for loading neighbour points is too small, the result of a macro step may not be satisfying because of lacking information from the surrounding of a block. If the distance is too big, there is a risk that too many points need to be loaded for processing a block. This may lead to memory issues and long processing duration, in worst case to the failure of the macro (step) execution.

Points from neighbour blocks do not belong to the active block and are not saved with the active block points.

Run a macro on loaded points

You can execute the active macro on the points that are loaded in TerraScan. To run the complete macro, select **On loaded points** command from the **Run** pulldown menu in the **Macro** dialog.

You can also execute a macro by entering a key-in command (MicroStation) or Spaccel (Spatix), such as:

scan run macro modelkey.mac

which would read and execute the file MODELKEY.MAC. If no directory is specified in the key-in command, the application searches the macro file in the directory specified in the TSCAN_MACRODIR environment variable in the Configuration Variables for MicroStation definition.

A macro can be executed step-by-step on loaded points using the **Step** button in the **Macro** dialog. This is useful to test the result of the separate processing steps when creating a macro.

After a macro has been executed on loaded points, the **Macro execution** window appears. It shows a report that lists all macro steps and a return value. The return value can be the amount of points that has been effected by the macro step or another value specifying the result of the macro step. If a

processing step failed, the line in the report appears in red color. A definition of the return values for many processing functions is given in Section Function Prototypes.

The macro report can be saved as text file using the **Save as** command from the **File** pulldown menu or printed using the **Print** command from the **File** pulldown menu in the **Macro execution** dialog.

Run a macro on selected files

Not UAV

A macro can be executed on selected files. You select a number of input files and optionally, additional transformation and/or file storage settings. When the macro is executed, the application loads one input file at a time, executes the macro steps on the points extracted from the input file and, if applicable, writes the results into an output file.

The input files can be in any format that TerraScan supports. Thus, the command provides a possibility to apply processing steps in batch mode to files that are not supported in TerraScan projects, such as text files.

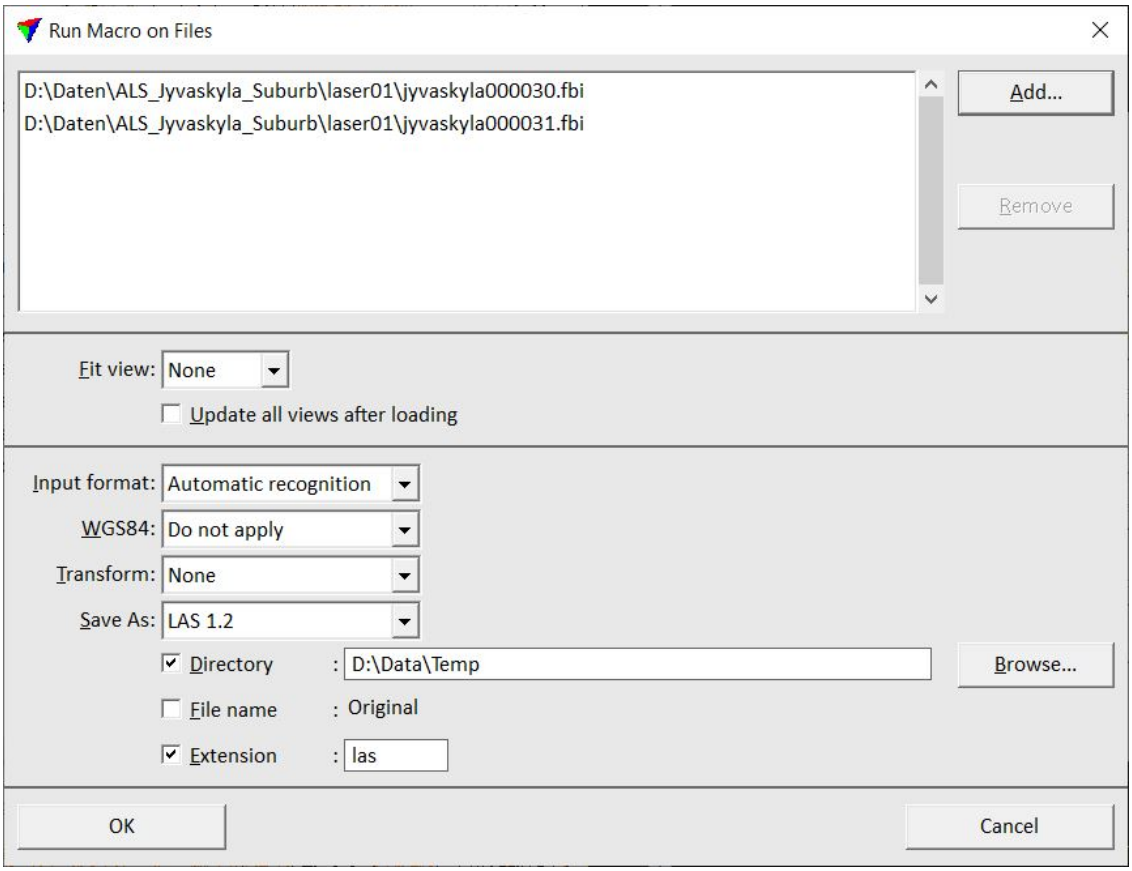
After a macro has been executed on loaded points, the **Macro execution** window appears. It shows a report that lists for each input file the amount of loaded points, the executed macro steps and a return value, as well as the amount of saved points, if applicable. The return value can be the amount of points that has been effected by the macro step or another value specifying the result of the macro step. If a processing step failed, the line in the report appears in red color. A definition of the return values for many processing functions is given in Section Function Prototypes.

The macro report can be saved as text file using the **Save as** command from the **File** pulldown menu or printed using the **Print** command from the **File** pulldown menu in the **Macro execution** dialog.

To run a macro on selected files:

1. Select **On selected files** command from the **Run** pulldown menu in the **Macro** dialog.

   This opens the **Run Macro on Files** dialog:



2. Click **Add** to add files to the list of files to process.

   This opens the **Select files to run macros on** dialog, a standard dialog for opening multiple input files.

3. Select files and click **Open**.

   You may continue with step 2 to add additional files.

4. Define additional settings in the **Run Macro on Files** dialog, if applicable.

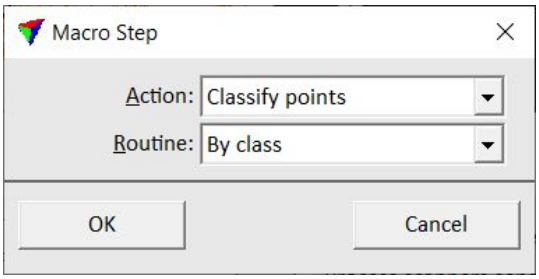
5. Click OK to start the process.

SETTING	EFFECT
Add	Add files for processing.
Remove	Remove selected files from the list of files to process.
Fit view	View to fit after loading an input file: **None**, **All**, or any CAD file view specified by its number.
Update all views after loading	If on, all views are updated after loading an input file.
Input format	Definition of the input file format. The list contains all binary and text formats that are implemented in TerraScan as well as formats

SETTING	EFFECT
	defined in File formats / User point formats category of TerraScan **Settings**. Implemented file formats are usually recognized automatically.
WGS84	Transformation from WGS84 coordinate values to another projection system applied to the output files. The list contains projection systems that are active in Coordinate transformations / Built-in projection systems, Coordinate transformations / US State Planes, and Coordinate transformations / User projection systems categories of TerraScan **Settings**.
Transform	Transformation applied to the output files. The list contains transformations defined in Coordinate transformations / Transformations category of TerraScan **Settings**.
Save As	Format of the output file(s). The list contains most of the binary and text formats that are implemented in TerraScan as well as formats defined in File formats / User point formats category of TerraScan **Settings**. Select **Do not save all points** option if there is a more selective output step included in the macro.
Directory	If on, you can specify the directory into which output files are written.
File name	If on, you can specify a name for the output file(s). You may include the **#name** Variable as part of the file name. This uses the name of the input file as name (or part of the name) for the corresponding output file. See File Name Variables for more information.
Extension	If on, you can specify the extension of output file(s).

Macro actions

When you click on the **Add** button in the **Macro** dialog, the **Macro Step** dialog opens:



The **Action** field includes a list of all available macro actions. Depending to the selected action, another dialog may open when OK is clicked or the dialog may be expanded when the action is selected. The settings for the selected macro action are then defined in the next or expanded dialog.

An overview of macro actions is given in the table below. There is a corresponding tool or menu command for many of the macro actions. The following Sections explain only the settings and parameters that are specific for the macro actions. Otherwise, there is a link to the description of the corresponding tool or command.

TO	USE ACTION
Apply a classification routine for single points	Classify points
Apply a classification routine for groups	Classify groups
Assign groups to laser points	Modify groups > Assign groups
Fix groups along block boundaries	Modify groups > Fix border groups
Clear group numbers of selected classes	Modify groups > Clear by class
Copy the group attribute from the closest neighbor point	Modify groups > Copy from closest
Split groups by class	Modify groups > Split groups by class
Draw polygons around point groups	Vectorize > Draw polygons
Detect paint markings from point cloud data	Vectorize > Find paint lines
Detect poles from point cloud data	Vectorize > Find poles
Detect overhead wires from point cloud data	Vectorize > Find wires
Create 3D models of buildings	Vectorize > Vectorize buildings
Call an Addon function	Addon command
Adjust misalignment angles	Adjust angles
Apply a TerraMatch corrections file to laser data	Apply correction
Assign a color to laser points	Assign color
Create a surface model in TerraModeler	Create model
Add a comment line to a macro	Comment
Compute normal vectors for laser points	Compute normal vectors
Compute parameters of road cross sections	Compute section parameters
Compute slopes on road surfaces	Compute slope arrows
Convert time stamps of laser data	Convert time stamps
Cut off points from long range measurements	Cut long
Cut off points from overlapping lines	Cut overlap

TO	USE ACTION
Assign line number from trajectories to laser points	Deduce line numbers
Delete points of certain class(es)	Delete by class
Delete points of certain line(s)	Delete by line
Execute a Windows command	Execute command
Export a lattice model	Export lattice
Extract additional points from waveform information	Extract echoes
Fix the elevation of points	Fix elevation
Execute a key-in command	Keyin command
Adjust color values of points	Modify color
Modify line numbers of points	Modify line numbering
Save points into new files	Output points
Save points into new files separated by line	Output by line
Save points into the original file	Save points
Manipulate intensity values	Scale intensity
Run a tie line search in TerraMatch	Search tie lines
Smooth the elevation of points	Smoothen points
Sort points by time stamps or coordinate values	Sort points
Thin points to a lower density	Thin points
Apply a transformation to the points	Transform points
Update the display of a CAD file view	Update views
Create a section points data set	Write section points
Write points into CAD file	Write to design

Each macro action returns a value that indicates whether a process was successful or failed. The return value can be the amount of points that has been effected by the macro action or another value specifying the result of the macro action. The return value is displayed in the **Macro execution** window if a macro was executed in TerraScan or it is written in the report text file if the macro was executed in TerraSlave. If a processing step failed, the line in the **Macro execution** window appears in red color and the return value is usually -1.

Classify points

Classify points action requires the selection of a classification routine in the **Routine** list of the **Macro step** dialog. The routines available in the list are basically the same as in the Routine sub-menu of the **TerraScan** window. However, a few classification routines can be executed in macros only.

The classification routines are described in detail in Section **Points** of Chapter Classification Routines.

Return value: Number of points that were effected by the process or a value < 0 if the process failed. The return values for failed processes can be found in Section Function Prototypes.

Classify groups

Classify groups action requires the selection of a classification routine in the **Routine** list of the **Macro step** dialog. The routines available in the list are basically the same as in the Group / Classify sub-menu of the **TerraScan** window.

The classification routines are described in detail in Section **Groups** of Chapter Classification Routines.

Return value: Number of points and groups that were effected by the process or a value < 0 if the process failed. The return values for failed processes can be found in Section Function Prototypes.

Assign groups

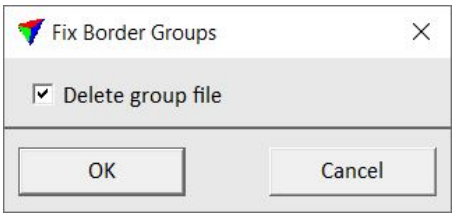
Assign groups action performs the same process as the Assign groups command.

If the macro step is performed on project level, the step creates a text file for each block. The text file stores the group number and center coordinates of each group that is cut by the block boundary. The enables the harmonization of group numbers along block boundaries with the Fix border groups macro step.

Return value: Number of points that were effected by the process or -1 if the process failed.

Fix border groups

Fix border groups action harmonizes group numbers along block boundaries. It requires that group numbers have been assigned to the points for all blocks of a project. The group assignment must be done with the Assign groups macro step on project level because **Fix border groups** relies on the text files that are created by the macro step.



SETTING	EFFECT
Delete group file	If on, the text files created by the Assign groups macro action are deleted after groups along block boundaries have been fixed.

Return value: Number of groups that were effected by the process or -1 if the process failed.

Clear by class

Clear by class action performs the same process as the Clear by class command.

Return value: Number of points that were effected by the process or -1 if the process failed.

Copy from closest

Copy from closest action performs the same process as the Copy from closest command.

Return value: Number of points that were effected by the process or -1 if the process failed.

Split by class

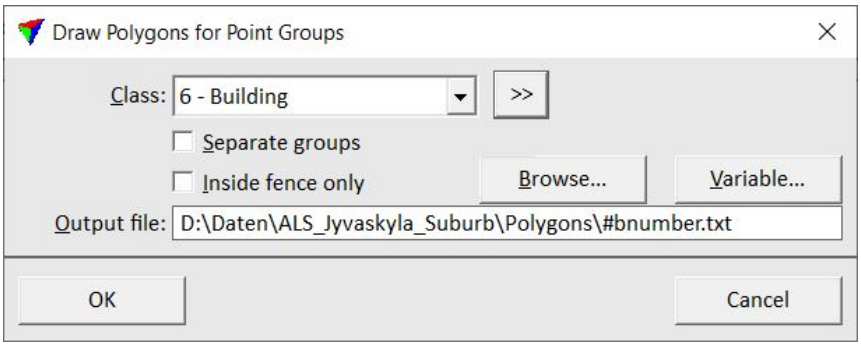
Split by class action performs the same process as the Split by class command.

Return value: Number of groups that were effected by the process or -1 if the process failed.

Draw polygons

Draw polygons action performs about the same process as the Draw Polygons command. However, instead of drawing polygons immediately into the CAD file, the process writes the coordinates of the polygon vertices into text files.

The text files can be read into the CAD file by using the Read / Polygons command in order to draw the polygons. The storage location and name of the output file(s) are defined in the action settings. If the macro runs on a project with several blocks, the file name should include a variable, such as the block name or number.



Return value: Number of polygons written in the text file, -1 if the process failed, or -2 if the output files could not be written.

Find paint lines

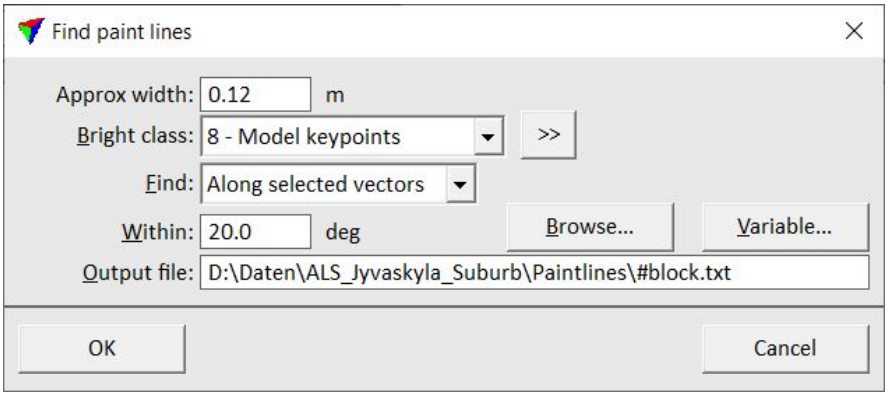
Find paint lines action finds linear chains of a given width from points. The linear chains may represent paint lines in the center or along the edges of roads.

A macro should first classify points with high intensity values using By intensity routine from the road surface into a separate class. If color values are stored for the points, By color routine could be used as an alternative to classify bright points. The result of the classification by intensity or color values should include points on paint markings but it will also include some noise points in the surrounding. Therefore, polygons for asphalt areas may be useful to exclude points outside the area of interest. In addition, the noise in intensity values can be reduced by using the Smoothen points routine.

The extraction of linear chains can be supported by an alignment element. Then, only chains within a given angular tolerance from the alignment are extracted.

The action writes the extracted paint line information into text files. The information includes the center point coordinates of a paint line, reliability value, normal direction in XY, and extension of the line.

The text files can be read into the CAD file by using the Read / Paint lines command in order to visualize the lines. The storage location and name of the output file(s) are defined in the action settings. If the macro runs on a project with several blocks, the file name should include a variable, such as the block name.



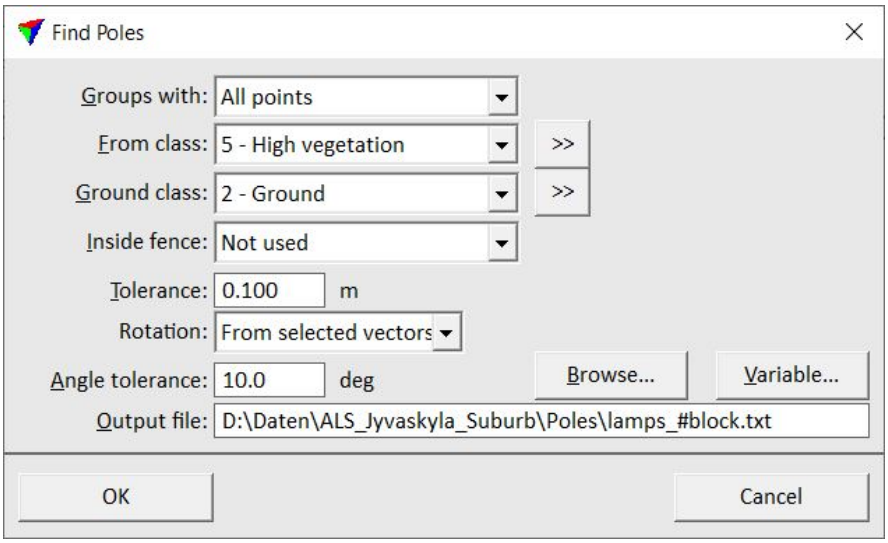
SETTING	EFFECT
Approx width	Approximate width of the linear chains in the point cloud. Refers to the width of the paint markings to extract.
Bright class	Point class(es) that are included in the linear chain extraction. The class(es) should include mostly points on paint markings.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Bright class** field.
Find	Specifies how the software extracts linear chains: • **All lines** - all linear chains. • **Along selected vectors** - only linear chains **Within** the given angular tolerance.
Output file	Storage location and name of the output text file. If the macro runs on a project with several blocks, the file name should include a variable, such as the block name.
Browse	Can be used to open a standard dialog for selecting a storage location for a file.
Variable	Can be used to define a variable as part of the output file name. See Variables for a list of available variables.

Return value: Number of line elements written in the text file.

Find poles

Find poles action performs about the same process as the Find Poles tool. However, instead of drawing cell elements for the poles immediately into the CAD file, the process writes the cell name and location into text files.

The text files can be read into the CAD file by using the Read / Poles command in order to visualize the objects. The storage location and name of the output file(s) are defined in the action settings. If the macro runs on a project with several blocks, the file name should include a variable, such as the block name.

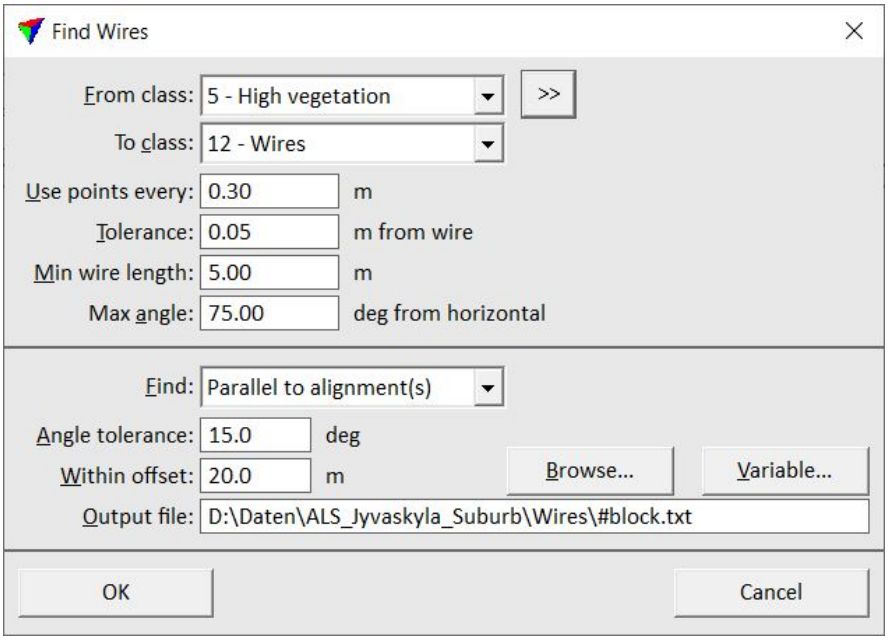


Return value: Number of groups that are effected by the process and classified into the target class.

Find wires

Find wires action performs about the same process as the Find Wires tool. However, instead of drawing lines for the wires immediately into the CAD file, the process writes the coordinates of vertices of each wire into text files.

The text files can be read into the CAD file by using the Read / Wires command in order to visualize the wires. The storage location and name of the output file(s) are defined in the action settings. If the macro runs on a project with several blocks, the file name should include a variable, such as the block name.

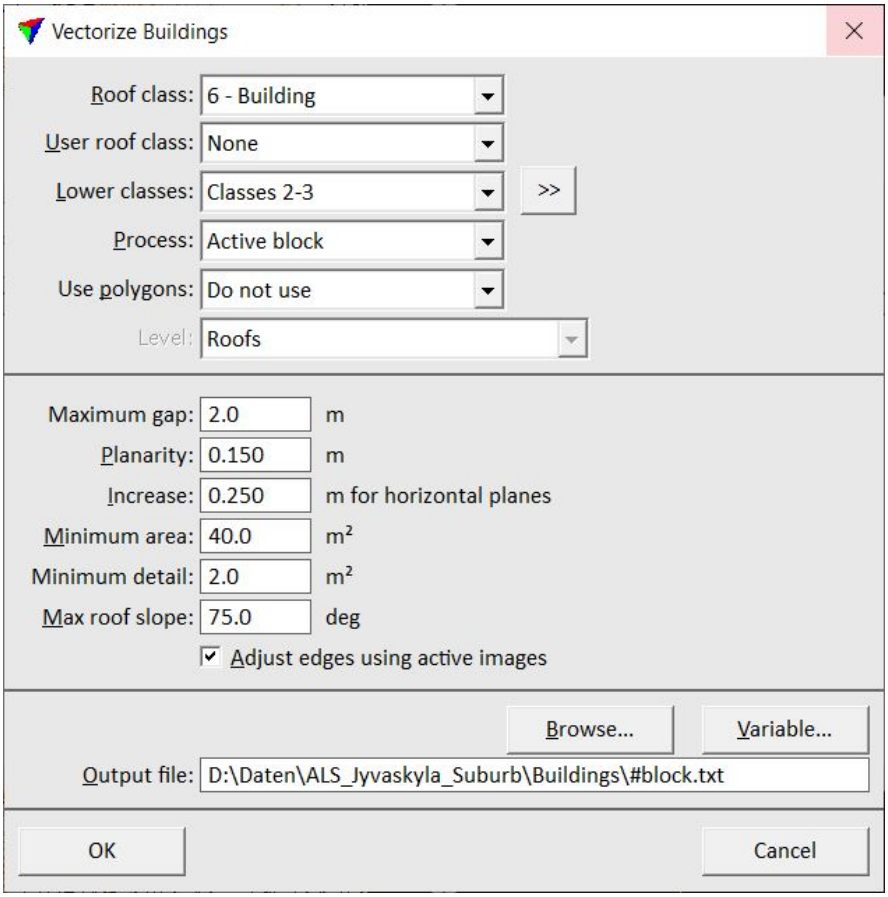


Return value: Number of points that are effected by the process and classified into the target class.

Vectorize buildings

Vectorize buildings action performs about the same process as the Vectorize Buildings tool. The main difference is that the building models are not drawn into the CAD file directly. Instead, text files are created that stores drawing parameters for each building model and the corner coordinates for all building model shapes.

The text files can be read into the CAD file by using the Read / Building models command in order to visualize the building models. The storage location and name of the output file(s) are defined in the action settings. If the macro runs on a project with several blocks, the file name should include a variable, such as the block name.

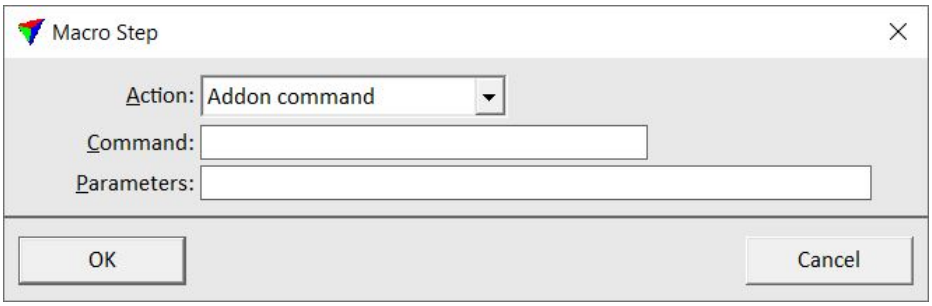


Return value: Number of building models written in the text file.

Addon command

Addon command action can be used to call a command of an addon function. Addons are the interface between TerraScan and user-developed DLL functions. Chapter DLL Interface provides more information about addons.

The **Macro step** dialog contains two input lines, one for the command syntax and another one for parameter values.



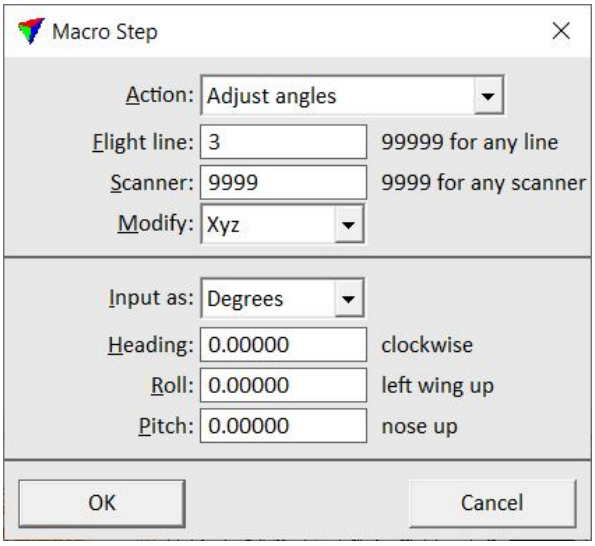
Adjust angles

Adjust angles action performs the same process as the Adjust laser angles command.

The only difference in the input dialog is that **Flight line** and **Scanner** are text fields in the **Macro step** dialog instead of selection lists in the command's dialog. This means that you need to type the number of a line and scanner for which you want to adjust angles. The following values are accepted:

- **Flight line** - a number between 0 and 65535 or 99999 for any line.
- **Scanner** - a number between 0 and 255 or 9999 for any scanner.

Trajectories must be loaded in TerraScan before starting a macro with the Adjust angles action.

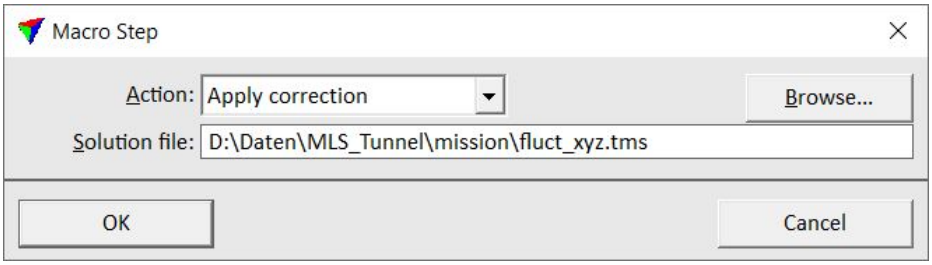


Return value: Number of points that were effected by the process or -1 if the process failed due to missing trajectory information.

Apply correction

Apply correction action applies corrections to points that were computed and stored by TerraMatch. It requires a TerraMatch **Solution file** as parameter. The action performs the same process as the Apply corrections tool of TerraMatch. The **TerraMatch User Guide** provides extensive information about TerraMatch functionality and data processing.

Apply correction action can not run in TerraSlave.



Apply correction is a processing step that can not be easily undone. Therefore, it is recommended to create a copy of the original block binary files if this action is included in a macro.

Assign color

Assign color action performs the same process as the Assign color to points command.

Return value: Number of points that were effected by the process or -1 if the process failed.

Create model

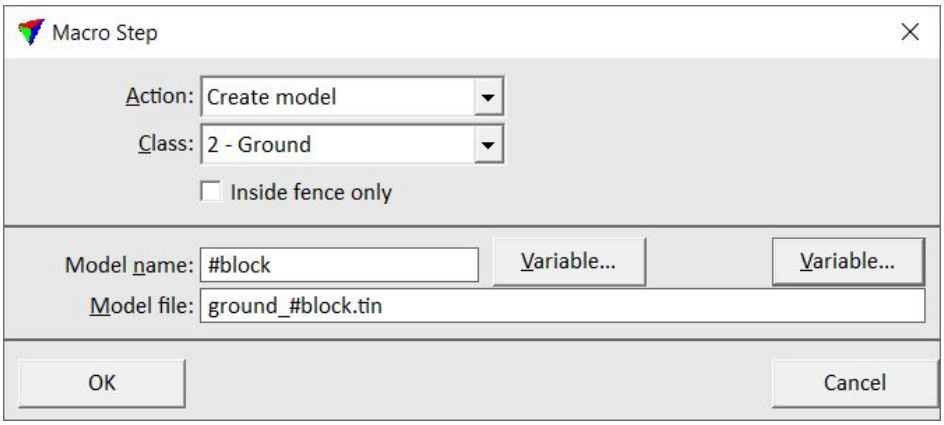
Create model action creates a TerraModeler surface model. Basically, it performs the same process as the Create surface model command. The action requires two additional parameters:

- **Model name** - name of the surface model in TerraModeler.
- **Model file** - name and extension (normally .TIN) of the surface model file that is stored on a hard disk. The file is stored in the same directory as the CAD file.

If the macro runs on a project with several blocks, the model name and model file name should include a variable, such as the block name. This allows the clear identification of a model loaded in TerraModeler and the storage of several model files in the same directory.

Before the action is executed, TerraModeler is launched automatically if it is not yet running. After execution, the surface model is active in TerraModeler and stored in the file on the hard disk. The **TerraModeler User Guide** provides extensive information about TerraModeler functionality and surface models.

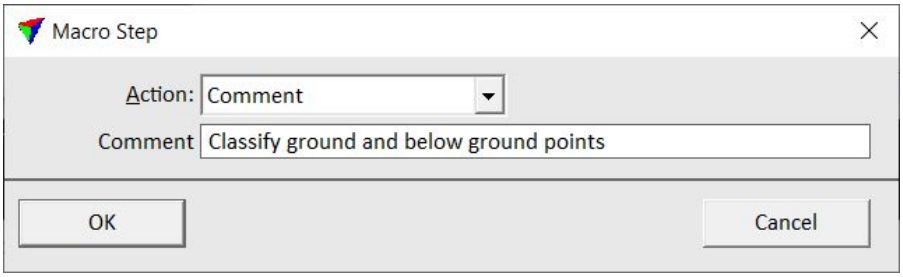
Create model action can not run in TerraSlave.



Return value: Internal number of the surface model (starting from 1 for the first model) or -1 if the process failed.

Comment

Comment action adds a comment line to the macro. Comment lines are introduced by a hash character (#) that is added automatically before the text of the comment. The length of the **Comment** text field is limited to 258 characters including spaces.



Compute distance

Compute distance action performs the same process as the Compute distance command. Elements drawn in the CAD file can not be used as reference for distance computation in the macro action.

Return value: Number of points that were effected by the process or -1 if the process failed.

Compute normal vectors

Compute normal vectors action performs the same process as the Compute normal vectors command.

Return value: Number of points that were effected by the process or -1 if the process failed.

Compute road parameters

Compute road parameters action extracts information from cross sections at regular spacing along a road. The extraction requires a high density of points on the road surface. Therefore, the action is only suited for MLS data sets. It requires the classification of points on the road surface using the Hard surface routine. Additionally, points close to the surface points should be classified using the By distance routine, for example points within an elevation range of ±1 cm.

The section parameter extraction relies on an alignment element, such as a line string representing the centerline of a road. An **Offset** parameter in the action settings defines on which side of the alignment (left or right) the cross sections are extracted. The left or right side is defined relatively to the digitization direction of the alignment element. To get cross section parameters for both sides, you need to add the **Compute section parameters** action two times to a macro.

The following list contains the cross section parameters that can be extracted:

- **Edge to edge slope** - slope from the middle of the two leftmost points to the middle of the two rightmost points.
- **Cross section roughness** - average distance of points to a line connecting the left- and rightmost points.
- **Maximum deviation** - maximum distance from a point to a line connecting the left- and rightmost points. The value is positive if the point is above the line and negative if the point is below the line.
- **Maximum rut depth** - maximum elevation distance from a point to a line connecting two other points in the section. The elevation distance is always measured downward from the line to the point.
- **Left rut depth** - the left rut is searched in an interval from the left edge to the first point on the right side of the cross section center. The left rut depth is the **Maximum rut depth** within this interval (see illustration below).
- **Right rut depth** - the right rut is searched in an interval from the right edge to the first point on the left side of the cross section center. The right rut depth is the **Maximum rut depth** within this interval (see illustration below).

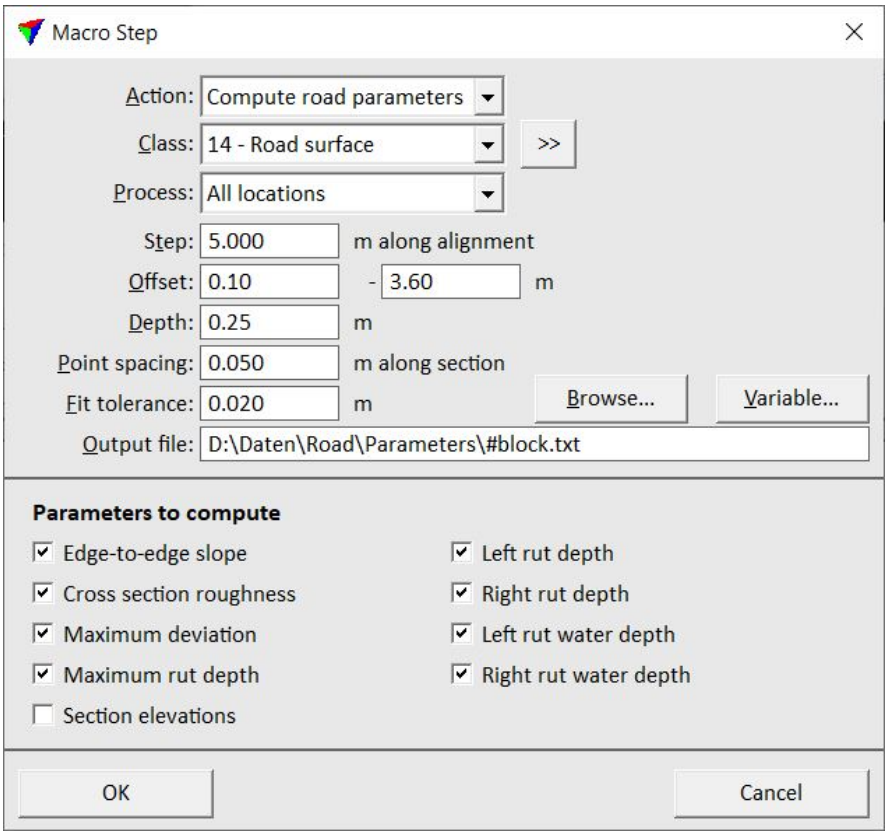
- **Left rut water depth** - searched in the same interval as the **Left rut depth**. The rut water depth is the maximum elevation difference from a point to a water surface in the left rut (see illustration below).
- **Right rut water depth** - searched in the same interval as the **Right rut depth**. The rut water depth is the maximum elevation difference from a point to a water surface in the right rut (see illustration below).

- **Section elevations** - elevation distances between points to the fitted section line. The line is fitted to all points of a cross section within a given tolerance. The elevation distance is measured at given regular steps along the cross section, for example every 5 cm.

The process writes the section parameters into text files, one for each block binary file of a TerraScan project. Depending on the selection of parameters for extraction, the text files contain more or less columns with the section parameter values.

The text files can be read into the CAD file by using the Read / Section parameters command in order to visualize the cross section parameters.

You must select the alignment element before starting a macro with the Compute road parameters action.



SETTING	EFFECT
Class	Source class(es) that are included in the section parameter extraction.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can

SETTING	EFFECT
	select multiple source classes from the list that are then used in the **From class** field.
Process	Area to process and include in the output file: • **All locations** - area covered by the active block binary file and possibly neighbouring areas, if neighbour points are loaded for the macro execution. • **Inside active block** - area covered by the active block binary file.
Step	Distance between consecutive cross sections. Measured along the alignment element.
Offset	Defines the area for extracting the cross section parameters relative to the alignment element. If the offset values are positive, the cross section are extracted on the right side of the alignment. If the offset values are negative, the cross section are extracted on the left side of the alignment.
Depth	Depth of a cross section. The points within the cross section depth are included in the parameter extraction.
Point spacing	Distance between points along the cross section that define the locations where the cross section is analyzed for the parameter computation.
Fit tolerance	Defines how well a fitted cross section line follows the points.
Output file	Storage location and name of the output text file. If the macro runs on a project with several blocks, the file name should include a **Variable**, such as the block name.
Variable	Opens the **Macro variable** dialog which contains the list of variables. Select the variable that you want to use as part of the output file name.
Parameters to write	Switch on all parameters that you want to include in the output file.

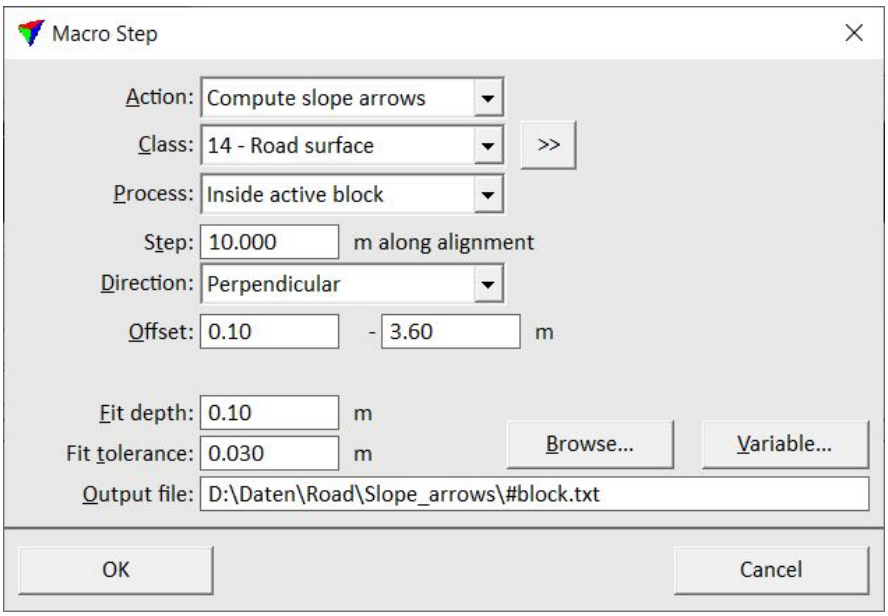
Return value: 0 if the process was successful, -1 if the process failed, or -2 if the output files could not be written.

Compute slope arrows

Compute slope arrows action performs about the same process as the Draw Slope Arrows tool. However, instead of drawing slope arrows immediately into the CAD file, the process writes the slope arrow parameters into text files.

The text files can be read into the CAD file by using the Read / Slope arrows command in order to visualize the slope arrows. The storage location and name of the output file(s) are defined in the action settings. If the macro runs on a project with several blocks, the file name should include a variable, such as the block name.

You must select the alignment element before starting a macro with the Compute slope arrows action.



Return value: Number of arrows written in the text file, -1 if the process failed, or -2 if the output files could not be written.

Convert time stamps

Convert time stamps action converts the format of time stamps. Supported conversions are:

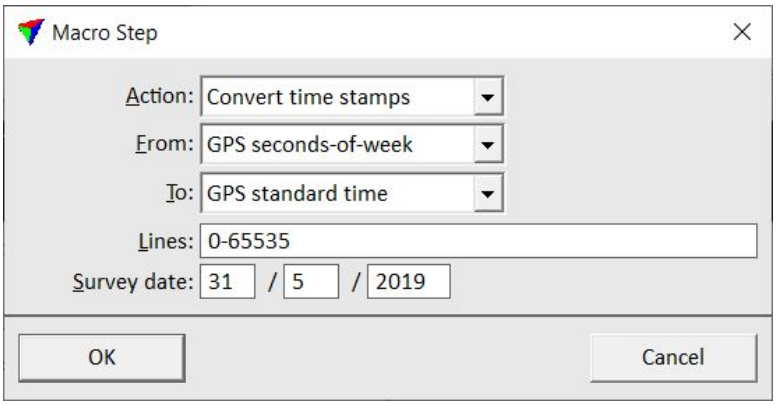
INPUT FORMAT	CONVERTED FORMAT
GPS seconds-of-week	GPS standard time
	GPS time
GPS standard time	GPS seconds-of-week
	GPS time
GPS time	GPS seconds-of-week

INPUT FORMAT	CONVERTED FORMAT
	GPS standard time
Unix time	GPS seconds-of-week
	GPS standard time
	GPS time

The conversion is necessary, for example, if data collected in several weeks is processed together in one project. Then, GPS seconds-of-week time stamps result in repeated values and GPS standard time must be used in order to provide unique time stamps for each laser point. This is a requirement for many processes that rely on the link between laser points and trajectory information. Therefore, it is essential that time stamps of trajectories and laser data are stored in the same GPS time format.

The conversion of time stamps can also be done when laser points are imported into a project. See Import points into project command and Section Attributes to store of New project command for more information.

Further, time stamps can be manipulated by using the Fix time macro action.



SETTING	EFFECT
From	Original time stamp format of the laser points.
To	Target time stamp format.
Lines	Only points in the given line(s) are effected. Use a comma or minus to separate several line numbers, for example 2-5,10. 0-65535 stands for all lines.
Survey date	Date when the laser data was captured. The format is day/month/year (dd/mm/yyyy). This is only active for the conversion from **GPS seconds-of-week** to **GPS standard time** or **GPS time**.

Return value: Number of points that were effected by the process or -1 if the process failed.

Cut long

Cut long action performs the same process as the Cut long range command.

Return value: Number of points that were effected by the process, -2 if the process failed due to invalid parameters.

Cut overlap

Cut overlap action performs the same process as the Cut overlap command.

Return value: Number of points that were effected by the process, -2 if the process failed due to invalid parameters, or -4 if the process failed due to missing trajectory information.

Deduce line numbers

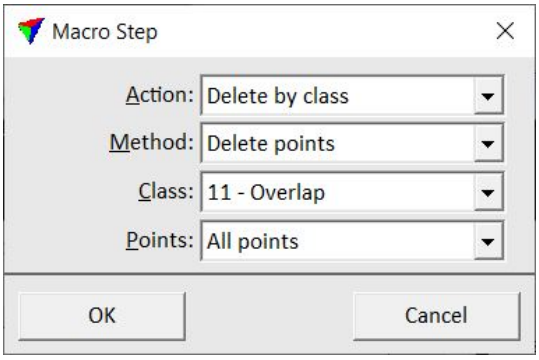
Deduce line numbers action performs the same process as the Deduce using time command.

Return value: Number of points that were effected by the process. An error message appears if the process fails, for example due to missing trajectory information.

Delete by class

Delete by class action performs about the same process as options of the Delete command. The action deletes points from the block binary files. You can select a certain class and specify the area from which to delete points.

For **LAS 1.4** files, you can specify that the withheld bit is set instead of removing points from the file.



SETTING	EFFECT
Method	Defines what method is applied to points:

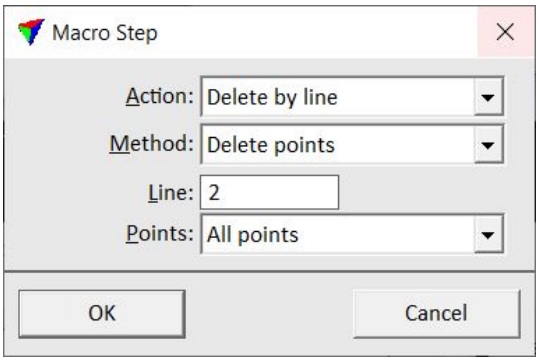
SETTING	EFFECT
	• **Delete points** - deletes the point. This is the normal method to remove points from a file. • **Set withheld bit** - sets the withheld bit for points in LAS 1.4 files. The point itself is kept in the file.
Class	Point class(es) to delete. Use **Any class** to delete all points.
Points	Specifies the area for which to delete points: • **All points** - all points are deleted. • **Inside fence only** - all points inside a MicroStation fence or selected polygons are deleted. • **Outside fence only** - all points outside a MicroStation fence or selected polygons are deleted. • **Outside block only** - all points outside the active block area are deleted.

Return value: Number of points that were effected by the process or -1 if the process failed due to missing fence information.

Delete by line

Delete by line action performs about the same process as the **By line** option of the Delete command. The action deletes points from the block binary files. You can define a certain line number and specify the area from which to delete points.

For **LAS 1.4** files, you can specify that the withheld bit is set instead of deleting points.



SETTING	EFFECT
Method	Defines what method is applied to points: • **Delete points** - deletes the point. This is the normal method to remove points from a file.

SETTING	EFFECT
	• **Set withheld bit** - sets the withheld bit for points in LAS 1.4 files. The point itself is kept in the file.
Line	Line number of points to be deleted. A number between 0 and 65535.
Points	Specifies the area for which to delete points: • **All points** - all points are deleted. • **Inside fence only** - all points inside a MicroStation fence or selected polygons are deleted. • **Outside fence only** - all points outside a MicroStation fence or selected polygons are deleted.

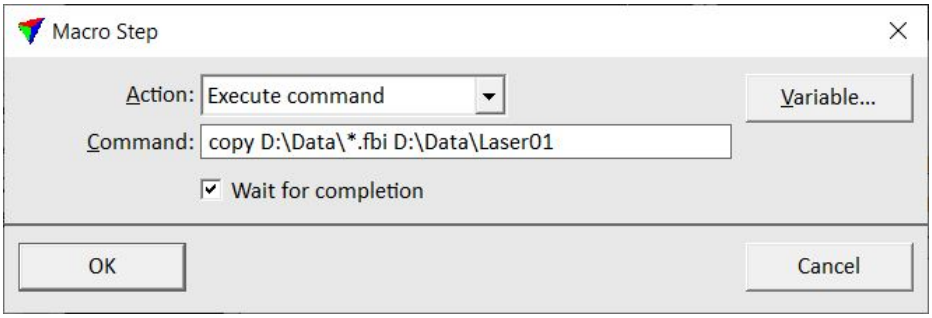
Return value: Number of points that were effected by the process or -1 if the process failed due to missing fence information.

Execute command

Execute command action calls a Windows batch command. It starts the **cmd.exe** application of Windows and executes the given command.

You can include variables in the batch command that refer to block binary files, such as the block file name or project directory.

If the **Wait for completion** option is switched on, the software waits until the Windows batch process is completed before starting any next macro step.

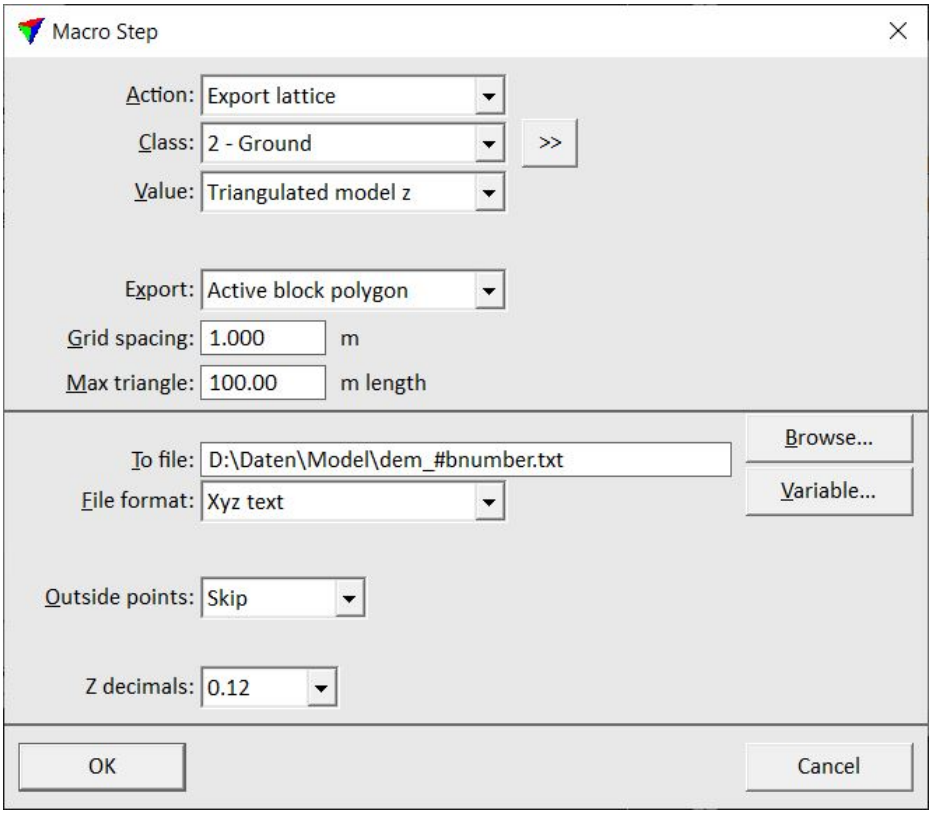


The **Command** given in the image above copies all files with the extension .FBI from the D:\DATA directory to the D:\DATA\LASER01 folder. Only when the copy process finished, the other steps in the macro are started.

Export lattice

Export lattice action performs about the same process as the Export lattice model command for loaded points and the Export lattice models command for projects. The major differences are:

- There are no selected shapes required that define the area for each lattice model. The area is defined by the boundaries of the block binary files.
- Instead of defining a model buffer area, set a good distance for loading neighbour points when the macro is executed.
- The storage location and name of the output file(s) are defined in the action settings. If the macro runs on a project with several blocks, the file name should include a variable, such as the block name. Text elements drawn in the CAD file can not be used in the macro action.



Return value: 1 if the process was successful, 0 if the process failed due to the lack of points in the source class, or -2 if the output files could not be written into the given directory.

Extract echoes

Extract echoes action performs the same process as the Extract Echoes command. The action requires that one or more polygons are selected before starting the macro.

Return value: Number of points that were effected by the process or -1 if the process failed.

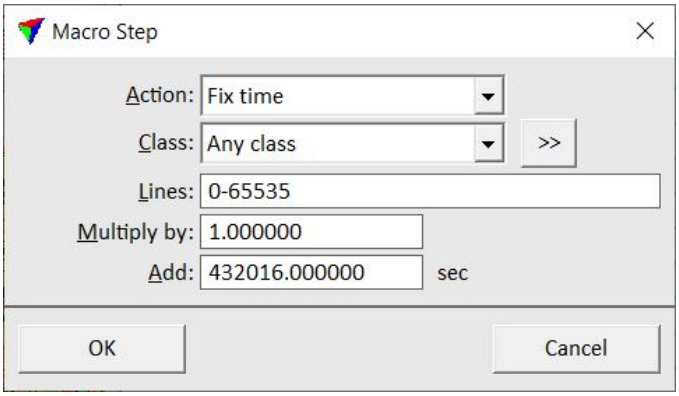
Fix elevation

Fix elevation action performs the same process as the Fix elevation tool.

Return value: Number of points that were effected by the process.

Fix time

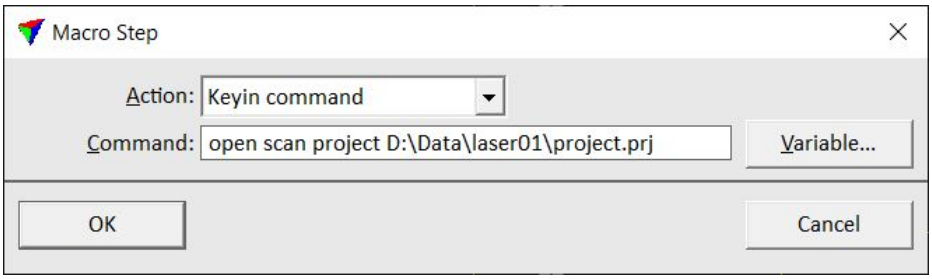
Fix time action manipulates the time stamps of laser points. You can multiply time stamps by a factor and/or add a value to the time stamps.



SETTING	EFFECT
Class	Point class(es) for which the time modification is computed.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Class** field.
Lines	Points of the given line(s) are effected by the time modification. Use a comma or minus to separate several line numbers, for example 2-5,10. 0-65535 refers to all lines.
Multiply by	Factor by which time stamps are multiplied. Example: convert millisecond time stamps to seconds.
Add	Value that is added to time stamps. Example: convert Unix seconds-of-day time stamps to GPS seconds-of-day time stamps.

Keyin command

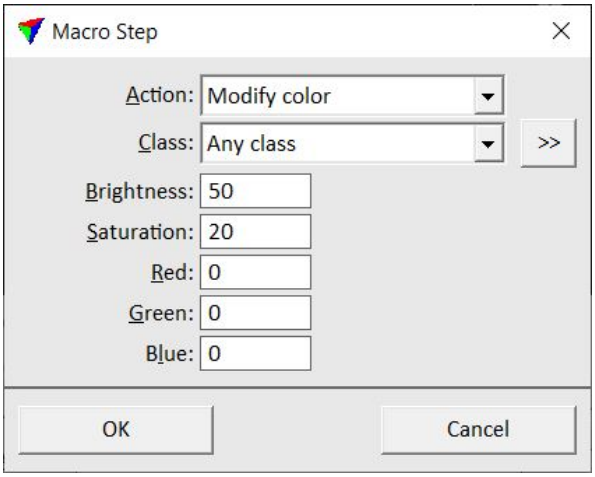
Keyin command action calls a key-in command of TerraScan. Key-in commands are described in Chapter Key-in commands.



The **Command** given in the image above opens the TerraScan project from file PROJECT.PRJ stored in folder D:\DATA\LASER01.

Modify color

Modify color action adjusts the color values of points. The components to adjust are brightness, saturation and gray balance. The action requires that color values are stored for the points. It is normally used after extracting color values from images, for example with Extract color from images command for projects or Extract color from images command for loaded points.



SETTING	EFFECT
Class	Point class(es) that are effected by the process.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Class** field.

SETTING	EFFECT
Brightness	Value to add to the brightness of color values. Accepts values between -100 and 500.
Saturation	Value to add to the saturation of color values. Accepts values between -100 and 200.
Red	Value to add to the current red value of a point. Accepts values between -65535 and 65535.
Green	Value to add to the current green value of a point. Accepts values between -65535 and 65535.
Blue	Value to add to the current blue value of a point. Accepts values between -65535 and 65535.

Return value: Number of points that were effected by the process.

Modify line numbering

Modify numbering action performs the same process as the Modify numbering command.

Return value: Number of points that were effected by the process.

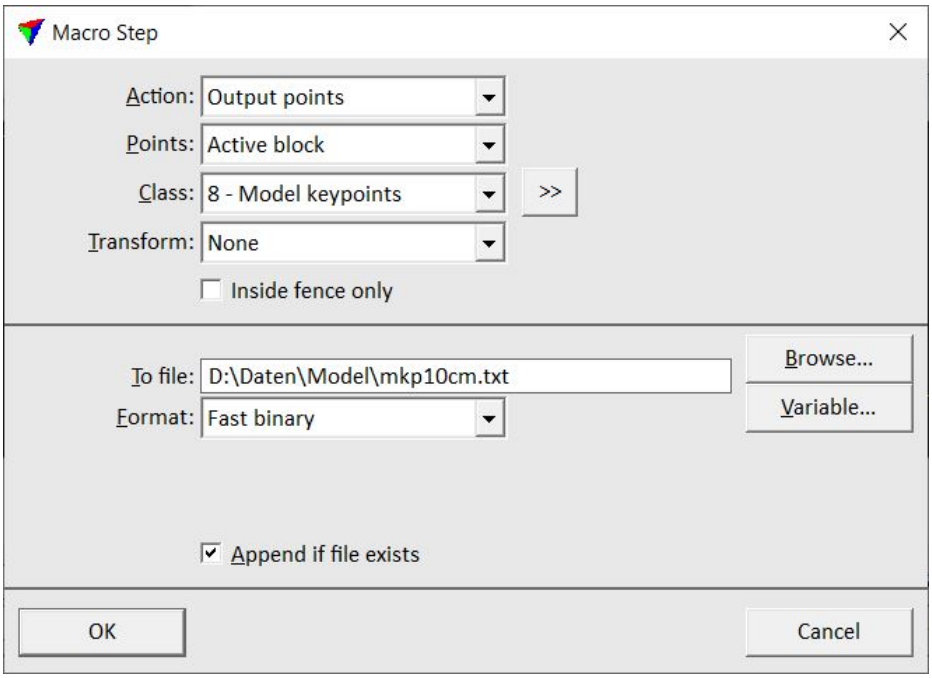
Output points

Output points action can be used to write points into a new file. A typical example is a macro that:

1. classifies model keypoints.
2. writes only model keypoints into one output file for all block binary files.
3. does not effect the original block binary files in any way.

The action can also be used to write points block-wise into new files that may have a different format than the original block binary files, for example for delivery purposes.

The output file format can be any of the Supported file formats implemented in TerraScan or a user-defined point file format.



SETTING	EFFECT
Points	Points for output: **All points** which would include neighbour points or **Active block**.
Class	Point class(es) that are included in the output file(s).
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Class** field.
Transform	Transformation applied to the points before an output file is written. The list contains all transformation defined in the Coordinate transformations / Transformations category of TerraScan **Settings**.
Inside fence only	If on, only points inside a fence or selected polygon(s) are included in the output file(s).
To file	Storage location and name of the output file. If it does not include a directory, the project directory is used. If there should be one output file for each block binary file, the file name must include a variable, such as the block name. Add the extension of the output file manually at the end of the file name according to the selected output **Format**.
Variable	Opens the **Macro variable** dialog which contains the list of variables. Select the

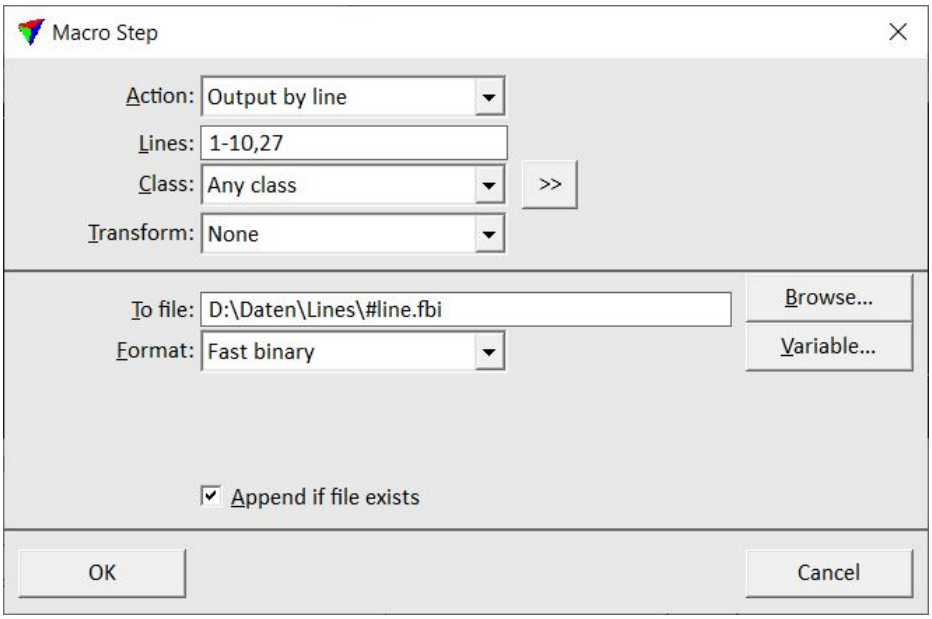
SETTING	EFFECT
	variable that you want to use as part of the output file name.
Format	Format of the output file(s).
Delimiter	Character that separates different fields in an implemented text file format: **Space**, **Tabulator**, or **Comma**. This is only active if Format is set to a text file format that does not explicitly define the delimiter itself.
Xyz decimals	Number of decimals stored for coordinate values. This is only active if **Format** is set to any text file format.
Append if file exists	If on, input files are written into one output file if the file name is the same. If the output file name does not contain a variable, all input files are written into a single output file. If off, input files are written into individual output file(s), if the file name contains a variable. If no variable is used that makes an output file name unique, the process might overwrite existing files of the same name.

Return value: Number of points that were written into the output file, -1 if the process failed due to a missing fence, -2 if the output file(s) could not be written into the given directory, -3 if the process failed due to invalid parameters, -4 if the process failed due to a missing transformation.

Output by line

Output by line action can be used to write points into output files per line number. You would typically use this to produce final output files for delivering each line in a separate file.

The output file format can be any of the Supported file formats implemented in TerraScan or a user-defined point file format.



SETTING	EFFECT
Lines	Points of the given line(s) are included in the output file(s). Use a comma or minus to separate several line numbers, for example 2-5,10. 1-65535 refers to all lines and creates one output file for each line.
Class	Point class(es) that are included in the output file(s).
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Class** field.
Transform	Transformation applied to the points before an output file is written. The list contains all transformation defined in the Coordinate transformations / Transformations category of TerraScan **Settings**.
To file	Storage location and name of the output file. If it does not include a directory, the project directory is used. If there should be individual output files for several lines, the file name must include the **#line** variable. Add the extension of the output file manually at the end of the file name according to the selected output **Format**.
Variable	Opens the **Macro variable** dialog which contains the list of variables. Select the

SETTING	EFFECT
	variable that you want to use as part of the output file name.
Format	Format of the output file(s).
Delimiter	Character that separates different fields in an implemented text file format: **Space**, **Tabulator**, or **Comma**. This is only active if **Format** is set to a text file format that does not explicitly define the delimiter itself.
Xyz decimals	Number of decimals stored for coordinate values. This is only active if **Format** is set to any text file format.
Append if file exists	If on, input files are written into one output file if the file name is the same. If the output file name does not contain a variable, all input files are written into a single output file. If off, input files are written into individual output file(s), if the file name contains a variable. If no variable is used that makes an output file name unique, the process might overwrite existing files of the same name.

Return value: Number of points that were written into the output file, -3 if the process failed due to invalid parameters, -4 if the process failed due to a missing transformation.

Save points

Save points action performs the same process as the Save points command. It saves the points into their original binary file. There is no restriction for executing the action as for the menu command.

Return value: Number of points that were effected by the process.

Scale intensity

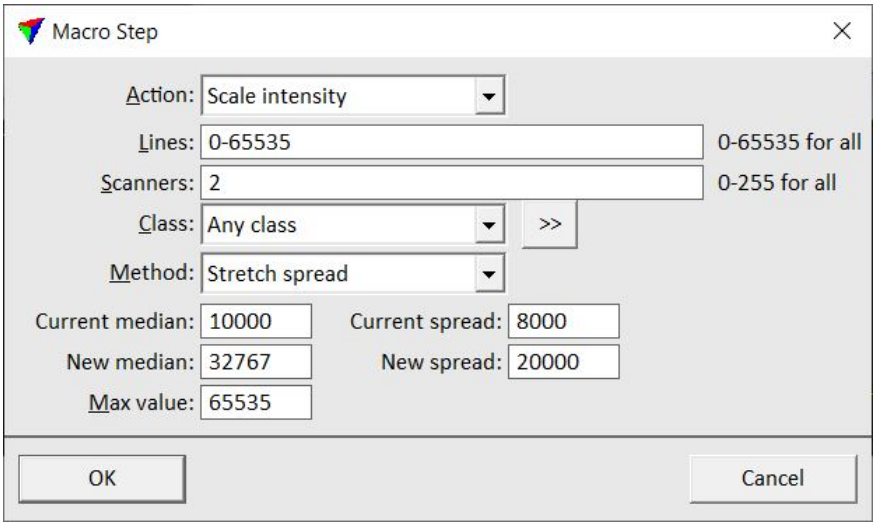
Scale intensity action manipulates the intensity values of points. It computes new intensity values according to a selected method:

- **Direct scale** - the new intensity value is the result of adding and/or multiplying the old value with given values.
- **Angle of incidence** - the new intensity value is computed based on the angle of incidence of a point and a given factor. The higher the factor, the stronger the effect of the intensity correction. The method requires that normal vectors are available, they have to be computed for each line separately, for example by the Compute normal vectors routine, before applying the method. The computation effects only points of planar dimension. The method is suited for MLS data sets for

which the intensity values vary systematically depending on the angle of incidence of the laser beam.

- **Stretch spread** - the new intensity value is computed based on the median and spread values of the old values.

The **Direct scale** and **Stretch spread** methods are useful for adjusting intensity values of different scanners, lines, or data collection days to each other. You can check the average, median, and spread values of loaded points by using the Addon / View histogram command. The values shown in the **Laser intensity histogram** window can be used as basis for the input values of the **Scale intensity** action.



SETTING	EFFECT
Lines	Only points in the given line(s) are effected. Use a comma or minus to separate several line numbers, for example 2-5,10. 0-65535 refers to all lines.
Scanners	Only points captured by the given scanner(s) are effected. Use a comma or minus to separate several scanner numbers, for example 1-3,5. 0-255 refers to all scanners.
Class	Point class(es) that are effected by the process.
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Class** field.
Method	Method of computing new intensity values: **Direct scale**, **Angle of incidence**, or **Stretch spread**.
New value	Values for computing the new intensity value of a point. This is only active if **Method** is set

SETTING	EFFECT
	to **Direct scale**.
Factor	Determines how strong the effect of intensity value computation based on angle of incidence is. This is only active if **Method** is set to **Angle of incidence**.
Current median	Current median of intensity values. This is only active if **Method** is set to **Stretch spread**.
Current spread	Current spread of intensity values. This is only active if **Method** is set to **Stretch spread**.
New median	Target median of intensity values. This is only active if **Method** is set to **Stretch spread**.
New spread	Target spread of intensity values. This is only active if **Method** is set to **Stretch spread**.
Max value	Maximum allowed intensity value.

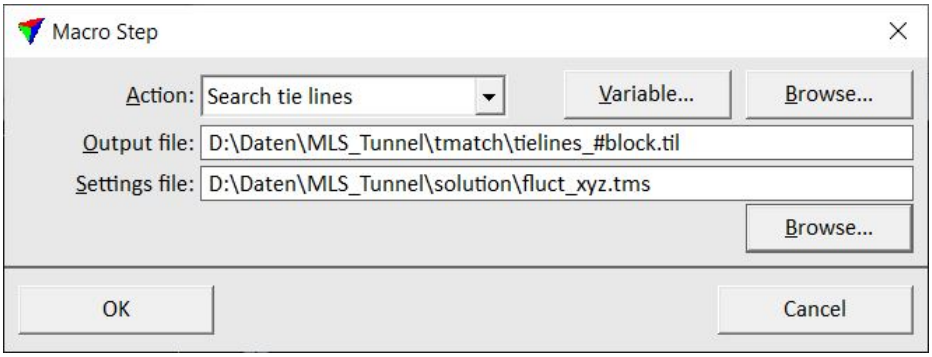
Return value: Number of points that were effected by the process.

Search tie lines

Search tie lines action performs a tie lines search in TerraMatch. It requires a TerraMatch tie line search **Settings file** as parameter. The settings file can be created in the Search tie lines dialog of TerraMatch. The action performs the same process as the Search tie lines command of TerraMatch. The **TerraMatch User Guide** provides extensive information about TerraMatch functionality and data processing.

Running the macro action requires that TerraMatch is available on the same computer. The correct trajectories must be active in TerraScan and laser data must be linked to the trajectories by using the Deduce using time command or Deduce line numbers macro action.

The macro action writes the tie lines into an output file. The storage location and name of the output file(s) are defined in the action settings. If the macro runs on a project with several blocks, the file name should include a variable, such as the block name.



SETTING	EFFECT
Output file	Storage location and name of the output tie line file. Use the **Browse** button above the text field in order to open the **Output file** dialog, a standard Windows dialog for defining a file location and name. Use the Variable button in order to select a variable for the tie line file name.
Settings file	Location and name of the tie line search settings file.

Smoothen points

Smoothen points action performs the same process as the Smoothen points command.

Return value: 1 if the process was successful, 0 if the process failed, or -2 if the process ran out of memory.

Sort points

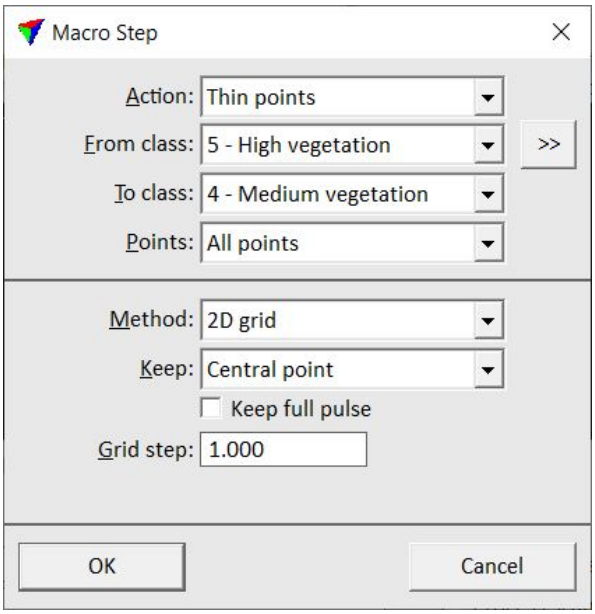
Sort points action performs the same process as the Sort command.

Return value: 1 if the process was successful or 0 if the process failed.

Thin points

Thin points action performs the same process as the Thin points command. The only difference is that you can define the area for thinning points in the macro action. The process either effects all points, points inside or points outside a fence or selected polygon(s).

If points inside or outside a fence or selected polygon(s) are thinned, the macro step can not be performed in TerraSlave.



Return value: Number of points that were effected by the process, -1 if the process failed due to a missing fence, or -3 if the process failed due to invalid parameters.

Transform points

Transform points action performs the same process as the Transform loaded points command.

If the transformation manipulates the Z values of the points in a way that they do not represent elevation values anymore but rather elevation differences, the macro must create a new copy of the data instead of overwriting the original files. This applies if you run the macro on a project or on selected files. These transformations include the types:

- **Height from ground** - elevation of each point is replaced by its height above ground. Ground is defined by points in one or more point classes.
- **Height from TIN** - elevation of each point is replaced by its height above a TIN model. The TIN model is defined by an active surface model in TerraModeler.
- **Add TIN** - elevation values of a TIN are added to the original elevation values of loaded points. The corresponds to an *Adjust to geoid* action which uses a TIN as geoid model. The TIN is defined by an active surface model in TerraModeler.
- **Scale dZ from TIN** - scaled elevation difference values between the loaded points and a TIN are added to the original elevation values of loaded points. The TIN is defined by an active surface model in TerraModeler.

Return value: Number of points that were effected by the process, -1 if the process failed due to a missing fence, -2 if the process was aborted, -3 if the process failed due to invalid parameters, -4 if the process failed due to a missing transformation.

Update views

Update views action redraws all CAD file views that display points.

Return value: 1.

Write section points

Write section points action produces new files that contain only points of cross sections at regular steps along an alignment element. The resulting files are suitable for visualizing data of a corridor object, such as a road or railroad.

The action relies on a linear element that is used as alignment element. The parameters of the action determine, how many points are written in the new files by settings like the width of the sections left and right of the alignment element, the step size along the alignment element, and the depth of the sections.

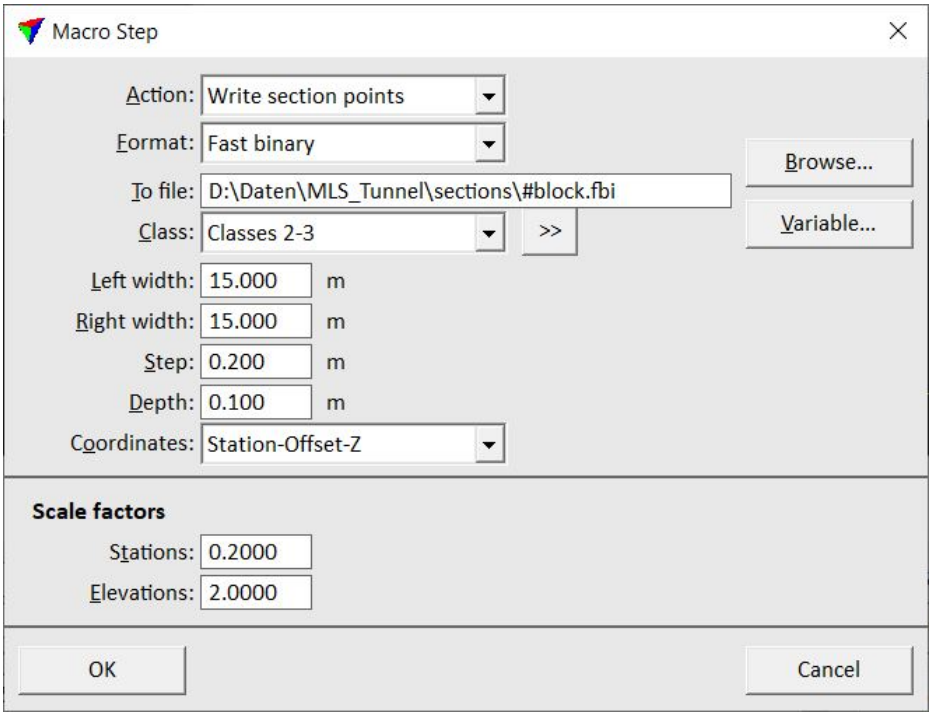
The output files can be written with either the original coordinates of the points or in an artificial coordinate system **Station-Offset-Z**. The artificial coordinate system is defined by:

- X = scaled station along the alignment element.
- Y = offset to the left and right sides of the alignment element.
- Z = scaled elevation values.

The artificial coordinate system should be used, for example, to produce a data set suited for digitization tasks along a corridor data set. The advantages are:

- faster viewing as the corridor is shortened.
- exaggerated elevation values make small elevation changes better visible.
- easier panning as the whole corridor runs along the view's X axis without curves.
- better shaded surface display as the direction of the corridor relative to the light source does not change.

You must select the alignment element before starting a macro with the Write section points action.



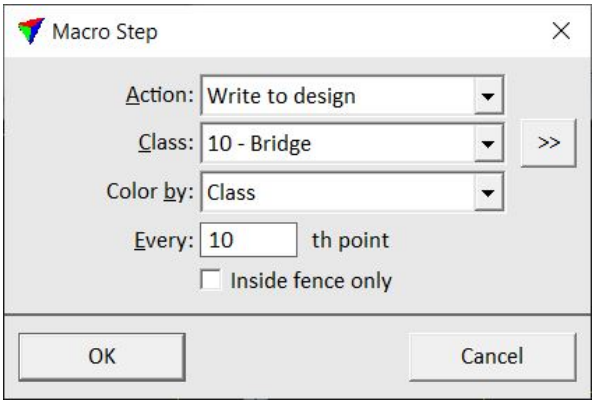
SETTING	EFFECT
Format	File format of the output files: **FastBinary**, **LAS**, **Scan binary 8 bit** or **16 bit**.
To file	Storage location and name of the output file(s). If there should be individual output files for several block binary files, the file name must include a **Variable**. Add the extension of the output file manually at the end of the file name according to the selected output **Format**.
Variable	Opens the **Macro variable** dialog which contains the list of variables. Select the variable that you want to use as part of the output file name.
Class	Point class(es) that are included in the output file(s).
>>	Opens the **Select classes** dialog which contains the list of active classes in TerraScan. You can select multiple source classes from the list that are then used in the **Class** field.
Left width	Width of the cross sections to the left side of the alignment element. The side is relative to the digitization direction of the alignment.

SETTING	EFFECT
Right width	Width of the cross sections to the right side of the alignment element. The side is relative to the digitization direction of the alignment.
Step	Distance between consecutive cross sections along the alignment element.
Depth	Depth of a cross section.
Coordinates	Defines the coordinate system of the output files: **Original** or **Station-Offset-Z**.
Stations	Scale factor for stations along the alignment element. Determines the compression of the original corridor length. This is only active if **Coordinates** is set to **Station-Offset-Z**.
Elevations	Scale factor for elevation values. Determines the elevation exaggeration. This is only active if **Coordinates** is set to **Station-Offset-Z**.

Return value: Number of points written into the output file or -1 if the process failed.

Write to design

Write to design action performs about the same process as the Write to design file command. The action includes only **Active symbology**, **Class**, **Line** and **Point color** as coloring options for drawing the points.



Return value: Number of points that were effected by the process.

Configuration Variables for MicroStation

MicroStation is able to locate TerraScan with the help of configuration variables. When you install TerraScan, the installation program will create a configuration file TERRA.CFG which defines the required environment variables. This file is placed in MicroStation's CONFIG\APPL sub-directory.

For example, C:\USTATION\CONFIG\APPL\TERRA.CFG may contain:

```
#-------------------------------------------------
#  TERRA.CFG - Configuration for Terra Applications
#-------------------------------------------------
TERRADIR=c:/terra/
TERRACFG=$(TERRADIR)config/
TERRADOCS=$(TERRADIR)docs/
MS_MDLAPPS < $(TERRADIR)ma/
%if exists ($(TERRACFG)*.cfg)
%   include $(TERRACFG)*.cfg
%endif
```

This configuration file will include all the configuration files in C:\TERRA\CONFIG directory. TerraScan's configuration file TSCAN.CFG contains:

```
#-----------------------------------------
#  TSCAN.CFG - TerraScan Configuration File
#-----------------------------------------
TSCAN_DATA=$(TERRADIR)data/
TSCAN_LICENSE=$(TERRADIR)license/
TSCAN_MACRODIR=$(TERRADIR)macro/
# Directory for user preferences (user has write access)
TSCAN_PREF=$(TERRADIR)tscan/
# Directory for settings (may point to read-only directory)
TSCAN_SET=$(TERRADIR)tscan/
# Files for settings (may be shared by organization)
TSCAN_ALIGNREP  = $(TSCAN_SET)alrepfmt.inf
TSCAN_OUTFMT    = $(TSCAN_SET)outfmt.inf
TSCAN_TRANSFORM = $(TSCAN_SET)trans.inf
TSCAN_TARGETS   = $(TSCAN_SET)targets.inf
TSCAN_CODES     = $(TSCAN_SET)codes.inf
```

In a default configuration, MicroStation automatically includes these settings as configuration variables. You can check the values for these variables in the **Configuration Variables** dialog of MicroStation. In case these variables have not been defined correctly, you should define them manually.

- MS_MDLAPPS should include the directory where TSCAN.MA is located.
- TSCAN_DATA defines a default directory for incoming laser points.
- TSCAN_LICENSE should point to the directory where user license TSCAN.LIC is located.
- TSCAN_MACRODIR defines a directory where macros are searched from.
- TSCAN_SET should point to a directory where settings can be stored. This directory may be shared by an organization and the user may lack write access to it.
- TSCAN_PREF should point to a directory where user preferences can be stored. The user must have write access to this directory.
- TSCAN_ALIGNREP defines the file in which alignment report formats are stored.
- TSCAN_OUTFMT defines the file in which output file formats are stored.
- TSCAN_TRANSFORM defines the file in which coordinate transformations are stored.
- TSCAN_TARGETS defines the file in which target object types are stored.
- TSCAN_CODES defines the file in which EarthData code translation table is stored.

DLL Interface

TerraScan installation includes an example dynamic link library together with source code illustrating how tools can be added into TerraScan.

These addon tools may perform such tasks as classifying points, writing points to output files or displaying statistical information. The names of the tools will appear in the Addon menu in the **TerraScan** window. The tools are responsible for providing modal or mode-less dialogs for the input of required user settings.

The example link library \TERRA\MA\TSCANADD.DLL has been built using Visual C++ 6.0 from source codes included in \TERRA64\ADDON directory. It provides one tool: the display of an intensity histogram.

You may modify the source code and build a dynamic link library to replace the one supplied during installation. The addon library must provide four predefined functions which TerraScan will call at different stages of application execution. These functions are:

- AddonFillCommands() is called at start-up so that the DLL will inform TerraScan about available tools.
- AddonLinkVariables() is called after start-up to provide the DLL with information about some of TerraScan's internal tables and variables.
- AddonRunCommand() is called whenever user starts an addon command.
- AddonEnd() function is called before the application is unloaded so that the DLL may release any resources it has allocated.

See \TERRA64\ADDON directory for example source code files.

The example source code uses C syntax and accesses WIN64 API to create the user interface items.

File Formats

TerraScan stores point clouds in its own proprietary point file formats. It also supports the standard format for point cloud data, LAS, and a number of text formats. See Supported file formats for an overview. In addition, user-tailored text formats for point clouds can be defined in the TerraScan **Settings**.

TerraScan can import trajectory information from several binary and ASCII formats. In addition, user-tailored text formats for trajectories can be defined in TerraScan **Settings**. TerraScan converts the files into its own trajectory format.

Supported point file formats

TerraScan has built-in support for the following point file formats:

T = text file format
B = binary file format

R = read files
W = write files

NAME	TYPE		CONTENT
FASTBINARY	B	R/W	TerraScan FastBinary (FBI).
SCAN8	B	R/W	TerraScan binary with 8 bit line numbers (BIN).
SCAN16	B	R/W	TerraScan binary with 16 bit line numbers (BIN).
TOPEYE	B	R	TopEye binary (DTE) with geocentric coordinates.
EARTHDATA	B	R/W	EarthData binary (EBN or EEBN).
LEICA	B	R	Leica-Helava binary (LDI).
LAS	B	R/W	Industry standard format, versions 1.0, 1.1, 1.2, 1.4*
LAS	B	R	Industry standard format, version 1.3
LAZ	B	R/W	Compressed LAS format, versions 1.0, 1.1, 1.2, 1.3
PTS	T	R	Leica PTS format.
XYZ	T	R/W	Easting Northing Elevation
CXYZ	T	R/W	Class Easting Northing Elevation
XYZI	T	R/W	Easting Northing Elevation Intensity
CXYZI	T	R/W	Class Easting Northing Elevation Intensity
PXYZI	T	R	Pulse number Easting Northing Elevation Intensity
XYZXYZII	T	R	East1 North1 Elev1 East2 North2 Elev2 Intensity1 Intensity2
TXYZXYZII	T	R	Time East1 North1 Elev1 East2 North2 Elev2 Intensity1 Intensity2

NAME	TYPE		CONTENT
TXYZI	T	R	Time Easting Northing Elevation Intensity
XYZIXYZI	T	R	East1 North1 Elev1 Intensity1 East2 North2 Elev2 Intensity2
TXYZIXYZI	T	R	Time East1 North1 Elev1 Intensity1 East2 North2 Elev2 Intensity2
Grass Sites	T	W	
XYZdZ	T	W	Easting Northing Elevation Elevation difference to a surface
IdXYZ	T	W	Index Easting Northing Elevation
IdXYZP	T	W	Index Easting Northing Elevation Pulse number

In some dialogs, E is used instead of X and N is used instead of Y in order to refer to the easting and northing coordinates, respectively.

*LAS 1.4 limitations:

- does not store coordinate system information

Binary file formats can be used to store point cloud data in TerraScan projects. More information about formats and their capabilities of storing attributes for points can be found in the description of project creation.

TerraScan binary files

The recommended format for processing point clouds in TerraScan is **FastBinary**. This format is not documented openly but information may be provided upon request.

The information below refers to file formats which were last revised on 12.07.2001 and on 15.07.2002. These version dates are stored in the file header. Future versions of TerraScan may store laser points into another format but will always recognize and read in the old files.

TerraScan reads and writes two versions of the old binary file format:

- **Scan binary 8 bit line** - a more compact version which can accommodate line numbers 0-255. Files with this format have HdrVersion field set to 20010712.
- **Scan binary 16 bit line** - a slightly bigger version which can accommodate line numbers 0-65535. Files with this format have HdrVersion field set to 20020715.

See \TERRA64\ADDON\ROUTINES.C for example source code for reading in TerraScan binary files.

File organization

TerraScan binary file consists of a file header of 48 bytes and a number of point records. The size of the point record is 16 bytes for file version 20010712 and 20 bytes for file version 20020715. Each

point record may be followed by an optional four byte unsigned integer time stamp and an optional four byte RGB color value.

For example, a file containing four laser points and their time stamps using format 20020715 would consist of:

- 48 byte header (ScanHdr)
- 20 byte record for first point (ScanPnt)
- 4 byte time stamp for first point
- 20 byte record for second point (ScanPnt)
- 4 byte time stamp for second point
- 20 byte record for first three (ScanPnt)
- 4 byte time stamp for three point

Structure definitions

The structure of the file header is:

```
typedef struct {
 int     HdrSize ;      // sizeof(ScanHdr)
 int     HdrVersion ;   // Version 20020715, 20010712, 20010129 or
     970404
 int     RecogVal ;     // Always 970401
 char    RecogStr[4];   // CXYZ
 long    PntCnt ;       // Number of points stored
 int     Units ;        // Units per meter = subpermast * uorpersub
 double  OrgX ;         // Coordinate system origin
 double  OrgY ;
 double  OrgZ ;
 int     Time ;         // 32 bit integer time stamps appended to points
 int     Color ;        // Color values appended to points
} ScanHdr ;
```

The structure of a point record for file version 20010712 is:

```
typedef struct {
 BYTE    Code ;         // Classification code 0-255
 BYTE    Line ;         // Line number 0-255
 USHORT  EchoInt ;      // Intensity bits 0-13, echo bits 14-15
 long    X ;            // Easting
 long    Y ;            // Northing
 long    Z ;            // Elevation
} ScanRow ;
```

The structure of a point record for file version 20020715 is:

```
typedef struct {
 Point3d Pnt ;          // Coordinates
 BYTE    Code ;         // Classification code
 BYTE    Echo ;         // Echo information
 BYTE    Flag ;         // Runtime flag (view visibility)
 BYTE  Mark ;           // Runtime flag
 USHORT  Line ;         // Line number
 USHORT  Intensity ;    // Intensity value
} ScanPnt ;
```

Coordinate system

Laser point coordinates are stored as integer values which are relative to an origin point stored in the header. To compute user coordinate values X, Y and Z (normally meters), use:

```
X = (Pnt.X - Hdr.OrgX) / (double) Hdr.Units ;
Y = (Pnt.Y - Hdr.OrgY) / (double) Hdr.Units ;
Z = (Pnt.Z - Hdr.OrgZ) / (double) Hdr.Units ;
```

Time stamps

Time stamps are assumed to be GPS week seconds. The storage format is a 32 bit unsigned integer where each integer step is 0.0002 seconds.

Echo information

TerraScan uses two bits for storing echo information. The possible values are:

```
0 Only echo
1 First of many echo
2 Intermediate echo
3 Last of many echo
```

Supported trajectory file formats

TerraScan has built-in support for the following trajectory file formats:

NAME	TYPE	CONTENT
TYXZRPH	Text	Time Northing Easting Elevation Roll Pitch Heading
TYXZSSSQSS	Text	Time Northing Easting Elevation Skip Skip Skip Quality Skip Skip
TYXZQSSSS	Text	Time Northing Easting Elevation Quality Skip Skip Skip Skip
TTXYZHPR	Text	Time Time Easting Northing Elevation Heading Roll Pitch
TXYZ	Text	Time Easting Northing Elevation
SBET.OUT	Binary	Proprietary file format of Applanix
SBTC / SBIC	Binary	Proprietary file format of NovAtel Inertial Explorer 8.6+
POF 1.1	Binary	Proprietary file format of Riegl.

TerraScan trajectory binary files

Imported trajectories are stored as binary files with TRJ extension. These files contain a header followed by a number of trajectory position records.

The structure of the file header is:

```
typedef struct {
 char    Recog[8] ;        // TSCANTRJ
 int     Version ;         // File version 20010715
 int     HdrSize ;         // sizeof(TrajHdr)
 int     PosCnt ;          // Number of position records
 int     PosSize ;         // Size of position records
 char    Desc[78] ;        // Description
 BYTE    SysIdv ;          // System identifier (for lever arms)
 BYTE    Quality ;         // Quality for whole trajectory (1-5)
 double  BegTime ;         // First time stamp
 double  EndTime ;         // Last time stamp
 int     OrigNbr ;         // Original number (before any splitting)
 int     Number ;          // Line number (in laser points)
 char    VrtVideo[400] ;   // Vertical facing video
 double  VrtBeg ;          // Start time of VrtVideo[]
 double  VrtEnd ;          // End time of VrtVideo[]
 char    FwdVideo[400] ;   // Forward facing video
 double  FwdBeg ;          // Start time of FwdVideo[]
 double  FwdEnd ;          // End time of FwdVideo[]
 char    WaveFile[400] ;   // Waveform data file
 char    Group[16] ;       // Group (session description)
} TrajHdr ;
```

The structure of the trajectory position records is:

```
typedef struct {
 double  Time ;            // Time stamp (seconds in some system)
 Dp3d    Xyz ;             // Position
 double  Head ;            // Heading (degrees)
 double  Roll ;            // Roll (degrees)
 double  Pitch ;           // Pitch (degrees)
 BYTE    QtyXy ;           // Quality for xy, 0=not set
 BYTE    QtyZ ;            // Quality for z, 0=not set
 BYTE    QtyH ;            // Quality for headingy, 0=not set
 BYTE    QtyRp ;           // Quality for roll/pitch, 0=not set
 short   Mark ;            // Run time flag
 short   Flag ;            // Run time flag
} TrajPos ;
```

where Dp3d structure is:

```
typedef struct {
 double  x ;
 double  y ;
 double  z ;
} Dp3d ;
```

and where xy and z quality values translate to meters as:

```
QualityInMeters = pow(QtyXy,1.5) * 0.001 m

(1=0.0010m, 2=0.0028m, 3=0.0052m, 4=0.0080m,... 255=4.072m)
```

and where heading, and roll/pitch quality values translate to degrees as:

```
QualityInDegrees = pow(QtyH,1.5) * 0.0001 deg

(1=0.00010, 2=0.00028, 3=0.00052, 4=0.00080,... 255=0.4072 deg
```

File Name Variables

TerraScan includes a number of commands that write output files, for example, for point clouds or for vector data derived from point clouds. In order to write separate files with specific names, there are different variables that can be used in the output file name definition. The variables are always introduced with a # followed by the variable name.

Possible variables are:

- **#bnumber** - block number.
- **#block** - block name or block binary file name without extension.
- **#bpath** - path of the block binary file. This includes the binary file name and its extension.
- **#bemin** - minimum easting value of block boundary vertices.
- **#bnmin** - minimum northing value of block boundary vertices.
- **#bemax** - maximum easting value of block boundary vertices.
- **#bnmax** - maximum northing value of block boundary vertices.
- **#pdir** - path of the storage directory of the block binary file. This does not include the name of the binary file.
- **#name** - active file name. Use this to replace the file name component when running a macro on selected files.
- **#line** - line number stored for a point. Use this variable with the Output by line action in order to store one file for each line.

Installation Directories

TerraScan shares the same directory structure with all Terra Applications. It is recommended that you install all Terra Applications in the same directory.

The list below shows a typical directory structure when TerraScan has been installed in path C:\TERRA64.

C:\TERRA64	installation directory for Terra applications
addon	example addon source files
app	application files for Spatix
tscan.ix	application
tscanadd.dll	library
cell	MicroStation cell libraries
karttali.cel	sample cell library with topographic objects, such as trees
config	application configuration files
tscan.cfg	defines environment variables for MicroStation
tscan.ini	defines environment variables for Spatix
coordsys	coordinate system definitions
docs	documentation, such as user guides in PDF format
example	sample files
dangerpoint_report.html	HTML template for creating danger object reports
license	user license files
tscan.lic	user license
ma	application files for MicroStation
seed	seed files, templates
seed3dcm.dgn	seed file for creating MicroStation 3D design files with centimeter resolution
seed3dcm.spx	template for creating Spatix files with centimeter resolution
seed3dmm.dgn	seed file for creating MicroStation 3D design files with millimeter resolution
seed3dmm.spx	template for creating Spatix files with millimeter resolution
tscan	user settings and configuration files
tscan.ptc	default list of point classes
tslave	TerraSlave application files and sub-folders

- A -

- B -

- C -

- D -

- E -

- F -

- G -

- I -

- L -

- M -

- N -

- O -

- P -

- R -

- S -

- T -

- U -

- V -

- W -