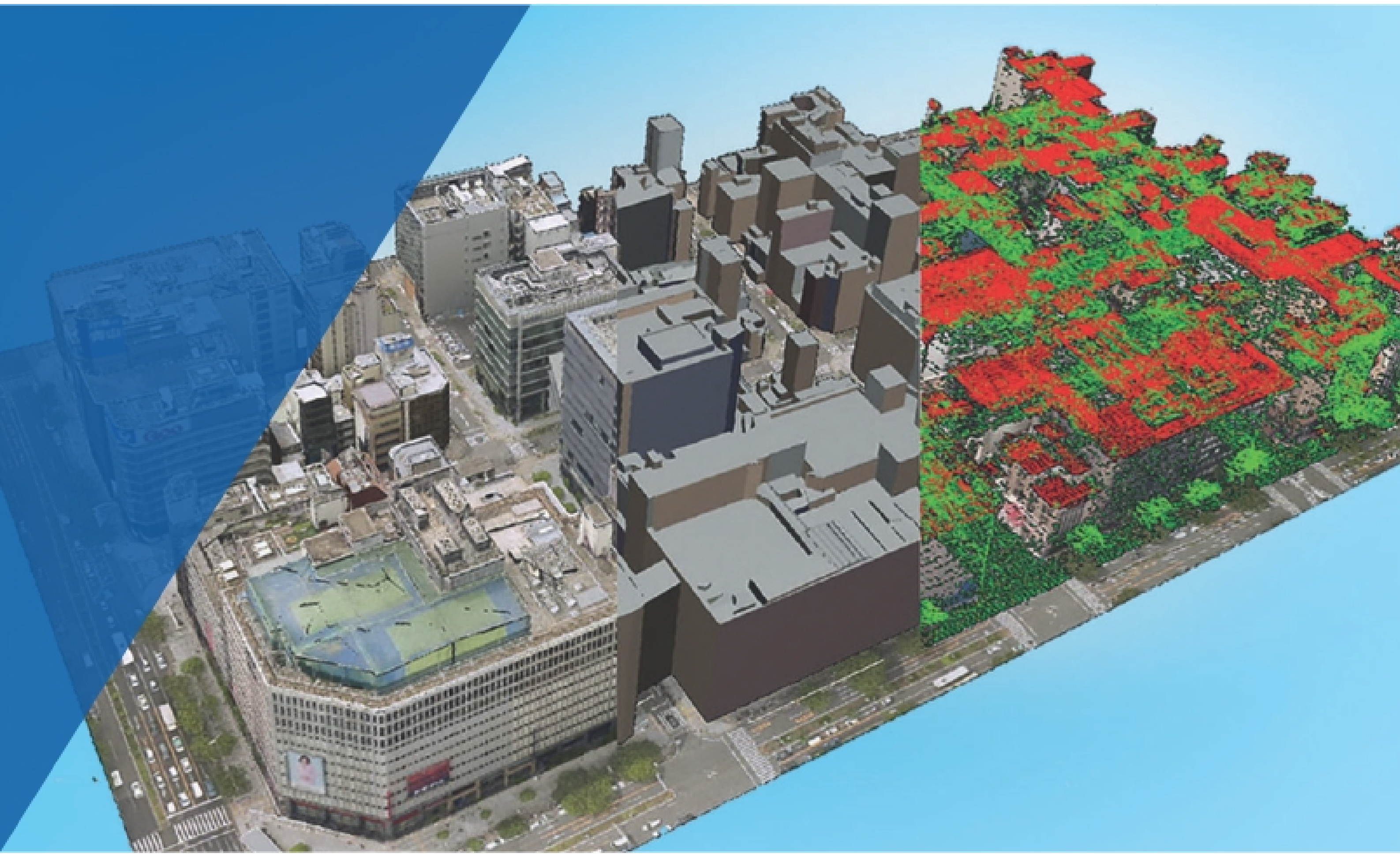
TerraModeler USER GUIDE

64-bit version



Document created in March, 2024

Contents

Contents

Contents

Contents

Acknowledgment

The picture on the title page was created from a CAD file provided as a courtesy of Soil and Water Ltd, Itälahdenkatu 2, 00210 Helsinki, Finland. The CAD file was created from an aerial photograph using TerraSurvey and TerraModeler.

© 2024 Terrasolid Ltd

Printed: March 2024 in Finland.

Trademarks

About this User Guide

This document serves as a user's guide for the 64-bit versions of TerraModeler. The entry-level version, TerraModeler Lite, is functionally a subset of the full version, TerraModeler. TerraModeler 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 TerraModeler, TerraModeler UAV, and/or TerraModeler Lite work identically in all versions. Tools that are not available in TerraModeler Lite are marked as "*Not Lite*" in the documentation. Tools that are not available in TerraModeler 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 Bentley CAD 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 Bentley CAD versions is necessary, this document refers to the CAD environment simply as "CAD platform".

Notes and hints are highlighted in light blue boxes.

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

Spatix documentation

The User Guide for Spatix is published by GISware Integro and delivered as PDF with the software. It can be opened with the **Manual** command from the **Help** menu of Spatix.

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.

Bentley CAD documentation

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

Terminology

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

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

BENTLEY	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	Big element • Construct Big Element / Big line	element type • tool • tool

BENTLEY	SPATIX	REMARK
• Create Complex Shape	• Construct Big Element / Big polygon	
Cell	Symbol	element type, also Cell library
Fence (= selected shape)	Clipper (= selected polygon)	many TerraScan functions consider a selected polygon as fence
Key-in command	Spaccel (**Sp**atix **accel**erate**s**)	typed command to call a function, also key-in window, key-in line

TerraModeler

Introduction

TerraModeler is a terrain modeling application built on top of a CAD platform. It enables the creation of triangulated models of ground, soil layers or design surfaces. The models can be created based on survey data, graphical elements, XYZ text files, point cloud data loaded in TerraScan or stored in LAS or TerraScan binary formats.

Theoretically, TerraModeler can handle an unlimited number of different surfaces in the same CAD file. In practice, however, the number and size of loaded models as well as the performance of TerraModeler and the CAD platform depend on the RAM of the computer.

You can edit any of the surface models interactively. Editing options include add, move or delete individual points; move, drop, or flatten all the points inside a fence; construct breaklines and add new elements to the models.

Surface models can be used, for example, as a design aid. You can place elements on the elevation of the surface model or drop existing elements to follow the surface.

Profile generation in TerraModeler includes separate tools for drawing 3D section views, profiles and alignment cross sections or tunnel sections. Additionally, 3D elements can be projected into a profile and elements drawn in a profile can be projected back to their true 3D positions.

Surface model visualization can be done by generating contours, drawing colored triangles, shaded surfaces, colored grid displays, slope arrows and numerical representations of elevation points. All of these visualization displays can be updated after you have made modifications to a surface model.

Tools for quantity computation can be utilized to calculate and report the volume between two surfaces and/or based on tunnel or trench sections. The area of computation can be limited to take place only inside a fence.

While many tools in TerraModeler are suited for traditional survey data types, TerraModeler can also work with point cloud data because it is tightly integrated with TerraScan. Point clouds loaded in TerraScan can be used to create a surface model in TerraModeler. This is especially useful for ground classification verification because the surface model is updated automatically according to point class changes as long as the surface model is linked with the point cloud. In addition, the point cloud can be used directly for quantity computations.

The full version of TerraModeler includes tools for producing lattice models, triangles, and contour lines in batch processes for large project areas. The processes can produce the output files based on point clouds organized in a TerraScan project. Additionally, breaklines can be included in the contour line/lattice model/triangle creation.

TerraModeler Lite

TerraModeler Lite is a light version of TerraModeler and provides a subset of the functionality of the full version. It can be used create and manually edit surface models from design data, to display models, to compute simple quantities and to create profiles.

TerraModeler Lite does not include advanced quantity computation tools, domain or region tools and it provides only a limited selection of display methods for surface models.

The Function matrix provides a complete overview of tools and commands available in the different TerraModeler versions.

TerraModeler UAV

TerraModeler UAV is another lighter version of TerraModeler dedicated for processing data that is collected by UAV-mounted devices.

TerraModeler UAV does not include domain or region tools. It is available only in a bundle with other Terra UAV applications, such as TerraScan UAV, TerraMatch UAV and/or TerraPhoto UAV.

The Function matrix provides a complete overview of tools and commands available in the different TerraScan versions.

Hardware and software requirements

TerraModeler 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 Bentley CAD. Only one installation of Terra applications is needed and files are shared by the CAD platforms, such as license and settings files.

To run TerraModeler, 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 TerraModeler requires about 5 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 website.

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 platform version. You install Terrasolid software from an USB-Stick probably only if you participate in a training event.

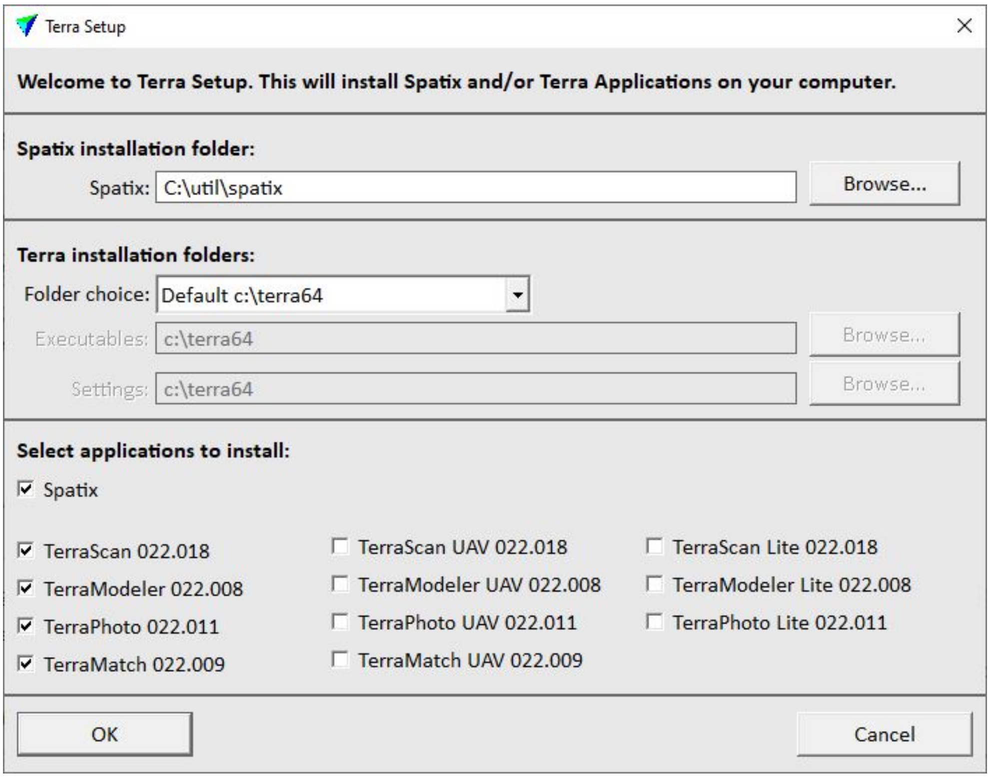
Terrasolid applications for Bentley products and Spatix may be installed on the same computer and run parallel. The applications should be installed in the same directory (e.g. c:\terra64). This enables the use of the same configuration files, settings, etc. for both platforms.

To install TerraModeler from a zip file together with 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 next to the input field in order to select a new installation folder for Spatix. The folder is created automatically, if it does not exist.

4. Define the installation folder(s) where to install TerraModeler and maybe other Terra applications.

   The default **Folder choice** is **Default c:\terra64**. This installs all executables and setting files into the same folder C:\TERRA64. The folder is created automatically, if it does not exist.

   As an alternative, if executables and settings files need to be separated, select another **Folder choice**:

   - **Default 'Program files'** - executables are installed into C:\PROGRAM FILES\TERRASOLID, setting files are installed into C:\TERRA64.
   - **Freely selectable folders** - the user defines a folder for **Executables** and **Settings** in the corresponding input fields. Click on the **Browse** button next to each input field in order to select a folder.

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.

> Install all Terrasolid applications into the same folder(s) in order to ensure interaction between the applications without trouble.

To install TerraModeler from a zip file on top of any Bentley product:

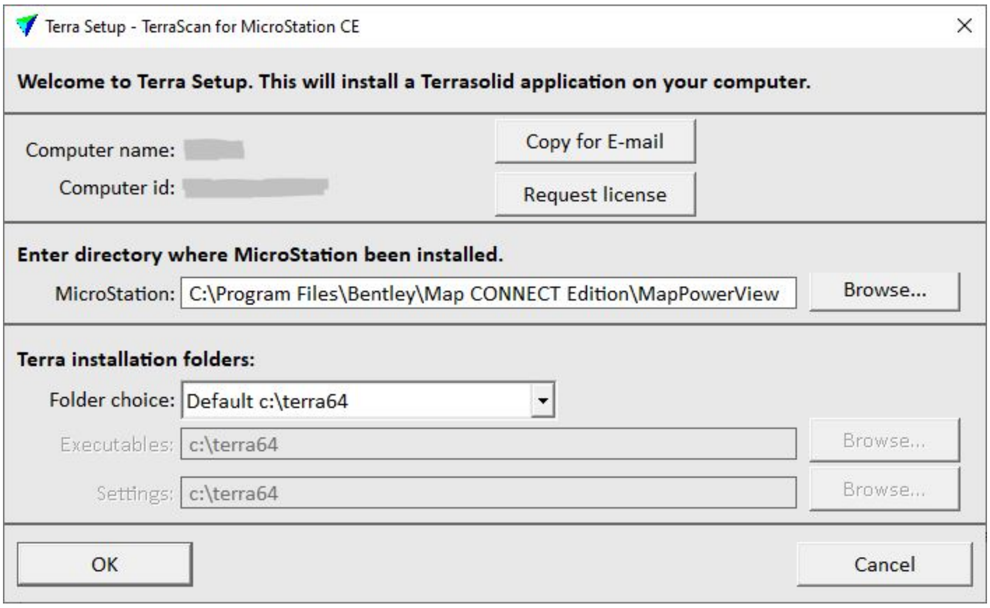
1. Unpack the zip archive with any zip file manager.

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 Bentley product (MicroStation, Map PowerView or any other compatible product) has been installed. It automatically searches all local hard disks to find the Bentley installation directory.

   The installation dialog opens:



The dialog is the same for all Bentley products and Terra applications. The labels in the dialog always refer to "MicroStation", no matter what Bentley product is used.

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

4. Define the installation folder(s) where to install TerraModeler and maybe other Terra applications.

   The default **Folder choice** is **Default c:\terra64**. This installs all executables and setting files into the same folder C:\TERRA64. The folder is created automatically, if it does not exist.

   As an alternative, if executables and settings files need to be separated, select another **Folder choice**:

   - **Default 'Program files'** - executables are installed into C:\PROGRAM FILES\TERRASOLID, setting files are installed into C:\TERRA64.
   - **Freely selectable folders** - the user defines a folder for **Executables** and **Settings** in the corresponding input fields. Click on the **Browse** button next to each input field in order to select a folder.

5. Click OK to start the installation.

   A message is displayed when the installation is finished.

> Install all Terrasolid applications into the same folder(s) in order to ensure interaction between the applications without trouble.

> 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 TerraModeler from USB-Stick on top of a Bentley product:

The process is the same for all Bentley products. The labels in all dialogs always refer to

"MicroStation", no matter what Bentley product is used.

1. Insert the USB-Stick.

2. Locate the correct installation directory on the stick.

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 TerraModeler 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.

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 Bentley CAD installation. Alternatively, you can use the **Browse** button to locate the Bentley CAD platform installation folder yourself.

6. Click OK to continue.

   This opens another **Terra Setup** dialog.

7. Select the TerraModeler **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 TerraModeler

TerraModeler is an application that runs on top of Spatix (Ix App) or Bentley CAD (MDL Application).

To start TerraModeler 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 **tmodel.ix** file.

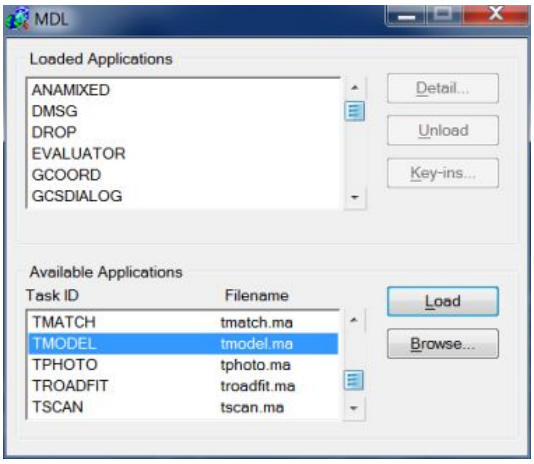
   You may select other applications as well.

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

To start TerraModeler in Bentley CAD:

1. Select **MDL Applications** command from the **Utilities** menu in Bentley CAD.

   The MDL dialog opens:



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

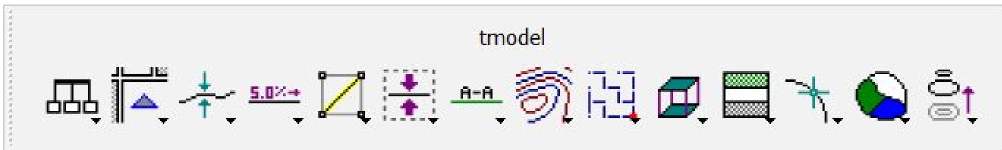
3. Click the **Load** button.

   OR

1. Key in *MDL LOAD TMODEL*.

The **Available Applications** list shows all MDL applications that Bentley CAD is able to locate. Bentley CAD tool searches for MDL applications in the directories listed in **MS_MDLAPPS** configuration variable. If Bentley CAD can not find TMODEL.MA, you should check the variable in the **Configuration Variable** dialog of Bentley CAD. Make sure the directory path of the TMODEL.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 **TerraModeler** toolbox:



If the **TerraModeler** toolbox is accidentally closed, it can be re-opened with the keyin command:

model app main

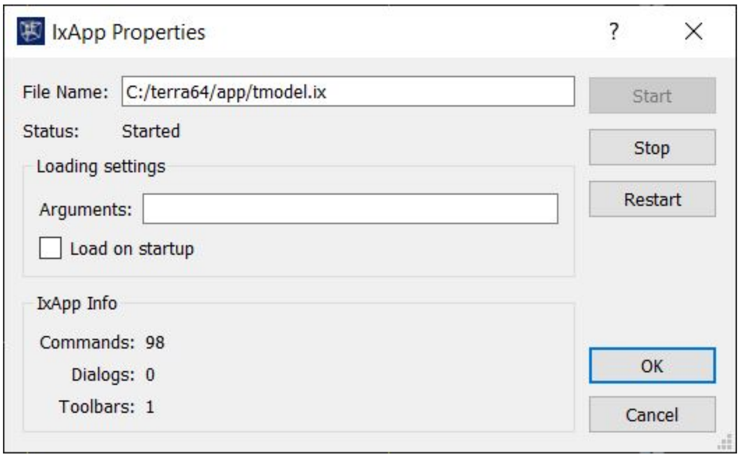
Unloading TerraModeler

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

To unload TerraModeler ***in Spatix:***

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

   The **IxApp Properties** dialog opens:



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

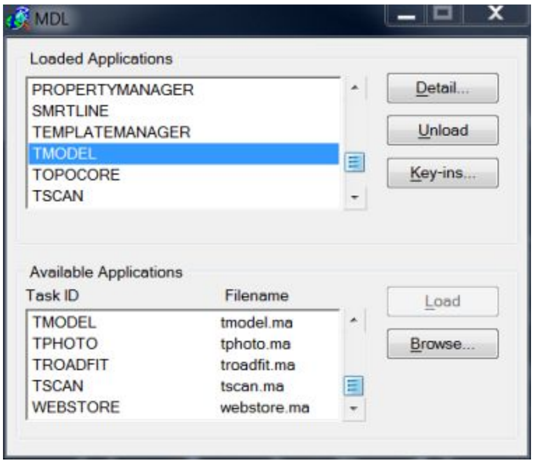
   This unloads TerraModeler, closes the TerraModeler toolbox and updates the Status and IxApp Info in the **IxApp Properties** dialog.

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

To unload TerraModeler ***in Bentley CAD:***

1. Select MDL **Applications** command from the **Utilities** pulldown menu in Bentley CAD.

   The **MDL** dialog opens:



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

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

   OR

1. Key in *MDL UNLOAD TMODEL*.

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

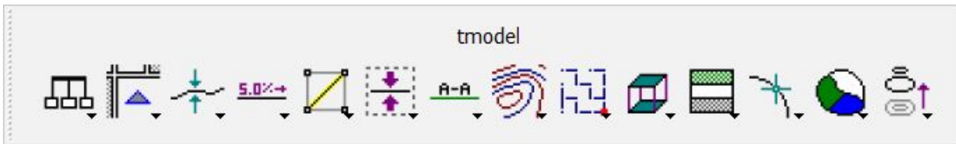
Tutorial

Not Spatix

This tutorial introduces some basic concepts and terminology used by TerraModeler. With the help of an example Bentley CAD design file, you are guided through a number of exercises using some of the features in the software. The tutorial assumes that you are already familiar with basic viewing and drawing tools in Bentley CAD tools.
This tutorial has been prepared with Bentley MicroStation. The tutorial can be run on other CAD platforms but some CAD platform tools used might be named differently. For the sake of clarity, this tutorial mentions when a tool used is a Bentley MicroStation tool.

TerraModeler interface

When you load TerraModeler, it opens its **TerraModeler** toolbox. All TerraModeler tools are located in this toolbox, which actually includes 14 toolbars. These toolbars can be dragged out from the toolbox. They contain icons representing the individual tools. To activate a tool, click the tool icon.



You can control the operation of a tool with tool settings. These are control items affecting the operation of that specific tool. Most of the tools display their settings in a separate dialog where you can enter the appropriate values.

Opening an example design file

This tutorial uses an example CAD design file that is included in the setup of TerraModeler. This example file has been provided by courtesy of Soil and Water Ltd, a company with expertise in many fields of civil engineering. The design file was created from an aerial photograph using TerraSurvey.

To open the example design file:

1. Select **Open** command from the **File** ribbon in MicroStation.

   The **Open Design File** dialog opens.

2. Locate the MODEL.DGN file by browsing your computer's directory structure.

   If you installed TerraModeler in the default directory C:\TERRA64, you can find the design file in C:\TERRA64\EXAMPLE\MODEL.DGN.

3. Select MODEL.DGN and click OK.

   After opening the design file, you can take a closer look at it. View 1 has been fitted to show all the elements in the design file. You can rotate this view or zoom in to see smaller details.

Starting TerraModeler

Start TerraModeler as described in Section Starting TerraModeler.

Creating a surface model

TerraModeler can create a surface model from a number of data sources. The best tool to use depends on the source and format of the data:

DATA SOURCE:	USE TOOL:
Survey data	Triangulate Survey after reading in the survey data with TerraSurvey.
Design file elements	Triangulate View or Triangulate Elements.
XYZ text file	Surfaces tool opens a window for managing surfaces. The **File** pulldown menu in this window has a command for importing xyz text files.

To create a surface model in the example design file:

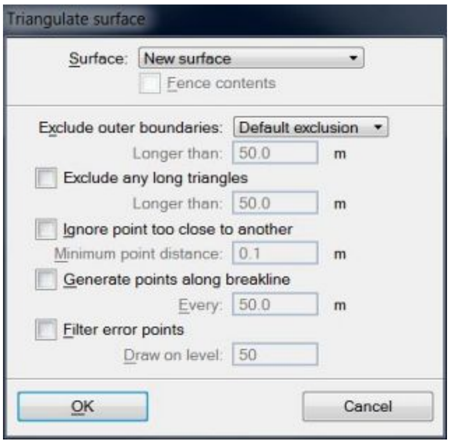
1. Bring View 2 in front of the other views.

   This view displays only selected levels in the design file. Elements that do not reside on the ground elevation have been hidden by switching off their levels.

2. Select Triangulate View tool from the **Create Surfaces** toolbox.

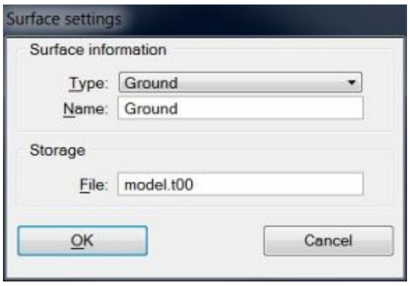
3. Identify view 2.

   The **Triangulate surface** dialog opens:



4. Click OK.

   The **Surface settings** dialog opens:

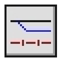


5. Enter **Ground** as the surface name.

6. Click OK.

   TerraModeler processes the information. A progress bar shows the progress of the process.

Viewing surface statistics



You just created your first surface model with TerraModeler. The model exists in two types of memory. It has been saved to a binary file on the hard disk for permanent storage. When you work with TerraModeler, the surface model is also loaded in the RAM of your computer.

You can use the Surfaces tool to view information about loaded surfaces.

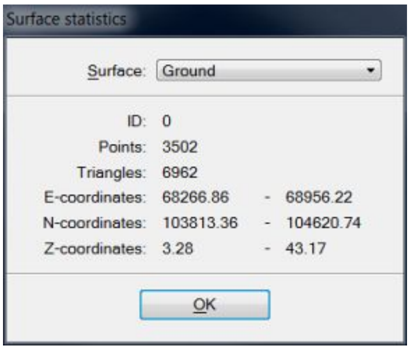
To view surface statistics:

1. Select Surfaces tool from the **General** toolbox.

   The **Surfaces** window opens showing a list of loaded surfaces. Select surface model **Ground**, if it is not selected already.

2. Select **View statistics** from the **Utility** menu.

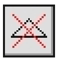
   The **Surface statistics** dialog opens:



   The dialog shows the internal ID of the surface model, the number of points and triangles, and the ranges of easting, northing, and elevation coordinates.

3. Click OK after reviewing the statistics.

Viewing triangulation



TerraModeler creates surface models by triangulation. This means that the application creates a network of triangles connecting the points in the model. The network is constructed in a way that there is one triangle at every XY location inside the surface area. Thus, in a surface model every XY location has one and only one elevation value.

To view the triangulation:

1. Zoom in on a position in View 2 inside the surveyed area.

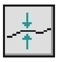
2. Select Exclude Triangle tool from the **Edit Point** toolbox.

   This tool can be used to excluding or including triangles. In TerraModeler terminology, to exclude a triangle means to mark it as invalid or unknown.

   This tool can also be used just to view the triangulation.

3. Move the mouse inside View 2. As you move the mouse, TerraModeler displays the triangle at the mouse pointer position.

Placing elements on surface elevation



A surface model can be used as a design aid. TerraModeler makes it easy to place elements at the elevation of a surface. You may use this capability, for example, to place items such as manholes, trees, or shrubs on the ground elevation.

To place elements on surface elevation:

1. Select View Elevation tool from the **Draw** toolbox.

   The **View elevation** dialog opens:

   As you move the mouse, the dialog dynamically displays the elevation value of the selected **Surface** at the mouse pointer position.

2. Set **Points on surface** lock on.

3. Start drawing an element using any MicroStation tool for drawing elements. Feel free to experiment with by drawing some lines, shapes, or placing a few cells.

   The elements are drawn as 3D elements using the elevation value from the **Ground** surface.

When **Points on surface** lock is on, TerraModeler calculates the elevation for all data points entered in a top view inside the surface area. Therefore, be sure to turn the lock off if you do not want to use it any longer.

Calculating slopes

When designing new construction surfaces, you may need to create a slope onto an existing surface. Calculate Slope tool supports this task. As a first step, you have to create a graphical element representing the upper or lower edge of a slope. The example design file has a suitable element for this purpose. It is a green shape element that represents the bottom of a flat rectangular excavation at elevation +29.50.

To locate the shape element:

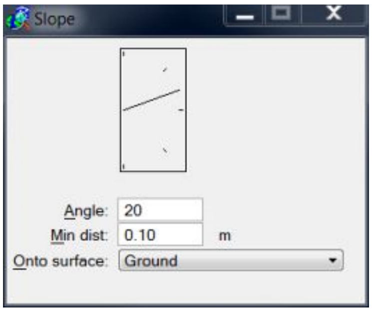
1. Bring View 4 in front of all other views.

   The view displays only the green shape element on level 56.

To calculate a slope upwards to the ground surface:

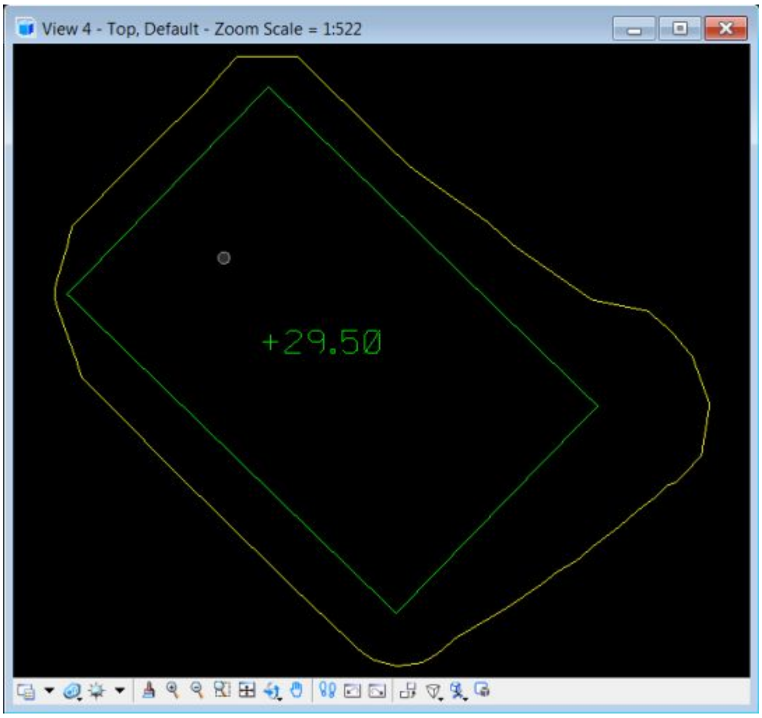
1. Select yellow as the active color.
2. Select the Calculate Slope tool.

   This opens the **Slope** dialog:

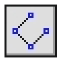
   

3. Enter 20 as the slope **Angle**.
4. Select **Ground** as the **Onto surface**.
5. Identify the green shape element as the element to start from.
6. Define the slope direction with a data point outside the shape element. This means that the slope is calculated outwards from the element.

   The application calculates a slope upwards from the element at an angle of 20 degrees. It draws the upper edge of the slope as a yellow line string.


View 4 - Top, Default - Zoom Scale = 1:522
+29.50

Creating a design surface



TerraModeler can be used to create design surfaces for earthworks projects. The best approach is to first create graphical elements representing the shape of the design surface. This is exactly what you did in the previous step with the Calculate Slope tool. You can now use the green shape element and the yellow slope element to create a new design surface.

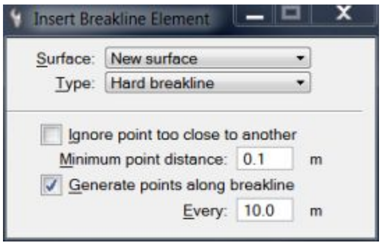
To create a design surface:

1. Select the elements in View 4.

   Your selection set should include the green shape element and the yellow line string representing the upper slope edge.

2. Select Insert Breakline Element tool from the **Create Surfaces** toolbox.

   This opens the **Insert Breakline Element** dialog:

3. Select **New surface** as the **Surface** to insert points into and **Type** as **Hard breakline** .

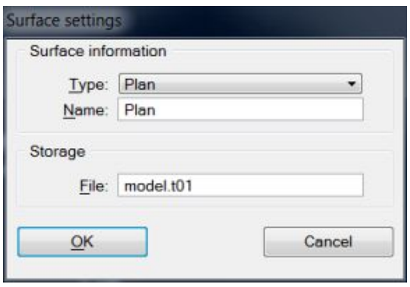
4. Switch **Generate points along breakline** on and enter 10.0 into the **Every** field.

   This setting generates new points along a long breakline if the distance between two breakline points is longer than 10 meters.

5. Accept the elements.

   This opens the **Triangulate surface** dialog.

6. Accept default settings by clicking OK which opens the **Surface settings** dialog:

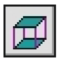
7. Select **Plan** in the **Type** field.

8. Enter **Plan** as the surface **Name**.

9. Click OK.

TerraModeler creates a new surface using the vertices of the selected elements.

Computing excavation quantities



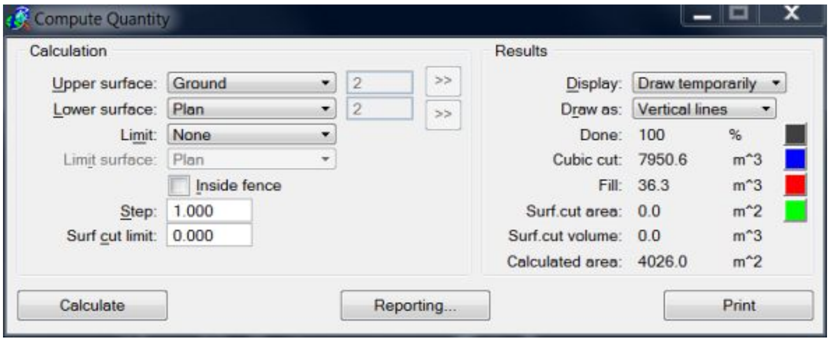
You have now two surface models. The first one, **Ground**, is based on survey information. The second one, **Plan**, is a design model of a planned excavation with slopes onto the ground surface.

You are ready to compute excavation quantities using the Compute Quantity tool. It computes the volume between two surfaces and produces results summing up both, cut and fill volumes. The calculation is based on a grid method.

To compute excavation quantities:

1. Select Compute Quantity tool from the **Quantity** toolbox.

   The **Compute Quantity** dialog opens:



2. Select **Ground** as the **Upper surface**.

3. Select **Plan** as the **Lower surface**.

4. Enter 1.0 as the **Step** value.

5. Make sure that **Draw temporarily** or **Write to file** is selected as the **Display option** and **Vertical lines** as **Draw as** option.

6. Click on the **Calculate** button to start computation.

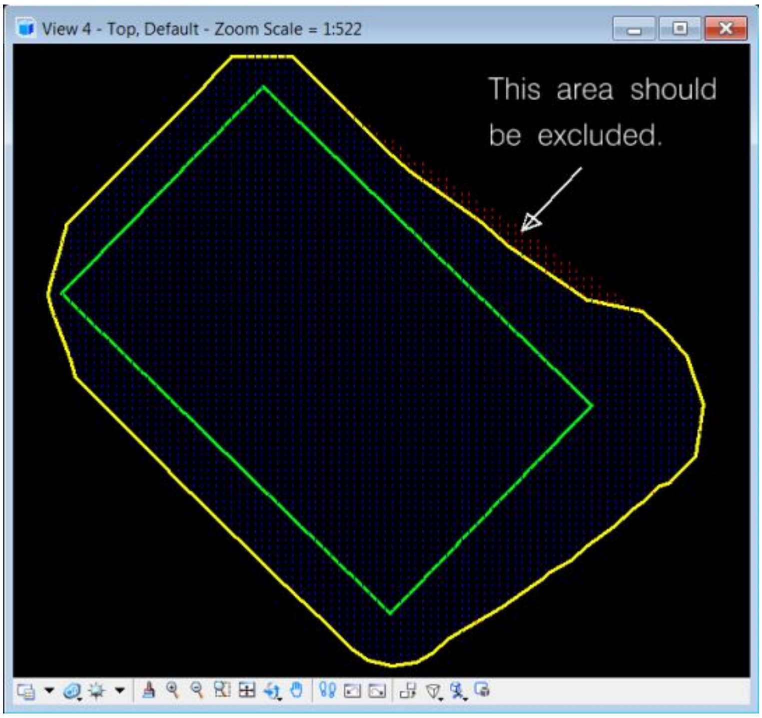
   The application starts calculating the volume between the two surfaces. It displays the progress of the calculation as vertical linear elements. The color of each line indicates if that grid position results in cut or fill volume.

   When the calculation is finished, check the results. TerraModeler reports about 7951 cubic meters of cut volume and about 36 cubic meters of fill volume. The cut volume is a correct value. The fill volume, however, is something you may want to take a closer look at.

   When TerraModeler creates a surface model, the triangulated area is a concave polygon which encloses all the points in the model. To control the actual area that should be covered by the model, you can exclude some of unnecessary triangles on the outer boundaries.

   When you create a surface with tools in TerraModeler, the application can automatically exclude narrow, long triangles from the outer boundaries. The same task can be done later by modifying an existing surface model.

In our excavation example, you may decide simply to ignore the fill volume reported by the Compute Quantity tool as you know for sure that the fill volume should be zero. Alternatively, you may decide to exclude the extra triangles in the **Plan** surface model.



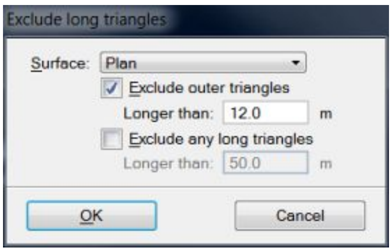
To exclude triangles from the excavation automatically:

1. Select Surfaces tool from **General** toolbox.

   The **Surfaces** window opens showing a list of loaded surfaces. Select surface **Plan** if it is not selected already.

2. Select Exclude long triangles command from the **Edit** pulldown menu.

   The **Exclude long triangles** dialog opens:



3. Switch on **Exclude outer triangles** and type 12.0 as the maximum length in the **Longer than** field.

4. Click OK.

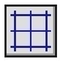
To exclude triangles manually:

1. Select Exclude Triangle tool from the Edit Point toolbox.

2. Select **Plan** as the active surface in the tool’s dialog.
3. Enter a data click inside each of the triangles to exclude.

   After excluding the extra triangles, you may recalculate the excavation quantities as described above.

Displaying a colored grid

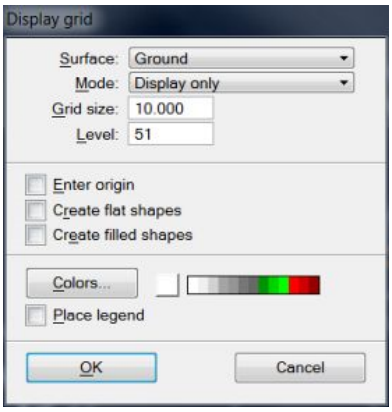


A surface model is often used for visualization purposes. TerraModeler is able to display a surface model with several display method.

To display a colored grid:

1. Select Display Grid tool from the **Display Surface** toolbox.

   The **Display grid** dialog opens:

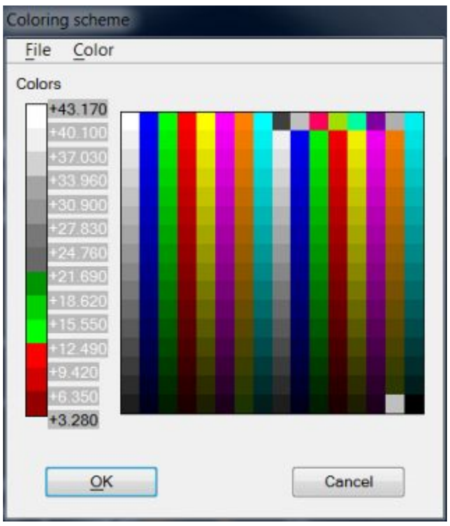


2. Select **Ground** in the **Surface** field and **Display only** in the Mode field.

3. Enter 10.0 in the **Grid size** field.

4. Click on the **Colors** button to define a coloring scheme.

   The **Coloring scheme** dialog opens:



5. Select **Open** command from the **File** menu to load a coloring scheme from a file.

6. Locate and open MODEL.CLR file.

If you installed TerraModeler in the default directory C:\TERRA64, you can find this file in C:\TERRA64\EXAMPLE\MODEL.CLR.

7. Click OK to close the **Coloring scheme** dialog.

8. Click OK to close the **Display grid** dialog.

   TerraModeler draws a colored grid. To view the grid, use view controls to change the rotation angle and to zoom in or out. Use view 2 for displaying the grid.

Make sure the level used in displaying the grid is switched on in View 2. The default level is 51. You may also switch off the **Clip Front** and **Clip Back** options in the View Attributes dialog for View 2.

The grid was drawn in **Display only** mode. This means that TerraModeler did not write the grid into the design file. The grid is erased when you unload TerraModeler or exit MicroStation. Alternatively, you can erase the grid with Erase Display tool.

Drawing a profile

You can create a profile along any linear element. The profile shows the shape of surface models along the alignment element. A profile is drawn as a cell element that can be freely positioned anywhere in the design file.

To create an alignment element:

1. Make an empty level the active level in MicroStation.

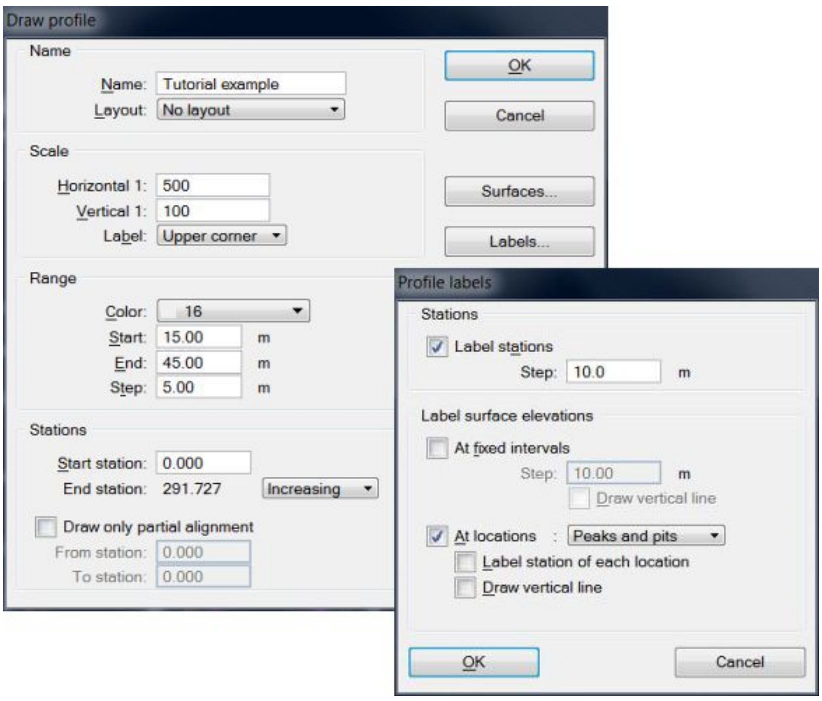
2. Place a line string element that runs across or inside the ground surface model using the Place Line String tool in MicroStation.

To draw a profile:

1. Select the Draw Profile tool.

2. Identify the alignment element you just created and accept the highlighted element.

   The **Draw profile** dialog opens:



3. Define settings in the dialog.

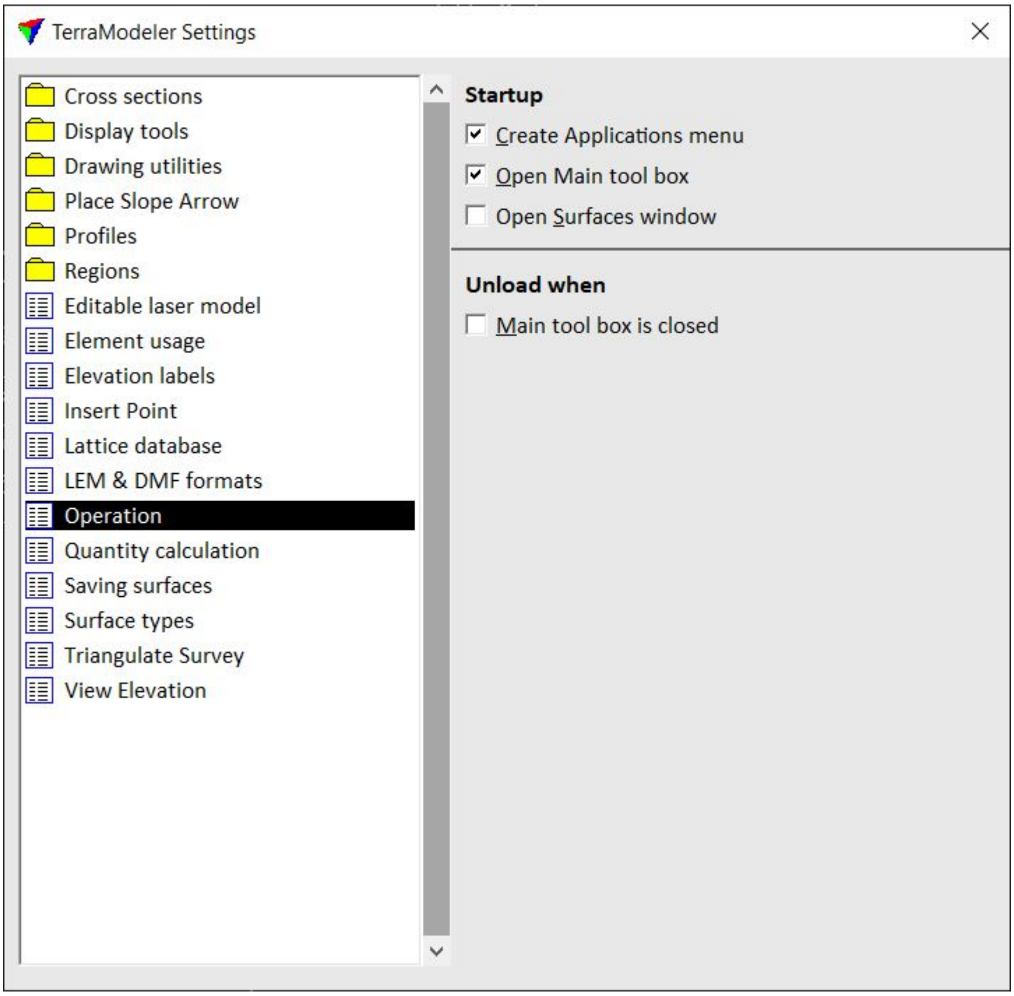
4. Click on the **Surfaces** button and select **Ground** as the only surface for being drawn in profiles in the Profile surfaces dialog.

5. Click on the **Labels** button and define settings in the Profile labels dialog.

6. Enter a data point to position the profile cell. It is a good idea to position the profile outside the surveyed site in order to avoid overlap with any existing elements.

TerraModeler Settings

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



SETTINGS FOLDER / CATEGORY	SETTINGS CATEGORY
Cross sections / Elevation grid	Editable laser model
Cross sections / Elevation labels	Element usage
Cross sections / Offset labels	Elevation labels
Cross sections / Placement	Insert point
Display tools / Color scheme legends	Lattice database *Not Lite, Not UAV*
Display tools / Contours	LEM & DMF formats
Display tools / Display shaded surface	Operation
Drawing utilities / Label area	Quantity calculation
Drawing utilities / Label elevation	Saving surface

SETTINGS FOLDER / CATEGORY	SETTINGS CATEGORY
Drawing utilities / Measure slope	Surface types
Place slope arrow / Arrows *Not Lite*	Triangulate survey
Place slope arrow / Labels *Not Lite*	View elevation
Profiles / Elevation grid	
Profiles / Labels	
Profiles / Layouts	
Profiles / Levels	
Profiles / Titles	
Regions / Region design *Not Lite, Not UAV*	
Regions / Region levels *Not Lite, Not UAV*	
Regions / Region types *Not Lite, Not UAV*	
Regions / Title formats *Not Lite, Not UAV*	

Cross sections / Elevation grid category

Elevation grid category in **Cross sections** folder defines the symbology of elevation grid range lines and labels in cross sections.

SETTING	EFFECT
Style	Defines the line style of range lines. Uses CAD file line styles.
Weight	Defines the line weight of range lines. Uses CAD file line weights.
Font	Font type of elevation grid labels.
Size	Text size of elevation grid range labels. Given in millimeters on paper.
On side	Defines the location of labels of range elevations: on the **Left**, on the **Right** or on **Both** sides of the range.

Cross sections / Elevation labels category

Elevation labels category in **Cross sections** folder defines the symbology of elevation labels in cross sections.

SETTING	EFFECT
Font	Font type of elevation labels. Uses font types available in the CAD platform.
Size	Text size of elevation labels. Given in millimeters on paper.
Direction	Text rotation: **horizontal** or **vertical**.
Digits	Number of full-number digits in elevation labels.
Decimals	Number of decimals in elevation labels.
Color	Text and note line color for binding elevation labels. Uses the active color table of the CAD file.

Cross sections / Offset labels category

Offset labels category in **Cross sections** folder defines the symbology of offset labels in cross sections.

SETTING	EFFECT
Font	Font type of offset labels. Uses font types available in the CAD platform.
Size	Text size of offset labels. Given in millimeters on paper.
Color	Color of offset labels. Uses the active color table of the CAD file.
Direction	Text rotation: **horizontal** or **vertical**.
Show sign in offsets	If on, the plus (+) sign is displayed for positive and the minus (-) sign for negative offset label values.

Cross sections / Placement category

Placement category in **Cross sections** folder defines how Draw Alignment Sections tool groups the cross sections for the display.

SETTING	EFFECT
Rows	Defines how many cross sections to draw on top of each other.
Order	Order of cross sections in a group: • **First one at the bottom** - station values increase upwards. • **First one at the top** - station values increase downwards.
X spacing	Defines the spacing between cross sections in X direction (between cross section columns).
Y spacing	Defines the spacing between cross sections in Y direction (between cross section rows).

Display tools / Color scheme legends

Color scheme legends category in **Display tools** folder defines the size and the symbology of color scheme legend bars, boundaries and labels.

SETTING	EFFECT
Scale	Method for scaling the color bar height: • **Fixed** - every color occupies the same height in the bar. You can define the height of one color as millimeters on paper. • **Relative** - height of a color is relative to its elevation or slope interval. You can define the height of the whole bar.
Height	Height of one color in the color bar or of the whole color bar. This depends on the setting for **Scale** described above. Given in millimeters on paper.
Width	Color bar width in millimeters on paper.
Draw border	If on, an outer border line is drawn around the color bar using the given **Border color** and line weight.
Draw boundary lines	If on, each color of the color bar is surrounded by a boundary line using the given **Boundary color** and line weight.
Symbology	Color and line weight of boundary labels of color schemes. Uses CAD file colors and line weights.
Font	Font type of color scheme labels. Uses font types available in the CAD platform.
Size	Text size of color scheme labels. Given in millimeters on paper.

Display tools / Contours category

Contours category in **Display tools** folder defines the default symbology of contour line labels.

SETTING	EFFECT
Font	Font type of contour labels. Uses font types available in the CAD platform.
Size	Text size of contour labels. Given in millimeters on paper.
Style	Line style of contour line labels. Uses CAD file line styles.
Weight	Line weight of contour line labels. Uses CAD file line weights.

Display tools / Display Shaded Surface category

Display Shaded Surface category in **Display tools** folder defines default settings for the shaded surface display mode.

SETTING	EFFECT
Sun azimuth	Azimuth angle of the sun. North direction is defined as 0 degree, angle values increase clockwise.
Sun angle	Angle of the sun above the horizon.
Color scheme	Determines default color scheme for shaded surface display
File	File location of custom color scheme to be used as default. Visible only when **Color scheme** is set to **Selected colors**
Color cycles	Number of color cycles. Use zero to create a gray scale display showing triangle slope only.
Fit to	Determines how colors are fit to elevation values: • **Whole surface** - colors of the color scheme are fit to the elevation range of the whole surface. • **View content** - colors of the color scheme are fit to the elevation range that is visible in a CAD file view. This results in a very detailed display for large-scale views.
Views	Views for displaying the shaded surface.

Drawing utilities / Label Area category

Label Area category in **Drawing utilities** folder defines the format and the symbology of area labels which Label Area tool creates.

SETTING	EFFECT
Font	Font type of area text elements. Uses font types available in the CAD platform.
Size	Text size of area labels. Given in millimeters on paper.
Accuracy	Number of decimals and unit for area labels.
2D area label prefix/suffix	Prefix/Suffix that is added to the area label of 2D areas.
3D area label prefix/suffix	Prefix/Suffix that is added to the area label of 3D areas.

Drawing utilities / Label Elevation category

Label Elevation category in **Drawing utilities** folder defines the format and the symbology of elevation labels which Label Elevation tool creates.

SETTING	EFFECT
Position	Position of the label related to the guiding line: **Above line**, **Extending from line** or **Below line**.
Font	Font type of elevation label text elements. Uses font types available in the CAD platform.
Size	Text size of elevation label texts. Given in millimeters on paper.
Accuracy	Number of decimals of elevation values.
Display plus	If on, the plus sign is displayed for positive elevations.
Display minus	If on, the minus sign is displayed for negative elevations.
Prefix	Text added to the label before the elevation value.
Suffix	Text added to the label after the elevation value.

Drawing utilities / Measure Slope category

Measure Slope category in **Drawing utilities** folder defines the symbology of labels which Measure Slope tool creates.

SETTING	EFFECT
Font	Font type of slope measurement labels. Uses font types available in the CAD platform.
Size	Text size of slope measurement labels. Given in millimeters on paper.
Color	Color of slope measurement labels. Uses the active color table of the CAD file.
Style	Line style of slope measurement labels. Uses CAD file line styles.
Weight	Line weight of slope measurement labels. Uses CAD file line weights.

Place Slope Arrow / Arrows category

Not Lite

Arrows category in **Place Slope Arrow** folder defines the length and the design of arrows drawn by Place Slope Arrow tool.

SETTING	EFFECT
Place by	Placement point of a slope arrow: **Arrow start** or **Arrow center**.
Arrowheads	Number of arrowheads in a slope arrow: • **One** - all arrows have one arrowhead. • **One to three** - number of arrowheads depends on the slope gradient as shown in the illustration of the arrow.
Length	Two fields specifying the maximum and the minimum length of a slope arrow. Given in millimeters on paper.
Slope	Gradient values at which the arrow length is equal to: - maximum length - two thirds of maximum length - one third of maximum length - minimum length

Place Slope Arrow / Labels category

Not Lite

Labels category in **Place Slope Arrow** folder defines the format and the symbology of labels drawn by Place Slope Arrow tool.

SETTING	EFFECT
Write slope label	If on, a label indicating the gradient in percent is placed along a slope arrow.
Level	Level number in the CAD file on which the arrow label is placed.
Font	Font type of slope arrow labels. Uses font types available in the CAD platform.
Size	Text size of slope arrow labels. Given in millimeters on paper.
Unit	Value label unit: • **Percentage** - percentage ratio between elevation change and horizontal distance. • **Degree** - slope angle in degrees.
Accuracy	Number of decimals of the slope arrow label.
Symbology	Color and line weight of slope arrow labels. Uses CAD file colors and line weights.

Profiles / Elevation grid category

Elevation grid category in **Profiles** folder defines the symbology of elevation grid lines and labels in profiles.

SETTING	EFFECT
Style	Line style of grid lines. Uses CAD file line styles.
Weight	Line weight of grid lines. Uses CAD file line weights.
Extend left	Defines how long grid lines are extended over the left boundary of the profile. Given in millimeters on paper.
Extend right	Defines how long grid lines are extended over the right boundary of the profile. Given in millimeters on paper.
Font	Font type of elevation grid labels in profiles. Uses font types available in the CAD platform.
Size	Text size of elevation grid labels in profiles. Given in millimeters on paper.
On side	Defines the location of grid labels: on the **Left**, on the **Right** or on **Both** sides of the profile.

Profiles / Labels category

Labels category in **Profiles** folder defines the format of station and elevation labels in profiles.

SETTING	EFFECT
Font	Font type of profile labels. Uses font types available in the CAD platform.
Size	Text size of profile labels. Given in millimeters on paper.
Color	Text color of station labels. Uses the active color table of the CAD file.
Direction	Text rotation of station labels: **horizontal** or **vertical**.
Digits	Number of full-number digits of elevation labels.
Decimals	Number of decimals of elevation labels.

Profiles / Layouts category

Layouts category in **Profiles** folder displays a list of user defined profile layouts. Each layout definition contains a list of data rows that appear below the profile.

You can edit existing layouts by using the **Edit** button in the **Settings** dialog. This opens the **Profile layout** dialog which shows data rows defined for this layout. The dialog lets you add, edit, or delete existing data rows for a profile layout.

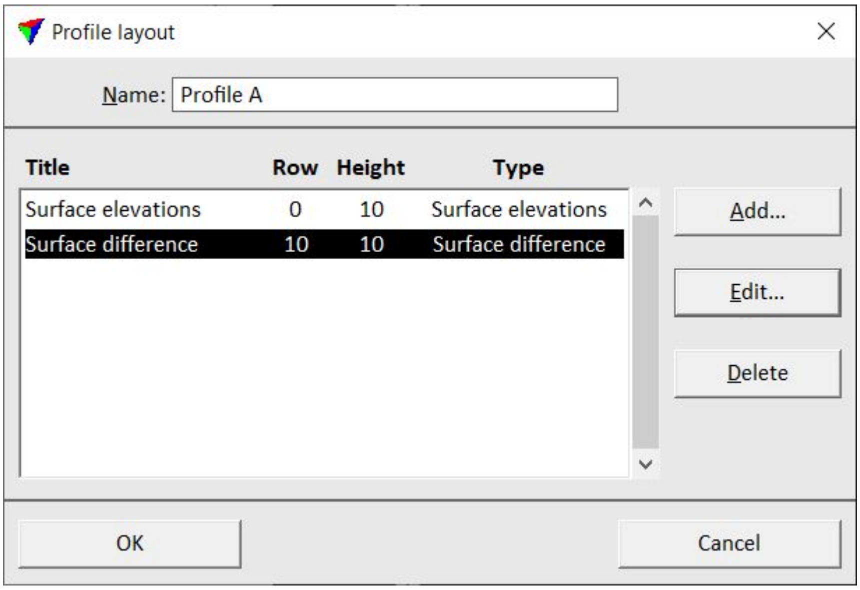
You can delete existing layouts by using the **Delete** button in the **Settings** dialog. The **Copy** button creates an identical copy of a selected layout definition.

To define a new profile layout:

1. Open the **Layouts** category in the **Profiles** folder.

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

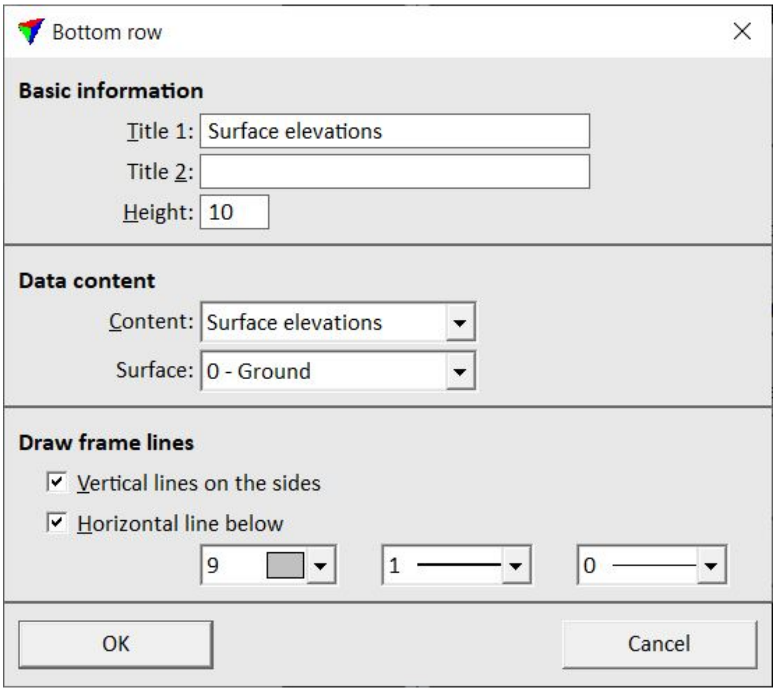
   The **Profile layout** dialog opens:



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 **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 frame lines.

8. Click OK in the **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 modified settings for TerraModeler.

SETTING	EFFECT
Title 1	Free text used as first line of a title in the bottom row.
Title 2	Free 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: • **Surface elevations** - elevations of surfaces of the given **Surface type**. • **Surface difference** - Difference between surfaces of the two given surface types.

SETTING	EFFECT
	• **Pipe** - space reservation for information of a given **Pipe type**. The content is filled by TerraPipe. • **Pipe stations** - space reservation for pipe stationing of a given **Pipe type**. The content is filled by TerraPipe. • **Stationing** - stations along the alignment element of the profile. • **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 given symbology.

The symbology of the bottom row titles below a profile can be set in Profiles / Titles category.

Profiles / Levels category

Levels category in **Profiles** folder defines the levels on which various parts of a profile cell are drawn.

SETTING	EFFECT
Use	Defines how to assign levels for profile cell components: • **Active level** - draws are components on the active level in the CAD file. • **Level settings** - you can assign levels for the different components of the profile cell. Referenced by CAD file level number.

Profiles / Titles category

Titles category in **Profiles** folder defines the symbology of profile titles and bottom row titles.

SETTING	EFFECT
Font	Font type of profile title and scale labels. Uses font types available in the CAD platform.
Size	Text size of profile title and scale labels. Given in millimeters on paper.
Width	Space reserved for bottom row titles on the left side of the profile.
Text color	Text color of bottom row titles. Uses the active color table of the CAD file.
Text weight	Line weight of bottom row titles. Uses CAD file line weights.
Font	Font type of bottom row titles. Uses font types available in the CAD platform.
Size	Text size of bottom row titles. Given in millimeters on paper.

Regions / Region design category

Not Lite, Not UAV

Region design category in **Regions** folder defines the display symbology and boundary settings for regions.

SETTING	EFFECT
Hilite	Color, line style and line weight for displaying a highlighted region. Uses CAD file symbology.
Display intersections	If on, boundaries of intersecting regions are displayed with the given color and line weight.
Dangle extension	Maximum gap that is closed between the end point of one region boundary line and another region boundary line when a region shape is generated.
Curve stroking	Maximum distance between a curve element used as region boundary line and the boundary of the generated region shape.
End point gap	Maximum gap that is closed between the end point of one region boundary line and the end point of another boundary line when a region shape is generated.

Regions / Region levels category

Not Lite, Not UAV

Region levels category in **Regions** folder defines the levels in the CAD file which are used for drawing the different components of regions.

SETTING	EFFECT
Level usage	Levels used for region components, referenced by level number: • **Center marker** - center point of a region. • **Solution shapes** - region shape boundary. • **Title text** - title of a region, CAD text element. • **Patterns** - region shape as polygon with pattern filling. • **Color fill** - region shape as filled-color polygon.

Regions / Region types category

Not Lite, Not UAV

Region types category in **Regions** folder displays a list of types of regions. By default there are two region types defined but more can be added using the **Add** button.

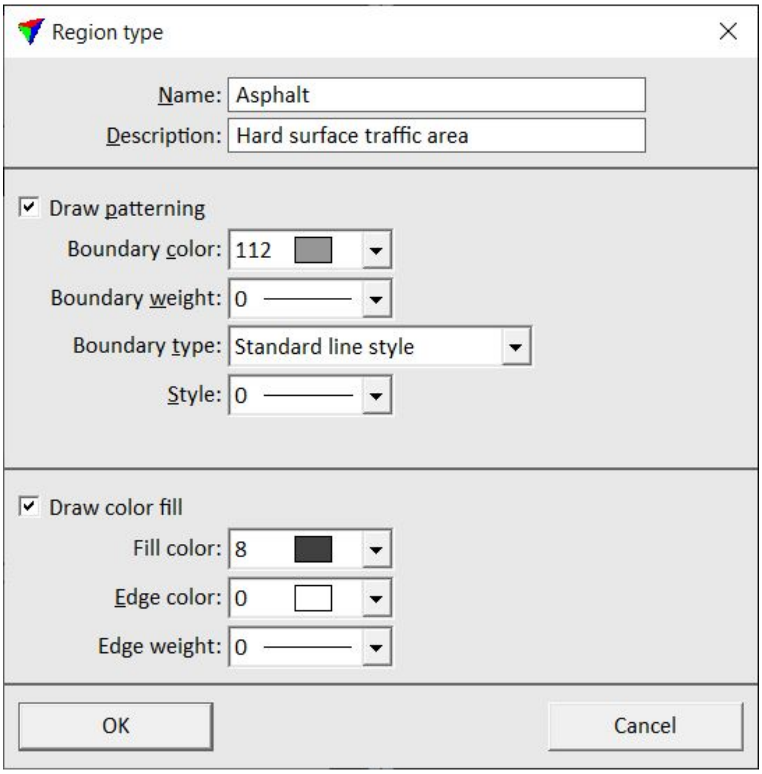
You can edit existing region types by using the **Edit** button in the **Settings** dialog. This opens the **Region type** dialog which lets you change the symbology settings for this region type. With the **Delete** button the selected region type is removed from the list.

To define a new region type:

1. Open the **Region types** category in the **Regions** folder.

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

   This opens the **Region type** dialog:



3. Type a **Name** and (Optional) a **Description** for the region type.

4. Define symbology settings for region pattern. The symbology settings become active if **Draw patterning** is switched on.

   Custom line styles are available only in Bentley CAD.

5. Define symbology settings for region color fill. The symbology settings become active if **Draw color fill** is switched on.

6. Click OK to the **Region type** dialog.

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

Regions / Title formats category

Not Lite, Not UAV

Title formats category in **Regions** folder displays a list of title formats for regions. By default there are two region title formats defined but more can be added using the **Add** button.

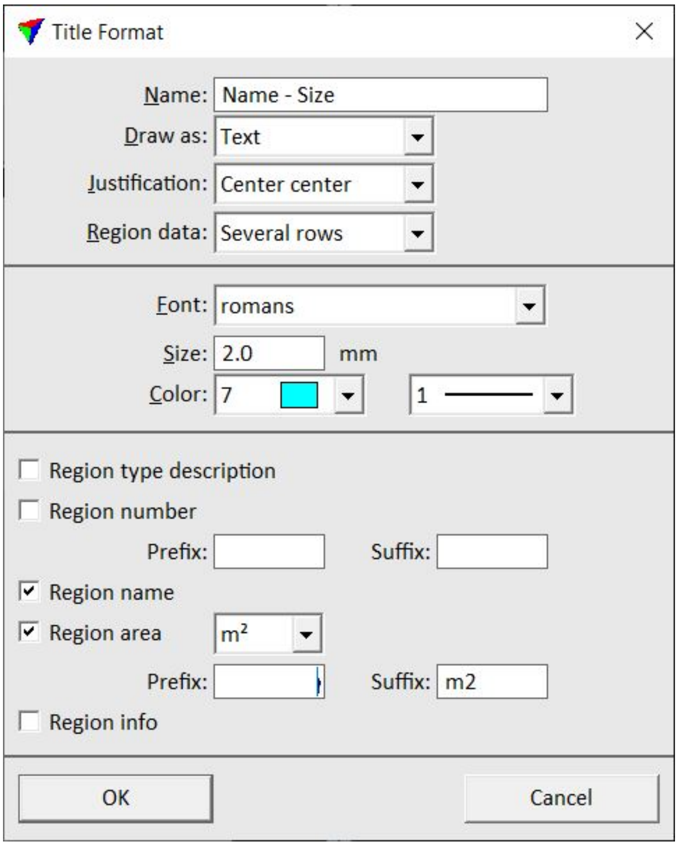
You can edit existing region title formats by using the **Edit** button in the **Settings** dialog. This opens the **Title format** dialog which lets you change the settings for this region title format. With the **Delete** button the selected region title format is removed from the list.

To define a new title format for regions:

1. Open the **Title formats** category in the **Regions** folder.

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

   This opens the **Title Format** dialog:



3. Type a **Name** for the title format and select settings for title placement.

4. Choose symbology settings for the title format.

5. Select auto-text options depending on what you want to display as a region title.

6. Click OK to the **Title Format** dialog.

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

Editable laser model category

Editable laser model category defines the default settings for surface models that are created from point cloud data.

SETTING	EFFECT
Naming	Naming method for new surface models: • **Ask name** - user types a name when a new model is created. • **Automatic** - the new model gets the name given in the **Name** field automatically.
Exclude	Method of exclusion of outer boundary triangles for new surface models: • **No exclusion** - no triangles are excluded. • **Default exclusion** - the software computes a reasonable triangle length for exclusion from the surface's point density. • **By key-in length** - triangles up to the length given in the **Longer than** field are excluded.
Exclude any long triangles	If on, any triangles with edge lengths longer than the value given in the following **Longer than** field are excluded (internal triangles as well as outer boundary triangles).

Element usage category

Element usage category defines how graphical elements are interpreted as points and how text elements are interpreted as elevation points. The settings are used by the Triangulate Survey tool.

SETTING	EFFECT
Radius	Circles smaller than defined in the **Radius** field are interpreted as a single points (circle center). Larger circles are stroked into several points along the circumference.
Accept single character symbols	If on, single character text elements are interpreted as elevation points.
Require +- sign in numbers	If on, numerical text elements must have a sign character in order to be interpreted as elevation labels.
Require decimal point in numbers	If on, numerical text elements must have a decimal point in order to be interpreted as elevation labels.
Accept elevation text cells	If on, MicroStation cell elements are interpreted as elevation marker and labels. The cell origin is used as xy location of the elevation point and the text as elevation label. *Bentley CAD only*

Elevation labels category

Elevation labels category defines the symbology if elevation labels. The settings effect the Insert Point tool, the Label Peaks and Pits tool, the Place Elevation Text tool, and the leveling texts for random and inferred points of the See Export / Graphical elements command.

SETTING	EFFECT
Justify	Location of the placement point (= point of elevation measurement): • **Left**, **Center**, **Right** | **Top**, **Center**, **Bottom** - placement point location relative to the label text box. • **Decimal point** - placement point is located at the decimal point.
Offset dx	Offset between the placement point and the label in x (left-right) direction. Given in millimeters on paper. Only active if **Justify** is not set to **Decimal point**.
Offset dy	Offset between the placement point and the label in y (up-down) direction. Given in millimeters on paper. Only active if **Justify** is not set to **Decimal point**.
Font	Font type of elevation label text elements. Uses font types available in the CAD platform.
Size	Text size of elevation label texts. Given in millimeters on paper.
Accuracy	Number of decimals of elevation labels.
Display plus	If on, the plus sign is displayed for positive elevations.
Display minus	If on, the minus sign is displayed for negative elevations.
Point marker	Type of the placement point: • **None** - no point marker is drawn. • **Character** - a given character is drawn as point marker. • **Zero length line** - a line element of zero length (= point) is drawn as point marker. • **Library cell** - a cell element from the active cell library is drawn as point marker. *Bentley CAD only*
Font	Font type of the point marker character. Uses font types available in the CAD platform. Only active if **Point marker** is set to **Character**.

SETTING	EFFECT
Size	Text size of point marker character. Given in millimeters on paper. Only active if **Point marker** is set to **Character**.
Character	Character used as point marker symbol. Only active if **Point marker** is set to **Character**.
Weight	Weight of the point marker. Uses CAD file line weights. Only active if **Point marker** is set to **Zero length line**.
Cell	Name of the cell used as point marker. Only active if **Point marker** is set to **Library cell**. *Bentley CAD only*
Size	Cell size give in millimeters on paper. Only active if **Point marker** is set to **Library cell**. *Bentley CAD only*

Insert Point category

Insert Point category defines whether Insert Point tool draws elevation labels permanently into the CAD file.

SETTING	EFFECT
Draw permanently	If on, the text element is written to the CAD file. If off, the text element is displayed temporarily.

Lattice database category

Not Lite, Not UAV

Lattice database category defines the storage location and elevation units of lattice database files. Additionally, it defines how lattice database files are drawn in profiles and cross sections.

SETTING	EFFECT
Directory	Defines the directory where lattice database files are located. This directory may contain Intergraph GRD, Disimp or Ordnance Survey NTF files.
Draw in profiles	If on, the lattice database files from the given **Directory** are drawn in profiles and cross sections.
Color	Color of the lattice database files. Uses the active CAD file color table. Only active if **Draw in profiles** is switched on.
Weight	Line weight of the lattice database files. Uses CAD file line weights. Only active if **Draw in profiles** is switched on.
Style	Line style of lattice data base files: • **Standard** - uses CAD file line styles. • **Custom** - uses user-defined line styles. *Bentley CAD only* Only active if **Draw in profiles** is switched on.
Style name	Name of a user-defined line style. The **Select** button opens a list of custom styles. Only active if **Style** is set to **Custom**. *Bentley CAD only*
Style scale	Scale of a user-defined line style. Expressed as relation between the original custom style and a scale factor. Only active if **Style** is set to **Custom**. *Bentley CAD only*
Intergraph	Elevation unit of Intergraph GRD files. Based on that, the software derives decimal elevation values from the integer values stored in the lattice files.
Disimp	Elevation unit of Disimp files. Based on that, the software derives decimal elevation values from the integer values stored in the lattice files.

LEM & DMF formats category

LEM & DMF formats category defines level and format settings for Japanese LEM & DMF format files.

SETTING	EFFECT
Search	Defines where to search for polygons: in the **Active design** file, **Reference files**, or **Both**.
Water bodies	Level in the CAD file(s) where water polygons are located.
Islands	Level in the CAD file(s) where island polygons are located.
Row order	Order of rows in LEM files: **Bottom first** or **Top first**.

Operation category

Operation category defines what menus the application opens at start-up and how the application can be closed.

SETTING	EFFECT
Create Applications Menu	Adds the **Application** pulldown menu to Spatix. This does not have any effect on Bentley CAD platforms.
Open Main tool box	If on, the application opens its **TerraModeler** toolbox at start-up.
Open Surfaces window	If on, the application opens the **Surfaces** window at start-up.
Main tool box is closed	If on, the application is unloaded when the **TerraModeler** toolbox is closed.

Quantity calculation category

Quantity calculation category defines how small elevation differences are ignored in quantity calculation and settings for quantity reports.

SETTING	EFFECT
Less than	Accuracy limit for quantity calculation tools. If the elevation difference between two surfaces is less than the limit, this column is ignored.
Volume	Accuracy of volume values in quantity reports.
Area	Accuracy of area values in quantity report.
Max usage	Maximum amount of memory used for creating quantity reports.

Saving surfaces category

Saving surfaces category defines when surface models are saved automatically to the surface file on a hard disk.

SETTING	EFFECT
When first created	Save surfaces automatically when the surface is first created.
When saving design file	Save surfaces automatically when saving the CAD file.

Surface types category

Surface types category displays a list of user defined surface types. Typical surface types are ground, plan, bedrock, and various soil layers. Whenever you create a new surface, you have to select a surface type from which the surface gets predefined symbology settings for profiles and colored triangles.

You can use the **Edit** button to modify settings for an existing surface type. The button opens the **Surface type** dialog which lets you adjust settings for this surface type. With the **Delete** button an existing surface type can be removed from the list.

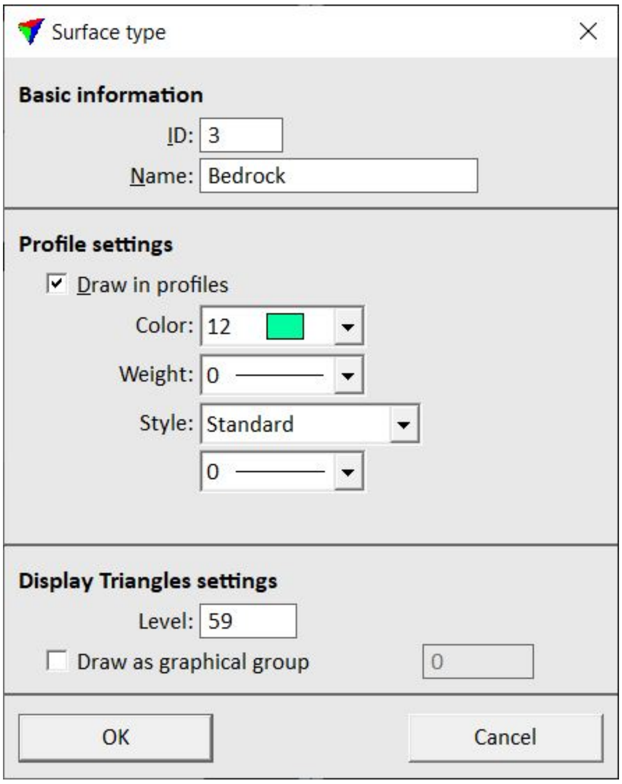
To add or edit a surface type:

1. Open the **Surface types** category. Click **Add** in the **Settings** dialog.

   OR

1. Open the **Surface types** category. Select a surface type definition and click **Edit** in the **Settings** dialog.

   This opens the **Surface type** dialog:



2. Type a **Name** for the surface type.

3. Select symbology settings for drawing the surface type into profiles.

4. Define settings for displaying the surface type as triangles.

5. Click OK to the **Surface type** dialog.

6. Close the **Settings** dialog in order to save the modified settings for TerraModeler.

Triangulate Survey category

Triangulate Survey tool creates a surface model from graphical elements created by a survey drawing or a mapping application. The feature coding of this application is used when deciding what elements can be used as valid source data for the selected surface. Other tools and commands using these settings are Triangulate Multiple Sources, Produce contours, Produce lattice models and Produce triangles.

The settings in **Triangulate Survey** category determine what applications can be called to inquire if an element represents valid data for the selected surface. An application can be called only if it is installed on the computer. An exception is TerraSurvey which does not have to be installed on the computer.

SETTING	EFFECT
TerraSurvey	If on, use elements created by TerraSurvey.
Accept by	How to determine suitability of elements: • **Active feature list** - call TerraSurvey to inquire suitability of an element based on the current feature list. • **Creation time settings** - do not call TerraSurvey. Elements created by TerraSurvey may be accepted for the surface model based on information stored with the elements at creation time.
Scan reference files	If on, the application checks reference files attached to the active CAD file for graphical elements in order to create a surface model.

You do not have to have TerraSurvey installed to use its feature coding to filter elements for a surface model. When TerraSurvey creates survey drawing elements, it stores the active modeling settings of this feature as attribute data. TerraModeler knows how to interpret the data and can select elements based on the creation time modeling settings.

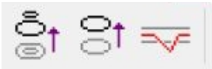
View Elevation category

View Elevation category defines how the **Points on surface** lock in View Elevation tool works if a new element is snapped to another CAD file element.

SETTING	EFFECT
Apply to snapped points	If on, the elevation value derived from the surface given in the View Elevation tool's dialog is applied for all data points, no matter if they are snapped to other 3D elements or not. Thus, the snap determines only the xy location of the data point.

2D Contours toolbox

The **2D Contours** toolbox is used to lift 2D contour elements into 3D or to validate elevations of linear elements.



TO	USE TOOL	
Lift 2D contours using contour labeling		Lift 2D Contours
Set elevation of a contour element		Set Contour Elevation
Validate elevations of linear elements		Check Linear Elevations

Check Linear Elevations

Not Spatix

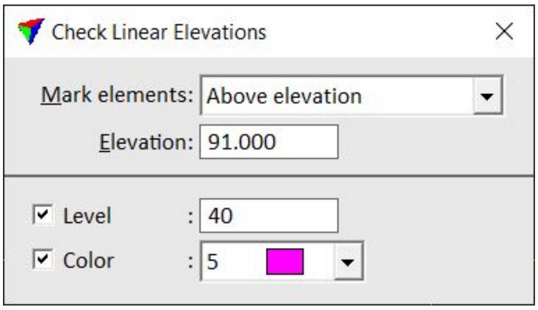
Check Linear Elevations tool validates the elevations of selected linear elements. It marks invalid elements by modifying level and color of the elements.

To check elevations of linear elements:

1. Select element(s) to check.

2. Select the **Check Linear Elevations** tool.

   This opens the **Check Linear Elevations** dialog:



3. Define settings.

4. Accept the selection with a data click.

   This modifies all elements defined by the fence mode that violate the given criteria.

SETTING	EFFECT
Mark elements	Defines invalid elements: • **Above elevation** - mark elements which extend above a given **Elevation**. • **Below elevation** - mark elements which extend below a given **Elevation**. • **With non-uniform elevations** - mark elements which have vertices at more than one elevation.
Elevation	Defines the elevation limit for valid elements. This is only active if **Mark elements** is set to **Above elevation** or **Below elevation**.
Find connected elements	If on, the application tries to find linear elements which connect exactly to the start or the end point of a line element. These connected elements are validated as well. This is only active if **Mark elements** is set to **With non-uniform elevations**.

SETTING	EFFECT
Level	If on, move invalid elements to the given level.
Color	If on, set invalid elements to the given color. Uses the active color table of the CAD platform.

Lift 2D Contours

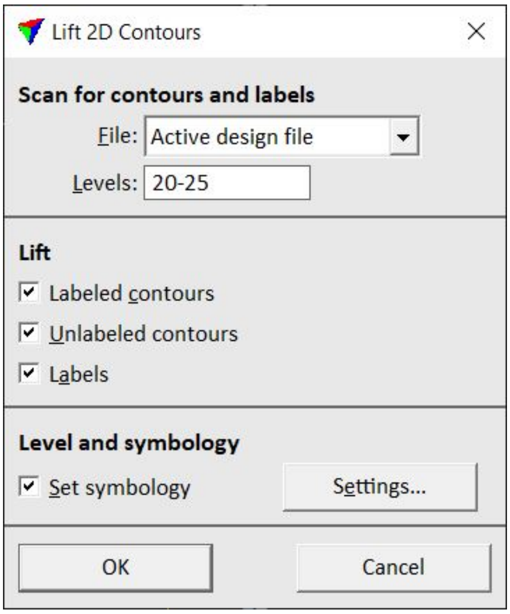
Lift 2D Contours tool converts two-dimensional contour elements into three-dimensional contours. The original elements are copied to their new elevation. The correct contour elevation is determined by a text element used as contour label.

The tool scans the given levels in either the active CAD file or in attached reference files *(Bentley CAD only)* for text and linear elements. It tries to find a matching contour elevation label for each contour chain. A match is found, if the start or the end point of a contour chain is close to an elevation label.

To lift 2D contours into 3D:

1. Select the **Lift 2D Contours** tool.

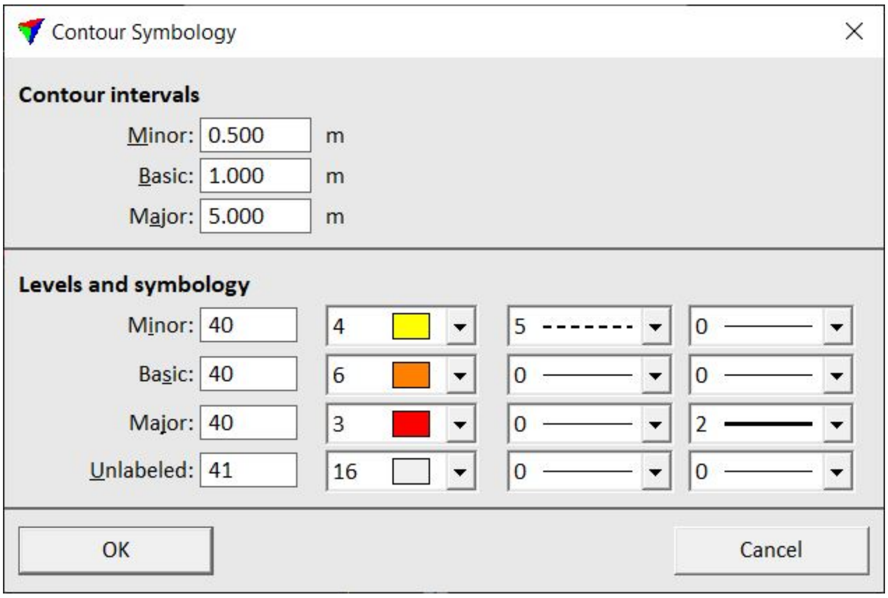
   This opens the **Lift 2D Contours** dialog:



2. Define settings for lifting 2D line and text elements.

3. Click **Settings** to define levels and symbology for lifted contours.

   This opens the **Contour Symbology** dialog:



4. Define settings for **Contour intervals**, **Levels and symbology** (color, line style, line weight).

5. Click OK to close **Contour Symbology** dialog.

6. Click OK in the **Lift 2D Contours** dialog.

TerraModeler scans the given levels in the specified files for text and linear elements. These are drawn at their new elevation in the active CAD file using the given contour symbology.

SETTING	EFFECT
File	Files from which to search for text and linear elements: • **Active design file** - scan the active CAD file. • **Reference files** - scan all reference files with Locate setting on in Bentley CAD Reference file manager. *(Bentley CAD only)*
Levels	List of levels from which to scan for elements. For example: • **61** - scan level 61. • **15,21-24** - scan levels 15, 21, 22, 23 and 24.
Labeled contours	If on, lift contours for which a matching label is available.
Unlabeled contours	If on, copy linear elements for which no elevation value is found. The elements are copied to zero elevation.
Labels	If on, the contour label text elements are lifted with the contour lines.
Set symbology	If on, modify the symbology of the lifted elements according to settings defined using

SETTING	EFFECT
	the Settings button.

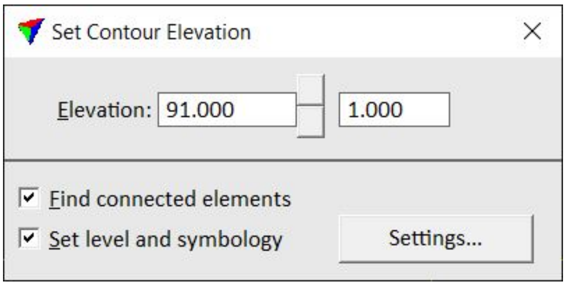
Set Contour Elevation

Set Contour Elevation tool lifts a linear element to a given elevation. It is designed to be used for lifting single two-dimensional contour elements to a correct elevation. It can be applied to line, shape and text elements. The original elements are moved to their new elevation.

To set elevation of contour element(s):

1. Select the **Set Contour Elevation** tool.

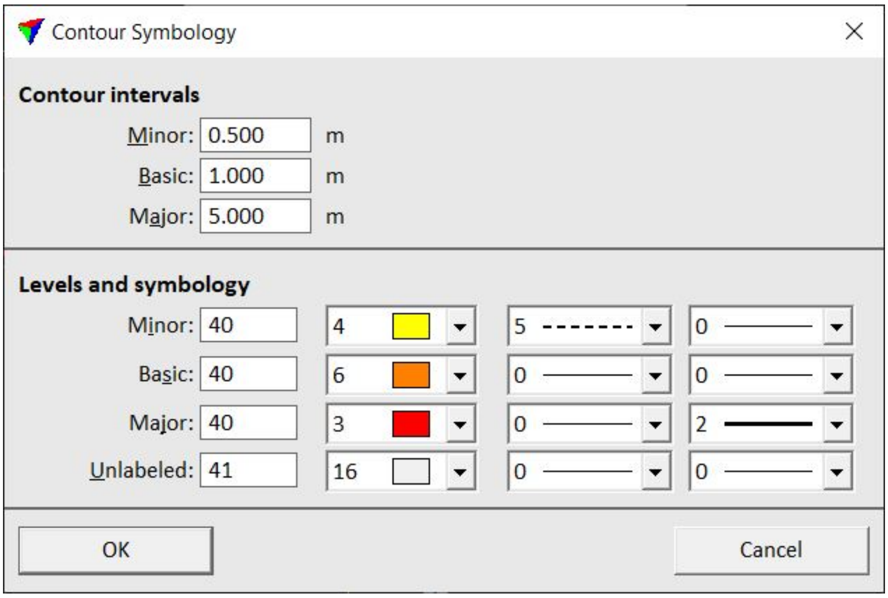
   The **Set Contour Elevation** dialog opens:



2. Set an elevation value in the **Elevation** field.

3. Click **Settings** to define level and symbology for the lifted contour.

   This opens the **Contour Symbology** dialog:



4. Define settings for **Contour intervals**, **Levels and symbology** (color, line style, line weight).

5. Click OK to close **Contour Symbology** dialog.

6. Identify the element to lift with a data click.

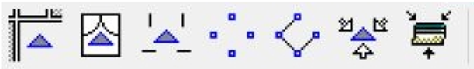
7. Accept the element with another data click.

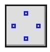
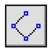
The element is lifted to the given elevation. You can continue with steps 2, 3, or 6.

SETTING	EFFECT
Elevation	Elevation value for the lifted contour element. Use the arrow buttons to change to a new elevation value in steps defined next to the arrow buttons.
Find connected elements	If on, the application tries to find linear elements which connect exactly to the start or the end point of the identified line element. These connected elements are lifted as well.
Set level and symbology	If on, modify the symbology of lifted elements according to settings defined using **Settings** button.

Create Surfaces toolbox

The tools in the **Create Surfaces** toolbox are used create surface models or to add new points to an existing surface model.



TO	USE TOOL	
Triangulate survey elements		Triangulate Survey
Triangulate elements visible in a view		Triangulate View
Triangulate elements by type and level		Triangulate Elements
Insert elements as random points		Insert Random Elements
Insert elements as breaklines		Insert Breakline Element
Triangulate breaklines and/or laser points		Triangulate Multiple Sources
Rebuild models		Rebuild Models

TerraModeler surface models are stored as triangulated surface files. A description of the file format can be found in Chapter TIN File Format Specification.

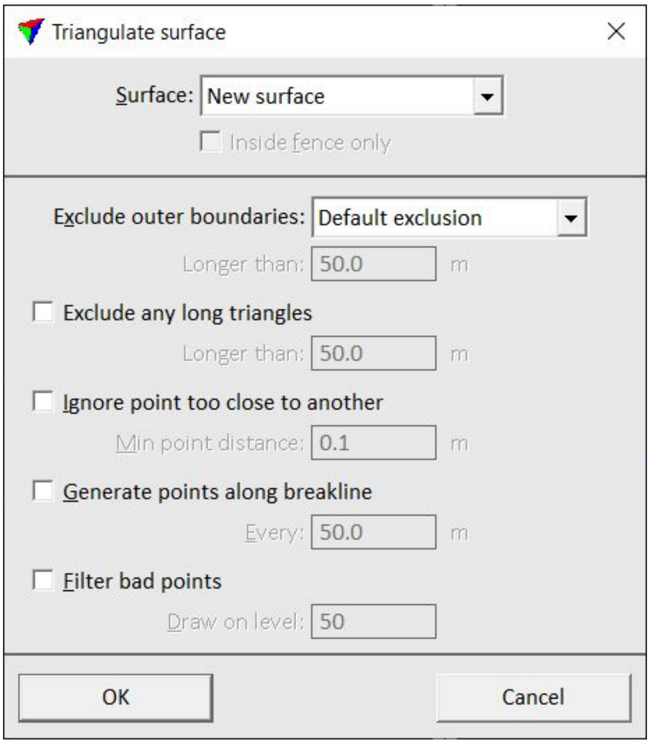
Creating a Surface Model

TerraModeler provides a number of tools for surface model creation. The source data may be different as well as the first steps of the creation workflow. They are described for the individual tools. At some point, the workflow becomes the same for all tools. The steps described here are the same for all model creation processes.

To continue creating a surface model:

1. Follow the steps for the individual tools to start the surface model creation.

   After the tool-specific steps, the **Triangulate surface** dialog opens:



2. Select the surface to be triangulated. You can select a new surface or an existing surface that is loaded in TerraModeler.

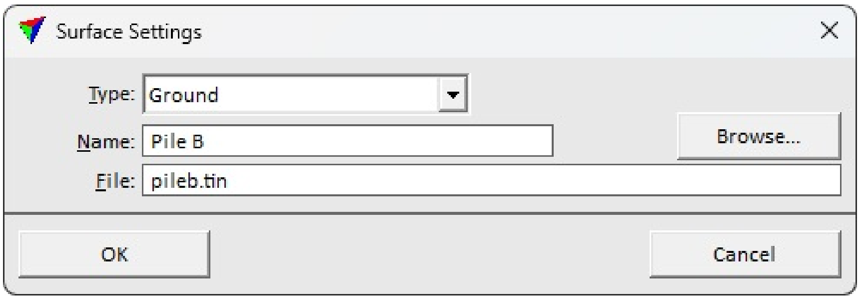
3. If required, select additional settings for excluding triangles and point handling.

4. Click OK.

SETTING	EFFECT
Surface	Determines how the surface is created: • **New surface** - a new surface is created. • ***<surface name>*** - new features are added to an existing surface model that is loaded in TerraModeler.

SETTING	EFFECT
Create as profile model	If on, the surface is created from a centerline element. TerraModeler internally copies the element to the left and right, and uses these elements for surface creation.
Exclude outer boundaries	Exclusion method of outer boundary triangles of the surface model: • **No exclusion** - no triangles are excluded. • **Default exclusion** - the software computes a reasonable triangle length for exclusion from the surface's point density. • **By key-in length** - triangles with edge lengths given in the subsequent **Longer than** field are excluded.
Exclude any long triangles	If on, any triangles with edge lengths longer than the value given in the subsequent **Longer than** field are excluded.
Ignore point too close to another	If on, the model is thinned by leaving out unnecessary points. If the distance between two points is less than **Minimum point distance**, one of the points is left out.
Generate points along breakline	If on, the model is improved by generating new points along any long breaklines. Points are generated at a distance given in the **Every** field.
Filter error points	If on, probable survey errors are filtered out from the model. Points that have been filtered out are drawn as small circles on the level defined in the **Draw on level** field.

If a new surface is created, the software opens the **Surface Settings** dialog:



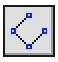
5. Select a surface **Type** and type a **Name** for the surface.

6. (Optional) Rename the **File** for storing the model on the hard disk. **Browse** storage destination for saving the model.

7. Click OK.

TerraModeler creates the surface model. A progress bar shows the process. After the process is finished, the time needed for the creation process is displayed in the status bar of the CAD platform.

Insert Breakline Element



Insert Breakline Element tool adds graphical elements as breakline points to a surface model. You can use this tool to create a surface model or to add points to an existing model loaded in TerraModeler.

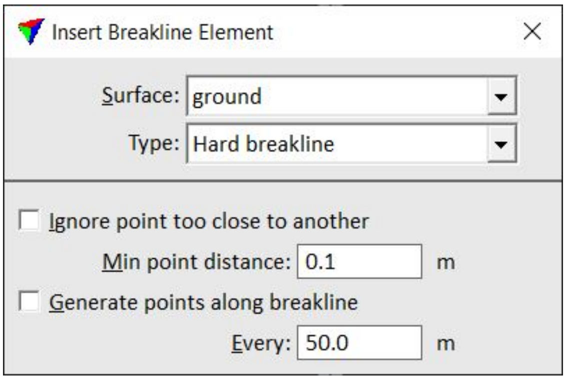
Before inserting a 3D breakline element to an existing surface model, the line element representing the breakline must be dropped to the surface elevation. This should be done with the Drop element to surface tool in order to ensure that the software can recognize the intersection points between the existing TIN and the new breakline element. Thus, TIN edges are broken correctly by the new breakline.

In general, it might be more convenient to create a new surface model from all available sources instead of inserting new elements to an already existing model.

To insert element(s) as breakline points to a surface:

1. Select the **Insert Breakline Element** tool.

   This opens the **Insert Breakline Element** dialog:



2. Select a **Surface** and a breakline **Type**.

3. (Optional) Define additional settings related to minimum distance between points or generation of additional points along the breakline.

4. Identify the element to insert with a data click.

5. Accept the element with another data click.

   TerraModeler inserts the element to the surface. You can continue to step 4.

   OR

1. Select element(s) to insert.

2. Select the **Insert Breakline Elements** tool.

   This opens the **Insert Breakline Element** dialog.

3. Define settings.

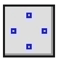
4. Accept the selected element(s) with a data click.

   TerraModeler inserts the element(s) to the surface.

   If a new surface is created, the **Triangulate surface** dialog opens. Follow the common steps for Creating a surface model.

SETTING	EFFECT
Surface	Surface to which breakline points are inserted: • **New surface** - a new surface is created. • <***surface name***> - points are added to an existing surface model that is loaded in TerraModeler.
Type	Defines how the element vertices are used as points in the surface model. See Breakline types.
Ignore point too close to another	If on, filter out unnecessary points. If the distance between two points is less than **Minimum point distance**, one of the points is left out.
Generate points along breakline	If on, the model is improved by generating new points along any long breaklines. Points are generated at a distance given in the **Every** field.

Insert Random Elements

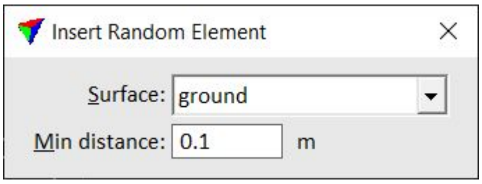


Insert Random Elements tool adds graphical elements as random points to a surface model. You can use this tool to create a surface model or to add points to an existing model loaded in TerraModeler.

To insert element(s) as random points to a surface:

1. Select the **Insert Random Elements** tool.

   This opens the **Insert Random Element** dialog:



2. Select a **Surface** and define a **Minimum distance** between points.

3. Identify the element to insert with a data click.

4. Accept the element with another data click.

   TerraModeler inserts the element to the surface. You can continue with step 3.

   OR

1. Select element(s) to insert.

2. Select the Insert **Random Elements** tool.

   This opens the **Insert Random Element** dialog.

3. Define settings.

4. Accept the selected element(s) with a data click.

   TerraModeler inserts the element(s) to the surface.

   If a new surface is created, the **Triangulate surface** dialog opens. Follow the common steps for Creating a surface model.

SETTING	EFFECT
Surface	Surface to which points are inserted: • **New surface** - a new surface is created. • ***<surface name>*** - points are added to an existing surface model that is loaded in TerraModeler.
Minimum distance	Thins the model by leaving out unnecessary points. If the distance between two points is

SETTING	EFFECT
	less than **Minimum distance**, one of the points is left out.

Rebuild Models

Rebuild Models tool is used to rebuild surface models that were created by the Triangulate Multiple Sources tool. The models must have been created as **Rebuildable models**.

The rebuild process deletes the old surface, reads the source information, triangulates the new surface, and redraws the display methods.

To rebuild a model:

1. Select the **Rebuild Models** tool.

   If there are one or more rebuildable models loaded in TerraModeler, the rebuilding process starts immediately for these models.

   If there is no rebuildable model loaded in TerraModeler, an information message appears.

Triangulate Elements

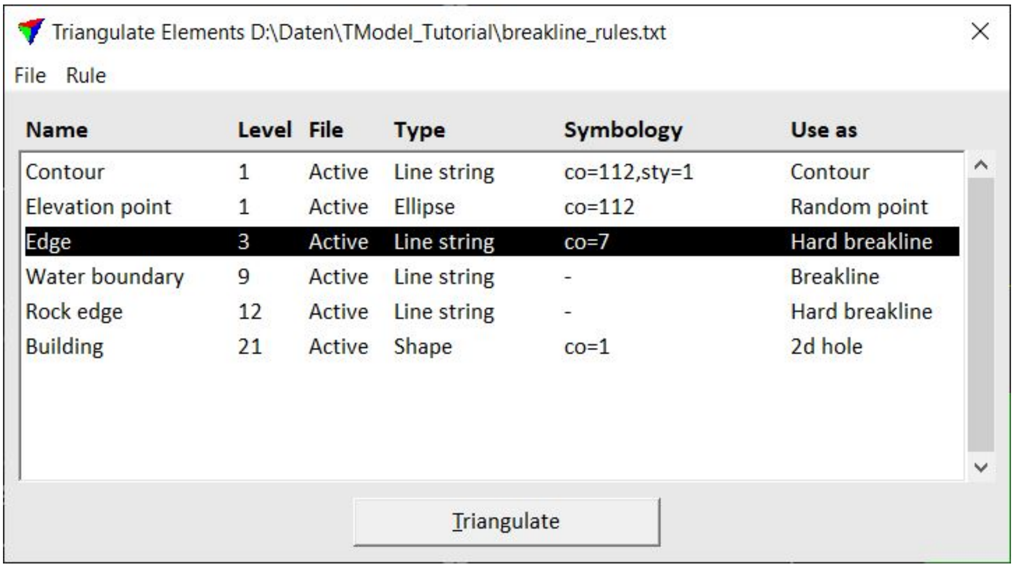
Triangulate Elements tool creates a surface model from graphical elements filtered by level, element type, and symbology. You can save the filtering rules to a text file for later use.

You can use this tool to create a surface model or to add points to an existing model loaded in TerraModeler.

General procedure for triangulating elements by level and type:

1. Select the **Triangulate Elements** tool.

   The **Triangulate Elements** window opens:



2. Create a list of Filtering rules for elements to be included in the model. Alternatively, you can load a previously saved set of rules from a file on the hard disk.

3. Click **Triangulate**.

   This opens the **Triangulate surface** dialog. Follow the common steps for Creating a surface model.

Filtering rules

The **Triangulate Elements** window lists filtering rules for graphical elements in a CAD file. Each filtering rule defines one element type located on a specific level to be accepted for the model. Points extracted from the element can be used as random, breakline, contour, hole or boundary points.

The filtering rules can be created manually or automatically by scanning the CAD file.

To save filtering rules into a text file, select **Save as** from the **File** pulldown menu.

To open a set of filtering rules from a file, select **Load** from the **File** pulldown menu.

To delete active rules or to create a new rule file, select **New** from the **File** pulldown menu. To delete a selected rule from the list, select **Delete** from the **Rule** pulldown menu.

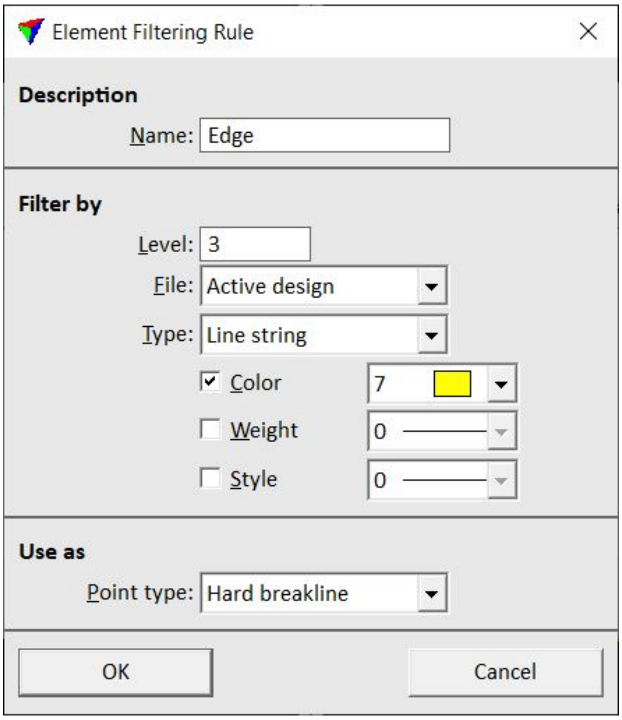
To add or edit a filtering rule to the list:

1. Select **Add** command from the **Rule** pulldown menu in the **Triangulate Elements** window.

   OR

1. Select an existing rule. Select **Edit** command from the **Rule** pulldown menu in the **Triangulate Elements** window.

   This opens the **Element Filtering Rule** dialog:



2. Type a **Name** for the rule.

3. Select settings for defining the rule.

4. Click OK.

   This adds the rule to the list or modifies the existing rule.

SETTING	EFFECT
Name	Description of the feature depicted by the elements.
Level	The rule applies to elements on this CAD file level.

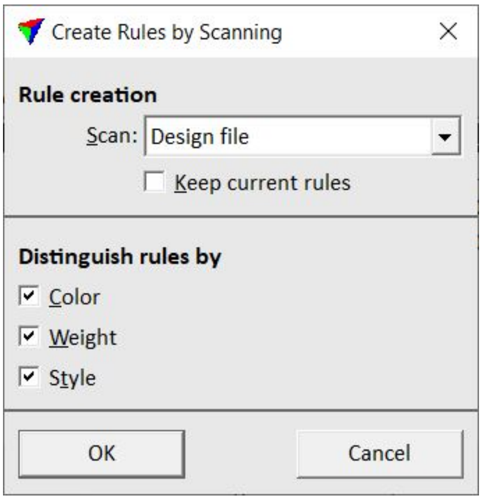
SETTING	EFFECT
Type	Element type the rule applies to.
Color	If on, the rule applies only to elements with a given line **Color**.
Weight	If on, the rule applies only to elements with a given line **Weight**.
Style	If on, the rule applies only to elements with a given line **Style**.
Point type	Defines how the element vertices are used as points in the surface model. See Breakline types.

You can use the **Selection** tool as an aid for creating new rules. Select an example element before choosing **Add** command from the **Rule** pulldown menu. This applies the type and symbology of the selected element to the **Element Filtering Rule** dialog.

To add rules to the list by scanning the CAD file:

1. Select **Create by scanning** command from the **File** pulldown menu in the **Triangulate Elements** window.

   This opens the **Create Rules by Scanning** dialog:



2. Select settings and click OK.

3. If **Scan** is set to **View contents**, select the view with a data click.

   The software adds the rules automatically to the list in the **Triangulate Elements** window. You may edit the rules, add additional rules or delete unnecessary rules.

SETTING	EFFECT
Scan	Defines where the software scans elements to create the rules: • **Design file** - in the active CAD file. • **Design and references** - in the active CAD file and in reference files attached to the active CAD file. *References are applicable in Bentley CAD only.* • **View contents** - in a selected view. Only elements visible in the selected view are used for rule creation.
Keep current rules	If on, the new rules are added to the existing rules. If off, existing rules are deleted.
Color	If on, rules are distinguished by element color.
Weight	If on, rules are distinguished by element weight.
Style	If on, rules are distinguished by element style.

Triangulate Multiple Sources

Triangulate Multiple Sources tool creates a new surface model from breakline elements and/or point cloud data in one operation. The surface model can be created as:

- **Normal model** - the model can not be rebuild when the source data has been changed.
- **Rebuildable model** - the surface model can be rebuild after the source data has been changed by using the Rebuild Models tool.
- **Editable model** - the surface model is updated if there are modifications of the laser points used in the model. This requires the usage of points loaded in TerraScan.

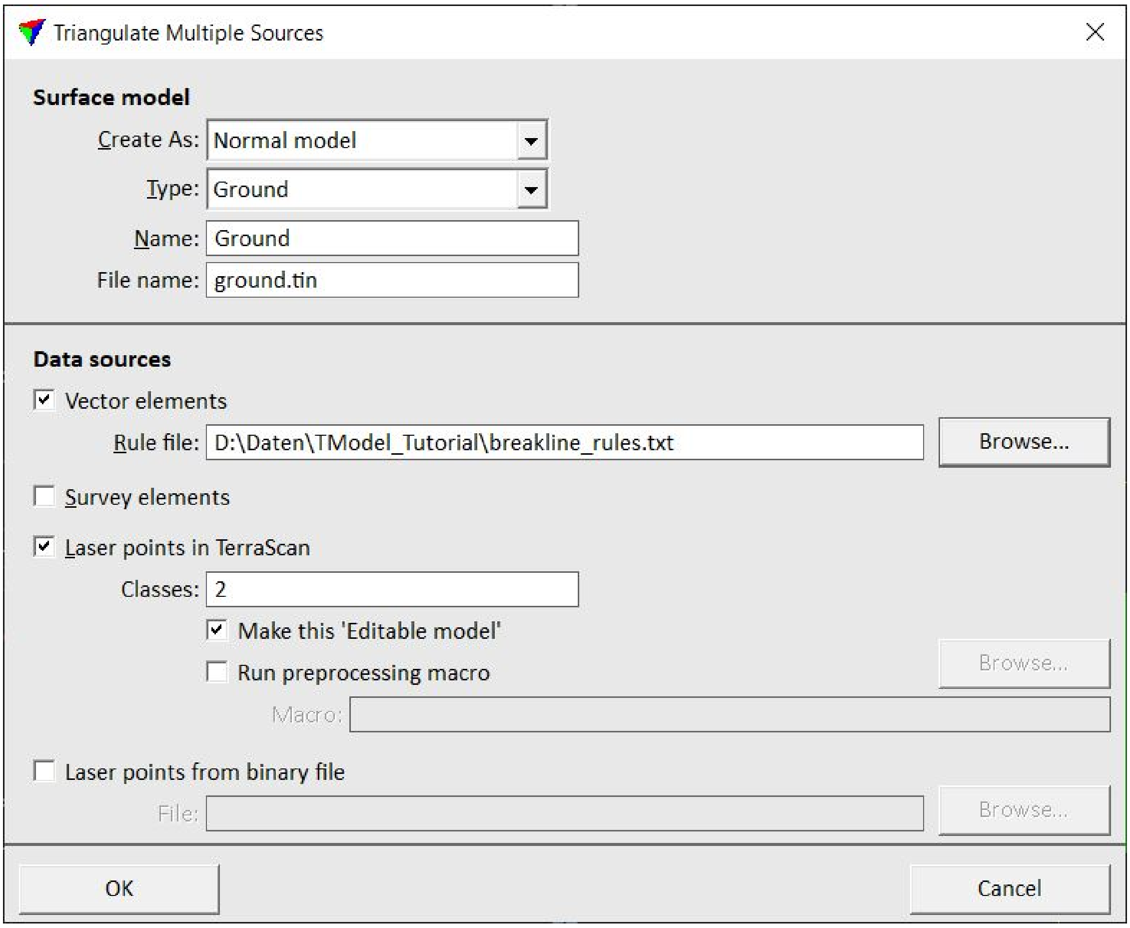
The breaklines can be defined as vector elements filtered by rules. See Filtering rules for more information about rule files and how to create them. Alternatively, breaklines can be derived from survey elements created by TerraSurvey or other applications enabled in Triangulate Survey category of the TerraModeler **Settings**.

Point cloud data can be used from points loaded in TerraScan or directly from a TerraScan binary file stored on a hard disk.

To create a surface model from multiple sources:

1. Select the **Triangulate Multiple Sources** tool.

   This opens the **Triangulate Multiple Sources** dialog:



2. Define settings for the surface model creation.

3. Select data sources and define settings.

4. Click OK.

This opens the **Triangulate surface** dialog. Follow the common steps for Creating a surface model.

SETTING	EFFECT
Create As	Defines surface model rebuild options: • **Normal model** - the model can not be rebuild. • **Rebuildable model** - the model can be rebuild after changes to the source data using the Rebuild Models tool.
Type	Surface model type.
Name	Name of the new surface model.
File name	Name of the surface model file stored on the hard disk.
Vector elements	If on, breaklines are included in the model. The breaklines are defined by vector elements and the given **Rule file**.
Survey elements	If on, breaklines are included in the model. The breaklines are defined by TerraSurvey or any other application enabled in TerraModeler **Settings**.
Laser points in TerraScan	If on, laser points loaded in TerraScan are included in the model. Only points of the given **Classes** are included.
Make this 'Editable model'	If on, the model created from the laser points is updated if the point class changes. Only active if **Laser points in TerraScan** is switched on.
Run preprocessing macro	If on, the given TerraScan **Macro** is applied to the loaded points before the surface model is created. Only active if **Laser points in TerraScan** is switched on.
Laser points from binary file	If on, laser points from the given TerraScan binary **File** stored on a hard disk are included in the model.

Triangulate Survey

Triangulate Survey tool creates a surface model from survey data that has been drawn into the CAD file with survey drawing or mapping applications. The feature coding of this other application is used to decide what elements can be accepted as valid data for the TerraModeler surface model.

The elements may originate from one of the supported applications:

- **TerraSurvey** - a survey drawing application developed by Terrasolid.
- **PRO600** - a mapping application developed by Leica.
- **DigiCad** - a map digitizing application developed by GeoCad.
- **Espa** - a mapping application developed by ESPA Systems Ltd.

The applications must be activated in Triangulate Survey category of TerraModeler **Settings** before the software can create surface models from the respective survey data. More information about the implementation of functions related to survey feature coding in other applications can be found in Section Triangulate Survey.

You can use this tool to create a new surface model or to add points to an existing model loaded in TerraModeler.

To triangulate survey elements:

1. Select the **Triangulate Survey** tool.

   The **Triangulate surface** dialog opens. Follow the common steps for Creating a surface model.

   TerraModeler scans the CAD file for survey elements and uses those to create a model.

Triangulate View

Triangulate View tool creates a surface model from graphical elements displayed in a view. This tool uses only elements residing on levels visible in the selected view and located inside the view range.

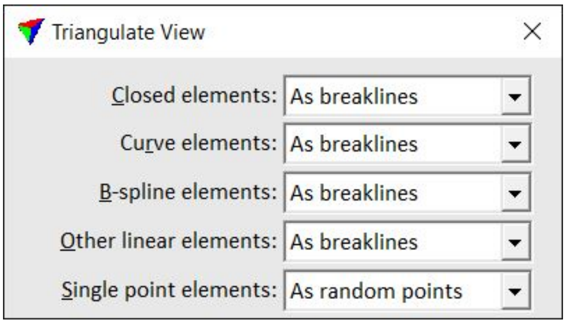
This tool includes closed elements as breakline points or holes, linear elements as breakline or contour points, and single point elements as random points into a surface model.

You can use this tool to create a new surface model or to add points to an existing model loaded in TerraModeler.

To triangulate view elements:

1. Select the **Triangulate View** tool.

   This opens the **Triangulate View** dialog:

2. Identify the view to triangulate by a mouse click inside the view.

   This opens the **Triangulate surface** dialog. Follow the common steps for Creating a surface model.

   The software creates the surface model from elements that are visible in the selected view.

SETTING	EFFECT
Closed elements	How to use closed elements: **As breaklines** , **As 3D holes** or **As 2D holes**.
Curve elements	How to use curve elements: **As breaklines** or **As contours**.
B-spline elements	How to use B-splines: **As breaklines** or **As contours**.
Other linear elements	Other linear elements are always used as breaklines.
Single point elements	Single point elements are always used as random points.

SETTING	EFFECT
Scan reference files	If on, elements in reference files that are attached to the active CAD file are used for surface creation as well.

Display Regions toolbox

Not Lite

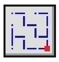
The tools in the **Display Regions** toolbox are used to generate region-based displays from a surface model.

These display methods rely on fairly complex computations which are performed when you start a tool. The created display elements are not updated after modifications of a surface if the tool is run again or by the **Update Displays** tool.

TO	USE TOOL	
Display rain water flow on surface		Display Drainage *Not Lite*
Display theme polygons colored by elevation, slope, or domain		Display Themes *Not Lite, Not UAV*
Display triangles colored by region boundaries		Display Region Triangles *Not Lite, Not UAV*
Display triangles with raster materials		Display Raster Triangles *Not Lite*

Display Drainage

Not Lite



Display Drainage tool computes and displays the flow of drainage water on a surface model. It uses a grid-based approach to determine the water flow.

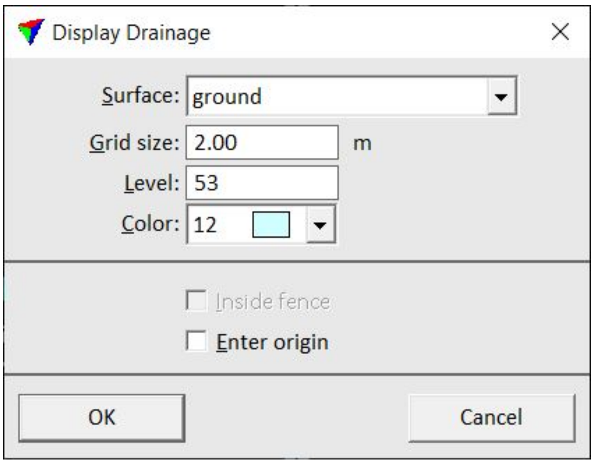
The process starts by placing an imaginary rain drop at each grid cell. Then the software determines where that rain drop would travel. It compares the elevation of the cell center point with the elevation of each of the neighboring cells. If a lower elevation is found, the rain drop travels to the neighboring cell to which the slope is steepest. This process continues until each of the rain drops have reached a pit or a local minimum elevation point.

Display Drainage tool creates CAD file elements which are not linked to a surface model. Thus, TerraModeler is not able to update or erase elements created by this tool.

To display drainage:

1. Select the **Display Drainage** tool.

   The **Display Drainage** dialog opens:



2. Define settings and click OK.

3. If **Enter origin** is switched on, define the origin point of a drainage grid cell with a data point.

   This computes the drainage water flow and displays the results temporarily in all open views. The display is updated every time when the view contents is updated with the corresponding CAD tool.

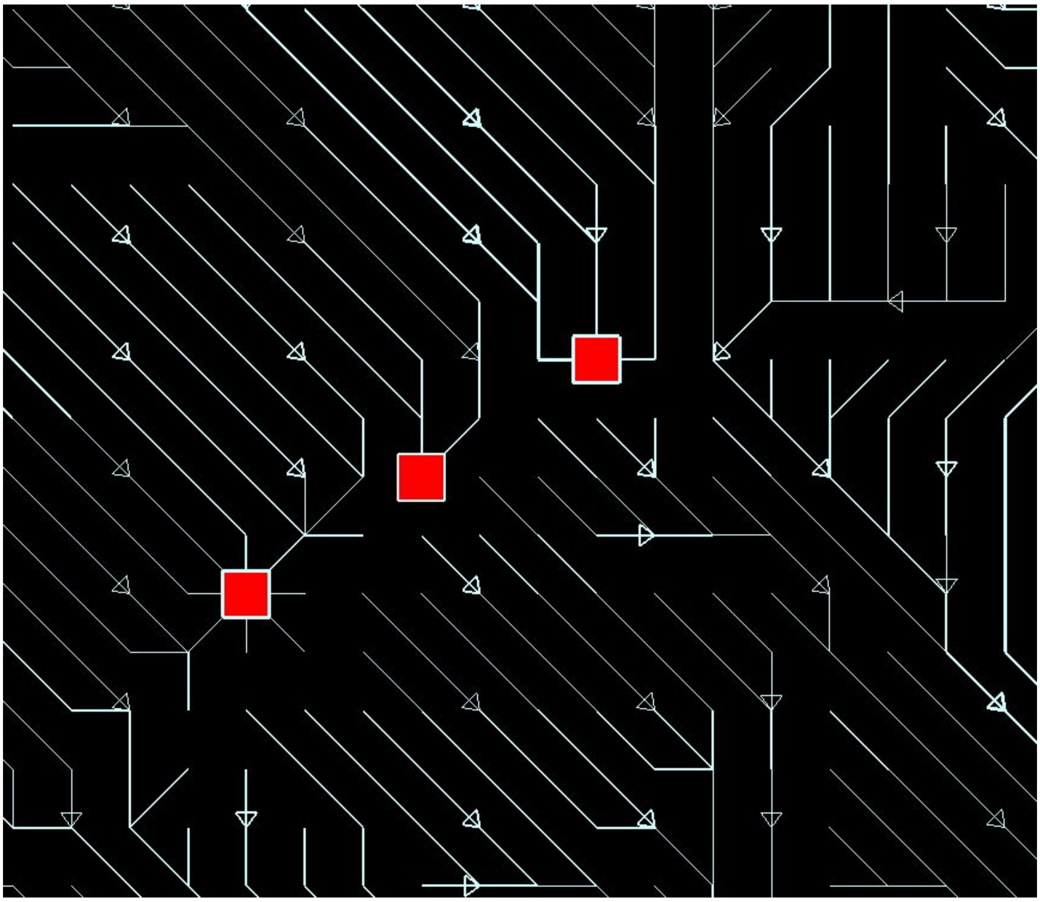
   In addition, the **Drainage Detail** dialog opens for Viewing the drainage system in detail.

SETTING	EFFECT
Surface	Name of the effected surface model.
Grid size	Size of the grid cells for drainage computation.

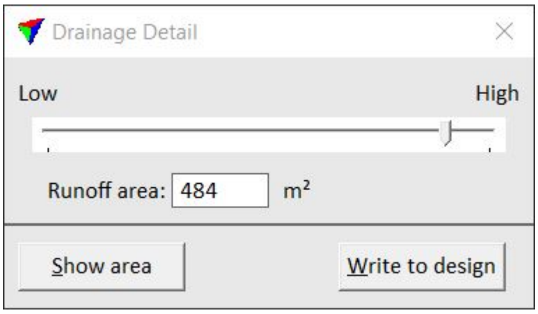
SETTING	EFFECT
Level	Number of the level in the CAD file on which the drainage network is drawn if it is drawn permanently into the CAD file.
Color	Color of the lines and arrows the represent the water flow. Uses the CAD file color table.
Inside fence	If on, the drainage system is only displayed inside a fence. This is only active if a polygon has been selected or a fence has been drawn before the tool is started.
Enter origin	If on, you can enter the origin point of a drainage grid cell with another data point.

Viewing the drainage system in detail

The drainage display shows the paths that rain drops would travel according to the grid-based calculation. The arrows show the flow direction and the line width gives an indication on how many rain drops travel along a path. Red squares mark places where drainage water would gather (outlet, pour point). As the rain drops can not travel outside the surface model, there may be several squares at the outer edges of the model.



The **Drainage Detail** dialog allows you to control the level of detail in the drainage display.



When the slider is set to **High**, the drainage display shows the paths of each single rain drop. If you move the slider closer to **Low**, the software leaves out smaller streams and only displays the major paths. The drainage network remains visible as long as you keep the **Drainage Detail** dialog open.

In addition, the **Show area** button in the **Drainage Detail** dialog can be used to identify the runoff area for a selected drainage grid cell.

Click on the **Write to design** button in order to draw the drainage display permanently into the CAD file.

To view a runoff area:

1. Click on the **Show area** button in the **Drainage Detail** dialog.

2. Move the mouse inside a view.

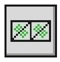
   The size of the **Runoff area** for the grid square at the mouse pointer location is displayed in the **Drainage Detail** dialog. This is the combined surface area of all the squares from which a rain drop travels to or through the selected grid cell.

3. (Optional) Enter a data point in order to display the boundaries of the runoff area.

   This draws the runoff area as a shape element into the CAD file using the active symbology.

Display Raster Triangles

Not Lite



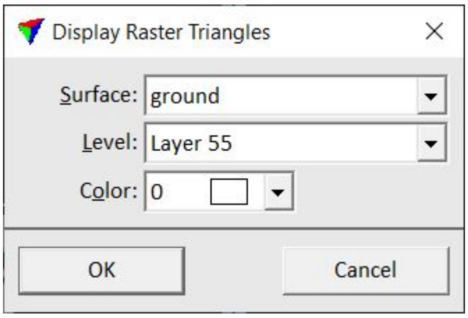
Display Raster Triangles tool displays triangles in a manner suitable for rendered images. The triangles are used by TerraPhoto in order to drape images on a surface model in rendered views. The images have to be attached as raster references in TerraPhoto.

The raster triangles are not drawn permanently into the CAD file and they remain linked to the surface model. This allows TerraModeler to update the display after modifications and to erase the display using the Erase Display tool.

To display raster triangles:

1. Select the **Display Raster Triangles** tool.

   This opens the **Display Raster Triangles** dialog:



2. Define settings and click OK.

   This displays the triangles. You can now use TerraPhoto tools to create rendered views.

SETTING	EFFECT
Surface	Name of the effected surface model.
Level	Name of the level in the CAD file on which the raster triangles are drawn.
Color	Color used to draw the raster triangles. Uses the active CAD file color table.

Display Region Triangles

Not Lite, Not UAV

Display Region Triangles tool displays the triangles of a surface model. The triangles are colored by regions which are defined by closed shape elements. The triangles and the parts of triangles falling inside each region are drawn using the color of the boundary shape. The tool may be used to visualize, for example, land use regions, soil type regions or property boundaries.

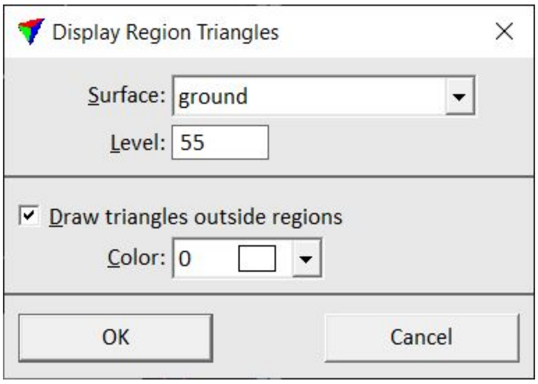
Display Region Triangles tool creates CAD file elements which are not linked to a surface model. Thus, TerraModeler is not able to update or erase elements created by this tool.

To display region triangles:

1. Select region boundary shapes.

2. Select the **Display Region Triangles** tool.

   This opens the **Display Region Triangles** dialog:



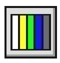
3. Define settings and click OK.

   This writes triangles colored by region into the CAD file.

SETTING	EFFECT
Surface	Name of the effected surface model.
Level	Number of the level in the CAD file on which region triangles are drawn.
Draw triangles outside regions	If on, triangles which are outside all selected boundary shapes are drawn using the given **Color**.

Display Themes

Not Lite, Not UAV



Display Themes tool creates colored shape elements to display theme areas in a surface model. Each theme area can represent an elevation range, a slope range, or a domain which has been assigned a distinctive color.

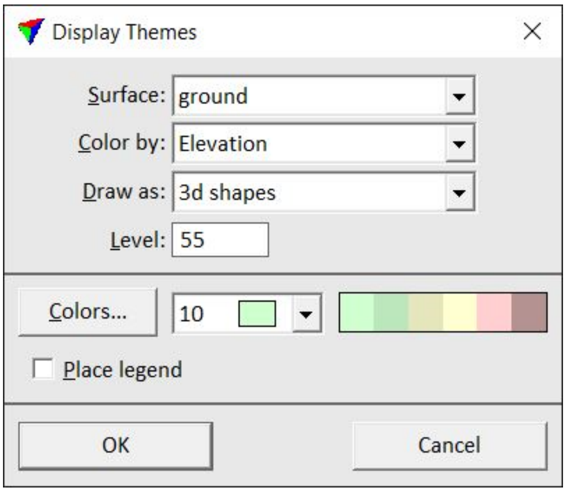
The result of theme coloring is very similar to the visualization created by Display Triangles tool. Both create elevation-based representations of a surface model which may look exactly the same when viewed in a top view. Display Themes tool has the advantage of creating only one shape element for each elevation or slope range. This results in a much smaller CAD file size compared with the triangle visualization approach which is advantageous especially in dynamic views.

Display Themes tool creates CAD file elements which are not linked to a surface model. Thus, TerraModeler is not able to update or erase elements created by this tool.

To display theme areas:

1. Select the **Display Themes** tool.

   The **Display Themes** dialog opens:



2. Define settings and click OK.

   The theme areas are drawn as filled shape elements into the CAD file.

3. If **Place legend** is switched on, define the location of drawing the legend with another data point.

SETTING	EFFECT
Surface	Name of the effected surface model.
Color by	Defines the content of the theme areas:

SETTING	EFFECT
	• **Elevation** - each theme area represents an elevation range. • **Slope** - each theme area represent a slope range. • **Domain** - creates theme polygons from neighboring triangles assigned to the same domain. See Define Domains tool for more information.
Draw as	Defines the shape type of theme areas: • **3d shapes** - the shapes follow the surface model elevation. • **2d shapes** - the shapes are plane and drawn at zero elevation.
Level	Number of the level in the CAD file on which the theme area shapes are drawn.
Colors	Opens the **Color scheme** dialog. See Creating a color scheme of discrete colors for more information.
Color list left of the color scheme preview	Opens the CAD file color table for single color selection.
Place legend	If on, a legend for the theme colors can be placed in the CAD file. This is only active if a color scheme is defined.

Display Single toolbox

Not Lite

The tools in the **Display Single** toolbox are used to place single contours, elevation texts, or slope arrows. There is another tool for modifying the symbology of contour lines.

TO	USE TOOL	
Place a contour at a single elevation		Place Contour
Place an elevation text		Place Elevation Text
Place a slope arrow		Place Slope Arrow
Modify symbology of contours at given intervals		Modify Contour Symbology

Modify Contour Symbology

Not Lite

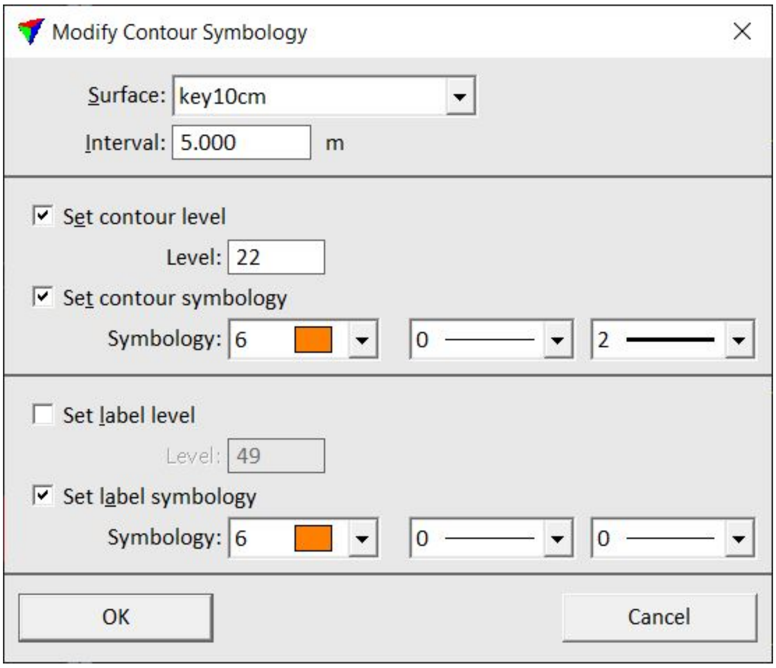
Modify Contour Symbology tool modifies the symbology of contour lines that are already drawn in the CAD file. It can also move contour lines to another level in the CAD file. The contour lines have to be created with **Mode** set to **Write to file** in the Display Contours tool in order to apply the tool.

The tool may be useful, for example, if contour lines need to be distinguished into more than three types that are supported by the Display Contours tool.

To modify the symbology of contours:

1. Select the **Modify Contour Symbology** tool.

   This opens the **Modify Contour Symbology** dialog:



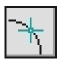
2. Define settings and click OK.

SETTING	EFFECT
Surface	Name of the effected surface model.
Interval	Interval of contour lines that are modified.
Set contour level	If on, the contour lines are moved to the given CAD file **Level**.
Set contour symbology	If on, the color, style and weight of the contour lines are modified to the given

SETTING	EFFECT
	Symbology. Uses the active color table of the CAD file, and CAD file line styles and weights.
Set label level	If on, the contour labels are moved to the given CAD file **Level**.
Set label symbology	If on, the color, style, and weight of the contour labels are modified to the given **Symbology**. Uses the active color table of the CAD file, and CAD file line styles and weights.

Place Contour

Not Lite



Place Contour tool draws a contour at a single elevation. The tool detects the surface elevation at the mouse pointer position and draws contour lines for this elevation.

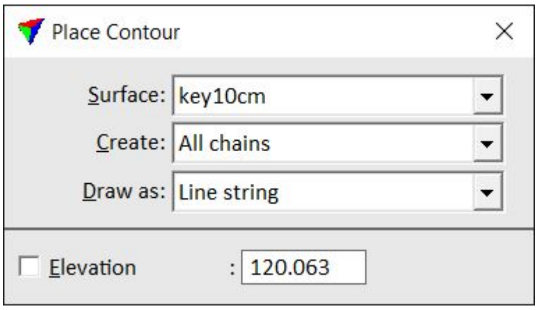
The contour lines are drawn using the active level and symbology settings in the CAD file.

You can place the contour at a precise elevation by locking the elevation to a key-in value.

To place a single contour:

1. Select the **Place Contour** tool.

   The **Place Contour** dialog opens:



2. Define settings.

3. Move the mouse inside a view.

   A preview of the contour running at the mouse pointer location is dynamically displayed.

4. Confirm the contour with a data click.

   This draws the contour(s) into the CAD file. You can continue with steps 2 or 3.

SETTING	EFFECT
Surface	Name of the effected surface model.
Create	Defines which contour lines are drawn: • **Only one** - only the contour line at the mouse pointer location is drawn. • **All chains** - all contour lines at the elevation of the mouse pointer location are drawn.
Draw as	Element type of the contour lines: • **Curves** - smooth elements for which the software computes the curvature, compact elements.

SETTING	EFFECT
	• **Line strings** - sharp elements, suitable for preview.
Elevation	If on, contours are drawn at the given elevation.

Place Elevation Text

Not Lite

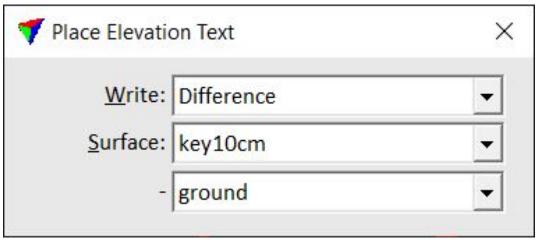
Place Elevation Text tool places a single elevation text. It shows the surface elevation at a given location. The tool can be used to label the difference between two surfaces as well.

Text font, size, and numeric format are determined by leveling point settings in Elevation labels category of the TerraModeler **Settings**. The settings also define the location of the label relative to the exact elevation point and if a point marker is set at the location of the elevation point. The label is drawn using active level and symbology settings in the CAD file.

To place a single elevation text:

1. Select the **Place Elevation Text** tool.

   This opens the **Place Elevation Text** dialog:



2. Define settings.

3. Move the mouse pointer inside a view.

   A preview of the label at the mouse pointer location is dynamically displayed.

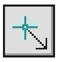
4. Confirm the elevation text location with a data click.

   The elevation label is drawn into the CAD file. You can continue with steps 2 or 3.

SETTING	EFFECT
Write	Defines the type of the label: • **Elevation** - the elevation value of a surface model is drawn. • **Difference** - the elevation difference between two surface models is drawn.
Surface	Surface model for which a label is placed. If **Write** is set to **Difference**, the surface is used as lower surface for the difference calculation.
-	If **Write** is set to **Difference**, the second surface is used as upper surface for the difference calculation.

Place Slope Arrow

Not Lite



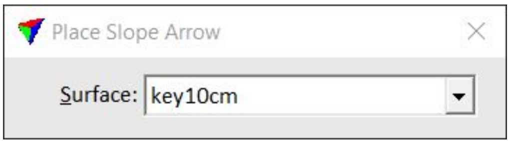
Place Slope Arrow tool places a single slope arrow. The slope arrow shows the direction of water flow at a given location. The arrow length indicates the steepness of the surface at that location.

The shape of the slope arrows can be defined in Place Slope Arrow / Arrows category of the TerraModeler **Settings**. It is further determined by the active symbology settings in the CAD file. In addition, a label showing the gradient of the slope can be placed for each slope arrow. The label settings are defined in Place Slope Arrow / Labels category of the TerraModeler **Settings**.

To place a single slope arrow:

1. Select the **Place Slope Arrow** tool.

   This opens the **Place Slope Arrow** dialog:

2. Select a **Surface** model for which slope arrows are placed.

3. Move the mouse pointer inside a view.

   The slope arrow is dynamically displayed at the mouse pointer location.

4. Confirm the arrow location with a data click.

   A slope arrow is drawn at the given location. You can continue with steps 2 or 3.

Display Surface toolbox

The tools in the **Display Surface** toolbox are used to generate displays from a surface model.

The first six tools generate different visualizations of a surface model. TerraModeler is capable of generating contours, colored triangles, a colored grid, elevation texts, slope arrows, or a shaded surface. The first five display methods can be drawn as permanent elements into the CAD file or as temporary elements which are not saved with the CAD file. A shaded surface is always drawn temporarily.

All six display methods can be updated after modifications to the surface model,using the Update Displays tool. The visualizations can be removed from display by using the Erase Display tool.

Display Boundary tool and Label Peaks and Pits tool create elements which are not updated with the **Update Displays** tool or removed by the **Erase Display** tool.

TO	USE TOOL	
Display contours		Display Contours
Display colored triangles		Display Triangles
Display colored grid		Display Grid
Display elevation as grid spaced texts		Display Elevation Texts
Display grid spaced slope arrows		Display Slopes
Display coloring by elevation and sun angle		Display Shaded Surface *Not Lite*
Update displays after surface modification		Update Displays
Erase contours, triangles or grid		Erase Display
Display surface model boundary		Display Boundary
Label local minimum and maximum points		Label Peaks and Pits

Display modes

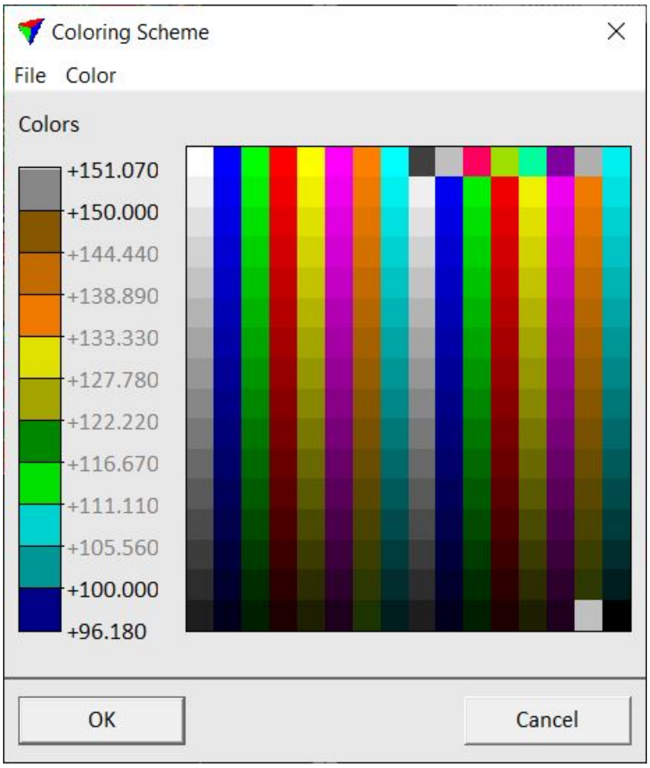
TerraModeler supports two different modes in which surface displays may be generated.

Write to file mode adds the created elements to the CAD file. This has the disadvantage of increasing the CAD file size quite substantially with some display methods. Very large surface models may result in CAD files which are very cumbersome for processing with CAD tools or for display in dynamic views. The advantages include the option to manipulate the elements using CAD tools and the ability to save the display on a hard disk as a part of the CAD file.

Preview mode recalculates and draws the display elements each time a view is updated. This display mode does neither increase the CAD file size nor consume any RAM space. The mode is supported only by tools which can quickly re-calculate the whole display.

Creating a color scheme of discrete colors

There are several display tools in TerraModeler that enable the definition of color schemes from discrete colors for surface model display. The **Coloring scheme** dialog is used to create such a coloring scheme by assigning colors for different elevations. The dialog uses the active color table of the CAD file.



The vertical bar on the left shows all the colors currently assigned in the scheme. Each color boundary has a label displaying the elevation value where that color change occurs. The text label is black, if the elevation value is fixed to the color boundary. A white elevation value has not been fixed and is recalculated if you change the number of colors in the scheme.

From the coloring scheme dialog, you can save a coloring scheme file to a hard disk.

To assign a new color:

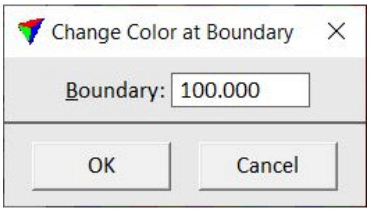
1. Click on one of the available colors in the color table on the right.

   This adds the new color at the lower end of the **Colors** bar and recalculates all unfixed elevation boundaries.

To fix an elevation boundary:

1. Click on the text label of the elevation boundary.

   This opens the **Change Color at Boundary** dialog:



2. Type an elevation value in the **Boundary** field.

   The color will change at this value.

To create a new color scheme:

1. Select **Remove all** command from the **Color** pulldown menu in the **Coloring scheme** dialog.

   This removes all assigned colors from the current color bar.

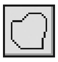
2. Assign new colors to the color bar as described above.

   Each color is added to the lower end of the bar, thus, the first color is assigned to the highest elevation value.

3. (Optional) Save the color scheme as a file on a hard disk using the **Save as** command from the **File** pulldown menu in the **Coloring scheme** dialog.

USE MENU ITEM	TO
File - Open	Open a previously saved coloring scheme file.
File - Save As	Save the coloring scheme to a file.
Color - Add empty	Add an empty color which is not drawn.
Color - Remove last	Remove last color from the coloring scheme.
Color - Remove all	Remove all colors from the coloring scheme.

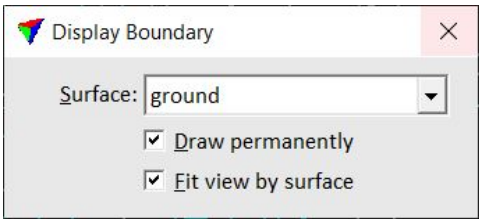
Display Boundary



Display Boundary tool draws a boundary shape of a surface model. This can be used to outline the area covered by the surface model.

To display the boundary of a surface model:

1. Select the **Display Boundary** tool.
2. This opens the **Display Boundary** dialog:



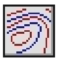
3. Select the **Surface** you want to display.
4. Move the mouse pointer inside a view.

   The boundaries of the selected surface model are highlighted.

5. (Optional) Select a view with a data click to draw the boundary shape permanently in to the CAD file and/or to fit the view to the surface model.

SETTING	EFFECT
Surface	Name of the effected surface model.
Draw permanently	If on, the surface boundary is drawn as shape into the CAD file when you enter a data point in a view.
Fit view by surface	If on, the software fits the view to the surface area when you enter a data point.

Display Contours



Display Contours tool draws contour lines for a surface model. You can generate quick contours which are suitable for verifying the surface model or higher-quality contours to be plotted on paper.

The tool can also be used to save a contour settings file that is required for producing contour lines automatically in a batch process. The settings file stores all information related to contour lines, labels, and ticks, except the name of the surface model. See Utility / Produce contours for more information.

Contours can be drawn as:

- **Curves** - smooth elements for which the software computes the curvature. This results in fairly compact elements but can cause contours to intersect each other.
- **Line strings** - sharp elements suitable for mathematical design surfaces or for surface verification.
- **Soft line strings** - smooth elements for which TerraModeler computes the smoothing by inserting additional vertices. This results in quite complex elements.

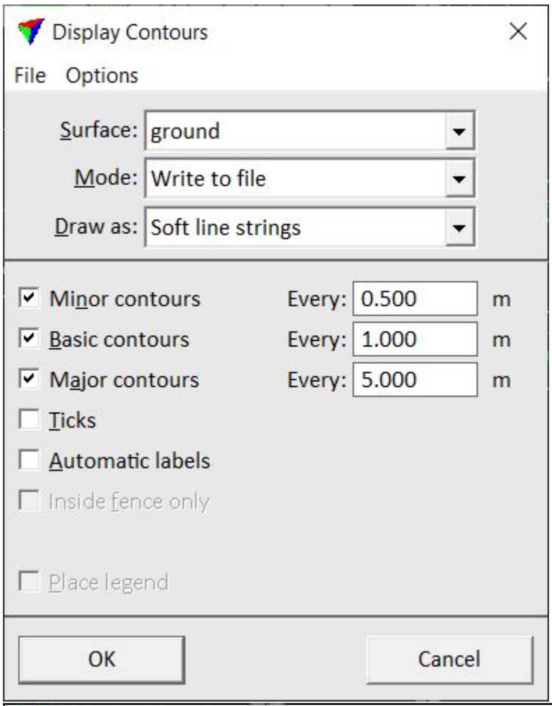
TerraModeler supports three different types of contours. **Minor**, **Basic** and **Major** contours can be distinguished from each other by line color, weight, or style. You can select which of the contour types is displayed and at what intervals. If more than three contour types have to be distinguished, you may use the Modify Contour Symbology tool in order to assign a different level and symbology to contours that are already drawn into the CAD file.

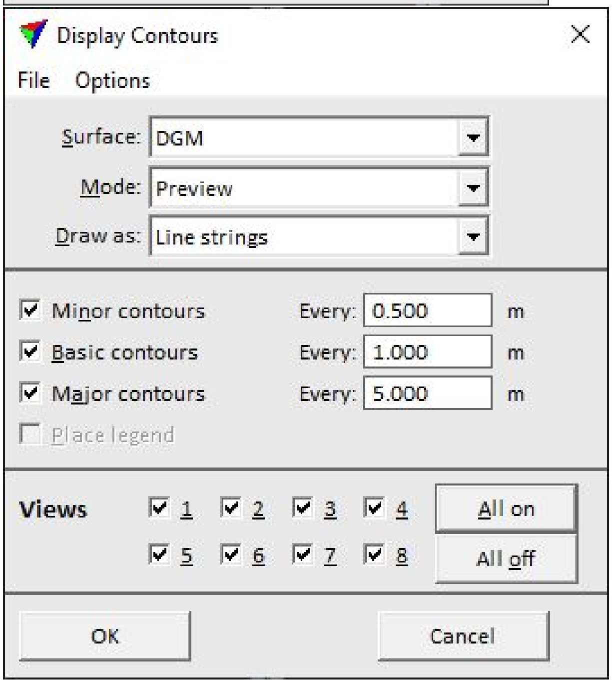
In addition to the contour lines, the software can create labels and ticks for the contours. Further, depressions can be displayed using different symbology settings than for hills. Labels, ticks, and depression line symbology are available in display modes **Write to file**, but not in **Preview mode**.

To display contours:

1. Select the **Display Contours** tool.

   The **Display Contours** dialog opens:





2. Define settings for contour generation.

3. Define settings for the symbology of contour lines, labels, and ticks. This can be done using commands Contour options, Contour label options, and Contour tick options from the **Options** pulldown menu in the **Display contours** dialog.

OR

3. Load a previously saved settings file into the dialog using the **Load settings** command from the **File** pulldown menu.

4. (Optional) Save the settings into a file on a hard disk by using the **Save settings As** command from the **File** pulldown menu.

Click **Cancel** to the **Display contours** dialog if you just want to save a contour settings file for automatic contour line production.

5. Click OK to the **Display contours** dialog.

6. If **Place legend** is switched on, define the location of drawing the legend with a data click.

The contours for the selected surface model are displayed.

SETTING	EFFECT
Surface	Name of the effected surface model.
Mode	Display mode for contour lines: • **Write to file** - elements are written and stored in the CAD file. • **Preview** - elements are recalculated and redrawn whenever the view is updated.
Draw as	Element type of the contour lines: • **Curves** - smooth elements for which the software computes the curvature, may overlap each other, compact elements. This is only active if **Mode** is set to **Write to file**. • **Line strings** - sharp elements, suitable for preview. • **Soft line strings** - smooth elements for which TerraModeler computes smoothing, best quality, large elements. This is only active if **Mode** is set to **Write to file**.
Minor contours	If on, minor contours are drawn at **Every** distance.
Basic contours	If on, basic contours are drawn at **Every** distance.
Major contours	If on, major contours are drawn at **Every** distance.
Ticks	If on, ticks are drawn along contour lines. This is only active if **Mode** is set to **Write to file**.
Automatic labels	If on, labels are drawn along contour lines. This is only active if **Mode** is set to **Write to file**.
Inside fence only	If on, contours are displayed only inside a selected polygon or fence. This is only active if **Mode** is set to **Write to file**.
Write attributes	If on, the software writes the contour elements in a way that enables the update of the contours using the Update Displays tool and the removal of contours using the Erase

SETTING	EFFECT
	Display tool. This is only active if **Inside fence only** is switched on.
Place legend	If on, a legend for contour line symbology is placed. This is only active if **Color by** is set to **Elevation** in the Contour options dialog.
Views	Switch on the CAD file views in which the contours should be visible. This is only active if **Mode** is set to **Preview**. Use the **All on** or **All off** buttons in order to switch on/off the visibility for all views.

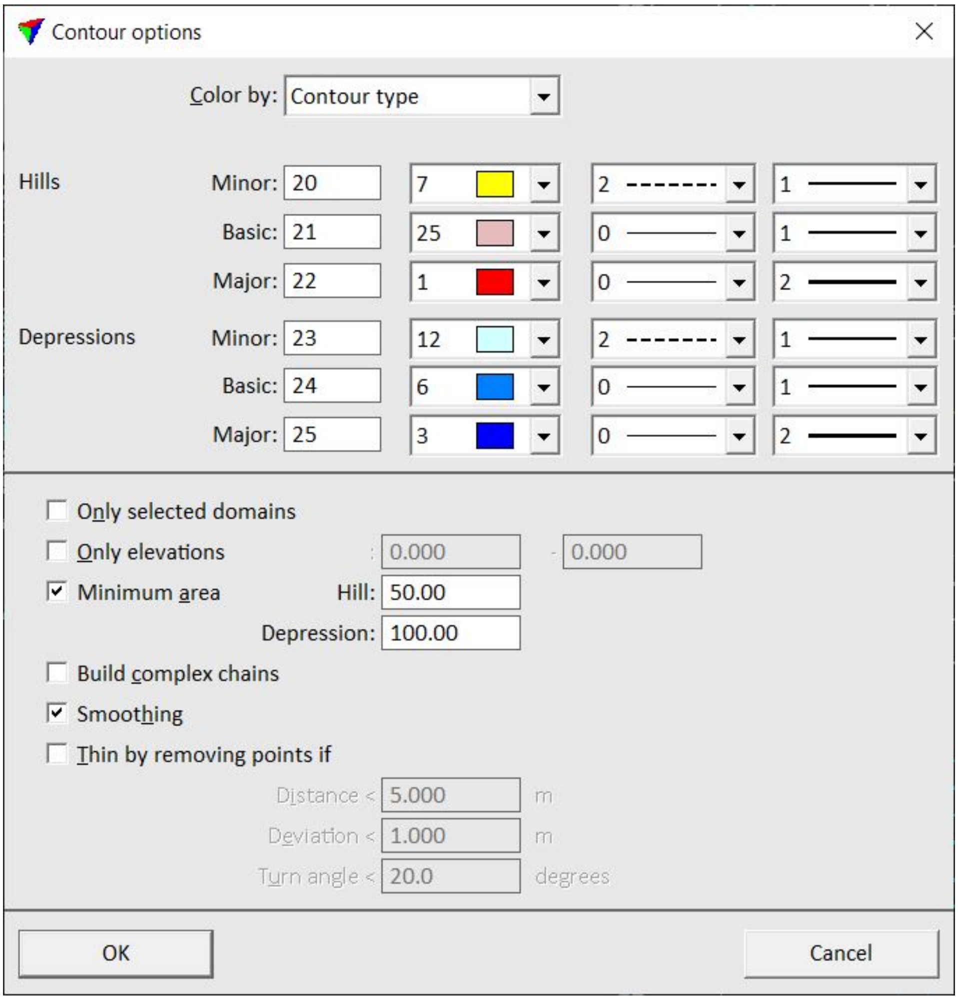
Contour options

Contour options define symbology settings for contour lines, such as level, coloring, line style, and line weight for the different contour types. They can also limit the contour display to specific domains or an elevation range, define the minimum areas for closed contours, or influence the way of contour line generation.

To define contour options:

1. Select **Contours** command from the **Options** pulldown menu in the **Display contours** dialog.

   The **Contour options** dialog opens:.



2. Define settings and click OK.

SETTING	EFFECT
Color by	Contour line coloring method: • **Contour type** - different colors according to the type of the contour line. • **Elevation** - a coloring scheme is used for coloring contour lines according to their elevation value.
Colors	Color scheme for coloring contours by their elevation value. The color scheme is defined using the **Define** button. See Creating a color scheme of discrete colors for information about how to create a coloring scheme. Only active if **Color by** is set to **Elevation**.
Minor	Level, color, line style and line weight for minor contours of hills/depressions.
Basic	Level, color, line style and line weight for basic contours of hills/depressions.

SETTING	EFFECT
Major	Level, color, line style and line weight for major contours of hills/depressions.
Only selected domains	If on, contours are only drawn for domains for which the **Draw contours** option is switched on. See Define Domains tool for more information.
Only elevations	If on, contours are drawn only for the given elevation range.
Minimum area	A closed contour line is not drawn, if the enclosed surface area is smaller than **Minimum area**. Separate settings for **Hills** and **Depressions**. Helps to leave out unnecessary detail.
Build complex chains	If on, contours are drawn as complex chains or as complex shapes.
Smoothing	If on, contour vertices are slightly adjusted to produce smoother lines. If off, contours pass through triangle edges at mathematically correct positions which may result in jagged contour lines.
Thin by removing points if	If on, some of the computed contour vertices may be left out. This helps to minimize the CAD file size when creating contours for large surface models.
Distance	A vertex can be left out if the distance between the previous and the next remaining vertex is smaller than this value. Only active if **Thin by removing points if** is switched on.
Deviation	A vertex can be left out if it is closer than this distance value to the line segment connecting the previous and the next remaining point. Only active if **Thin by removing points if** is switched on.
Turn angle	A vertex can be left out if the contour direction changes less than this value at the vertex location. Only active if **Thin by removing points if** is switched on.

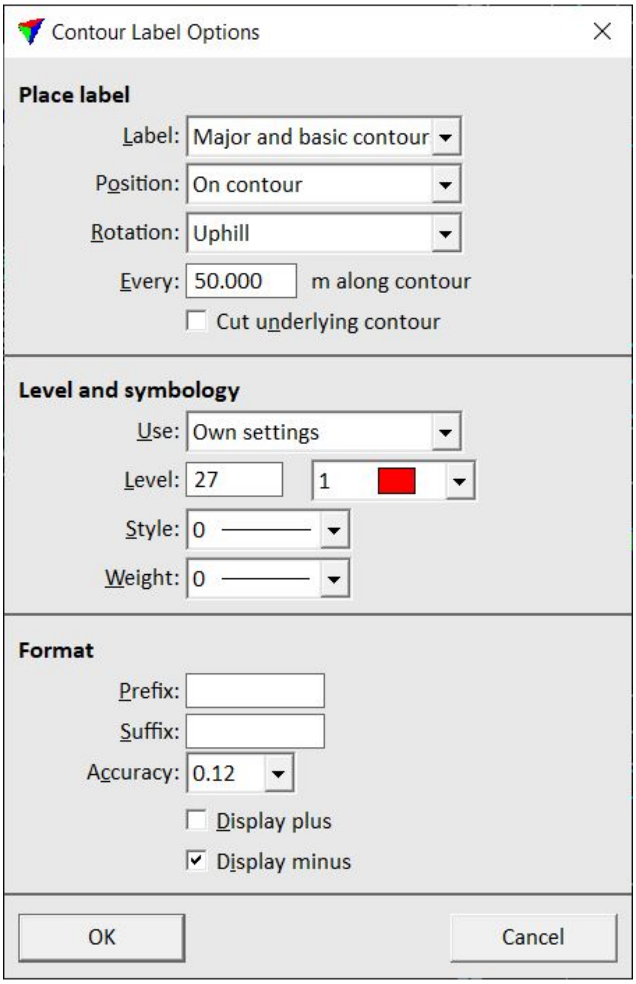
Contour label options

Contour label options define the position of contour labels in relation to the contour lines, as well as their symbology settings, and their text format.

To define contour label options:

1. Select **Contour labels** command from the **Options** pulldown menu in the **Display contours** dialog.

   The **Contour Label Options** dialog opens:



2. Define settings and click OK.

SETTING	EFFECT
Label	Defines the contour line type(s) for labeling: **All contours**, **Major and basic contours**, or **Major contours only**.
Position	Location where label text elements are placed along a contour line: • **Above contour** - above the contour line at specified intervals. • **On contour** - overlapping the contour line at specified intervals. • **Below contour** - below the contour line at specified intervals.

SETTING	EFFECT
	• **At contour start** - at the start point of a contour chain.
Rotation	Rotation of the label text element: • **Uphill** - label is readable when looking uphill. • **Downhill** - label is readable when looking downhill. • **North up** - label is readable when looking in north direction.
Every	Distance between consecutive contour labels. Given in meters along the contour.
Cut underlying contour	If on, the contour line is cut under the label text element. On Bentley CAD platforms, the line under a label text is drawn as a construction class element. You can use view attributes to control the visibility of the lines under label texts. Further, you can use filtered selection in order to select and delete the lines under label texts.
Use	Defines level and color settings for contour labels: • **Contour level and color** - a label is drawn using level and color of its contour line type (major, basic, or minor). • **Own settings** - all labels are drawn using the same level and color assigned to labels.
Level	Level and color of contour labels. This is only active if **Use** is set to **Own settings**.
Style	Line style of contour labels. This is only active if **Use** is set to **Own settings**.
Weight	Line weight of contour labels. This is only active if **Use** is set to **Own settings**.
Prefix	Prefix added at the beginning of each contour label.
Suffix	Suffix added at the end of each contour label.
Accuracy	Number of decimals shown in contour labels.
Display plus	If on, the plus sign is displayed for positive elevations.
Display minus	If on, the minus sign is displayed for negative elevations.

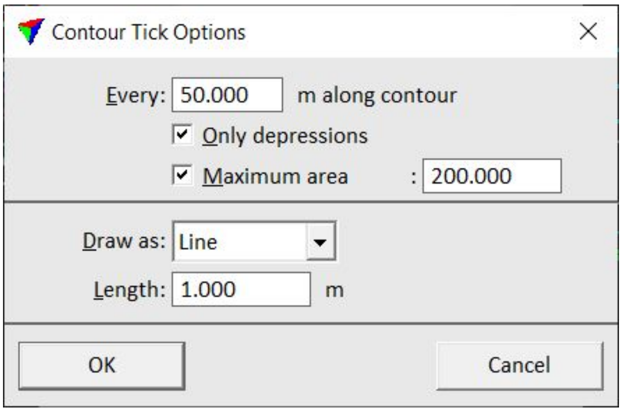
Contour tick options

Contour tick options define the position of ticks in relation to the contour lines, as well as the format of the ticks.

To define contour tick options:

1. Select **Contour ticks** command from the **Options** pulldown menu in the **Display contours** dialog.

   The **Contour Tick Options** dialog opens:



2. Define settings and click OK.

SETTINGS	EFFECT
Every	Distance between consecutive contour ticks given in meters along the contour.
Only depressions	If on, ticks are drawn only for closed depressions.
Maximum area	If on, ticks are drawn only for closed depressions smaller than a given area.
Draw as	Shape of ticks: **Line** or **Triangle**.
Length	Length of the tick line or triangle.

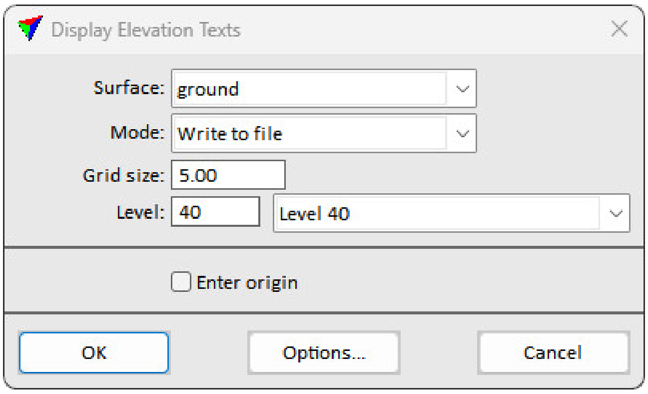
Display Elevation Texts

Display Elevation Texts tool draws a grid of labels which display the surface elevation. The exact location of the elevation can be marked by a point marker or by the decimal point of the elevation text.

To display elevation texts:

1. Select the **Display Elevation Texts** tool.

   The **Display Elevation Texts** dialog opens:



2. Define settings.

3. (Optional) Click on the **Options** button in order to open the Elevation text settings dialog and define settings for elevation label symbology.

4. Click OK to the **Display elevation texts** dialog.

5. If **Enter origin** is switched on, enter the origin point of the grid with a data click.

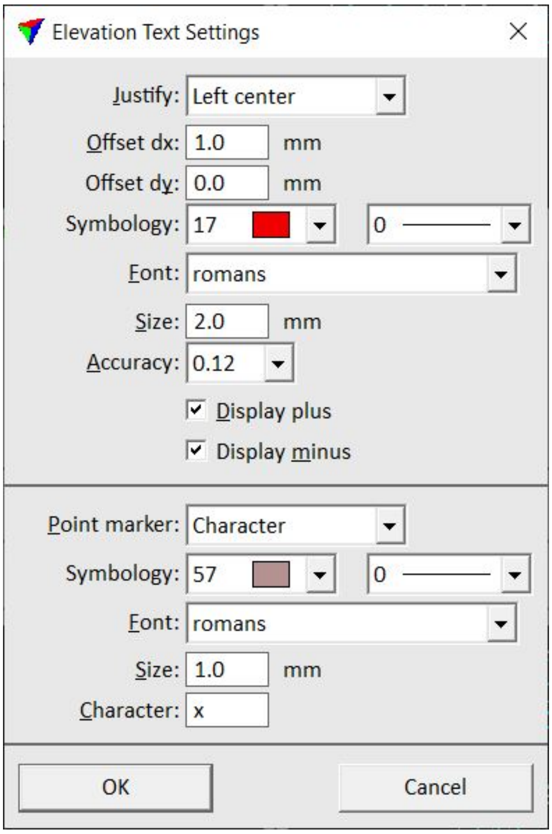
   This draws elevation texts for the selected surface model on the given level.

SETTING	EFFECT
Surface	Name of the effected surface model.
Mode	Display mode for elevation texts: • **Write to file** - elements are written and stored in the CAD file.
Grid size	Distance between elevation text locations.
Level	Number of the level in the CAD file on which elevation texts are drawn.
Enter origin	If on, you can enter the origin point of the grid with another data point. If the point is inside the surface area, an elevation text is drawn at this location.

SETTING	EFFECT
Options	Opens the **Elevation Text Settings** dialog. See Elevation text settings for more information.

Elevation text settings

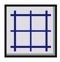
The **Elevation Text Settings** dialog defines the symbology and other display options for elevation texts created by the Display Elevation Texts tool.



SETTING	EFFECT
Justify	Location of the placement point (= point of elevation measurement): • **Left**, **Center**, **Right** | **Top**, **Center**, **Bottom** - placement point location relative to the label text box. • **Decimal point** - placement point is located on the decimal point.
Offset dx	Offset between the placement point and the label in x (left-right) direction. Given in millimeters on paper. Not active if **Justify** is set to **Decimal point**.

SETTING	EFFECT
Offset dy	Offset between the placement point and the label in y (up-down) direction. Given in millimeters on paper. Not active if **Justify** is set to **Decimal point**.
Symbology	Color and line weight of the elevation text. Uses the CAD file color table and line weights.
Font	Font type of elevation texts. Uses font types available in the CAD platform.
Size	Text size of elevation texts. Given in millimeters on paper.
Accuracy	Number of decimals of elevation texts.
Display plus	If on, the plus sign is displayed for positive elevations.
Display minus	If on, the minus sign is displayed for negative elevations.
Point marker	Type of the placement point marker: • **None** - no point marker is drawn. • **Character** - a given character is drawn as point marker. • **Zero length line** - a line element of zero length (= point) is drawn as point marker.
Symbology	Color and line weight of the point marker. Uses the CAD file color table and line weights. This is only active if **Point marker** is set to **Character** or **Zero length line**.
Font	Font type of the point marker character. Uses font types available in the CAD platform. This is only active if **Point marker** is set to **Character**.
Size	Text size of the point marker character. Given in millimeters on paper. This is only active if **Point marker** is set to **Character**.
Character	Character used as point marker symbol. This is only active if **Point marker** is set to **Character**.
Weight	Weight of the point marker zero length line. Uses CAD file line weights. This is only active if **Point marker** is set to **Zero length line**.

Display Grid

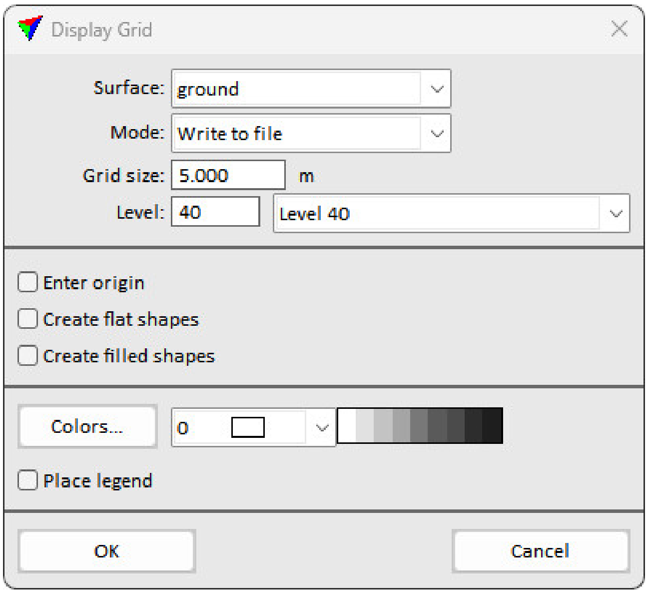


Display Grid tool generates a grid presentation of a surface model. You can create a coloring scheme where the color of the grid squares changes according to the surface elevation.

To display a grid:

1. Select the **Display Grid** tool.

   The **Display Grid** dialog opens:



2. Define settings and click OK.

3. If **Enter origin** is switched on, define the origin point of the grid with a data click.

   This draws a squared grid for the selected surface model on the given level.

4. If **Place legend** is switched on, define the location of drawing the legend with another data click.

SETTING	EFFECT
Surface	Name of the effected surface model.
Mode	Display mode for the grid: • **Write to file** - elements are written and stored in the CAD file.
Grid size	Size of rectangular squares in the grid.
Level	Number of the level in the CAD file on which the grid is drawn.
Enter origin	If on, you can enter the origin point of the grid with another data point. If this point is inside

SETTING	EFFECT
	the surface area, it becomes a corner point of the grid square at this location.
Create flat shapes	If on, each grid square is drawn as flat square at the elevation of its center point.
Create filled shapes	If on the grid squares are filled with color.
Colors	Opens the **Color scheme** dialog. See Creating a color scheme of discrete colors for more information.
Color list left of the color scheme preview	Opens the CAD file color table for single color selection.
Place legend	If on, a legend for the triangle colors can be placed in the CAD file. This is only active if a color scheme is defined.

Display Shaded Surface

Not Lite

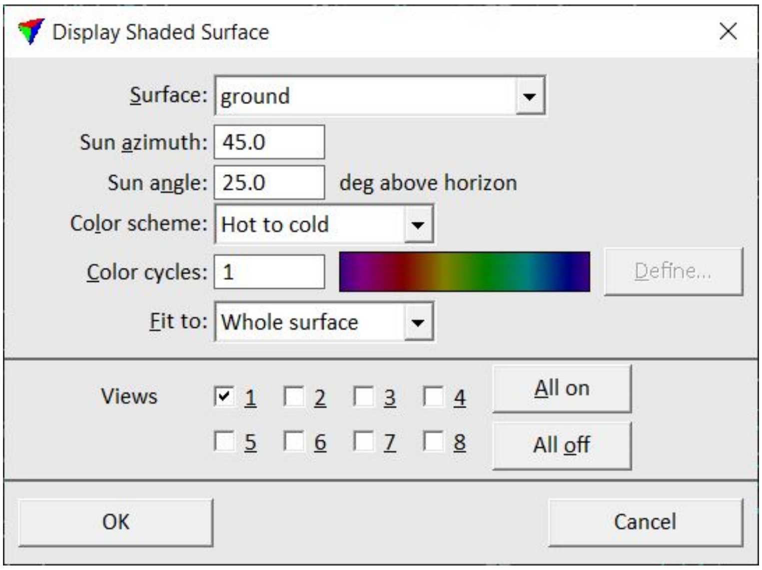
Display Shaded Surface tool draws the surface model colored by elevation and by triangle slope at the same time. The display method utilizes the Hue-Saturation-Value (HSV) color model in order to visualize the surface. The elevation value is represented by the color hue while the triangle slope is determined by the color value. The sun angle above the horizon further influences the brightness of colors in a shaded surface display.

The display is drawn temporarily as a raster in CAD file views, it is not written into the CAD file.

To display a shaded surface:

1. Select the **Display Shaded Surface** tool.

   This opens the **Display Shaded Surface** dialog:



2. Define settings and click OK.

   This creates a shaded surface display in the selected views.

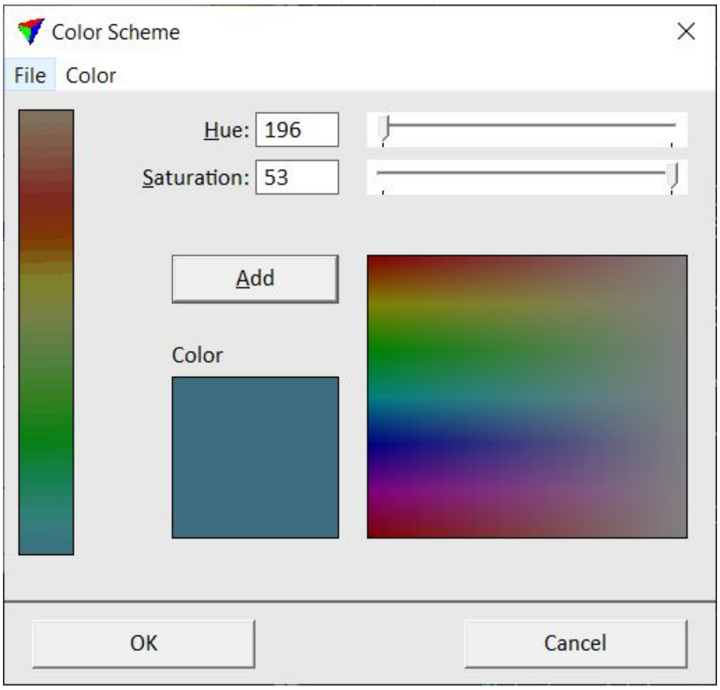
SETTING	EFFECT
Surface	Name of the effected surface model.
Sun azimuth	Direction from which the sun illuminates the model. Zero is north and angles increase clockwise.
Sun angle	Sun angle above the horizon.
Color scheme	Color scheme to use:

SETTING	EFFECT
	• **Hot to cold** - color scheme containing magenta, red, yellow, green, cyan, and blue hues. • **Earth tones** - color scheme containing gray, brown, yellow, and green hues with low saturation. • **Selected colors** - use the **Define** button to create a customized color scheme. See Define a customized color scheme for more information.
Color cycles	Number of color cycles. Use zero to create a gray scale display showing triangle slope only.
Fit to	Determines how colors are fit to elevation values: • **Whole surface** - colors of the color scheme are fit to the elevation range of the whole surface. • **View content** - colors of the color scheme are fit to the elevation range that is visible in a CAD file view. This results in a very detailed display for large-scale views.
Views	View(s) in which the shaded surface is displayed.

Shaded surface GeoTIFF files can be produced automatically in batch mode using the Utility / Produce lattice models command from the **Surfaces** window.

Define a customized color scheme

The **Color Scheme** dialog allows you to create a color scheme for a shaded surface display. The dialog uses a Hue-Saturation color model. The software creates smooth transitions between the colors assigned to the color scheme.



The vertical bar on the left shows the colors currently assigned to the scheme. The color field on the right shows color hue of the color cycle from top to bottom and color saturation from left to right.

To assign a new color:

1. Click in the color field on the right at the location of the color you want to add to the color scheme.

   OR

1. Define **Hue** and **Saturation** values by typing a number or moving the slider. The hue value can range from 0 to 359, the saturation value from 0 to 100.

   This selects the color and adjusts the value in the **Hue** and **Saturation** fields. The color is displayed in the **Color** field.

2. Click **Add** in order to add the selected color to the color scheme.

   The color is added to the lower end of the bar, thus, the first color is assigned to the highest elevation values.

To create a new color scheme:

1. Select **Remove all** command from the **Color** pulldown menu in the **Coloring scheme** dialog.

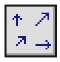
   This removes all assigned colors from the current color scheme.

2. Assign new colors to the color scheme as described above.

3. (Optional) Save the color scheme as a file on a hard disk using the **Save as** command from the **File** pulldown menu in the **Color Scheme** dialog.

USE MENU ITEM	TO
File - Open	Open a previously saved color scheme file.
File - Save As	Save the color scheme to a file.
Color - Remove last	Remove last color from the color scheme.
Color - Remove all	Remove all colors from the color scheme.

Display Slopes

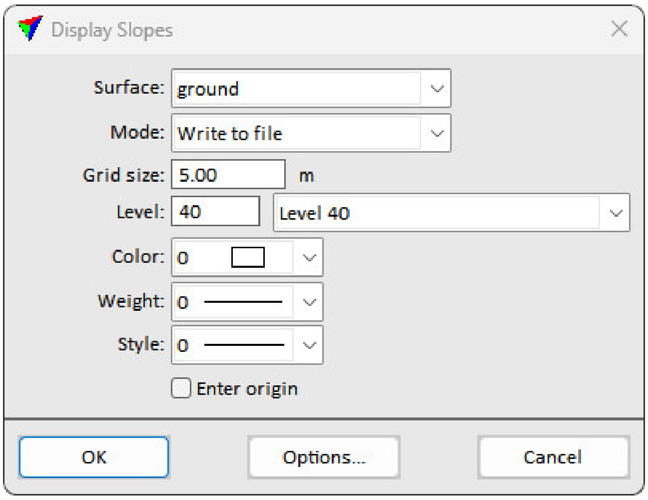


Display Slopes tool draws a grid of slope arrows. Each slope arrow shows the direction of water flow at this location. The arrow length indicates the steepness of the surface at this location. The tool can also create labels along the arrows which show the gradient of the slope.

To display slope arrows:

1. Select the **Display Slopes** tool.

   The **Display Slopes** dialog opens:



2. Define settings.

3. (Optional) Click on the **Options** button in order to open the [Slope Settings](#) dialog and define settings for slope arrow symbology.

4. Click OK to the **Display slopes** dialog.

5. If **Enter origin** is switched on, enter the origin point of the grid with a data click.

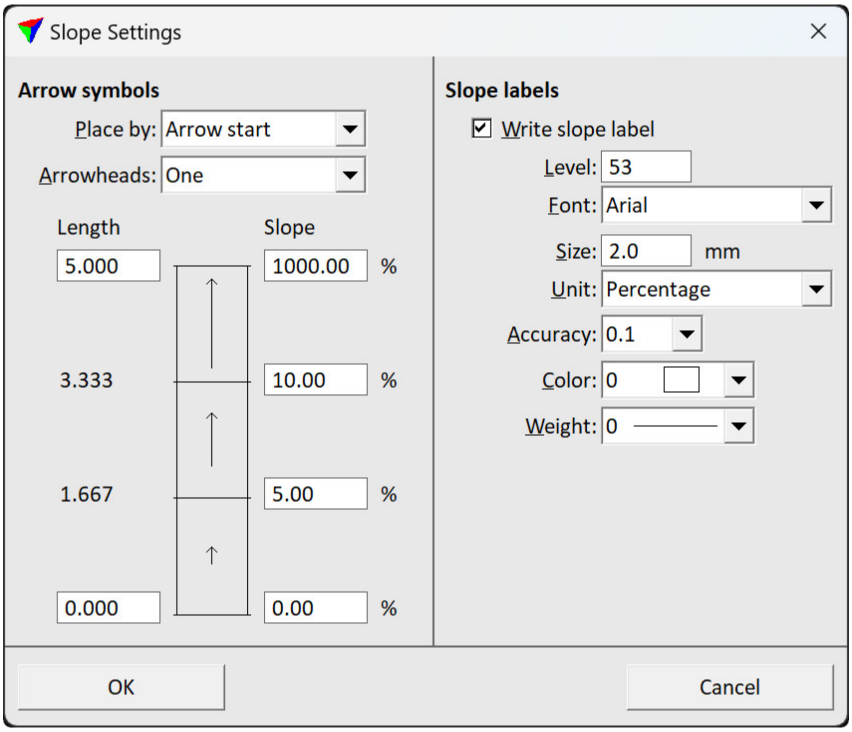
   This draws slope arrows for the selected surface model on the given level.

SETTING	EFFECT
Surface	Name of the effected surface model.
Mode	Display mode for slope arrows: • **Write to file** - elements are written and stored in the CAD file.
Grid size	Distance between slope arrow locations.
Level	Number of the level in the CAD file on which slope arrows are drawn.

SETTING	EFFECT
Color	Color of the arrows. Uses the CAD file color table.
Weight	Line weight of the arrows. Uses CAD file line weights.
Style	Line style of the arrows. Uses CAD file line styles.
Enter origin	If on, you can enter the origin point of the grid with another data point. If the point is inside the surface area, an elevation text is drawn at this location.
Options	Opens the **Slope Settings** dialog. See Slope settings for more information.

Slope settings

The **Slope Settings** dialog defines the symbology and label display options for slope arrows created by the Display Slopes tool.



SETTING	EFFECT
Place by	Placement point (= point of elevation measurement) of a slope arrow: **Arrow start** or **Arrow center**.

SETTING	EFFECT
Arrowheads	Number of arrowheads of a slope arrow: • **One** - all arrows have one arrowhead. • **One to three** - number of arrowheads depends on the slope.
Length	Two fields specifying the maximum and the minimum length of a slope arrow.
Slope	Slope values at which the arrow length is equal to: - maximum length - two thirds of maximum length - one third of maximum length - minimum length
Write slope label	If on, a text label showing the slope gradient is placed along a slope arrow.
Level	Level number in the CAD file on which the label is placed.
Font	Font type of slope arrow labels. Uses font types available in the CAD platform.
Size	Text size of slope arrow labels. Given in millimeters on paper.
Unit	Measurement unit of label value **Percentage** or **Degree**.
Accuracy	Number of decimals in the slope arrow label.
Color	Color of slope arrow labels. Uses the active color table of the CAD file.
Weight	Line weight of slope arrow labels. Uses the CAD file line weights.

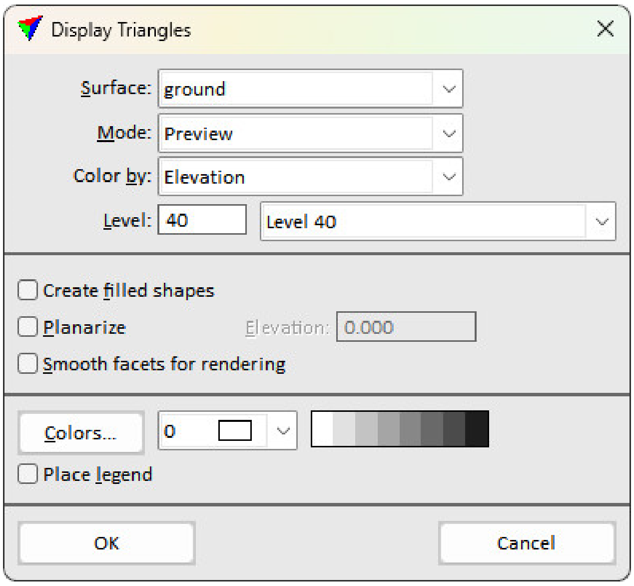
Display Triangles

Display Triangles tool draws colored triangles for a surface model. You can create a coloring scheme where the color changes according to the surface elevation or according to the slope gradient of the triangles.

To display triangles:

1. Select the **Display Triangles** tool.

   The **Display Triangles** dialog opens:



2. Define settings and click OK.

   This draws triangles for the selected surface model on the given level.

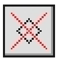
3. If **Place legend** is switched on, define the location of drawing the legend with another data point.

SETTING	EFFECT
Surface	Name of the effected surface model.
Mode	Display mode for triangles: • **Write to file** - elements are written and stored in the CAD file. A triangle boundary is drawn in counter-clockwise direction. • **Preview** - elements are recalculated and redrawn whenever the view is updated.
Color by	Attribute used for coloring: • **Elevation** - elevation value of the triangle determines its color.

SETTING	EFFECT
	• **Slope** - slope gradient of the triangle determines its color.
Level	Number of the level in the CAD file on which triangles are drawn.
Create filled shapes	If on, triangles are filled with color.
Planarize	If on, triangles are drawn as planarized triangles at the given **Elevation**.
Smooth facets for rendering	If on, triangles are displayed with a smoother change in coloring if CAD rendering is applied (No effect in MicroStation CE).
Inside fence only	If on, triangles are drawn if their center point is located inside a selected polygon or fence.
Colors	Opens the **Color scheme** dialog. See Creating a color scheme of discrete colors for more information.
Color list left of the color scheme preview	Opens the CAD file color table for single color selection.
Place legend	If on, a legend for the triangle colors can be placed in the CAD file. This is only active if a color scheme is defined.

Triangles can be produced automatically in batch mode using the See Utility / Produce triangles command from the **Surfaces** window.

Erase Display



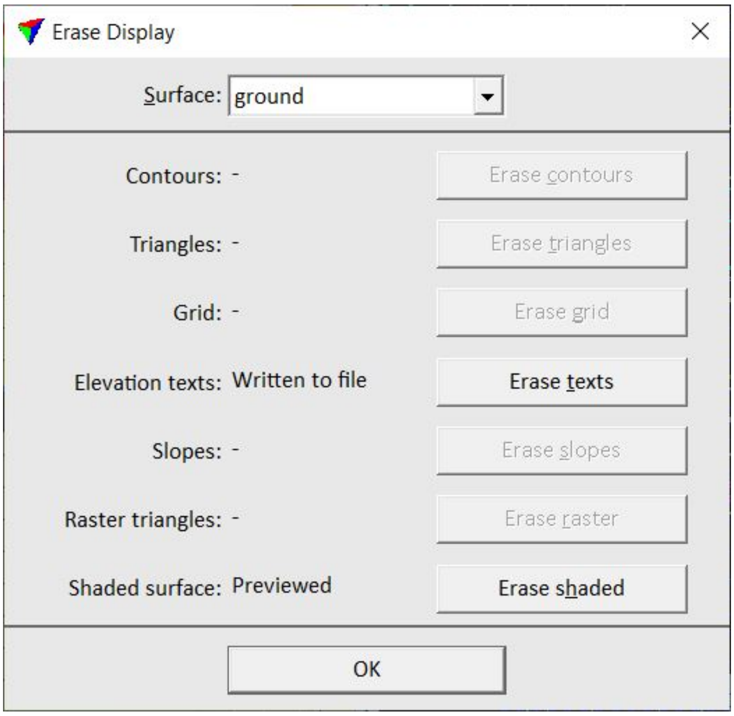
Erase Display tool removes previously displayed contours, triangles, a grid, elevation texts, slope arrows, raster triangles, or shaded surfaces.

The tools opens a window which shows the status of the display methods for the selected surface. In the example below, for the surface named **ground**, elevation texts are drawn in **Written to file** mode and a shaded surface is displayed as well. Contours, triangles, a grid, slope arrows or raster triangles are not displayed.

To erase display methods:

1. Select the **Erase Display** tool.

   This opens the **Erase Display** dialog:



2. Select the **Surface** for which to remove a display method.

3. Use the **Erase** button next to a display method to erase the corresponding display.

4. Click **OK** in order to close the **Erase display** dialog.

Label Peaks and Pits

Label Peaks and Pits tool draws labels for the highest point on hill tops and the lowest points inside depressions into the CAD file. The label contains a text showing the elevation value and a point marker placed at the location of the elevation point.

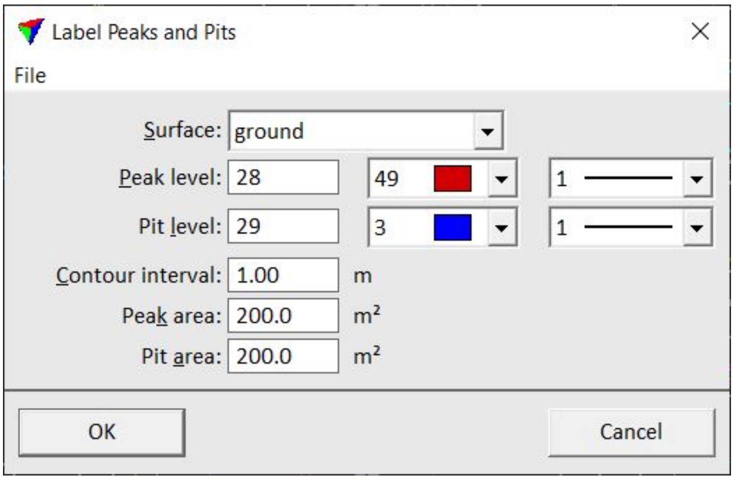
The layout of the label is determined by the settings in Elevation labels category of the TerraModeler **Settings**.

From the dialog of the tool, a peaks and pits settings file can saved for being used in automatic contour production. See Utility / Produce contours for more information. The settings file stores all information given in the dialog except the surface model name.

To draw peaks and pits into the CAD file:

1. Select the **Label Peaks and Pits** tool.

   This opens the **Label Peaks and Pits** dialog:



2. Define settings for peaks and pits display.

   OR

2. Load a previously saved settings file into the dialog using the **Load settings** command from the **File** pulldown menu.

3. (Optional) Save the settings into a file on a hard disk by using the **Save settings As** command from the **File** pulldown menu.

   Click **Cancel** to the **Label Peaks and Pits** dialog if you just want to save a peaks and pits settings file for automatic contour line production.

4. Click OK to the **Label Peaks and Pits** dialog.

   The software searches for peaks and pits in the surface model and writes the labels into the CAD file.

SETTING	EFFECT
Surface	Name of the effected surface model.
Peak level	Level, color, and line weight of labels for peaks. Uses the active CAD file color table and line weights.
Pit level	Level, color, and line weight of labels for pits. Uses the active CAD file color table and line weights.
Contour interval	Distance between two consecutive contour lines.
Peak area	A peak label is placed if the area enclosed by the biggest closed contour line around a hill top is at least as big as the given value.
Pit area	A pit label is placed if the area enclosed by the biggest closed contour line around a depression is at least as big as the given value.

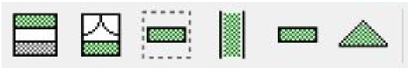
Update Displays

Update Displays tool updates contours, triangles, a grid, elevation texts, slope arrows, and shaded surfaces after surface models were modified. This tool updates all display methods for all modified surfaces immediately after clicking on the tool icon.

Domains toolbox

Not Lite, Not UAV

The tools in the **Domain** toolbox are used to define domains, to assign domain information for triangles, and to display domains.



Domains are related to the triangles in TerraModeler surface models. The domain number is stored for each triangle in the surface model TIN file.

You can use domains to separate different types of surface regions from each other. By defining a list of domains, you can create your own classification system for surface regions. This classification system may be used for visualization purposes. It allows to distinguish different-looking surfaces regions from each other.

Typically, you create a list of domains for various regions of the ground surface. The list could include domains such as ground, grass, rock, forest, or road. For each domain a unique coloring scheme and level should be assigned.

Display Domain Triangles tool draws surface model triangles using coloring schemes and levels that are assigned for each domain. If each domain is drawn on a unique CAD file level, it is easier to do further manipulation with CAD tools, such as:

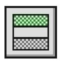
- Apply area pattern on all triangles belonging to a certain domain using the Bentley **Pattern Area** tool.
- Assign Bentley CAD rendering materials to triangles belonging to a certain domain.
- Drape an aerial photograph on a surface model using material assignment.

There are also a few tools in TerraModeler which enable the application of processes to certain domains only. Examples are Display Contours, Display Themes, Draw Profile, Compute Area.

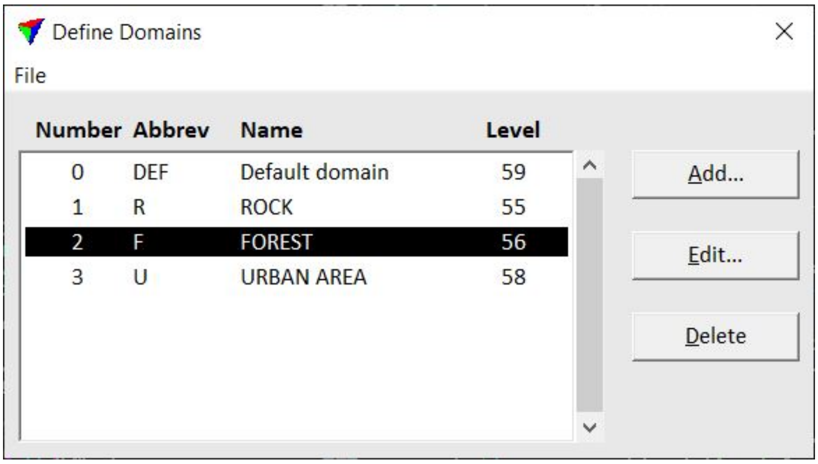
TO	USE TOOL	
Define domains		Define Domains *Not Lite, Not UAV*
Assign a view for dynamic domain display		Domain View *Not Lite, Not UAV*
Set domain of triangles inside an area		Set Area Domain *Not Lite, Not UAV*
Set domain of triangles between two linear elements		Set Domain Between Lines *Not Lite, Not UAV*
Set domain of triangles		Set Triangle Domain *Not Lite, Not UAV*
Display triangles using domain coloring		Display Domain Triangles *Not Lite, Not UAV*

Define Domains

Not Lite, Not UAV



Define Domains tool opens a dialog for defining domains. The dialog contains buttons for adding new domains, editing a domain, and deleting a domain.



The domain definitions can be saved into a text file or loaded from a previously saved file by using the commands from the **File** pulldown menu. When you open a CAD file, the software automatically loads the domain definition file that was used last with this CAD file.

For each domain, a single color or a color scheme can be chosen for visualization. If a color scheme is defined, the elevation differences of a surface model within the domain area are represented. Areas for which no specific domain is assigned are displayed using the **Default domain** settings.

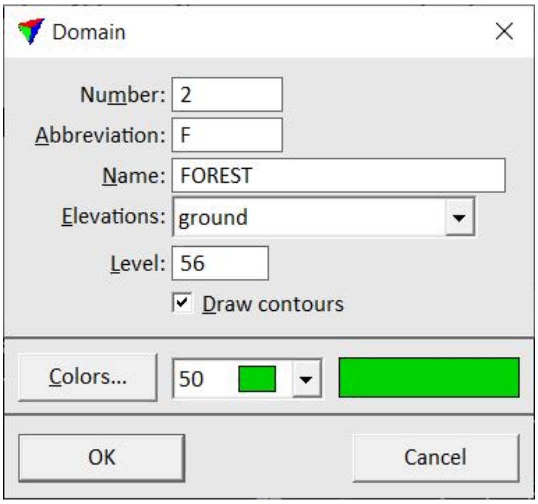
To add or edit a domain:

1. Click **Add** in the **Define Domains** dialog.

   OR

1. Select a domain definition and click **Edit** in the **Define Domains** dialog.

   This opens the **Domain** dialog:



2. Define settings for the domain.

3. Click OK.

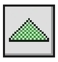
The new domain definition is added to the list in the **Define Domains** dialog or the selected domain is modified.

A selected domain can be removed from the list by using the **Delete** button in the dialog.

SETTING	EFFECT
Number	Number of the new domain. The domain number is stored for each triangle in the surface model TIN file if a domain is assigned to it.
Abbreviation	A free text that can be used as abbreviation for the new domain.
Name	A free text used as name for the domain.
Elevations	Name of the surface model that is used for determining the elevation ranges for the color scheme.
Level	Level number for displaying triangles in the CAD file.
Draw contours	If on, contour lines are drawn for this domain. This effects the display of contour lines with Display Contours tool.
Colors	Opens the **Color scheme** dialog which is used to define a coloring scheme. See Creating a color scheme of discrete colors for more information.
Color button	Opens the active CAD file color table for single color selection.

Display Domain Triangles

Not Lite, Not UAV



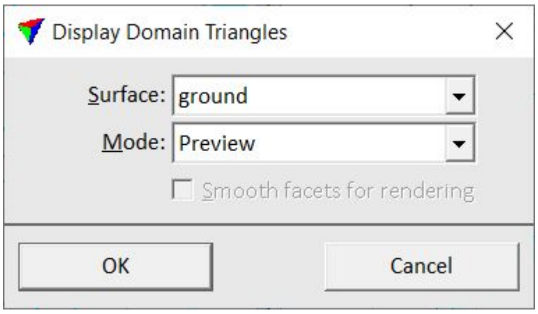
Display Domain Triangles tool draws triangles for a surface model using domain color schemes and levels.

If there are areas of the surface model where no specific domain is assigned, the display options of the default domain are used.

To display domain triangles:

1. Select the **Display Domain Triangles** tool.

   The **Display Domain Triangles** dialog opens:



2. Define settings and click OK.

   The display is created.

SETTING	EFFECT
Surface	Name of the effected surface model.
Mode	Display method for drawing domains: • **Preview** - displays triangles in preview mode. • **Write to file** - draws triangles permanently into the CAD file.
Smooth facets for rendering	If on, triangles are displayed with a smoother change in coloring if CAD rendering is applied (No effect in MicroStation CE or Spatix).

This tool is very similar to the Display Triangles tool. In fact, display domain triangles is an alternate way of displaying surface model triangles. You can update domain triangles with the Update Displays tool or erase them with the Erase Display tool.

Domain View

Not Lite, Not UAV

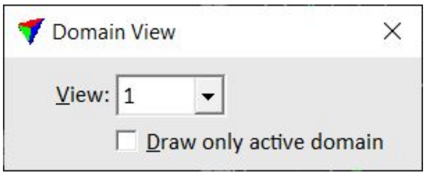
Domain View lets you select a view which displays domain assignments of the surface model you are working on. Whenever a CAD tool redraws the contents of this view, TerraModeler draws the surface model triangles as temporary elements using domain colors.

It is recommended to select a domain view when you are assigning domains to triangles. It provides the best way of seeing the effects of your work.

To assign a domain view:

1. Select the **Domain View** tool.

   The **Domain View** dialog opens:

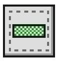


2. Select the **View** in which you want to see domain assignments.

SETTING	EFFECT
Draw only active domain	If on, it speeds up the view update by drawing only the active domain.

Set Area Domain

Not Lite, Not UAV



Set Area Domain tool defines a domain for all triangles inside or outside a fence. The tool uses the center point of a triangle to decide if a triangle is inside or outside the fence.

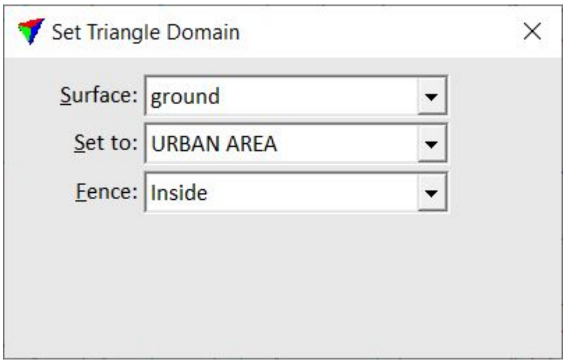
The area can be defined by a fence or selected shape element(s).

To set domain of fence contents:

1. Place a fence or select shape element(s) that define the area.

2. Select the **Set Area Domain** tool.

   This opens the **Set Area Domain** dialog:



3. Define settings.

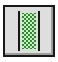
4. Accept the fence contents with a data click.

   The selected domain is assigned to the triangles inside or outside the fence.

SETTING	EFFECT
Surface	Name of the effected surface model.
Set to	Domain that is assigned to the triangles.
Fence	Fence mode: **Inside** or **Outside**.

Set Domain Between Lines

Not Lite, Not UAV

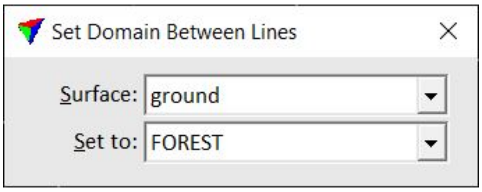


Set Domain Between Lines tool assigns a domain to all triangles between two linear elements. This tool is designed to be used with linear elements that are close to parallel to each other. TerraModeler uses the center point of a triangle to decide if the triangle is between the two linear elements.

To set domain between two linear elements:

1. Select the **Set Domain Between Lines** tool.

   This opens the **Set Domain Between Lines** dialog:



2. Define settings.

3. Identify the first linear element with a data click.

4. Identify the second linear element with a data click.

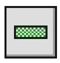
5. Accept the two highlighted linear elements with another data click.

   The selected domain is assigned to the triangles between the two linear elements.

SETTING	EFFECT
Surface	Name of the effected surface model.
Set to	Domain that is assigned to the triangles.

Set Triangle Domain

Not Lite, Not UAV

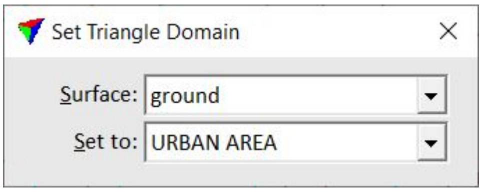


Set Triangle Domain tool assigns a domain to individual triangles. The tool can assign a domain to single triangles or to multiple triangles using a paint method.

To set triangle domains one triangle at a time:

1. Select the **Set Triangle Domain** tool.

2. This opens the **Set Triangle Domain** dialog:



3. Define settings.

4. Move the mouse pointer in a CAD file view.

   The shape of the triangle at the mouse pointer location is dynamically highlighted.

5. To set the domain of a single triangle, place a data click.

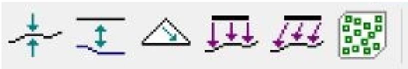
6. If the **Method** is set to **Paint**, move the mouse pointer.

   The domain is assigned to all triangles the mouse pointer passes over. The end the domain assignment, place another data click.

SETTING	EFFECT
Surface	Name of the effected surface model.
Set to	Domain that is assigned to the triangles.
Method	Defines how triangles are selected for domain assignment: • **One at a time** - each triangle is defined by a data click in order to assign the domain. • **Paint** - all triangles the mouse pointer passes over between two data clicks are assigned to the domain.

Draw using Surface toolbox

The tools in the **Draw using Surface** toolbox are used to view surface elevations or slopes, and to draw 3D elements based on a surface model.



TO	USE TOOL	
View surface elevation / place elements on surface		View Elevation
Compare elevations of two surfaces		Compare Elevations
View slope of triangles		View Slope
Drop elements to follow surface		Drop Element On Surface
Calculate and draw slope from an element		Calculate Slope
Fill area with cells		Fill Area With Cells

Calculate Slope

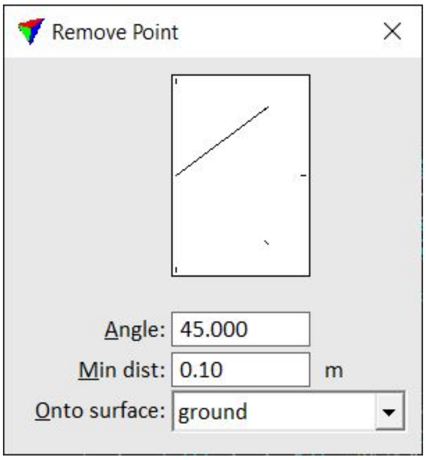
Calculate Slope tool calculates a slope from a graphical element onto a surface model. The starting element is either the top or the bottom edge of a slope. The tool calculates the other edge of the slope and draws it as a new line string element. The new element follows the elevation of a selected surface model.

The starting element has to be linear. Valid CAD element types include lines, line strings, curves, arcs, ellipses, shapes, complex chains and complex shapes.

To calculate slope from an element:

1. Select the **Calculate Slope** tool.

   This opens the **Slope** dialog:



2. Enter a slope angle in the **Angle** field and a minimum distance value in the **Min dist** field.

3. Select a surface to be used for slope calculation.

4. Identify the element to start from.

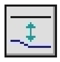
5. Define the slope direction with a data point. The direction is either left or right from the starting element.

   If found, the other edge of the slope is drawn. You can continue to step 2, 3 or 4.

SETTING	EFFECT
Angle	Slope angle in degree. A positive value calculates a slope upwards from the starting element. A negative value calculates a slope downwards. You can enter the angle as ratio of vertical and horizontal distances, for example, 1:3.

SETTING	EFFECT
Min dist	Minimum distance of vertices in the new line string. A mathematical slope may have some vertices very close to each other. For practical reasons, it is recommended to remove some of these vertices.
Onto surface	Name of the surface used to compute the other edge of the slope.

Compare Elevations



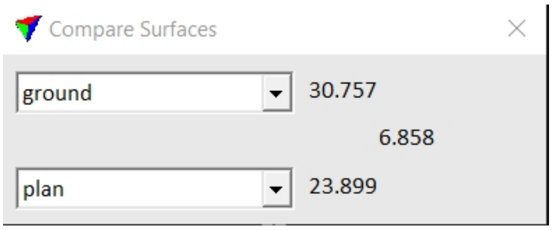
Compare Elevations tool compares elevations of two surfaces at the mouse pointer location. It opens a dialog which shows the elevations of two surfaces and their difference.

The tool requires at least two surfaces loaded in TerraModeler.

To compare surface elevations:

1. Select the **Compare Elevations** tool.

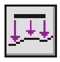
   The **Compare Surfaces** dialog opens:



2. Select the surfaces for comparison.

   If you move the mouse pointer into a top view, the surface elevations at the mouse pointer position are displayed. The dialog also shows the difference between the two selected surfaces.

Drop Element On Surface



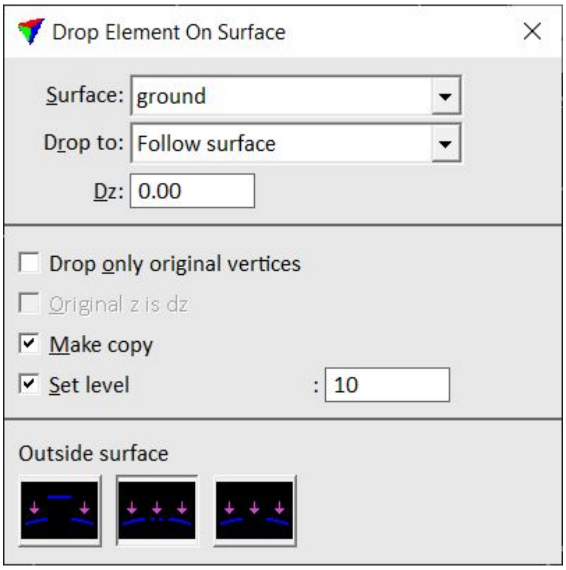
Drop Element On Surface tool drops elements to follow a surface model. The XY shape of an element remains unchanged but its Z shape is forced to follow the elevation of a surface model.

The tool works for all linear elements and cells. A linear element is converted into a line string or a complex chain of line strings. The elevation of the line string vertices is calculated from the surface model. A cell element remains as a cell. It is dropped on the surface by calculating the elevation of its origin point.

To drop element(s) on a surface:

1. Select the **Drop Element On Surface** tool.

   This opens the **Drop Element On Surface** dialog:



2. Define settings in the dialog.

3. Identify the element to drop.

3. Accept the element.

   The element is dropped on the surface. You can continue to step 2 or 3.

   OR

1. Select element(s) to drop.

2. Select the **Drop Element On Surface** tool.

3. Define settings in the dialog.

4. Accept element(s).

   All selected elements are dropped on the surface.

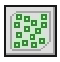
SETTING	EFFECT
Surface	Name of the surface to which elements are dropped.
Drop to	Defines how elevations for the element are calculated: • **Follow surface** - each vertex gets the elevation value of the surface at its location. • **Highest elevation** - all vertices get the elevation value of the highest vertex. • **Lowest elevation** - all vertices get the elevation value of the lowest vertex.
Dz	Elevation offset added to the dropped element. A positive value drops the element above the surface model elevation. A negative value drops the element below the surface model elevation.
Drop only original vertices	If on, only the existing vertices of the element are dropped on the surface. If off, the software adds intermediate vertices to the element to make it follow the surface elevation more accurately. This is only active if **Drop** to is set to **Follow surface**.
Original z is dz	If on, the original elevation values of vertices are applied as elevation offset to the dropped element. This is only active if **Drop only original vertices** is switched on.
Make copy	If on, the original element is not effected. Instead, a copy of the element is created.
Set level	If on, a dropped element is placed on the given level.
Use fence *(Bentley CAD only)*	If on, a fence is used to determine what elements are effected. Bentley CAD fence modes are used: • **Inside** - only elements that are fully inside a fence are dropped. • **Overlap** - elements that overlap with a fence are dropped. • **Clip** - the part of an element that is overlapped by a fence is dropped, other parts are not effected. • **Void** - only elements that are fully outside a fence are dropped. • **Void overlap** - elements that do not overlap with a fence are dropped.

SETTING	EFFECT
	• **Void clip** - the part of an element that is not overlapped by a fence is dropped, overlapped parts are not effected.
Outside surface	Defines the way an element is drawn outside the surface model area. The methods are: • Outside part remains unchanged in elevation but is drawn with active color, line style and line width. • Outside part gets interpolated elevation values and is drawn with active color, line style and line weight. • Outside part is not drawn and thus, results in a gap.

If there are parts of a linear element outside the surface model, the element is split into separate dropped elements. Thus, a line string or a shape is split into single line strings.

Fill Area With Cells

Not Spatix



Fill Area With Cells tool fills a bounded area with randomly placed cells. This tool can be used, for example, to generate a random fill pattern for natural areas such as forests.

The area has to be bounded either by placing a fence or by selecting a shape element.

The tool can be used with two- or three-dimensional cells. You can use surface models to define the bottom elevation or the top elevation of the cells. Typically, a model of the ground is used for the cell bottom elevation.

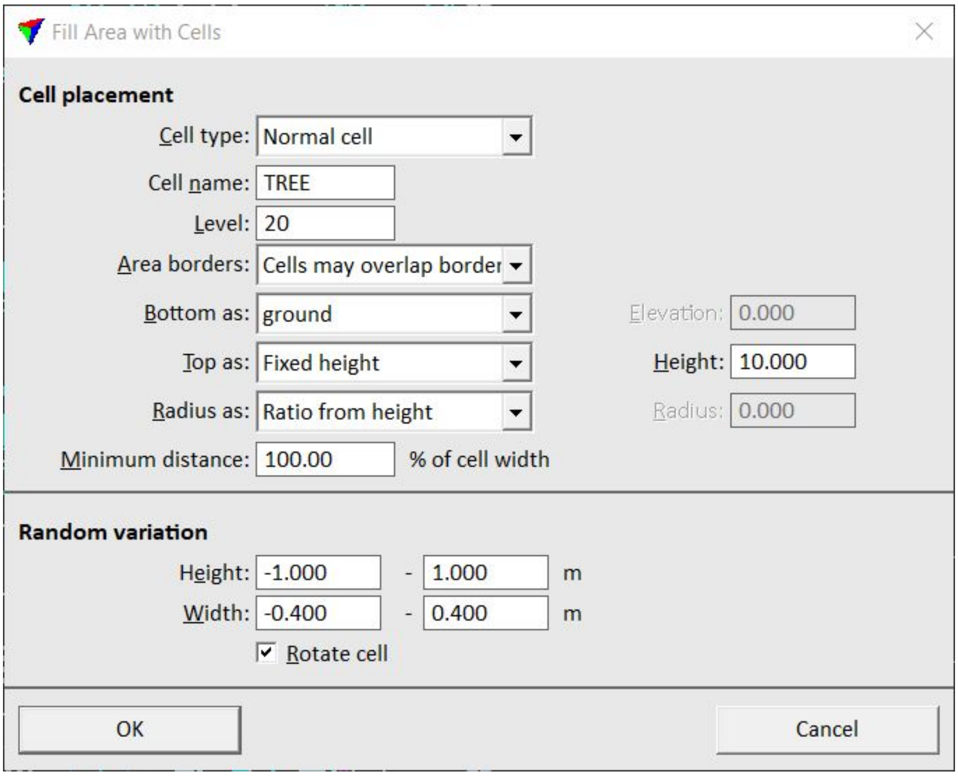
You can define some random variation for the cell dimensions to better resemble naturally grown plants. Variation in cell height, width, and rotation angle may result in a more realistic image.

To fill an area with cells:

1. Define the area by placing a fence or by selecting a shape element.

2. Select the **Fill Area With Cells** tool.

   The **Fill Area With Cells** dialog opens:



3. Define settings in the dialog.

4. Click OK.

5. Accept the area with another data click.

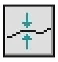
The application fills the area with cells. The process stops when the area is filled.

SETTING	EFFECT
Cell type	Type of the cell for area fill: • **Normal cell** - cells from a MicroStation cell library are used. • **RPC cell** - RPC cells are used.
Cell name	Name of a two- or three-dimensional cell. This is only active if **Cell type** is set to **Normal cell**.
RPC file	Location where RPC files are stored on a hard disk. RPC files are used to replace RPC cells in rendered views. This is only active if **Cell type** is set to **RPC cell**.
Level	Level on which cells are placed.
Area borders	Defines how cells are placed close to the area boundaries: • **Cells completely within borders** - cells have to be completely within the area boundaries. • **Cells may overlap borders** - cells may partially overlap area boundaries.
Bottom as	Name of a surface model for the bottom elevation of cells. If set to **Fixed elevation**, all cells are placed at the given **Elevation**.
Top as	Name of a surface model for the top elevation of cells. If set to **Fixed height**, all cells are drawn with the given **Height**.
Radius as	Defines the width of cells: • **Constant** - use the given **Radius** for all cells. • **Ratio from height** - calculates the width as a direct ratio from the cell height defined in the cell library.
Minimum distance	Defines how close to each other cells are placed. If less than 100 %, some cells may partially overlap each other. For forests, recommended values are between 60 and 120 %.
Height	Random variation for the cell height.
Width	Random variation for the cell width.
Rotate cells	If on, cells are rotated randomly.

RPC cells are not part of Terra applications. They are purchased by Archvision (www.archvision.com) and used in rendered views. These cells include texture maps of an object from different directions and angles. They can be used to render objects like trees, cars, road furniture etc. in a realistic and correct way no matter from which direction the object is viewed.

At the moment, cells and thus, the **Fill Area With Cells** tool does only work in Bentley CAD. There is not yet any corresponding element type in Spatix.

View Elevation

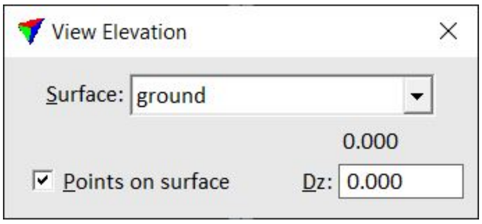


View Elevation tool shows the elevation of a surface model at the mouse pointer location. It also supports the placement of 3D vector elements by deriving surface elevations for the elements.

To view surface elevation:

1. Select the **View Elevation** tool.

   The **View Elevation** dialog opens:

   If you move the mouse pointer inside a view, the **Surface** elevation at the mouse pointer location is displayed in the dialog.

To place elements on surface elevation:

1. Select the **View Elevation** tool.

   The **View Elevation** dialog opens.

2. Select a **Surface** from which to derive the elevation values.

3. Switch on **Points on surface** lock.

   If you want to place elements above or below the surface, enter a **Dz** value. A positive value places the elements above the surface and a negative value below the surface.

4. Start the digitization of a vector element.

   When **Points on surface** lock is on, TerraModeler calculates the elevation from the given surface for all data points entered in a CAD file top view. The data point is effected if the XY location falls inside the surface model area.

   The view used for entering data points can be a top view which has been rotated around the Z-axis.

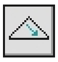
Make sure to turn off the **Points on surface** lock if you do not need it any longer. As it effects all data points, it may interfere with your normal work. The lock is turned off when the **View Elevation** dialog is closed.

The **Points on surface** lock can be switched on/off by using key-in commands/Spaccels:

- *Points On Surface on=0* - switches off the **Points on surface** lock.

- *Points On Surface on=1/dz=0.0* - switches on the **Points on surface** lock and sets the **Dz** value to 0.0.
- *Points On Surface on=1/dz=2.0* - switches on the **Points on surface** lock and sets the **Dz** value to 2.0.

View Slope

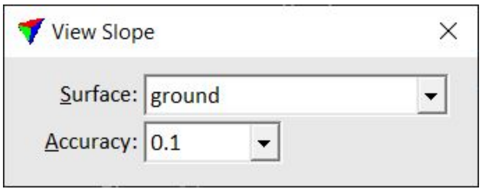


View Slope tool shows the slope gradient and the slope direction of a surface model triangle at the mouse pointer location.

To view slope directions and gradients:

1. Select the **View Slope** tool.

   This opens the **View Slope** dialog:



2. Select a **Surface** model and **Accuracy** setting.

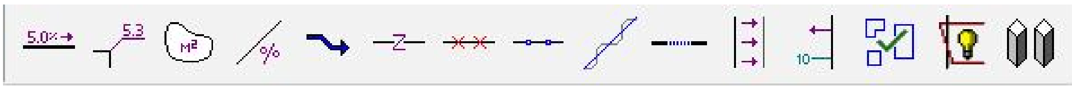
3. Move the mouse pointer inside the surface area.

   If the mouse pointer is inside a surface model, the triangle at the mouse pointer position is highlighted, its slope direction is displayed as an arrow, and its gradient is displayed as numerical percentage value in the status bar of the CAD platform.

SETTING	EFFECT
Surface	Name of the surface for slope gradient and direction display.
Accuracy	Number of decimals for the slope gradient display.

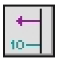
Drawing Utilities toolbox

The tools in the **Drawing Utilities** toolbox are used to measure, label and manipulate linear elements. The tools do not necessarily rely on surface models.



TO	USE TOOL
Measure slope of an element	Measure Slope
Label elevation of a point	Label Elevation
Label surface area inside element	Label Area
Place line at a fixed slope	Place Sloped Line
Modify linear element elevations to ensure downstream flow	Force Downstream Flow
Set linear element to given elevation	Set Linear Elevation
Remove unnecessary vertices from linear element	Thin Linear Element
Add intermediate vertices to linear element	Densify Linear Element
Apply smoothing to linear element	Smoothen Linear Element
Connect linear elements that have small gaps in between	Connect Linear Elements
Create a parametrized copy of linear element	Copy Linear Element
Apply alignment offset and station locks	Alignment Offset
Check linear elements for geometrical flaws	Validate Linear Elements
Fix vertices almost touching another element	Fix Touching Elements
Import and draw piles into the CAD file	Read Piles From File

Alignment Offset



Alignment Offset tool provides an easy way to place elements at a specific station or at a specific 3D offset from an alignment element. The tool can use any linear element as the alignment.

Alignment Offset tool opens a dialog which dynamically displays station and offset values at the mouse pointer location. You can lock data points by station, offset, elevation difference, or gradient from the selected alignment.

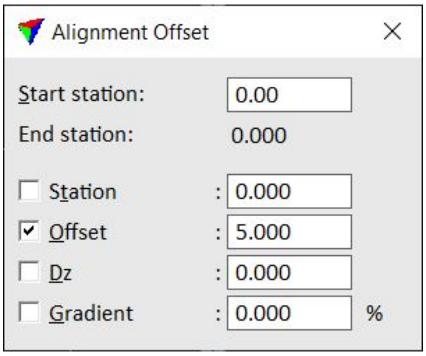
General procedure for applying alignment locks:

1. Select the **Alignment Offset** tool.

2. Identify the alignment element.

3. Accept the element.

   This opens the **Alignment Offset** dialog:



4. Switch on the locks which you want to apply.

5. Select any CAD element placement tool you want to use.

   When you enter a data point, its coordinates are adjusted according to the alignment offset locks and related to the selected alignment element.

SETTING	EFFECT
Start station	Station value for the start point of the alignment. Default is zero.
Station	If on, the XY location of a data point is adjusted to the given station position.
Offset	If on, the XY location of a data point is adjusted to the given offset from the alignment. The data point determines whether the offset is computed to the left or to the right of the alignment.

SETTING	EFFECT
Dz	If on, the Z value of a data point is adjusted by adding this elevation difference to the elevation of the alignment element.
Gradient	If on, the Z value of a data point is adjusted by using this gradient percentage from the elevation of the alignment element.

Make sure to turn off **Alignment Offset** locks if you do not need it any longer. As it effects all data points, it may interfere with your normal work. The lock is turned off when the **Alignment Offset** dialog is closed.

Connect Linear Elements

Connect Linear Elements tool creates a continuous line string from two or more single line (string) elements. The line string creation considers the XY and Z coordinates of the vertices of the original line elements. The new line string is created on the active CAD file level using the active symbology settings.

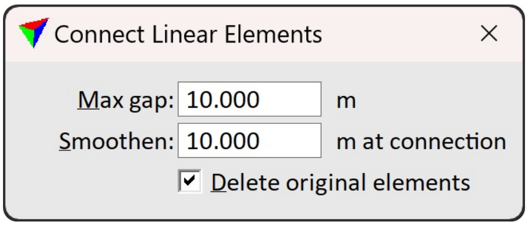
The tool can be used, for example, to create a continuous line from separate line elements that were extracted automatically, such as paint markings extracted from mobile laser scanner data in TerraScan.

To connect linear elements:

1. Select line elements.

2. Select the **Connect Linear Elements** tool.

   This opens the **Connect Linear Elements** dialog:



3. Define settings.

4. Place a data click inside a CAD file view in order to accept the line string creation.

   This draws the new line string into the CAD file.

SETTING	EFFECT
Max gap	Maximum gap between two adjacent line elements that is closed by the tool. Measured as true 3D distance between vertices.
Smoothen	Determines the level of smoothing in XY and Z applied to the new line string. The larger the value, the more smoothing is applied.
Delete original elements	If on, the original segments are removed when the continuous line is created.

Each data click inside a CAD file view creates a new line string from the selected elements as long as the tool is active. This may lead easily to duplicated lines created from the same selection set. You can undo the creation of the new line string by using the **Undo** command from the **Edit** pulldown menu of the CAD platform.

Copy Linear Element

Copy Linear Element tool creates a parametrized copy of a linear element. Furthermore, the tool can be used to move a linear element in xy and/or z direction without keeping the original element.

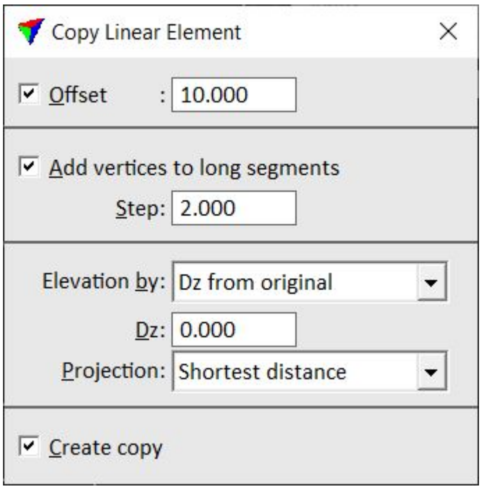
The elevation of the new element may be derived in several ways:

- no change in elevation
- set elevation to a fixed value
- use a given elevation difference from the original element
- use a given elevation difference or a gradient from another linear element

To create a copy of a linear element:

1. Select the **Copy Linear Element** tool.

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



2. Define settings.

3. Identify the original element which determines the horizontal shape.

4. Identify another element if the vertical shape of the new element is determined from another element.

5. Select the offset side (left or right) with a data click.

   A linear copy of the original element is created or the original element is moved to its new location.

SETTING	EFFECT
Offset	If on, the horizontal offset from the original element to the new element is fixed. If off,

SETTING	EFFECT
	the offset is determined by the last data click of the workflow.
Add vertices to long segments	If on, the application adds intermediate vertices to long line segments. This is useful to ensure that the created copy follows both the horizontal shape of the original element and the vertical shape of another element.
Elevation by	Defines the method of elevation value derivation for the new element: • **Keyin value** - constant elevation given as a numerical value in the **Elevation** field. • **Elevation point** - constant elevation given by a data click. • **Dz from original** - elevation difference from the original element given in the **Dz** field. • **Dz from another element** - elevation difference from another element given in the **Dz** field. • **Gradient from another element** - gradient percentage from another element given in the Gradient field.
Projection	Defines how the elevation of the new element is derived from another element: • **Shortest distance** - each vertex is compared against the closest position on the other element. • **Perpendicular** - each vertex is projected perpendicularly to the other element and the elevation is derived from the projected location. Only active if **Elevation by** is set to **Dz from another element** or **Gradient from another element**.
Create copy	If on, a copy of the original element is created. The original element remains unchanged.

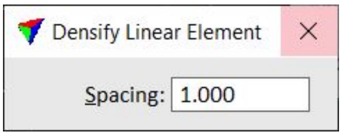
Densify Linear Element

Densify Linear Element tool adds intermediate vertices to linear elements. A spacing value determines the desired distance between consecutive vertices. The tool adds vertices if the distance between two consecutive vertices in the original element is longer than **1.333 * Spacing**.

To densify linear element(s):

1. Select the **Densify Linear Element** tool.

   The **Densify Linear Element** dialog opens:

2. Define a **Spacing** value that determines the distance between consecutive vertices.

3. Identify the element to densify.

4. Accept the element.

   Intermediate vertices are added to the selected element.

   OR

1. Select linear element(s) to thin.

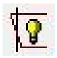
2. Select the **Densify Linear Element** tool.

3. Define a **Spacing** value that determines the distance between consecutive vertices.

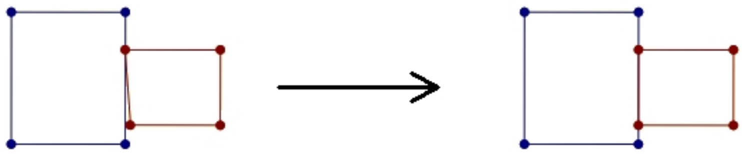
4. Accept the element(s) with a data click.

   Intermediate vertices are added to all selected element(s).

Fix Touching Elements



Fix Touching Elements tool tries to fix selected linear elements in places where elements intersect or almost touch each other. As a result, the elements are snapped to each other as shown in the following illustration.

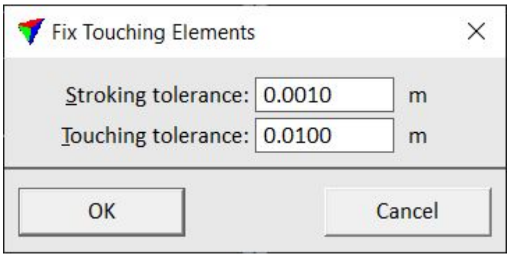


You may run the Validate Linear Elements tool with validation method **Touching** on in order to find out if there are such issues in a set of vector elements.

To fix touching elements:

1. Select elements.
2. Select the **Fix Touching Elements** tool.

   This opens the **Fix touching elements** dialog:



3. Define settings and click OK.

   This fixes touching elements according to the given settings and moves them to the active CAD file level. An information dialog shows the number of effected elements.

SETTING	EFFECT
Stroking tolerance	Tolerance distance for checking intersecting elements. Two elements are considered as touching if the location of a vertex of one element differs more than the given value from a vertex or line of the other element.
Touching tolerance	Minimum distance between separate elements. If the distance is smaller, elements are considered as touching elements.

Force Downstream Flow

Force Downstream Flow tool modifies the elevation values of vertices of linear elements to enforce a consistent slope direction down for the linear element. The tool is useful to ensure that water stream breaklines run down all the time without small upward changes. The tool only modifies elevations of vertices if this is required to enforce the consistent downward direction.

To enforce that linear element(s) go down consistently:

1. Select the **Force Downstream Flow** tool.

2. Identify the element with a data click.

3. Accept the element selection with another data click.

   This modifies the elevation values of the vertices of the selected linear element.

   OR

1. Select linear element(s).

2. Select the **Force Downstream Flow** tool.

3. Accept the selected element(s) with a data click.

   This modifies the elevation values of the vertices of all selected elements.

Label Area

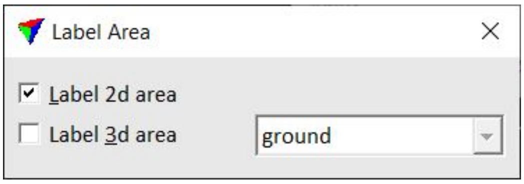
Label Area tool measures and labels the size of a bounded area. The tool can be used to label the area of closed elements in the CAD file.

The tool places the area size value and possibly some prefix and/or suffix. The settings for the tool are defined in Drawing utilities / Label Area category of the TerraModeler **Settings**. The symbology of the label is further determined by the active symbology settings in the CAD file.

To create area labels:

1. Select the **Label Area** tool.

   This opens the **Label Area** dialog:



2. Select the area type for labeling in the dialog.

3. Identify the bounding element which defines the area.

   The label is displayed at the mouse pointer location.

4. Define the location of the label with another data click.

   This draws the label in the CAD file. You can continue with step 2 or 3.

   OR

1. Select the bounding element(s) which defining the area.

2. Select the **Label Area** tool. This opens the **Label Area** dialog.

3. Select the area type for labeling in the dialog.

4. Accept the selected element(s) with a data click.

   This draws the label(s) in the CAD file. Application places labels inside bounding elements.

SETTING	EFFECT
Label 2d area	The tool measures and labels the 2d area defined by the closed element.
Label 3d area	The tool measures and labels the 3d area defined by the closed element that is projected on the given **Surface**.

Label Elevation

Label Elevation tool creates an elevation label for a data point. The tool can be used to label the elevation of points on vector elements in the CAD file.

The tool places the elevation value of the data point and a guiding line which connects the data point and the text. The settings for the tool are defined in Drawing utilities / Label Elevation category of the TerraModeler **Settings**. The symbology of the label is further determined by the active symbology settings in the CAD file.

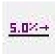
To create elevation labels:

1. Select the **Label Elevation** tool.
2. Identify the data point, for example, by snapping to a linear element.

   The label is displayed at the mouse pointer location.
3. Enter the location of the label with another data click.

   This draws the label in the CAD file. You can continue with step 2 or 3.

Measure Slope



Measure Slope tool measures and labels the average vertical slope of a linear element. The slope is calculated using the following mathematical expression:

```
Slope = 100 * (End z - Start z) / Horizontal length of element
```

Even if the element is a line string, the tool does not use the individual segments of the element. The slope is always calculated from the element's start point to its end point.

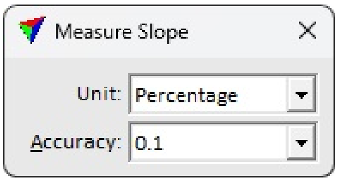
The element to be measured has to be linear. Valid CAD element types include lines, line strings, curves and arcs.

The tool places the slope gradient as a percentage value and the slope direction as an arrow close to the linear element and at the mouse click location. The settings for the label symbology are defined in Drawing utilities / Measure Slope category of the TerraModeler **Settings**.

To measure the slope of an element:

1. Select the **Measure Slope** tool.

   This opens the **Measure Slope** dialog:



2. Define settings.

3. Identify the element. The data click also defines the location of the label.

   The slope percentage and the slope direction are displayed in the status bar of the CAD platform.

4. Place a data click to accept the labeling or a reset click to reject the labeling. You can continue to step 2 or 3.

SETTING	EFFECT
Unit	Determines the unit of the label value: • **Percentage** - percentage ratio between elevation change and horizontal distance. • **Degree** - slope angle in degrees. • **Ratio 1:x** - ratio between elevation change and horizontal distance.
Accuracy	Decimal precision of the label. Options from zero up to four decimals.

Place Sloped Line

Place Sloped Line tool places a 3D line at a fixed vertical slope. You enter the start point of the line as a normal data point. The second data point defines the XY position of the end point. The elevation of the end point is calculated using the given slope or the given end point elevation.

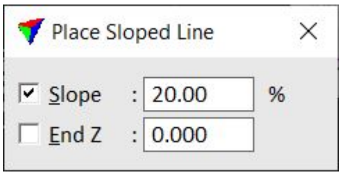
The slope is expressed as a percentage by the following mathematical expression:

```
Slope = 100 * Vertical distance / Horizontal distance
```

To place a line at a fixed slope:

1. Select the **Place Sloped Line** tool.

   This opens the **Place Sloped Line** dialog:

2. Fix the **Slope** and/or **End Z** value. If both values are fixed, the length of the line is defined by the two values. If only one value is fixed, step 4 defines the end point of the line.

3. Place the start point of the line with a data click.

4. Place the end point of the line with a data click.

   You can continue with step 2 or 4. The finish the line, click the reset mouse button.

Read Piles From File

Read Piles From File tool imports and draws pile locations into the CAD file. "Pile" refers to piles or pillars that are slammed into the ground for stabilizing ground, e.g. close to construction sides, roads, etc. When such an object is placed, its location is also surveyed with either top + bottom Z or top Z + length. TerraModeler can read a text file with the following attributes for each pile:

- **Easting** - X coordinate of the pile. Mandatory attribute.
- **Northing** - Y coordinate of the pile. Mandatory attribute.
- **Top Z** - top elevation coordinate of the pile. Mandatory attribute.
- **Base Z** - base elevation coordinate of the pile. Mandatory attribute if length is not provided.
- **Length** - length of the pile. Mandatory attribute if Base Z is not provided.
- **Width** - width of the pile.
- **Number** - number of the pile.

The tool draws a cylindrical cell element in the CAD file if the pile has a circular base shape or a cell with polygons on each sides if the base shape is rectangular.

The tool aims for checking tasks (are the piles placed correctly) and for monitoring. Later ground surface surveys may reveal changes of the ground and then, the pile drawings can be used to check if the ground changes because of problems with the piles.

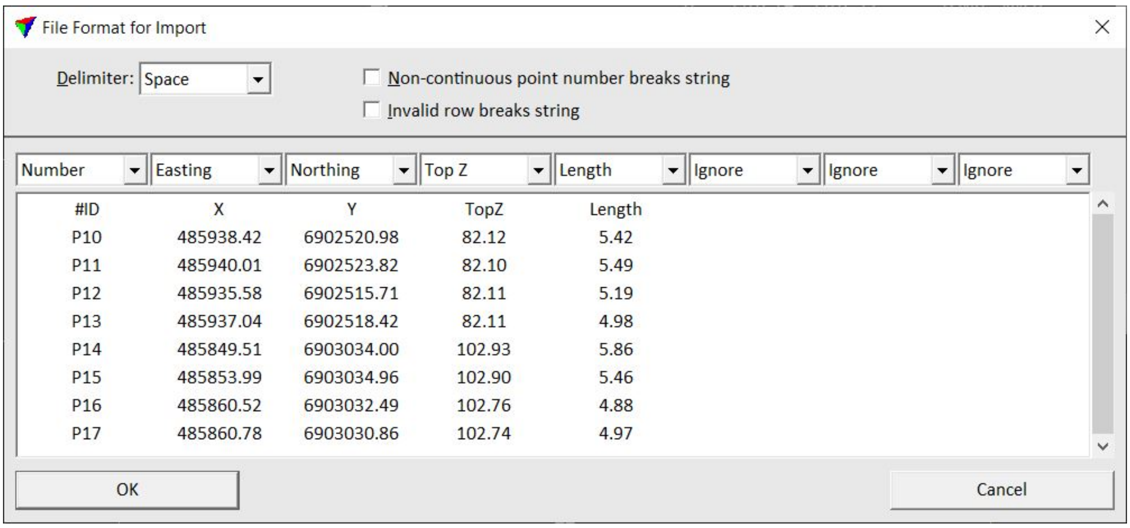
To read piles from a text file:

1. Select the **Read Piles from File** tool.

   This opens the first **Read piles from file** dialog, a standard dialog for selecting files.

2. Select the text file that stores the pile attributes and click **Open**.

   This opens the **File Format for Import** dialog:



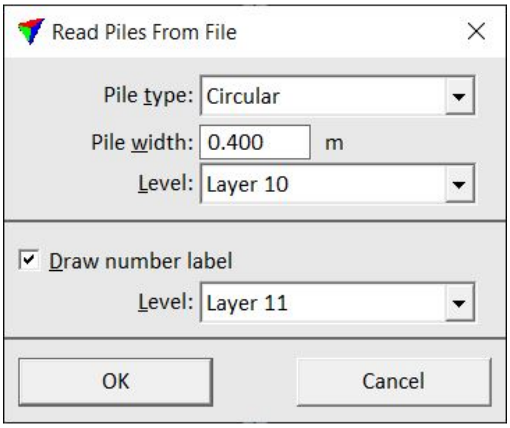
The dialog shows the first rows of the text file.

3. Check the **Delimiter** and select the correct one if the automatic selection is incorrect.

4. Select the correct attributes for each column in the text file. Select **Ignore**, if a column from the text file must not be imported.

5. Click OK.

   This opens another **Read Piles From File** dialog:

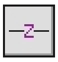


6. Define settings and click OK.

   This draws the pile elements into the CAD file. The elements (and labels) are drawn on the given level(s) using the active symbology of the CAD file.

SETTING	EFFECT
Pile type	Shape of the pile: **Circular** or **Rectangular**.
Pile width	Diameter or edge length of a pile. The value is used if no width value for each pile is provided in the text file.
Level	CAD file level on which the pile elements are drawn.
Draw number label	If on, the number attribute is drawn on the given **Level**. This requires a number attribute for each pile in the text file.

Set Linear Elevation

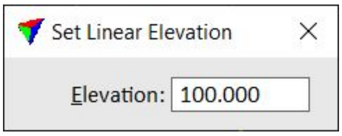


Set Linear Elevation tool adjusts a linear element to a given elevation. It sets a constant elevation for each vertex of the element.

To set the elevation of linear element(s):

1. Select the **Set Linear Elevation** tool.

   The **Set Linear Elevation** dialog opens:

2. Define an **Elevation** value.

3. Identify the element to adjust with a data click.

4. Accept the element with another data click.

   The element is adjusted to the given elevation. You can continue to step 2 or 3.

   OR

1. Select element(s) to adjust.

2. Select the **Set Linear Elevation** tool.

3. Define an **Elevation** value.

4. Accept element(s) with a data click.

   All selected element(s) are adjusted to the given elevation.

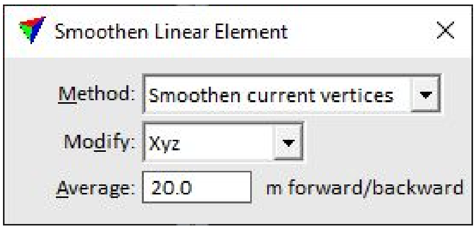
Smoothen Linear Element

Smoothen Linear Element tool can be used to smooth linear elements with a high vertex density. It has been developed to smoothen automatically extracted rail lines created by the Find rails tool of TerraScan. However, it may be used for any linear element. The smoothing is done by fitting vertices to an average line or circular arcs within a given distance.

To smooth linear element(s):

1. Select element(s) to smooth.
2. Select the **Smoothen Linear Element** tool.

   The **Smoothen Linear Element** dialog opens:



3. Define settings.
4. Accept element(s) with a data click.

   All selected element(s) are smoothed.

SETTING	EFFECT
Method	Determines the smoothening method: • **Smoothen current vertices** - based on line fitting equation. • **Add vertices at spacing** - based on B-Spline curvature. • **Add vertices by turn angle** - based on B-Spline curvature. The method is best suited for creating smooth-looking turns with a small radius.
Modify	Element dimension to modify in the smooth process: • **Xyz** - horizontal and vertical variation of vertices is modified. • **Xy** - only horizontal dimension. The vertical variation of vertices is not modified. • **Z** - only vertical dimension. The horizontal dimension of vertices is not modified. This is only active if **Method** is set to **Smoothen current vertices**.

SETTING	EFFECT
Average	Forward/Backward distance for the fitting algorithm. Within this distance the vertices are fitted to an average line or circular arc. This is only active if **Method** is set to **Smoothen current vertices**.
Spacing	Distance between consecutive vertices along the smoothed element. This is only active if **Method** is set to **Add vertices at spacing**.
Max turn	Maximum angle between consecutive line segments along the smoothed element. This is only active if **Method** is set to **Add vertices by turn angle**.

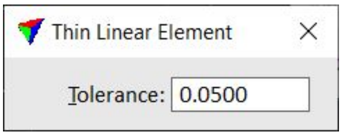
Thin Linear Element

Thin Linear Element tool removes unnecessary vertices from linear elements. A tolerance value determines the largest allowed 3D distance between the original and the resulting linear element.

To thin linear element(s):

1. Select the **Thin Linear Element** tool.

   The **Thin Linear Element** dialog opens:

2. Define a **Tolerance** value that determines how much the element is allowed to move.

3. Identify the element to thin with a data click.

4. Accept the element with another data click.

   All unnecessary vertices are removed from the selected element.

   OR

1. Select linear element(s) to thin.

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

3. Define a **Tolerance** value that determines how much the element(s) are allowed to move.

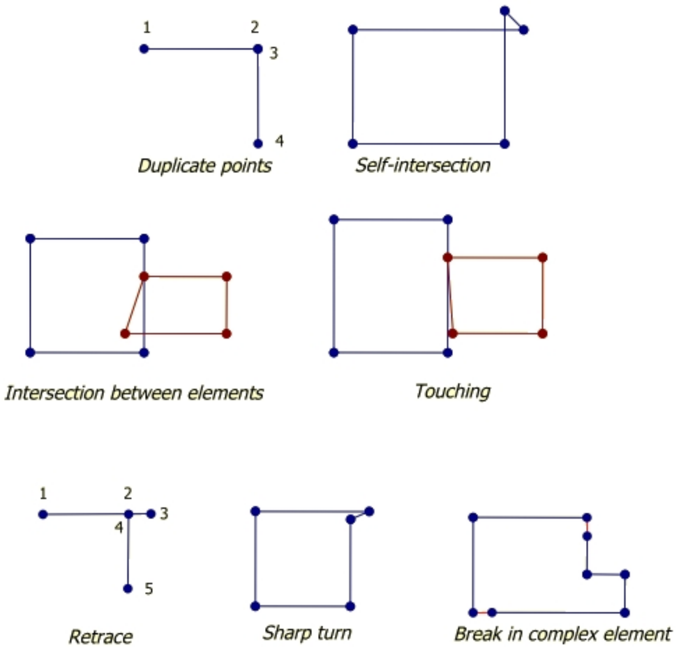
4. Accept the element(s) with a data click.

   All unnecessary vertices are removed from selected element(s).

Validate Linear Elements

Validate Linear Elements tool can be used for checking linear elements for geometrical flaws. It is intended to be used, for example, for checking vector data from existing maps (such as building footprints) before it is used for further processing or for validating newly created vector data before delivery.

The following figure illustrates the geometrical flaws that can be found with the tool.



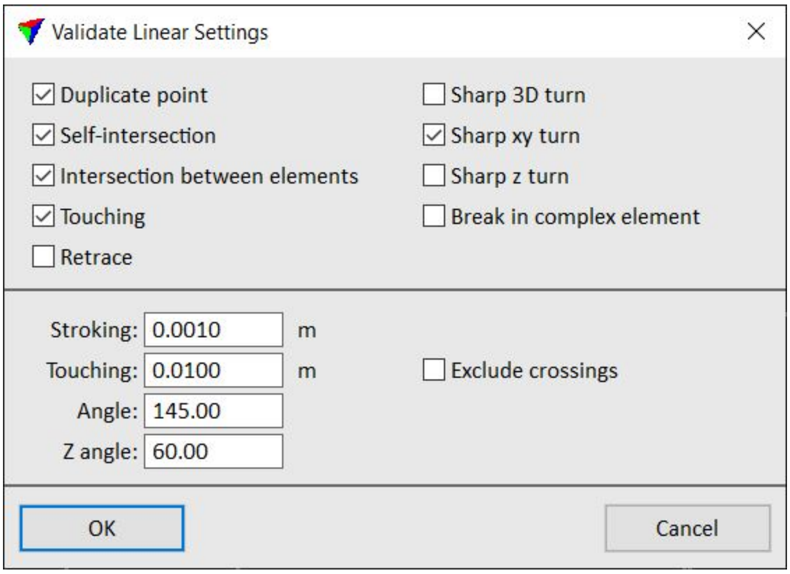
The tool opens a dialog with a list of all geometry flaws that it found according to the selected settings. In this dialog, there are user controls for checking the issues in an organized way.

To validate linear elements:

1. Select elements.

2. Select the **Validate Linear Elements** tool.

   This opens the **Validate Linear Settings** dialog:



3. Select which geometrical flaws you want to validate.

4. If applicable, define additional settings.

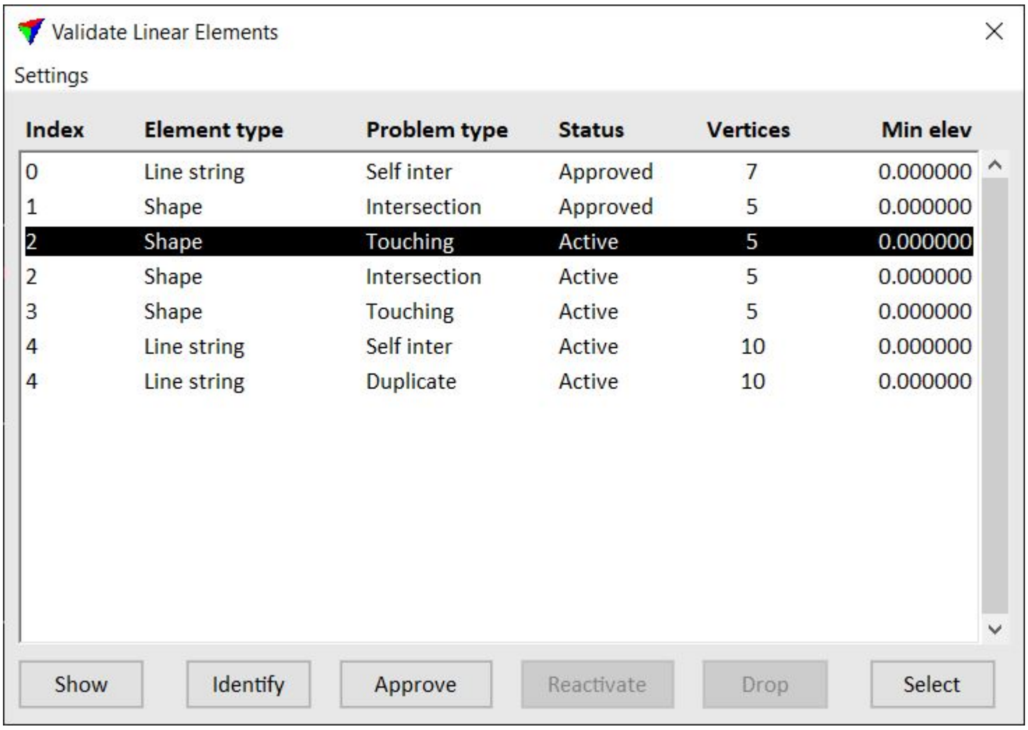
5. Click OK.

This opens the Validate Linear Elements dialog, a dialog that helps you to check the geometrical flaws in the vector data.

SETTING	EFFECT
Stroking	Tolerance distance for checking the geometry. A geometrical flaw is detected if the location of a vertex differs more than the given value from other vertices or lines.
Touching	Minimum distance between separate elements. If the distance is smaller, elements are detected as touching elements.
Angle	Determines what is a sharp XY turn in the geometry. A value of 145 means that the smallest allowed angle between two lines at a vertex is 35 degrees. Smaller angles are considered a sharp turn. Values can range from 0 to 180.
z-Angle	Determines what is a sharp Z turn in the geometry. Defined in the same way as **Angle**. Values can range from 0 to 180 degree.
Exclude crossings	If on, the **Touching** validity check ignores places where one line string element crosses another line string element at least two times.

Validate linear elements dialog

The **Validate Linear Elements** dialog shows a list of geometrical flaws that the tool found in the selected elements. Further, it provides user controls for checking the problems in an organized way. CAD tools can be used to fix the geometry problems.



Index - unique ID number of a vector element. One element can have several geometry flaws.

Element type - CAD element type. Valid element types include shapes, complex shapes, ellipses, lines, line strings.

Problem type - geometry flaw.

Status - status of the element. Can be **Active**, **Approved**, or **Dropped**.

Vertices - number of vertices of the element.

Min. elevation - minimum elevation of an element. This column can be changed to show the maximum elevation or the elevation range of vertices of an element.

Show - highlights the selected problem location or element. Select a row in the list and click on the button. Move the mouse pointer inside a CAD file view. This highlights a problem location by drawing small squares around vertices. Place a data click inside a view in order to center the problem location/element in the view.

Identify - selects a problem location or element in the list. Click on the button and place a data click close to an element in a CAD file view. The problem location or element closest to the data click is selected in the list.

Approve - sets the status of a problem element to **Approved**. This should be used after a geometrical flaw has been fixed.

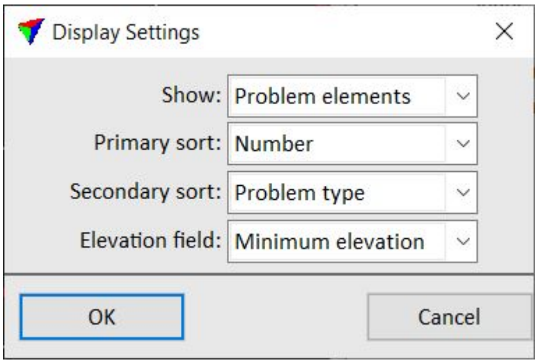
Reactivate - sets the status of a problem element to **Active**. The button is only available if an element with status **Approved** is selected in the list.

Drop - drops a complex element into its single elements. Dropping a complex element is necessary to fix a gap (break). The button is only available if an element with problem type **Break** and status **Active** is selected. The status of the dropped element is changed to **Dropped**.

Select - selects the geometry of the element that is selected in the list. Select a row in the list and click on the button. This selects the element. You may use the **Show** button in order to move the view to the location of the element.

Processing settings command in the **Settings** pulldown menu lets you view and change the settings for validating elements. The command opens the **Validate linear elements processing** dialog described above. After changing settings, the list of elements in the **Validate linear elements** dialog is updated automatically.

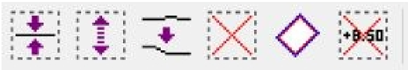
View settings command in the **Settings** pulldown menu define the content and order of elements in the **Validate linear elements** dialog.



SETTING	EFFECT
Show	Determines what elements are displayed in the list: **All elements** or **Problem elements**.
Primary sort	The elements are sorted primarily according to the given attribute.
Secondary sort	The elements are sorted secondarily according to the given attribute.
For elevation show	Determines what vertex elevation value is shown in the last column of the list: **Minimum elevation**, **Maximum elevation** or **Elevation change**.

Edit Area toolbox

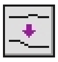
The tools in the **Edit Area** toolbox are used to modify surface model points or triangles inside or outside an area. The area is defined by a fence or a shape element. Valid element types include shapes, complex shapes and ellipses.



TO	USE TOOL
Flatten points to a fixed elevation	Flatten Area
Raise / lower points by a vertical distance	Move Area
Drop / lift points to follow another surface	Drop Area
Exclude triangles	Exclude Area
Insert elements as internal hole boundaries	Insert Hole Element
Remove points from the model	Remove Area

These tools considers the area boundary usually as a two-dimensional boundary. The Z coordinates of both, boundary vertices and surface model points are ignored when deciding what points are inside or outside the area.

Drop Area



Drop Area tool modifies the elevations of surface model points by dropping them to other selected surfaces. The direction of movement is either up or down:

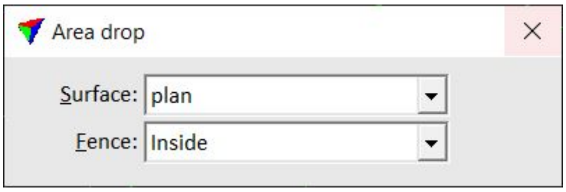
- **Down** - a point in the active surface is modified if one of the selected surfaces is located below that point's elevation. The elevation of the point is set to the highest elevation of the surfaces below the point.
- **Up** - a point in the active surface is modified if one of the selected surfaces is located above that point's elevation. The elevation of the point is set to the lowest elevation of the surfaces above the point.

The tool drops points inside or outside a fence. The fence can be defined by a fence or selected shape element. Valid element types include shapes, complex shapes and ellipses.

To drop area points:

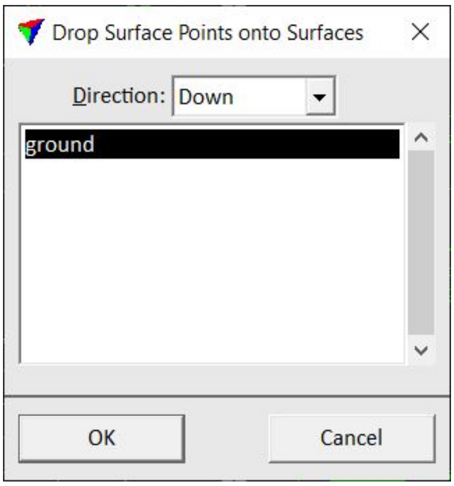
1. Place a fence or a shape element to define the area. (Optional) Select the shape.
2. Select the **Drop Area** tool.

   The **Area drop** dialog opens:



3. Select the active **Surface** from which to drop points and a **Fence** mode.
4. Identify the bounding shape for the area, if a shape element is used and not selected already.
5. Accept the fence contents with a data click.

   The **Drop Surface Points onto Surfaces** dialog opens:



1. Select a **Direction** and surface(s) to which points are dropped.
2. Click OK.

   The points are dropped to the surface(s).

Exclude Area

Exclude Area tool excludes or includes surface model triangles. The tool changes the status of all the triangles inside or outside a fence. The application uses the center point of a triangle to determine whether a triangle is inside or outside the fence.

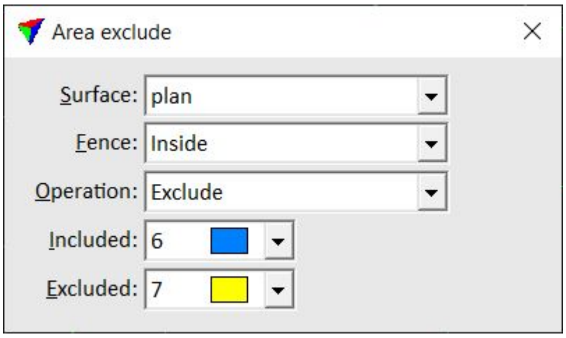
The fence can be defined by a fence or one or multiple shape element(s). Valid element types include shapes, complex shapes and ellipses.

To include/exclude triangles inside an area:

1. Place a fence or a shape element(s) to define the area. (Optional) Select the shape(s).

2. Select the **Exclude Area** tool.

   The **Area exclude** dialog opens:



3. Select settings for modifying the status of triangles.

4. Identify the bounding shape for the area, if a shape element is used and not selected already.

5. Accept the fence contents with a data click.

   The status of the triangles is modified.

SETTING	EFFECT
Surface	Name of the effected surface model.
Fence	Defines where triangles are effected: • **Inside** - status of triangles inside the fence is changed. • **Outside** - status of triangles outside the fence is changed.
Operation	Status to which the triangles are changed: **Exclude** or **Include**.
Included	Highlight color for included triangles. Uses the active color table of the CAD file.

SETTING	EFFECT
Excluded	Highlight color for excluded triangles. Uses the active color table of the CAD file.

The tool effects triangles in the same way as the Exclude Triangle tool. Thus, you may include triangles with the **Exclude Area** tool that were previously excluded with the **Exclude Triangle** tool or vice versa.

Flatten Area

Flatten Area tool flattens surface model points to a fixed elevation. The tool sets the elevation of all points inside or outside a fence.

You can enter the new elevation as a key-in value or as a data point.

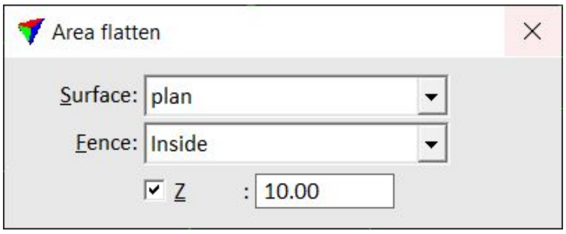
The fence can be defined by a fence or shape element. Valid element types include shapes, complex shapes and ellipses.

To flatten area points to a fixed elevation:

1. Place a fence or a shape element to define the area. (Optional) Select the shape.

2. Select the **Flatten Area** tool.

   The **Area flatten** dialog opens:



3. Define settings.

4. Identify the bounding shape for the area, if a shape element is used and not selected already.

5. Accept the fence contents with a data click. The data point determines the new elevation value for the surface points if **Z** is switched off.

   The points are moved to the given elevation.

SETTING	EFFECT
Surface	Name of the effected surface model.
Fence	Defines where points are effected: • **Inside** - points inside the fence are modified in elevation. • **Outside** - points outside the fence are modified in elevation.
Z	Elevation value to which points are moved. If on, the points are fixed to the given elevation value. If off, points are fixed to an elevation value given by the data point in step 5.

Flatten Area tool can be used for creating a flat plane inside a model. However, this operation alone does not produce a flat area with an exact boundary because the tool does not clip the triangles overlapping the fence border. The tool effects only the points inside the area. After the operation the surface model may start sloping down or up within the area boundary.

In order to create an exactly bounded flat area, you have to add the boundary lines to the surface model and make sure that triangles do not overlap the boundary.

General procedure for creating a flat plane:

1. Place a shape element to define the area.

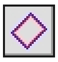
2. Set the elevation of the shape element using the Set Linear Elevation tool.

3. Add the shape element as **Hard breakline** points to the surface model using the Insert Breakline Element tool.

4. Select the shape element.

5. Start the **Flatten Area** tool to fix all the points inside the bounding element to the desired elevation.

Insert Hole Element



Insert Hole Element tool creates a void area inside the surface model by excluding all triangles inside a closed element. The bounding element can serve as a 2d boundary or as a 3d boundary:

- **2d boundary** - the elevation of the element has no effect. The two dimensional shape of the element is combined with the existing elevation of the surface model to create a boundary.
- **3d boundary** - the three dimensional vertices of the element are combined with the surface model to create a boundary.

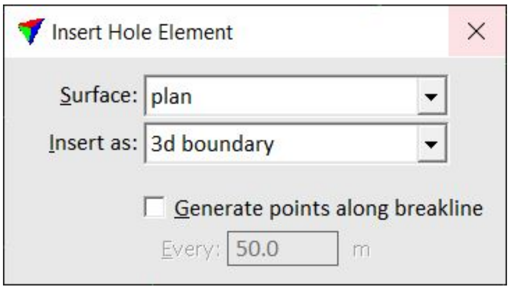
The boundary element has to be a closed linear element. Valid element types include shapes, complex shapes and ellipses.

The tool modifies the triangle network of the surface by adding addition vertices along the boundary element.

To insert element(s) to a surface as hole boundaries:

1. Select the **Insert Hole Element** tool.

   The **Insert Hole Element** dialog opens:



2. Define settings for the hole area.

3. Identify the boundary element to insert with a data click.

4. Accept the element with another data click.

   TerraModeler inserts the element to the surface and creates a void area inside. You can continue to step 2 or 3.

   OR

1. Select element(s) to insert as holes.

2. Select the **Insert Hole Element** tool.

3. Accept the element(s) with a data click.

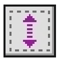
   TerraModeler inserts the element(s) to the surface and creates void area(s) inside.

SETTING	EFFECT
Surface	Name of the effected surface model.

SETTING	EFFECT
Insert as	Set the boundary type: • **2d boundary** - use only the two dimensional shape of the element. • **3d boundary** - insert vertices using the elevation of the element.
Generate points along breakline	If on, additional intermediate vertices are added to the boundary element at the given **Every** distance. This is only active if **Insert as** is set to **3d boundary**.

The tool changes the status of the triangles inside the void area to excluded. You may include triangles with the Exclude Area tool or the Exclude Triangle tool in order to remove the void area.

Move Area



Move Area tool moves surface model points up or down by a given distance. The tool modifies the elevation of all points inside or outside a fence.

You can enter the distance as a key-in value or with a data point.

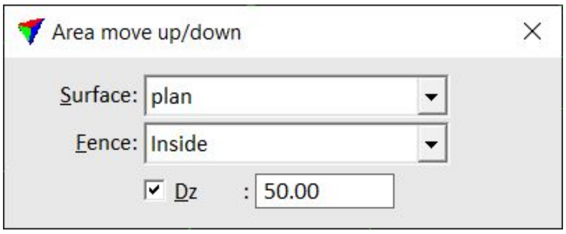
The fence can be defined by a fence or shape element. Valid element types include shapes, complex shapes and ellipses.

To move area points:

1. Place a fence or a shape element to define the area. (Optional) Select the shape.

2. Select the **Move Area** tool.

   The **Area move up/down** dialog opens:



3. Define settings.

4. Identify the bounding shape for the area, if a shape element is used and not selected already.

5. Accept the fence contents with a data click. The data point determines the distance for moving the points if **Dz** is switched off.

   The points are moved up or down.

SETTING	EFFECT
Surface	Name of the effected surface model.
Fence	Defines where points are effected: • **Inside** - points inside the fence are modified in elevation. • **Outside** - points outside the fence are modified in elevation.
Dz	Vertical distance by which the points are moved. A positive value moves points upwards, a negative value downwards. If on, the points are moved by the given elevation difference value. If off, points are

SETTING	EFFECT
	moved by the distance defined with the last data click.

Remove Area

Remove Area tool deletes points from a surface model. This tool removes all points inside or outside a fence.

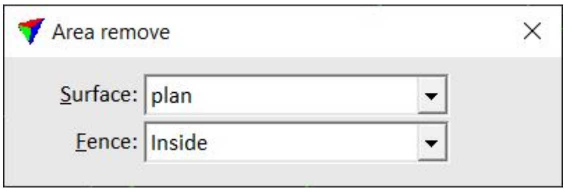
The fence can be defined by a fence or shape element. Valid element types include shapes, complex shapes and ellipses.

To remove area points:

1. Place a fence or a shape element to define the area. (Optional) Select the shape.

2. Select the **Remove Area** tool.

   This opens the **Area remove** dialog:



3. Select a **Surface** from which to remove points.

4. Select a **Fence** mode to determine where points are effected.

5. Identify the bounding shape for the area, if a shape element is used and not selected already.

6. Accept the fence contents.

   The points are removed and the triangle network is updated.

Edit Point toolbox

The tools in the **Edit Point** toolbox are used to modify a surface model on point-by-point basis.

TO	USE TOOL	
Construct/remove a breakline between points		Construct Breakline
Insert point along a breakline		Insert Point Along Breakline
Insert a point to the model		Insert Point
Move a point in the model		Move Point
Remove a point from the model		Remove Point
Exclude/include triangles		Exclude Triangle

Construct Breakline

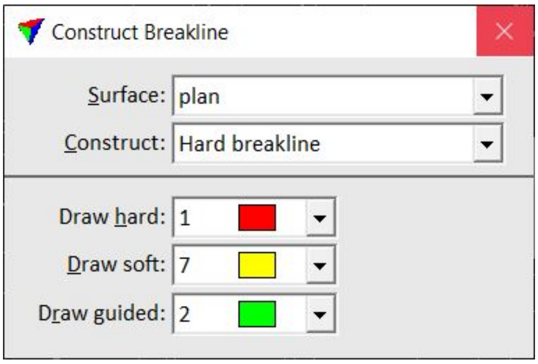
Construct Breakline tool constructs or removes breaklines between existing points in a model. Breaklines effect the way a model is triangulated. In this context, a breakline is a triangle edge connecting two points in the surface model. Thus, constructing or removing breaklines does not effect the actual points in a model, but it changes the way those points are connected by the triangle network.

The tool prompts you to identify two points in a surface model. A breakline is constructed, if the two points are not already connected by a breakline of the same type. If the two points are connected by a breakline of the same type, this tool removes the existing breakline.

To construct or remove breaklines:

1. Select the **Construct Breakline** tool.

   The **Construct Breakline** dialog opens:



2. Select a view for breakline display with a data click inside the view.

   Breaklines inside the selected view are highlighted with colors according to the breakline type.

3. Define settings in the dialog.

4. Identify a first point in the model.

5. Identify a second point in the model.

   A breakline is constructed or removed. You can continue with steps 3 or 4.

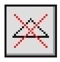
SETTING	EFFECT
Surface	Name of the surface for which to add or remove breaklines.
Construct	Type of the new breakline points: **Hard breakline**, **Soft breakline** or **Guided breakline**. See also Breakline types.
Draw hard	Preview color for hard breaklines.

SETTING	EFFECT
Draw soft	Preview color for soft breaklines.
Draw guided	Preview color for guided breaklines.

Attempting to construct a breakline may fail because:

- The new breakline would intersect an existing breakline. Breaklines can not intersect each other. Try to remove the old breakline first.
- The two points are far apart. Try to construct a shorter breakline first.
- The breakline would break through a very large number of triangle edges. Try to construct a shorter breakline first.

Exclude Triangle



Exclude Triangle tool excludes or includes triangles in a surface model. An included triangle is considered to be valid information about a surface. An excluded triangle is invalid, the surface does not exist at this location or the elevation is unknown. An excluded triangle is neither drawn in profiles, used in volume calculations, nor displayed in surface representations.

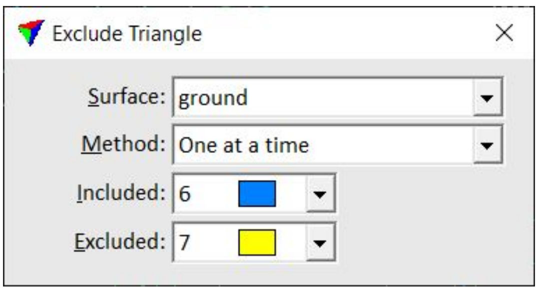
There are several methods of changing the status of a triangle with the tool:

- **One at a time** - triangles are defined one-by-one with data clicks inside the triangles. Can toggle between included and excluded status.
- **Paint** - several triangles are defined by touching them with the mouse pointer between two data clicks. Can toggle between included and excluded status.
- **Boundary line** - triangles are defined by the area between a boundary line and a line determined by a data click. The boundary line can be any linear element. The status can be changed from included to excluded only.
- **Between lines** - triangles are defined by two linear elements. The status can be changed from included to excluded only.

To include or exclude triangles:

1. Select the **Exclude Triangle** tool.

   The **Exclude Triangle** dialog opens:



2. Select a **Surface** and colors for highlighting **Included** and **Excluded** triangles.

3. Select a **Method** for changing the status of triangles.

   Continue depending on the selected **Method**.

Method: One at a time

4. Identify the triangle. The effected triangle is highlighted in either **Included** color or **Excluded** color if the mouse point is inside the triangle area.

   A data click toggles the status of the triangle.

Method: Paint

4. Identify a view for triangle display.

5. Enter a data click in the first triangle. The effected triangle is highlighted in either **Included** color or **Excluded** color if the mouse point is inside the triangle area.

   This mouse click determines whether triangles are included or excluded. If the first triangle is currently included, the tool excludes triangles the mouse pointer passes through. If the first triangle is currently excluded, the tool includes triangles.

6. Move the mouse pointer over all triangles that are to be excluded/included.

7. To finish changing the status of triangles, enter a second data click.

   The status is toggled for all triangles that were touched by the mouse pointer.

Method: Boundary line

4. Identify a linear element as boundary line.

5. Identify the direction to which triangles are excluded by moving the mouse pointer away from the linear element.

   A line is displayed at the mouse pointer location which defines the other boundary for excluding triangles.

6. Accept the area for triangle exclusion with a data click.

   The status is toggled to excluded for all triangles that are between the boundary line and the line defined by the second data click in step 6. A triangle is effected if its center point is between the boundary lines.

Method: Between lines

4. Identify a linear element as first line.

5. Identify another linear element as second line.

   The status is toggled to excluded for all triangles that are between the two selected linear elements. A triangle is effected if its center point is between the two lines.

To exclude/include triangles within a specific area, see Exclude Area tool for another option. Long triangles inside a surface model or along outer boundaries of a surface model can be excluded automatically during surface model creation or using the Edit / Exclude long triangles command in the Surfaces window.

Insert Point

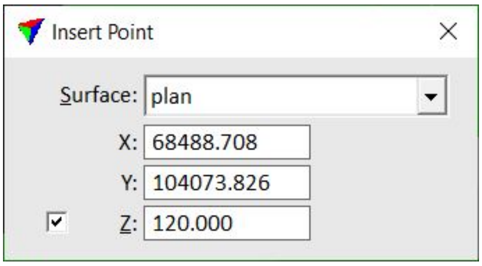
Insert Point tool adds a new point to a surface model. You enter the X and Y location as a data point. The elevation can be a key-in value or the elevation of the data point.

In addition, the tool can add a leveling text for the new point and optional, a point marker. The settings for the label are defined in Elevation labels category and Insert Point category of the TerraModeler **Settings**. The symbology of the label is further determined by the active symbology settings in the CAD file.

To insert a point to a surface model:

1. Select the **Insert Point** tool.

   This opens the **Insert Point** dialog:

2. Select a **Surface** model for which to insert new points.

3. Switch on **Z** and enter an elevation value in order to use a key-in elevation for new points.

   OR

   Switch off **Z** in order to use the data point elevation for new points.

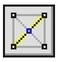
4. Move the mouse pointer inside a view.

   The XY coordinates a of the mouse pointer location are dynamically displayed in the dialog. The elevation of the data point is displayed at the mouse pointer location.

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

   The point is added to the model and possibly, a leveling text is drawn at the location. You can continue to step 2, 3, or 4.

Insert Point Along Breakline



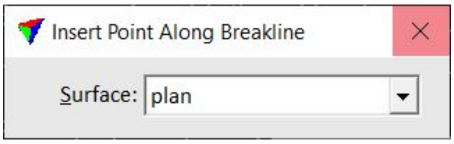
Insert Point Along Breakline tool inserts new points along an existing breakline. This is useful if the distance between two breakline points is very long compared to the average distance between surface points. A long breakline produces long narrow triangles in the surface model. Interpolating new points along the breakline may improve the surface model by making the triangles more uniform in size and closer to equilateral.

The elevation along the breakline does not change as the elevation of the new point is interpolated from the two end points of the breakline.

To insert a point along a breakline:

1. Select the **Insert Point Along Breakline** tool.

   This opens the **Insert Point Along Breakline** dialog:

2. Select a view for breakline display.

   Breaklines inside the selected view are highlighted as colored lines according to the breakline type.

3. Select a **Surface** model for which to enter breakline points.

4. Place a new point with a data click. The data point is projected to the closest breakline. The effected breakline is highlighted as blue line if the mouse pointer comes close to it.

   A new point is interpolated and inserted to the model. You can continue with steps 3 or 4.

Move Point

Move Point tool moves an existing point in a surface model. The tool can change a point location in different ways:

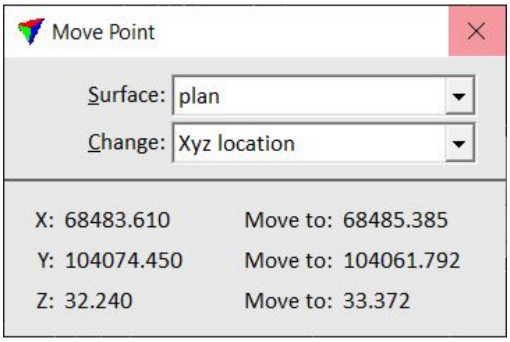
- **Elevation** - only the elevation of the point changes. The XY location remains the same. You can enter the new elevation as a key-in value or as a data point.
- **Xy location** - only the XY location changes, the elevation remains the same.
- **Xyz location** - all of the coordinate values are modified.

The tool opens a dialog which shows dynamically the current coordinate values of the point to be moved and the new coordinate values close to the mouse pointer location to which the point is moved.

To move a point in a surface model:

1. Select the **Move Point** tool.

   The **Move Point** dialog opens:



2. Define settings for moving a point.

3. Identify the point to move. The effected point is highlighted by a circle if the mouse pointer comes close to it.

   The XY and/or Z coordinates of the effected point and the mouse pointer location are dynamically displayed in the dialog.

4. Define the new location of the point with a data click.

   You can continue with steps 2 or 3.

SETTING	EFFECT
Surface	Name of the effected surface model.
Change	Defines what coordinates are changed for the point: • **Elevation** - the Z coordinate of the point is modified.

SETTING	EFFECT
	• **Xy location** - the XY coordinates of the point are modified. • **Xyz location** - the XYZ coordinates of the point are modified.
Move to	If on, the point is moved to the given elevation value. If off, the point is moved to an elevation given by the data point. This is only active if **Change** is set to **Elevation**.

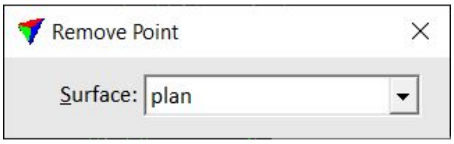
Remove Point

Remove Point tool deletes a point from a surface model. It automatically reorganizes the triangulation at the modified location.

To remove a point from a model:

1. Select the **Remove Point** tool.

   This opens the **Remove Point** dialog:

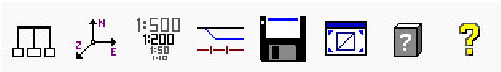
2. Select a **Surface** model from which to remove points.

3. Identify a point. The effected point is highlighted by a circle if the mouse pointer comes close to it.

   The point is removed from the model. You can continue with steps 2 or 3.

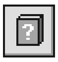
General toolbox

The tools in the **General** toolbox are used to define user settings, to set intended plotting scale, to manage surfaces, to save surface models, to view and send license information, and to access the on-line help.



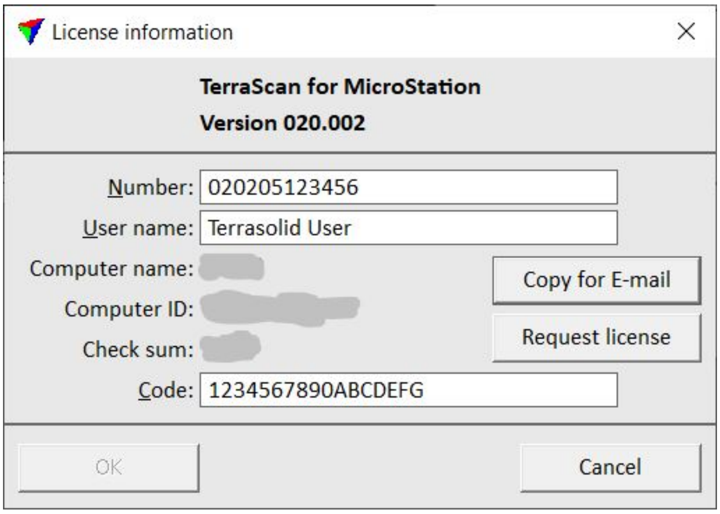
TO	USE TOOL	
Change user settings		Settings
Define coordinate range and resolution		Define Coordinate Setup
Set intended plotting scale / rescale elements		Set Scale
Manage surfaces		Surfaces
Save all modified surface models		Save Surfaces
Adjust CAD viewport to surface coverage		Fit View To Surface
View information about TerraModeler		About TerraModeler
View on-line help		Help On TerraModeler

About TerraModeler



About TerraModeler tool opens a dialog which shows information about TerraModeler and about the license.

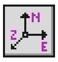
From this dialog, you can open the **License information** dialog which looks the same for all Terra Applications:



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 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.

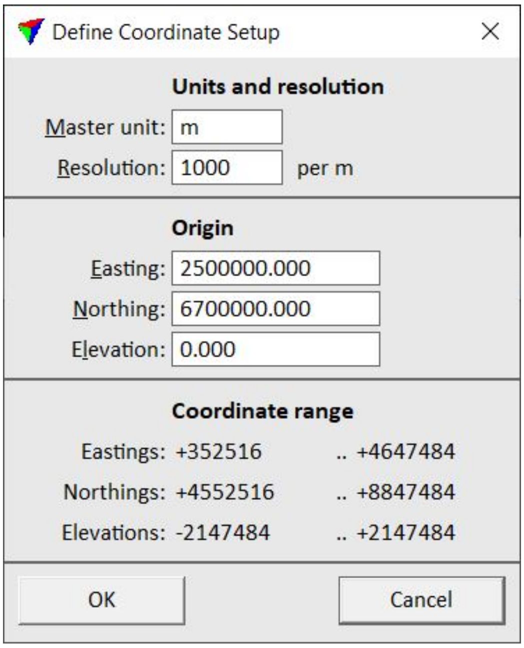
If, for example, one integer step is equal to one millimeter, all coordinate values are rounded to the closest millimeter. At the same time it would impose a limitation on how far apart points can be or how big the coordinate ranges are. Millimeter steps produce a coordinate cube which has a size of 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.

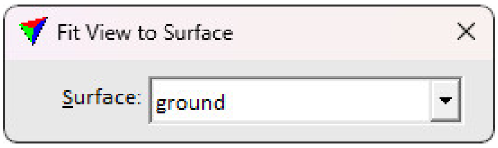
Fit View To Surface

Fit View To Surface tool adjusts the defined view to show the defined surface model entirely.

To fit a view to a surface:

1. Select the **Fit View To Surface** tool.

   The **Fit View To Surface** dialog opens:



2. Select the desired surface model.

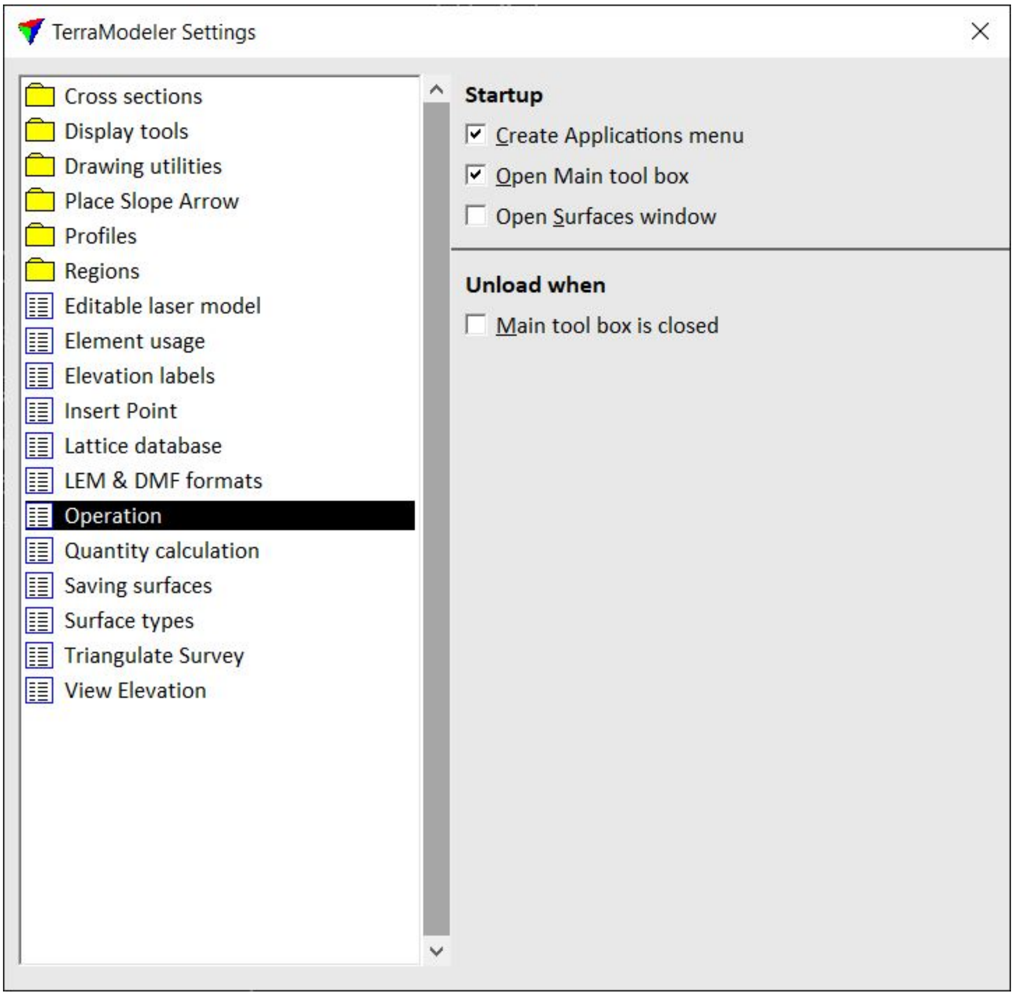
3. Identify the view to fit with a data click.

Help on TerraModeler

Help on TerraModeler tool opens the online help in the standard web browser.

Model Settings

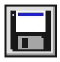
Model Settings tool lets you change a number of settings that control how TerraModeler works. Selecting this tool opens the **TerraModeler Settings** window.



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 Chapter TerraModeler Settings.

Save Surfaces



Save Surfaces tool saves all modified surface models to the hard disk. When you work on a surface model, all the surface data is kept in the computer's RAM. This tool ensures that all modifications are saved permanently to the surface model file.

You can use Surfaces tool to check what surfaces have been modified. The **Status** field in the **Surfaces** window indicates the storage status of each surface.

In the TerraModeler **Settings**, there is the Saving surfaces category for controlling when surface models are saved automatically.

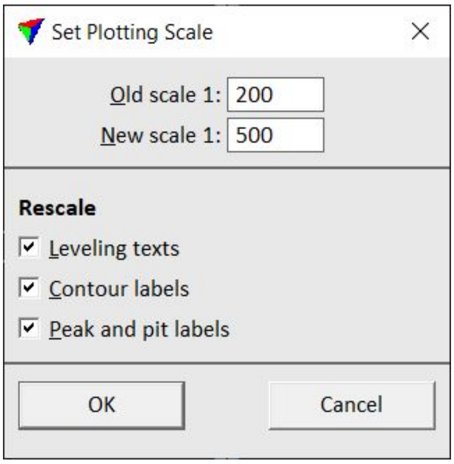
Set Scale

Set Scale tool sets the intended plotting scale of the design and rescales elements. Plotting scale affects the size of text elements that TerraModeler creates.

To set the intended plotting scale:

1. Select the **Set Scale** tool.

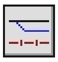
   The **Set Plotting Scale** dialog opens:



2. Enter a value for the **New scale**.
3. Select elements to be rescaled.
4. Click OK.

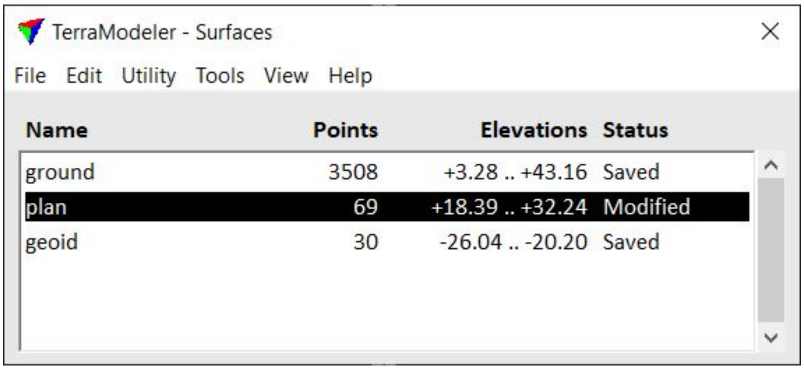
   The selected element types are rescaled.

Surfaces



Surfaces tool opens a window for managing surface models. You can use it, for example, to open, close, and delete surfaces, to rename surfaces, view surface statistics, perform mathematical operations with surfaces, and to import and export surface data.

The **Surfaces** window shows a list of available and active surfaces. For every existing surface model the list shows the number of points in the model and the elevation range. You may choose additional fields to be shown in the window.



See Chapter Managing Surfaces for a complete description of the menu commands.

Profiles toolbox

The tools in the **Profiles** toolbox are used to draw profiles and cross sections. Profiles and cross sections rely on alignment elements. The tools for updating profiles, projecting elements to and from profiles, labeling profile elevations, and creating output reports from profiles only work as long as the alignment element is available in the CAD file.

For reasons of simplicity, the following text refers to profiles and cross sections with the term “profile” if no differentiation between the two elements is necessary. This corresponds to the way of naming the tools in the software. The tools **Update Profile**, **Project into Profile**, **Project From Profile** and **Label Profile Elevations** can be applied to cross sections as well.

TO	USE TOOL	
Draw cross section view		Draw Section View
Draw profile along an alignment		Draw Profile
Draw cross sections along an alignment		Draw Alignment Sections
Update profile or cross section		Update Profile
Setup views to automatically display surface sections		Setup Automatic Sections
Project 3d elements into a profile		Project Into Profile
Project elements from a profile to 3d		Project From Profile
Label profile with an element's elevations		Label Profile Elevations
Output surface elevations along an alignment		Output Profile Elevations *Not Lite*
Output surface elevations for cross sections		Output Section Elevations *Not Lite*

Draw Alignment Sections

Not Spatix

Draw Alignment Sections tool draws cross sections along an alignment. Each cross section shows the shape of surface models at a position along the alignment. You can specify the width of the cross sections as a left and a right width perpendicular from the alignment.

The alignment element can be any linear element. Valid element types include lines, line strings, curves, arcs, ellipses, shapes, complex chains and complex shapes.

The cross sections are drawn as a group of cells that can be freely positioned anywhere in the CAD file. The arrangement of the cross sections in the group can be defined in Cross sections / Placement category of the TerraModeler **Settings**.

The design of the elevation range grid drawn for each cross sections is defined by settings in the **Draw sections along alignment** dialog as well as in Cross sections / Elevation grid category of the TerraModeler **Settings**. The cross section cells are drawn into the CAD file on the active level or on several levels that are defined in Profiles / Levels category of the TerraModeler **Settings**.

Optionally, the tool draws markers in the CAD files at places where a cross section has been generated. The section markers are placed perpendicular to the alignment element and may include a line and text elements showing the station along the alignment. The symbology of the line and text element is defined in the **Alignment Section Markers** dialog opened by the **Markers** button of the **Draw Sections Along Alignment** dialog.

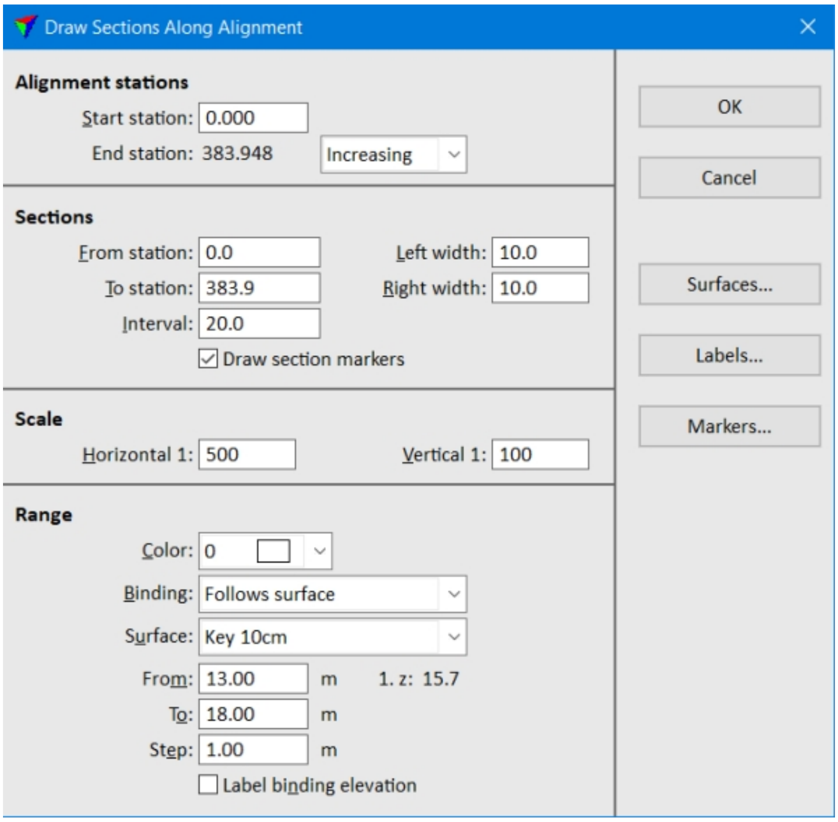
To draw alignment sections:

1. (Optional) Select the alignment element.

2. Select the **Draw Alignment Sections** tool.

3. Identify the alignment element if it has not been selected in step 1.

   The **Draw Sections Along Alignment** dialog opens:



4. Define settings in the dialog.

5. (Optional) Select surfaces for being displayed in cross sections using the **Surfaces** button. See Profile surfaces for more information.

6. (Optional) Define settings for cross section labels using the **Labels** button. See Alignment section labels for more information.

7. (Optional) Define settings for cross section markers using the **Markers** button. See Alignment section markers for more information.

8. Click OK in the **Draw Sections Along Alignment** dialog.

   7The cross section outlines are displayed at the mouse pointer position.

9. Define the location of the group of profile cells in a CAD file top view with a data click.

   This draws the cross sections into the CAD file.

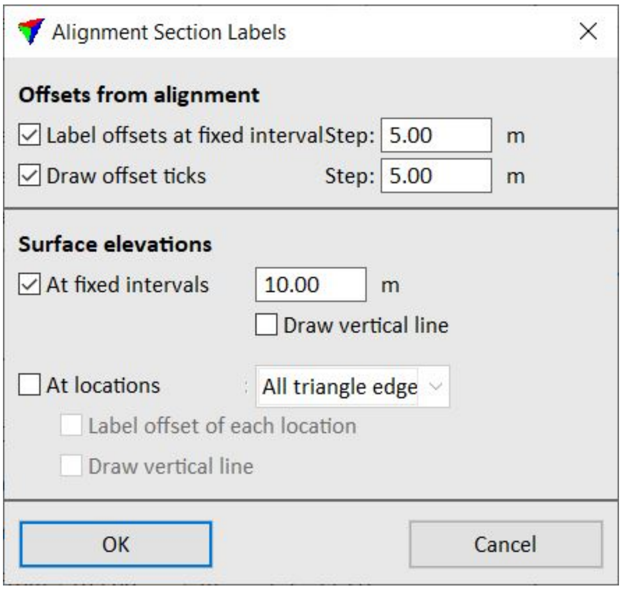
SETTING	EFFECT
Start station	Start station value of the alignment element.
End station	End station value defined by the length of the alignment element: • **Increasing** - stations values are increasing from start to end station. • **Decreasing** - station values are decreasing from start to end station

SETTING	EFFECT
From station	First station on which a cross section is drawn.
To station	Last station from which a cross section can be drawn.
Interval	Station interval between two successive cross sections. Determines the number of cross sections drawn.
Left width	Cross section width perpendicular to the left from the alignment.
Right width	Cross section width perpendicular to the right from the alignment.
Draw section markers	If on, the location of cross sections are marked with line and text elements along the alignment element. The text elements show the station number and are placed at both end points of the marker lines.
Horizontal	Horizontal scale which defines the intended plotting scale.
Vertical	Vertical scale which defines the elevation exaggeration as the ratio of horizontal / vertical scale.
Color	Color of the elevation range grid lines and labels.
Binding	Defines how the elevation range changes along the alignment: • **Stays constant** - the range stays the same as for the first cross section. • **Follows 3d alignment** - the range is adjusted to the elevation of the alignment at the cross section location. • **Follows surface** - the range is adjusted to the elevation of the selected **Surface** at the cross section location.
From	Lowest elevation range value for the first cross section. The **1. z** value next to the field shows the elevation value at the first cross section location.
To	Highest elevation range value for the first cross section.
Step	Vertical step size of the elevation range.

SETTING	EFFECT
Label binding elevation	If on, the binding elevation is labeled by a text and a guide line in each cross section.

Alignment section labels

The **Alignment Section Labels** dialog defines settings for labeling offsets and surface elevations in cross sections. The symbology for offset labels can be defined in Cross sections / Offset labels category and for surface elevation labels in Cross sections / Elevation labels category of TerraModeler **Settings**.

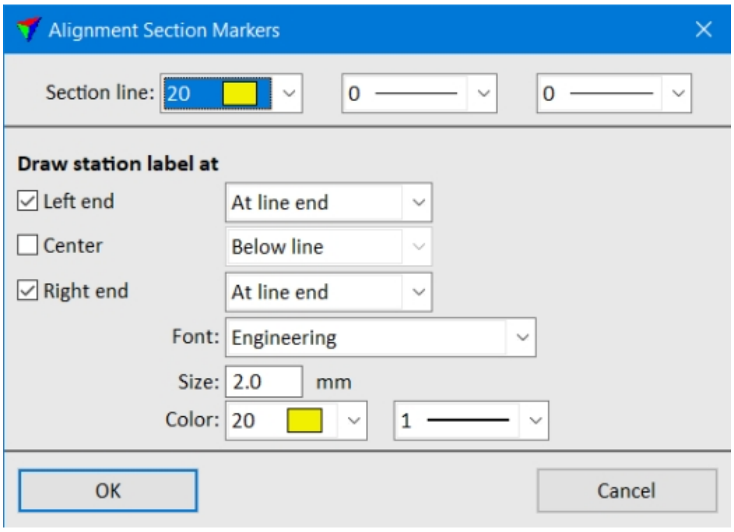


SETTING	EFFECT
Label offsets at fixed intervals	If on, labels for the offset from the alignment are drawn below each cross section. The distance between offset labels is defined in the **Step** field. Offset values are negative to the left and positive to the right of the alignment.
Draw offset ticks	If on, small lines for the offset from the alignment are drawn at the bottom of each cross section. The distance between offset ticks is defined in the **Step** field.
At fixed intervals	If on, surface elevations labels at fixed **Step**s are drawn below the cross section.
Draw vertical line	If on, a vertical line is drawn at the location of an elevation label. This is only active if **At fixed intervals** is switched on.

SETTING	EFFECT
At locations	If on, elevation labels are drawn at locations where there is a change in the surface model: • **All triangle edges** - locations where the alignment intersects a triangle edge. • **Breakline edges** - locations where the alignment intersects a breakline edge. • **Slope changes** - locations where the slope changes more than a given **Change limit**. • **Peaks and pits** - local maximum and minimum elevations.
Label offset of each location	If on, an offset label is placed at the location of an elevation label. This is only active if **At locations** is switched on.
Draw vertical line	If on, a vertical line is drawn at the location of an elevation label. This is only active if **At locations** is switched on.

Alignment section markers

The **Alignment Section Markers** dialog defines settings for markers that are placed at locations where cross sections have been generated. The markers may include text elements that indicate the station along the alignment.



SETTING	EFFECT
Section line	Color, weight and style of the marker line. Uses the active color table and line weights/styles of the CAD file.

SETTING	EFFECT
Left end	If on, a text element indicating the station along the alignment is placed at the left end of the section marker line.
Center	If on, a text element indicating the station along the alignment is placed in the center of the section marker line.
Right end	If on, a text element indicating the station along the alignment is place at the right end of the section marker line.
Font	Font type used for the text element. The list contains the fonts available in the CAD file.
Size	Size of the text element. Given in CAD file secondary units.
Color	Color and weight of the text element. Uses the active color table and line weights of the CAD file.

Draw Profile

Draw Profile tool draws a profile along an alignment element. The profile shows the shape of surface models along the alignment.

The alignment element can be any linear element. Valid element types include lines, line strings, curves, arcs, ellipses, shapes, complex chains and complex shapes.

A profile is drawn as a cell element that can be freely positioned anywhere in the CAD file.

The horizontal length of the profile cell is equal to the 2D length of the alignment element. The horizontal scale represents the intended plotting scale of the profile. All text items in the profile are scaled according to the plotting scale.

The design of the elevation range grid for the profile is defined by settings in the **Draw profile** dialog as well as in Profiles / Elevation grid category of the TerraModeler **Settings**. The profile cell is drawn into the CAD file on the active level or on several levels that are defined in Profiles / Levels category of the TerraModeler **Settings**.

A profile can have a name written on top of the profile. The symbology settings for the name as well as for bottom row titles of specific layouts can be found in Profiles / Titles category of the TerraModeler **Settings**. There is also the option to add additional information at the bottom of the profile. This is defined in Profiles / Layouts category of the TerraModeler **Settings**.

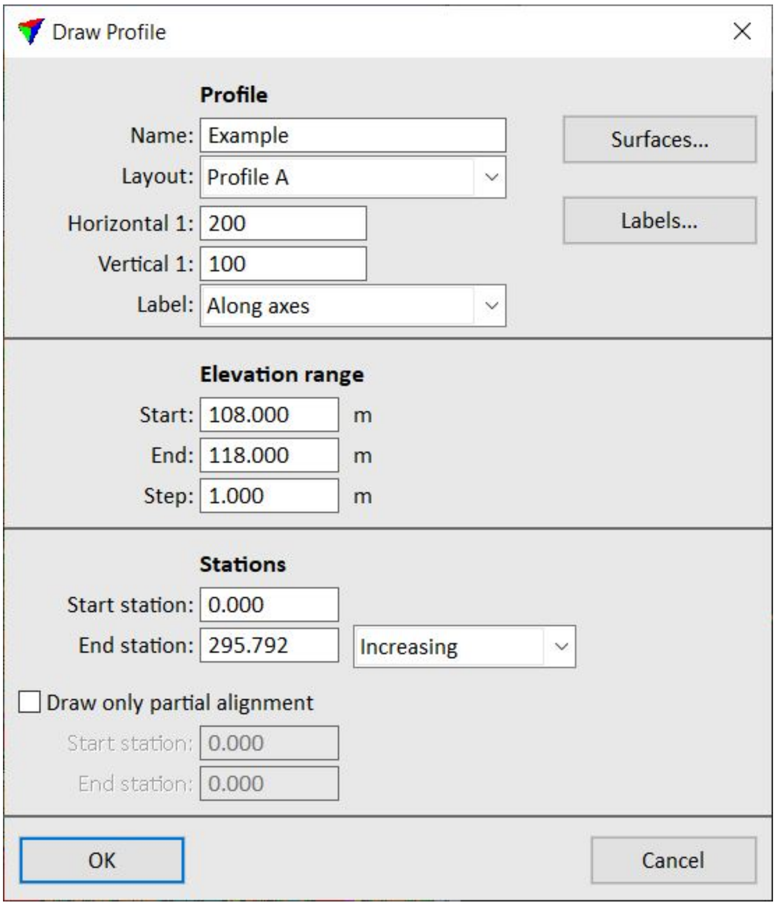
To draw a profile:

1. (Optional) Select the alignment element.

2. Select the **Draw Profile** tool.

3. Identify the alignment element if it has not been selected in step 1.

   The **Draw profile** dialog opens:



4. Define settings in the dialog.

5. (Optional) Select surfaces for being displayed in profiles using the **Surfaces** button. See Profile surfaces for more information.

6. (Optional) Define settings for profile labels using the **Labels** button. See Profile labels for more information.

7. Click OK in the **Draw profile** dialog.

   The profile outline is displayed at the mouse pointer position.

8. Define the location of the profile cell in a CAD file top view with a data click.

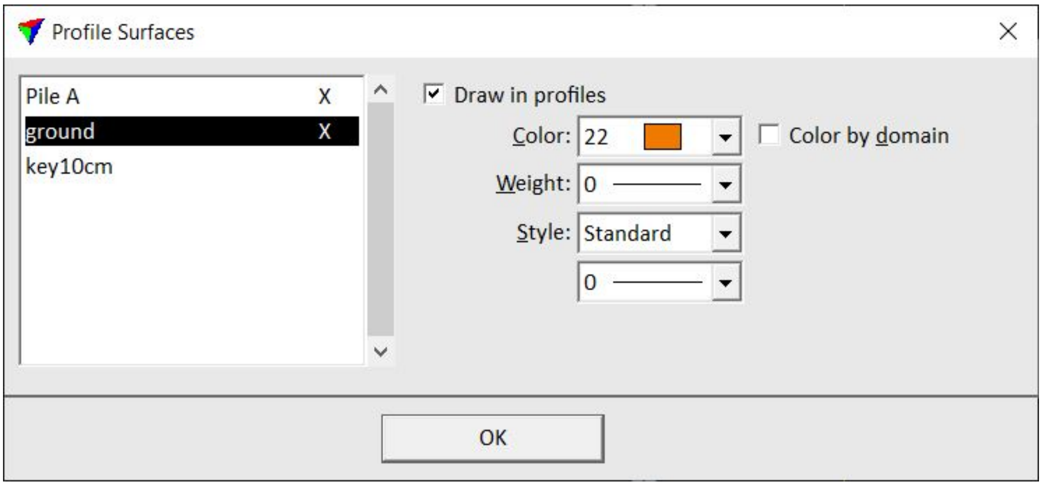
   This draws the profile into the CAD file.

SETTING	EFFECT
Name	Name drawn on top of the profile.
Layout	Layout scheme which defines the data rows that appear below the elevation grid of the profile: • **No layout** - no data row is drawn below the profile.

SETTING	EFFECT
	• ***<layout name>*** - uses the selected layout scheme which has been defined in Profiles / Layouts category of TerraModeler **Settings**.
Horizontal	Horizontal scale that defines the intended plotting scale.
Vertical	Vertical scale that defines the elevation exaggeration as the ratio of horizontal / vertical scale.
Label	Position where scale labels are drawn: **Along axis** or **Upper corner**.
Color	Color of the elevation range grid lines and labels.
Start	Lowest elevation of the elevation range.
End	Highest elevation of the elevation range.
Step	Vertical step size of the elevation range.
Start station	Start station value of the alignment element.
End station	End station defined by the length of the alignment element: • **Increasing** - station values are increasing from start to end station. • **Decreasing** - station values are decreasing from start to end station.
Draw only partial alignment	If on, the profile is only drawn for a part of the alignment defined by **From station** and **To station**.

Profile surfaces

The **Profile Surfaces** dialog provides a list of surface models available for display in profiles. You can define which profiles are displayed and the symbology for each surface.

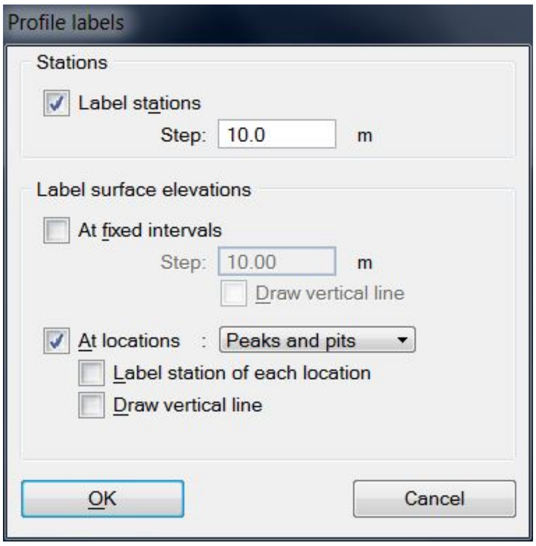


SETTING	EFFECT
Draw in profiles	If on, the selected surface in the list on the left side of the dialog is drawn in the profile.
Color	Line color for the selected surface. Uses the active CAD file color table.
Color by domain	If on, the color of domain(s) is used. See Define Domains tool for more information.
Weight	Line weight for the selected surface. Uses CAD file line weights.
Style	Line style for the selected surface: • **Standard** - uses CAD file standard line styles. • **Custom** - the user can define an own line style based on line styles and a scale. *(Bentley CAD only)*
Style name *(Bentley CAD only)*	Name of a custom line style. Use the **Select** button to select a line style definition. This is only active if **Style** is set to **Custom**.
Style scale *(Bentley CAD only)*	Scale of a custom line style. This is only active if **Style** is set to **Custom**.

The same dialog can be opened using the Edit / Profile settings command from the Surfaces window.

Profile labels

The **Profile labels** dialog defines settings for labeling stations and surface elevations in a profile. The symbology for bottom row labels, station labels, and elevation labels can be defined in Profiles / Labels category of TerraModeler **Settings**.



SETTING	EFFECT
Label stations	If on, the station values along the alignment are drawn below the profile.
Step	Horizontal step size of alignment stations that are labeled.
At fixed intervals	If on, surface elevation labels are drawn below the profile at fixed Steps .
Draw vertical line	If on, a vertical line is drawn at the location of an elevation label. This is only active if At fixed intervals is switched on.
At locations	If on, elevation labels are drawn at locations where there is a change in the surface model: • **All triangle edges** - locations where the alignment intersects a triangle edge. • **Breakline edges** - locations where the alignment intersects a breakline edge. • **Slope changes** - locations where the slope changes more than the given **Change limit** value. • **Peaks and pits** - local maximum and minimum elevations.
Label station of each location	If on, a station label is placed at the location of an elevation label. This is only active if **At locations** is switched on.
Draw vertical line	If on, a vertical line is drawn at the location of an elevation label. This is only active if **At locations** is switched on.

Draw Section View

Draw Section View tool creates a 3D section view that can be used for design purposes. A section view is a rotated view along a section line.

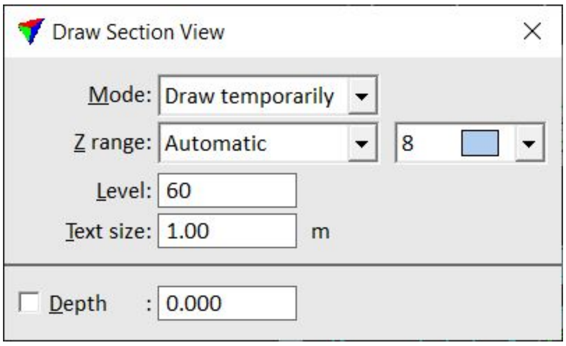
A section view is not meant to be plotted on paper. As its name implies, it is a rotated cross section view drawn at the true 3D position of the section line. This makes it ideal for design purposes. If you use a section view to place elements, they are drawn to a true 3D position.

The tool draws a line for each surface at the section's location if the surface type is selected to be drawn in profiles. The symbology of the line is related to the surface type and determined by settings in Surface types category in TerraModeler **Settings**. Optionally, the elevation range can be drawn including grid lines and labels.

To draw a section view:

1. Select the **Draw Section View** tool.

   This opens the **Draw Section View** dialog:



2. Define settings in the dialog.

3. Enter the start point and the end point of the section line in a top view.

   Continue with step 4 below.

   OR

1. Select the center line element of the section view.

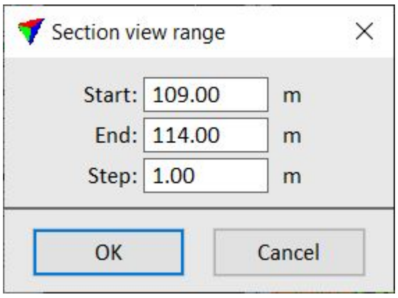
2. Select the **Draw Section View** tool.

3. Define settings in the dialog.

4. Define the section view depth by entering a data click in the top view or by defining the value in the **Depth** field.

5. Identify the view to be used as the section view with a data click.

6. If **Z range** is set to **Prompt for**, the **Section view range** dialog opens:



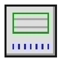
Define a **Start elevation**, **End elevation**, and vertical **Step** size and click OK.

The view is rotated to show the section with the section line start point on the left and the section line end point on the right. The depth of the view corresponds to the given depth value of the section.

SETTING	EFFECT
Mode	Drawing mode for surface section lines and elevation range in the section view: • **Write to file** - drawn as permanent elements.
Z range	Defines how the elevation range is set: • **None** - the elevation range is not drawn. • **Automatic** - TerraModeler decides the elevation range based on surface elevations. • **Prompt for** - the elevation range can be defined in **Section view range** dialog.
Level	CAD file level for drawing surface section lines and elevation range.
Text size	Text size of elevation range labels.
Depth	Section depth on both sides of the section line. If on, the depth is fixed to the given value.

Label Profile Elevations

Not Spatix



Label Profile Elevations tool labels a profile with elevation values. The tool can label a linear element drawn in the profile or single points identified with data points in the profile.

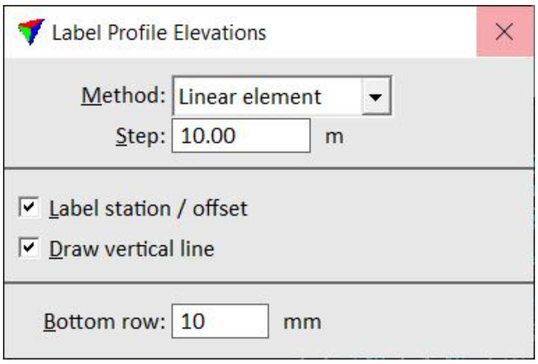
This tool can be used, for example, with an element representing a vertical alignment that has been designed in the profile. The element has to be linear. It should not extend beyond the profile's left or right margin. The tool translates the y-coordinates of the element into elevation values and writes these values below the profile.

The font, size, and number settings for the labels can be defined in Profiles / Labels category of TerraModeler **Settings**. The symbology of the labels is determined by the active symbology settings in the CAD file.

To label a profile with elevations:

1. Select the **Label Profile Elevations** tool.

   This opens the **Label Profile Elevations** dialog:

2. Define settings in the dialog.

3. Identify the profile to label.

   Continue according to the selected **Method**.

Method: Linear element

4. Identify a linear element to be labeled.

5. Accept the element.

   The element's elevations are labeled. You can continue with step 4.

Method: Single location

4. Move the mouse pointer inside the profile area.

   The elevation at the mouse pointer location is displayed.

5. Enter a data point in order to draw the label into the CAD file.

   You can continue with step 5.

SETTING	EFFECT
Method	Labeling method: • **Linear element** - labels elevations of a linear element at fixed **Step**s. • **Single location** - labels the elevation of a single data point.
Label station / offset	If on, the station value (profiles) or the offset from an alignment (cross sections) is drawn for each elevation. For station values the labeling relies on the settings for bottom row contents defined in Profiles / Layouts category of the TerraModeler **Settings**.
Draw vertical line	If on, a vertical line is drawn at each elevation label location.
Bottom row	Distance between the lower boundary of the profile's elevation range and the line of elevation labels. Given as millimeters on paper.

The **Label Profile Elevations** tool works only as long as the alignment element for a profile is drawn in the CAD file. Once the alignment element is deleted, labeling profile elevations is no longer possible.

Output Profile Elevations

Not Lite

Output Profile Elevations tool creates a text report which contains surface model elevations along an alignment element. The elevation positions for the output can be at fixed intervals or only at locations where there is a change in the shape of the surface model. You can view the report, print it out, or save it as a text file.

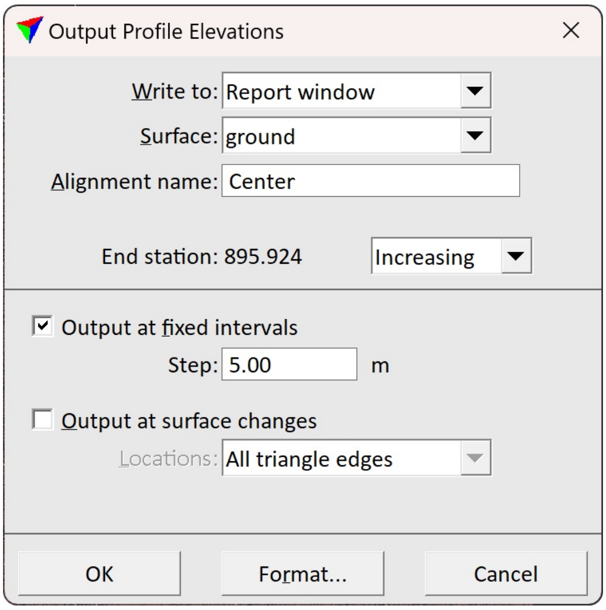
To output elevations along an alignment:

1. Select the **Output Profile Elevations** tool.

2. Identify the alignment to output elevations along.

3. Accept the highlighted alignment.

   This opens the **Output Profile Elevations** dialog:



4. Define settings.

5. (Optional) Click on the **Format** button to select options controlling the Profile Output format.

6. Click OK to the **Output profile elevations** dialog.

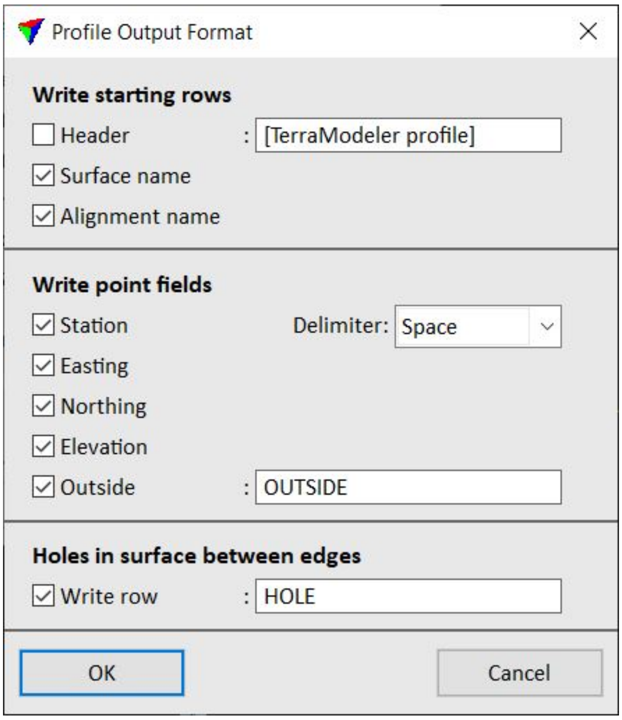
   Depending on the selected **Write to** option this either stores the report to a file or opens a report window which contains the output information. Use the commands from the **File** pulldown menu to print the report or save it into a text file.

SETTING	EFFECT
Write to	Result destination:

SETTING	EFFECT
	• **Text file** - store the results directly to a text file. More efficient with large output. • **Report window** - open report window displaying the results. The results can be printed or saved through the window.
Surface	Name of the surface for which a report is created.
Alignment name	Descriptive name for the alignment in the report.
Start station	Station value at the start point of the alignment element.
End station	End station value defined by the length of the alignment element: • **Increasing** - station values are increasing from start to end station. • **Decreasing** - station values are decreasing from start to end station.
Output at fixed intervals	If on, output surface elevations at a fixed interval. The **Step** field value determines the size of the interval.
Output at surface changes	If on, output surface elevations at locations where there is a change in the surface model: • **All triangle edges** - locations where the alignment intersects a triangle edge. • **Breakline edges** - locations where the alignment intersects a breakline edge. • **Slope changes** - locations where the slope changes more than a given limit. • **Peaks and pits** - local maximum and minimum elevations.

Profile output format

The **Profile Output Format** dialog provides some choices for formatting the output of the **Output Profile Elevations** tool.



The output file may contain some starting rows including **Header**, **Surface name**, and **Alignment name**. Each generates one output line if selected.

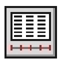
Each elevation position is written in one line. You can select what information fields are included in the output: **Station**, **Easting**, **Northing**, and **Elevation**.

When using **Output at fixed intervals**, the application computes elevation positions at fixed intervals along the alignment. Some of these positions may be outside the surface model. The **Outside** field determines if outside positions are written and what text is used for the output.

When using **Output at surface changes** with **All triangle edges** as locations, the application generates output positions only at triangle edges. The alignment may go outside the surface model in between the generated positions. If **Write row** is on, the application outputs a line between the positions to indicate that the alignment is outside the surface model or goes through a hole in the model.

Output Section Elevations

Not Lite



Output Section Elevations tool creates a text report which contains surface model elevations from cross sections along an alignment element. The cross section positions for the output can be at fixed intervals or only at locations where there is a change in the shape of the surface model. You can view the report, print it out, or save it as a text file.

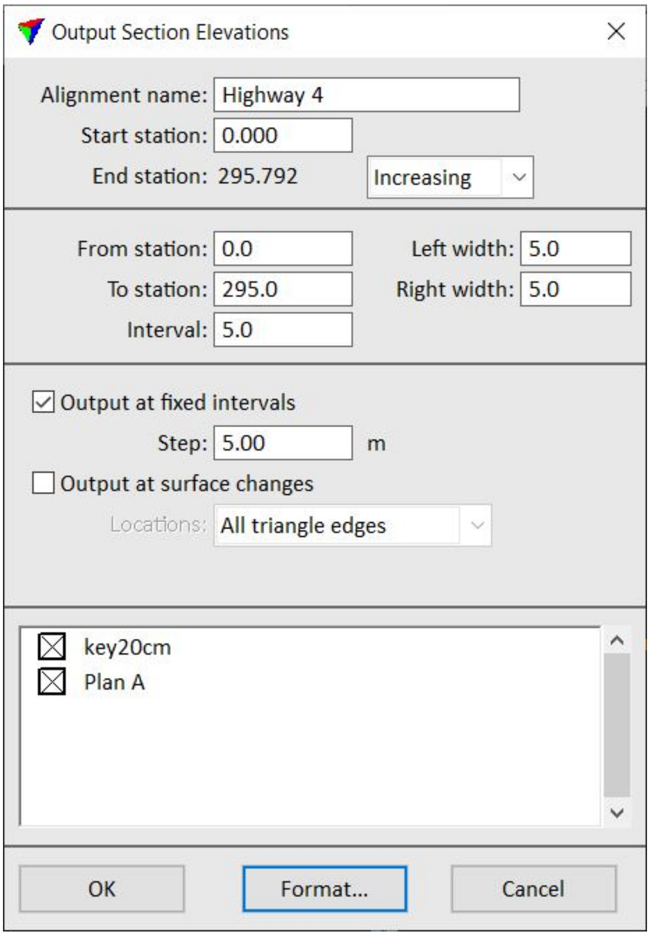
To output elevations from alignment sections:

1. Select the **Output Section Elevations** tool.

2. Identify the alignment for the sections.

3. Accept the highlighted alignment.

   This opens the **Output Section Elevations** dialog:



4. Define settings.

5. Select surfaces from the list in the lower part of the dialog. Only selected surfaces are included in the output.

6. (Optional) Click on the **Format** button to select options controlling the Section output format.

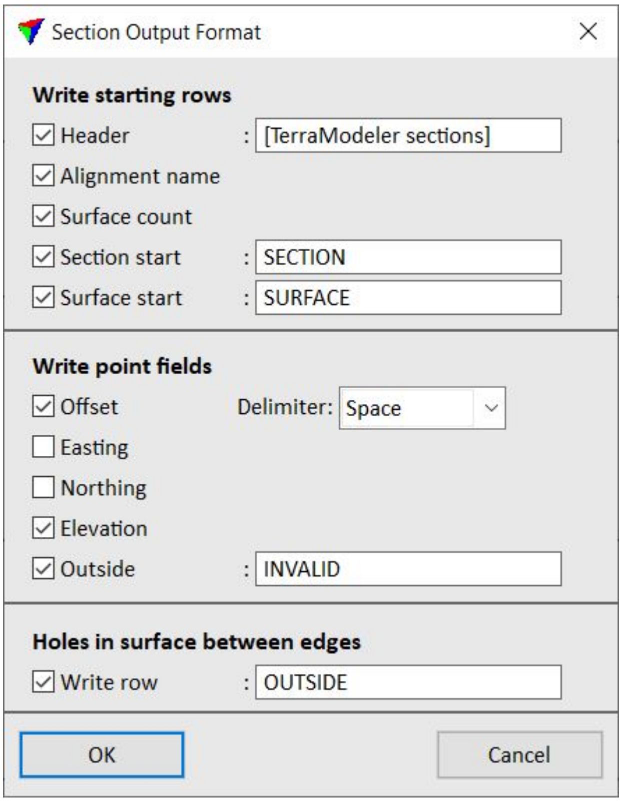
7. Click OK to the **Output Section Elevations** dialog.

   This opens a report window which contains the output information. Use the commands from the File pulldown menu to print the report or save it into a text file.

SETTING	EFFECT
Alignment name	Descriptive name for the alignment in the report.
Start station	Station value at the start point of the alignment element.
End station	End station value defined by the length of the alignment element: • **Increasing** - station values are increasing from start to end station. • **Decreasing** - station values are decreasing from start to end station.
From station	First station on which a cross section is included in the output.
To station	Last station from which a cross section can be included in the output.
Interval	Station interval between two successive cross sections. Determines the number of cross sections in the output.
Left width	Cross section width perpendicular to the left from the alignment.
Right width	Cross section width perpendicular to the right from the alignment.
Output at fixed intervals	If on, output surface elevations at a fixed interval along the cross section. The **Step** field determines the size of the interval.
Output at surface changes	If on, output surface elevations at locations where there is a change in the surface model: • **All triangle edges** - locations where the alignment intersects a triangle edge. • **Breakline edges** - locations where the alignment intersects a breakline edge. • **Slope changes** - locations where the slope changes more than a given limit. • **Peaks and pits** - local maximum and minimum elevations.

Section output format

The **Section Output Format** dialog provides some choices for formatting the output of the **Output Section Elevations** tool.



The output file may contain some starting rows including **Header**, **Alignment name** and **Surface count**. Each generates one output line if selected.

If **Section start** is switched on, a line is written to indicate the start of a new cross section. You can specify the prefix text appearing before the station value.

If **Surface start** field is switched on, a line is written to indicate the start of a new surface. You can specify the prefix text appearing before the surface name.

Each elevation position of a cross section is written in one line. You can select what information fields are included in the output: **Offset**, **Easting**, **Northing**, and **Elevation**.

When using **Output at fixed intervals**, the application computes elevation positions at fixed intervals along the cross section. Some of these positions may be outside the surface model. The **Outside** field determines if outside positions are written and what text is used for the output.

When using **Output at surface changes** with **All triangle edges** as locations, the application generates output positions only at triangle edges. The alignment may go outside the surface model in between the generated positions. If **Write row** is on, the application outputs a line between the positions to indicate that the cross section is outside the surface model or goes through a hole in the model.

Project From Profile

Not Spatix

Project From Profile tool projects elements from a profile into their 3D position. The tool creates a copy of the elements that run along the profile's alignment.

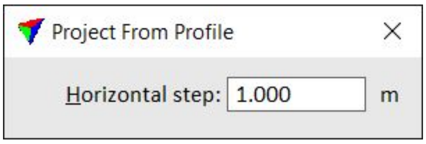
The tool can be used, for example, with an element representing a vertical alignment that has been designed in the profile. The element has to be linear. It should not extend beyond the profile's left or right margin.

The tool creates a new 3D element of type complex chain running along the profile's alignment. The symbology of the new element is determined by the active symbology in the CAD file.

To project element(s) from a profile:

1. Select the **Project From Profile** tool.

   The **Project From Profile** dialog opens:

2. Enter a **Horizontal step** value in order to define the maximum distance between consecutive vertices along the new 3D element.

3. Identify the profile from which the element(s) are projected.

4. Identify an element to project.

5. Accept the element with another data click.

   The element is projected into a new 3D complex chain running along the profile's alignment. You can continue with step 3.

   OR

1. Select element(s) to project.

2. Select the **Project From Profile** tool.

3. Enter a **Horizontal step** value in order to define the maximum distance between consecutive vertices along the new 3D element.

4. Identify the profile from which the element(s) are projected.

5. Accept the element(s) with another data click.

The **Project from Profile** tool works only as long as the alignment element for a profile is drawn in the CAD file. Once the alignment element is deleted, the projection of element from a profile is no longer possible.

Project Into Profile

Not Spatix

Project Into Profile tool projects 3D elements into a profile. This tool creates a copy of the element with its coordinates translated into the profile's coordinate system.

The element to be projected has to be a cell or a linear element. The element should be positioned within the extents of the profile alignment.

Linear elements are projected as a line string or a complex chain into profile cells. The application calculates a line string that approximates an arc, an ellipse, or a curve. The symbology of the elements in the profile is determined by the symbology of the original element.

To project element(s) into a profile:

1. Select the **Project Into Profile** tool.
2. Identify a profile into which the element(s) are projected.
3. Identify an element to project.
4. Accept the element with another data click.

   The element is projected into the profile. You can continue with step 3.

   OR

1. Select element(s) to project.
2. Select the **Project Into Profile** tool.
3. Identify a profile into which the element(s) are projected.
4. Accept the element(s) with another data click.

   The element(s) are projected into the profile.

The **Project Into Profile** tool works only as long as the alignment element for a profile is drawn in the CAD file. Once the alignment element is deleted, the projection of element into a profile is no longer possible.

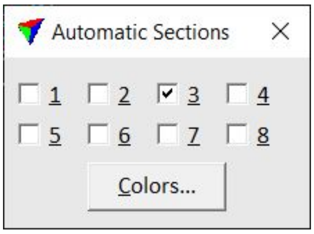
Setup Automatic Sections

Setup Automatic Sections tool defines views and symbology settings for displaying surface models in section views automatically. The settings are used by several tools in TerraScan, such as Draw Vertical Section, Rotate Section, Cut Section, Synchronize Views and Travel Path. The surface(s) are drawn as lines in the section views. The display option can be used, for example, to compare laser data elevations with surface elevations.

To setup automatic sections:

1. Select the **Setup Automatic Sections** tool.

   This opens the **Automatic Sections** dialog:

2. Select view(s) in which the surfaces are to be displayed.

3. Click on the **Colors** button.

   This opens the Profile surfaces dialog.

4. Define symbology settings for surfaces that are drawn in section views.

5. Click OK to the **Profile surfaces** dialog.

   The surface(s) are now displayed in section views that are created by TerraScan tools. The display is active as long as a view is selected in the **Automatic Sections** dialog. Thus, you may close the **Automatic Sections** dialog.

Update Profile

Not Spatix

Update Profile tool updates profile cell(s) after surface models or alignment elements were modified. The tool redraws the selected cells.

To update a profile:

1. Select profile cell(s) to update.

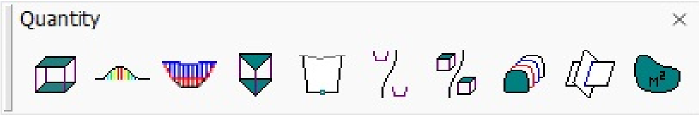
2. Select the **Update Profile** tool.

   The selected cell(s) are redrawn.

The **Update Profile** tool works only as long as the alignment element for a profile is drawn in the CAD file. Once the alignment element is deleted, the update is no longer possible.

Quantity toolbox

The **Quantity** toolbox contains tools that can be used to compute quantities between surface models or between surface models and cross sections of trenches, to find the intersection between two surfaces or to compute surface model areas. Further, there is a tool for computing tunnel volumes based on tunnel cross sections.



Compute Quantity is probably used most often. It calculates volumes between two surfaces using the grid method. It can calculate volumes from entire surface models or from areas restricted by a fence or selected polygon.

Define Section Templates and Compute Section Quantity tools can be used for calculating volumes along an alignment. The calculation is based on the traditional end-area method.

TO	USE TOOL	
Compute quantities using grid method		Compute Quantity
Compute stockpile volumes quickly using a point cloud or a surface model		Compute Stockpile Volume
Compute several		Compute Multiple Surface Volume *Not Lite*
Compute quantities using prismoidal method		Compute Prismoidal Quantity *Not Lite*
Compute quantities along an alignment		Compute Alignment Quantity *Not Lite*
Define section templates for trenches		Define Section Templates *Not Lite*
Compute trench quantities using section templates		Compute Section Quantity *Not Lite*
Compute tunnel volume from section elements		Compute Tunnel Volume *Not Lite*
Draw intersection of two surfaces		Draw Surface Intersection
Compute surface area		Compute Area *Not Lite*

Compute Alignment Quantity

Not Lite

Compute Alignment Quantity tool computes quantities along an alignment element. The tool uses the grid method for computing quantities between two surfaces.

The alignment can be defined by a linear element, such as a 3D line string or complex chain. The area of calculation is limited by the extension of the alignment element. It may be further limited by another surface model.

The surfaces can be defined by surface models in TerraModeler or by points loaded in TerraScan.

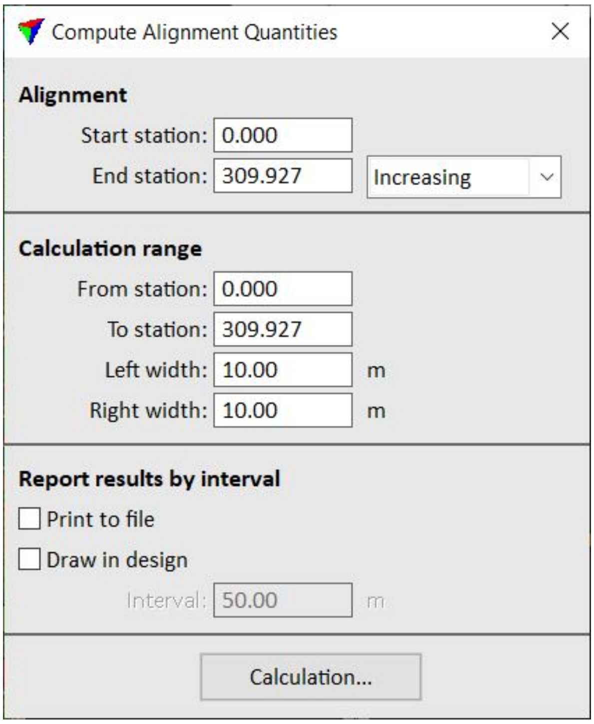
The results of the computation are shown in the tool's dialog. They can be presented graphically in the CAD file and a text report can be stored as well.

To compute quantities along an alignment:

1. Select an alignment element.

2. Select the **Compute Alignment Quantity** tool.

   The **Compute Alignment Quantities** dialog opens:

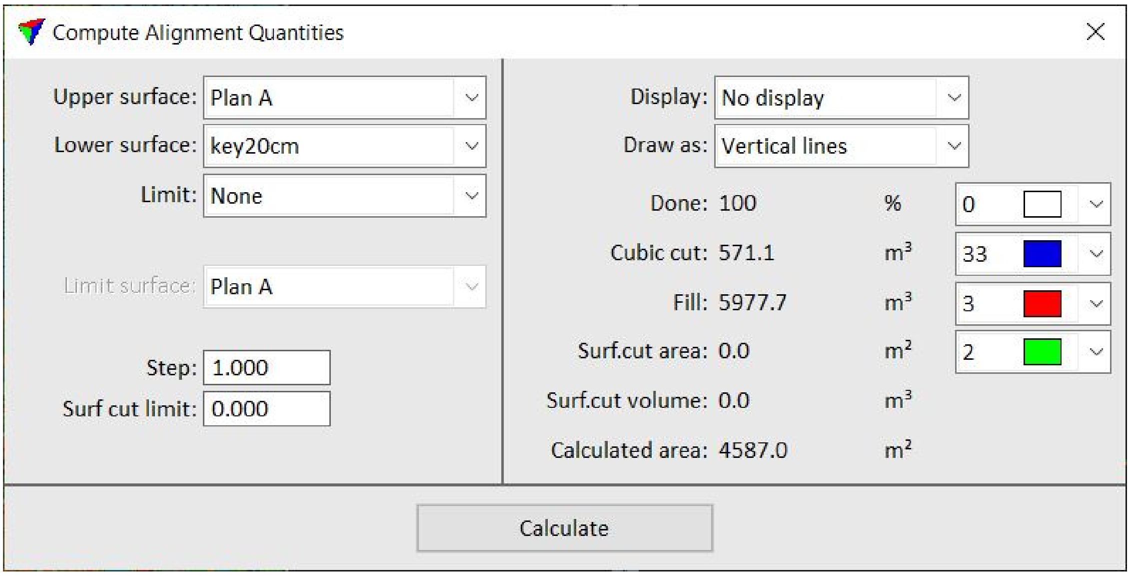


3. Define settings for the computation related to the alignment element.

SETTING	EFFECT
Start station	Station value at the start point of the alignment element.
End station	End station value defined by the length of the alignment element: • **Increasing** - station values are increasing from start to end station. • **Decreasing** - station values are decreasing from start to end station.
From station	Start station value for the quantity computation.
To station	End station value for the quantity computation.
Left width	Defines the computation area boundary parallel to the left from the alignment.
Right width	Defines the computation area boundary parallel to the right from the alignment.
Print to file	If on, the results of the computation are saved into a simple text file.
Draw into design	If on, the boundaries of the computation area and a linear scale of the alignment are drawn into the CAD file. The scale is divided into sections of the length defined by the Interval value. For each section, the rounded calculated quantity value is displayed.

4. Click on the **Calculation** button.

   This opens another **Compute Alignment Quantities** dialog:



5. Define settings related to surface models, computation steps, and display of the results.

6. Click on the **Calculate** button.

   The computation of the quantities starts. The results are displayed in the **Compute alignment** dialog on the right side and possibly in the CAD file. If **Print to file** was selected in the first dialog, a standard Windows dialog opens to define the location for storing the text file on a hard disk.

SETTING	EFFECT
Upper surface	Name of the upper surface model for quantity calculation. The use of **Laser points** requires that the points which define the surface are loaded into TerraScan.
Lower surface	Name of the lower surface for quantity calculation. The use of **Laser points** requires that the points which define the surface are loaded into TerraScan.
Limit	Limiting surface to restrict the calculation area: • **None** - no limiting surface used. • **Area** - calculate only within the **Limit surface** area. • **Lower limit** - calculate only above **Limit surface**. • **Upper limit** - calculate only below **Limit surface**.
Limit surface	Surface to limit the calculation area. This is not active if **Limit** is set to **None**.
Step	Defines the grid size for the calculation. A small step produces more accurate results but

SETTING	EFFECT
	the calculation process takes longer.
Surf cut limit	The software computes the cut volume separately for places where the cut depth is less than this value. This may be used for billing purposes if the price is not based on the cut volume but on the cut area.
Display	Display method for presenting the calculation results graphically in the CAD file: • **No display** - nothing is drawn into the CAD file. • **Display only** - draws temporary elements. • **Write to file** - draws permanent elements into the CAD file on the active level.
Draw as	Elements used to present the results graphically: • **Vertical lines** - a vertical line is drawn at the center of each calculated grid cell. This is the only available option if **Display** is set to **Display only**. • **Horizontal lines** - a horizontal lines is drawn for each calculated grid cell row. • **Filled shapes** - a shape is drawn for each calculated grid cell row. • **Pillars** - a 3D block is drawn for each calculated grid cell. Each block is build from 4 shape elements which may end up with a big number of elements to draw.
Color list next to calculation result line	Defines the display color for drawing the results in the CAD file. Uses the active color table of the CAD file.

TerraModeler has two tools for computing quantities along an alignment. **Compute Alignment Quantity** tool calculates volumes using the grid method and requires that you have an existing surface model for the excavation/accumulation. Compute Section Quantity tool uses section templates to define the excavation/accumulation and calculates volumes using the end-area method.

Compute Area

Not Lite

Compute Area tool computes the area covered by a surface model. It calculates values for both, the two-dimensional and the three-dimensional area.

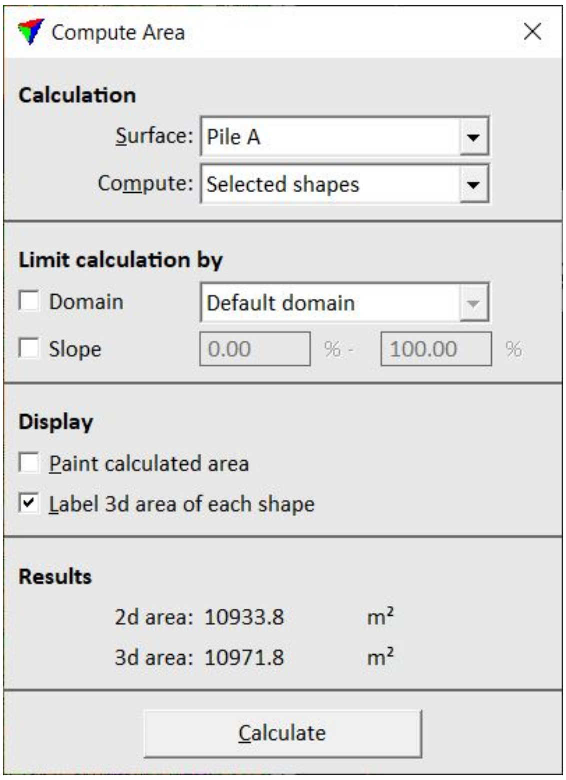
The 3D value is the true surface area of the triangles. The 2D value is computed using only x and y coordinates of the triangle vertices.

You can use a fence or selected shape element(s) to define the area for calculation. The area of computation can be further limited to a specific domain or to a given slope range.

To compute surface area:

1. Select the **Compute Area** tool.

   The **Compute Area** dialog opens:



2. Define settings and click on the **Calculate** button.

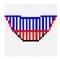
   The computation starts. The results are displayed in the lower part of the tool’s dialog and possibly in the CAD file.

SETTING	EFFECT
Surface	Name of the surface model for area calculation.

SETTING	EFFECT
Compute	Area to compute: • **Whole surface** - whole surface model. • **Fence contents** - only inside a fence. • **Selected shapes** - only inside selected shape element(s).
Domain	If on, compute only triangles assigned to the selected domain.
Slope	If on, compute only triangles within a given slope range. The lower and upper limits are defined as slope percentages.
Paint calculated area	If on, draw each calculated triangle as a temporary shape element into the CAD file. Doesn't work in Bentley CONNECT.
Label 3d area of each shape	If on, the computed 3D area value of each shape is drawn inside the shape as a permanent text element. This is only active if **Compute** is set to **Selected shapes**.

Compute Multiple Surface Volume

Not Lite



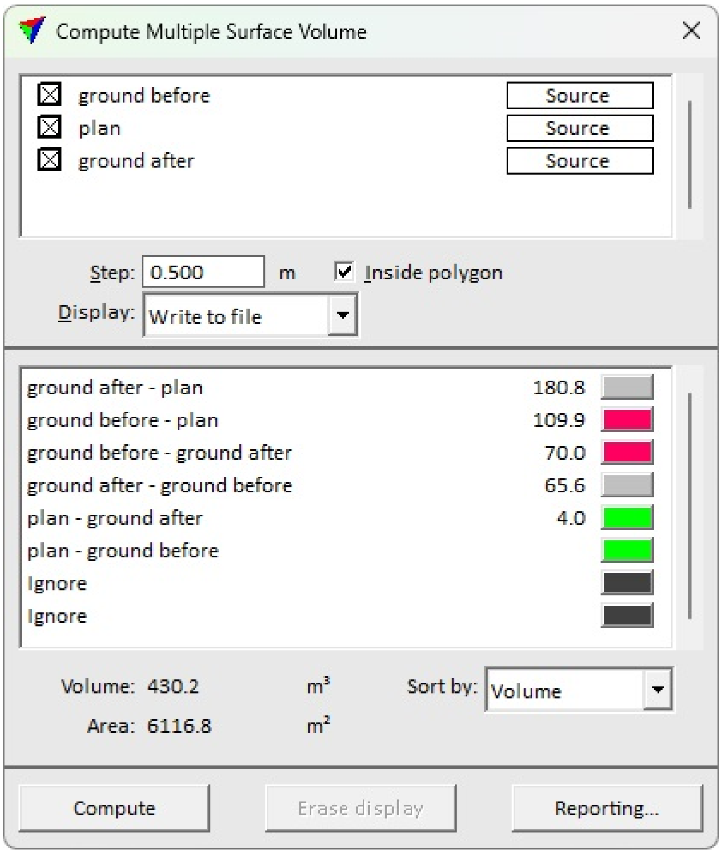
Compute Multiple Surface Volume tool computes and reports volumes between several surface models. The tool is useful for computing volumes when the complete area is modeled with several surface models describing regions, or when the aim is generating a comparison between different moments in time. The calculation is based on a grid method. You can control the accuracy and speed of calculation by defining the grid size.

You can use selected shape element(s) to limit the area for calculation.

To compute volumes from multiple surface models:

1. Select the **Compute Multiple Surface Volume** tool.

   The **Compute Multiple Surface Volume** dialog opens:



2. Define settings and Reporting options. click **Compute**.

   The computation results are displayed in the **Compute Multiple Surface Volume** dialog and possibly in the CAD file.

SETTING	EFFECT
<surface model>	Enabling a surface model includes the model in the calculation. If all surfaces are used as **Source** models, each model is used both as low and high surface. Setting a surface model as **Target** uses the specified model only as the low surface, and only fill volumes above the model are calculated.
Step	Defines the grid size for the calculation. A small step produces more accurate results but the calculation process takes longer.
Inside polygon	If on, the calculation takes place only inside a selected shape element.
Display	Display method for presenting the calculation results graphically in the CAD file: • **No display** - nothing is drawn into the CAD file. • **Draw temporarily** - temporary display of vertical line elements. Disappears if the tool is deactivated or display cleared with **Erase display**. • **Write to file** - draws triangles as permanent elements into the CAD file on the active level.
<low surface> - <high surface>	Report listing of fill volumes.
Color list next to calculation result line	Defines the display color for drawing the results in the CAD file.
Sort by	Sort key for arranging report rows: • **Volume** - sort report rows by calculated volume. • **Surface** - report rows appear in the same order surface models appear in the Surfaces list.

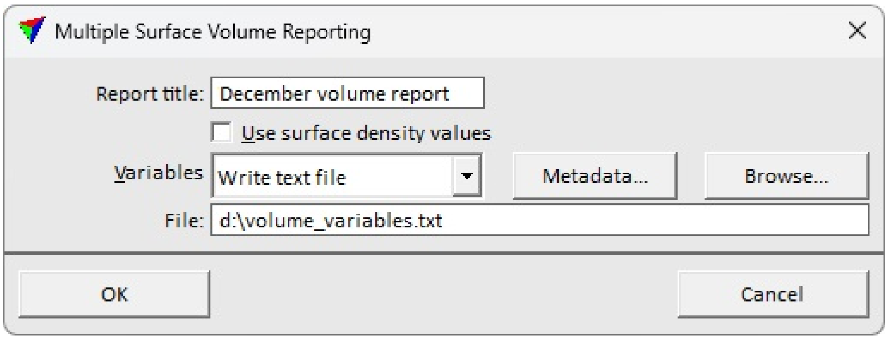
Reporting options

The **Multiple Surface Volume Reporting** dialog provides an option to create text file reports storing variable - value pairs. These reports can be further used to form efficiently pre-formatted report documents filling calculated values.

To setup multiple surface volume reporting:

1. Click on the **Reporting** button in the **Compute Multiple Surface Volume** dialog.

   The **Multiple Surface Volume Reporting** dialog opens:



2. Define settings according to your needs for the output report.

3. Click OK.

4. You can continue with **Calculate** in the **Compute Multiple Surface Volume** dialog.

If **Variables** is set to **Write text file** in the **Quantity reporting** dialog, the file storing variable - value pairs is created and saved at the given location on a hard disk.

SETTING	EFFECT
Report title	Free text that is used as title in the report.
Labels end with density	If on, a numeric value at the end of a text element inside a region is used as a multiplier to all volume values calculated for this region.
Variables	Determines if a variable text file is written and saved: • **Do not write** - no text file is written. • **Write text file** - a text file written and saved at the given **File** location.
Metadata	The button opens the **Quantity Metadata** dialog. The metadata is included at the beginning of the variable text file.
File	If on, region shapes without a text element inside are highlighted.

Compute Prismoidal Quantity

Not Lite

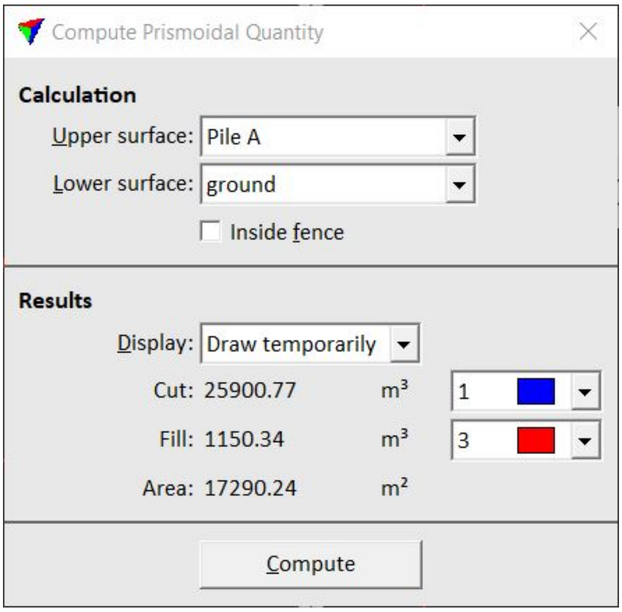
Compute Prismoidal Quantity tool computes quantities between two surfaces and produce results summing up both, cut and fill volumes. The calculation is based on a prismoidal method: the application compares the triangles of the two surfaces. This is the mathematically accurate method for computing quantities.

You can use a fence or a selected shape element to limit the area for calculation.

To compute quantities using prismoidal method:

1. Select the **Compute Prismoidal Quantity** tool.

   The **Compute Prismoidal Quantity** dialog opens:



2. Define settings and click **Compute**.

   The computation results are displayed in the **Compute prismoidal quantity** dialog and possibly in the CAD file.

SETTING	EFFECT
Upper surface	Name of the upper surface model for the quantity calculation.
Lower surface	Name of the lower surface model for the quantity calculation.
Inside fence	If on, the calculation takes place only inside a fence or a selected shape element.

SETTING	EFFECT
Display	Display method for presenting the calculation results graphically in the CAD file: • **No display** - nothing is drawn into the CAD file. • **Draw temporarily** - temporary display of triangles. Disappears if the tool is deactivated. • **Write to file** - draws triangles as permanent elements into the CAD file on the active level.
Color list next to calculation result line	Defines the display color for drawing the results in the CAD file. Uses the active color table of the CAD file.

Compute Quantity

Compute Quantity tool computes quantities between two surfaces and produces results summing up both, cut and fill volumes. The calculation is based on a grid method. You can control the accuracy and speed of calculation by defining the grid size.

You can use a fence or a selected shape element to define the area for calculation. The computation area can be further limited by another surface.

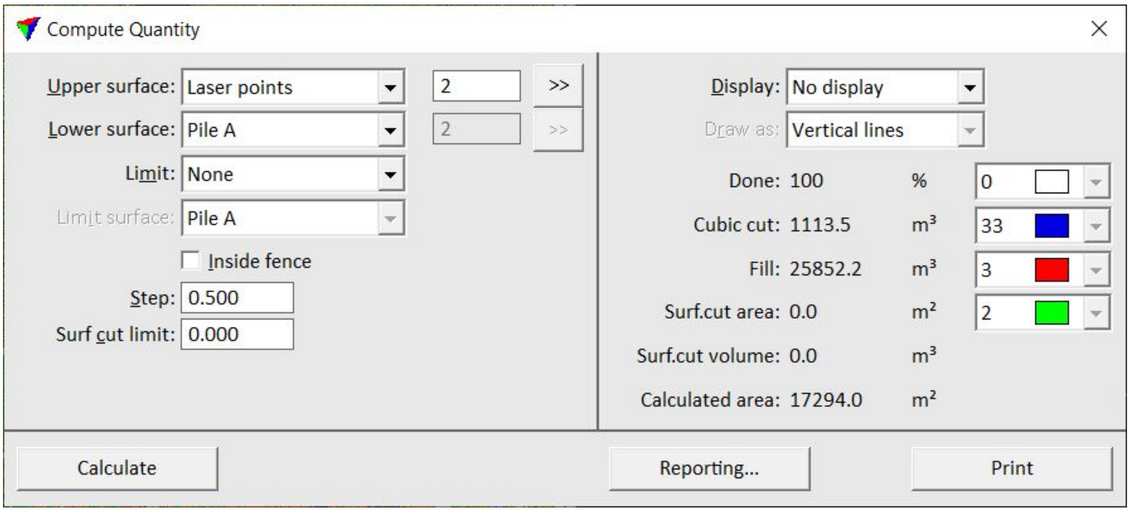
The surfaces for volume calculation can be defined by surface models loaded in TerraModeler or by points loaded in TerraScan.

The tool provides several options for creating text reports of the quantity calculation results. The results are presented in a separate report windows from which a text file can be saved or the report can be printed directly. Besides a simple report for the overall calculation results, there is also the possibility to output See Quantity reports for subregions.

To compute quantities using the grid method:

1. Select the **Compute Quantity** tool.

   The **Compute Quantity** dialog opens:



2. Define settings.

3. If a text report for subregions should be created, click on the **Reporting** button to define settings for the output. See See Quantity reports for subregions for more information.

4. Click on the **Calculate** button to start the computation.

   The computation results are displayed on the right side of the **Compute Quantity** dialog, and possibly in the CAD file and a separate report window.

   The results of the calculation for the whole area can be saved into a simple text report using the **Print** button in the **Compute Quantity** dialog.

The **Erase display** button appears if the results of the quantity computation have been displayed with setting **Display only**. Then, the button can be used to remove the temporary elements that illustrate the computation result. The temporary elements also disappear if the **Compute Quantity** dialog is closed.

SETTING	EFFECT
Upper surface	Name of the upper surface model for the quantity calculation. If **Laser points** is selected, the field for entering class number(s) becomes active. This requires that respective laser points are loaded 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 number field of the **Upper surface**.
Lower surface	Name of the lower surface model for the quantity calculation. If **Laser points** is selected, the field for entering class number(s) becomes active. Use the >> button in order to open the **Select classes** dialog for class selection. This requires that respective laser points are loaded 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 number field of the **Lower surface**.
Limit	Limiting surface to restrict the calculation area: • **None** - no limiting surface used. • **Area** - calculate only within the **Limit surface** area. • **Lower limit** - calculate only above **Limit surface**. • **Upper limit** - calculate only below **Limit surface**.
Limit surface	Surface to limit the calculation area. This is not active if **Limit** is set to **None**.
Inside fence	If on, the calculation takes place only inside a fence or a selected shape element.
Step	Defines the grid size for the calculation. A small step produces more accurate results

SETTING	EFFECT
	but the calculation process takes longer.
Surf cut limit	The software computes the cut volume separately for places where the cut depth is less than this value. This may be used for billing purposes if the price is not based on the cut volume but on the cut area.
Display	Display method for presenting the calculation results graphically in the CAD file: • **No display** - nothing is drawn into the CAD file. • **Display only** - temporary display of vertical line elements. Disappears if the tool is deactivated. • **Write to file** - draws permanent elements on the active level.
Draw as	Elements used to present the results graphically: • **Vertical lines** - a vertical line is drawn at the center of each calculated grid cell. This is the only available option if **Display** is set to **Display only**. • **Horizontal lines** - a horizontal lines is drawn for each calculated grid cell row. • **Filled shapes** - a shape is drawn for each calculated grid cell row. • **Pillars** - a 3D block is drawn for each calculated grid cell. Each block is build from 4 shape elements which may end up with a big number of elements to draw.
Color list next to calculation result line	Defines the display color for drawing the results in the CAD file. Uses the active color table of the CAD file.

Quantity reports for subregions

The **Quantity reporting** dialog provides an option to create output reports of quantity computations for subregions. In addition, a text file can be created that stores variable - value pairs.

Subregions and their names in the report can be defined in several ways:

- **Coordinate blocks** - the software divides the computation area automatically into squared blocks of a given size. The corner coordinates of the blocks are full meter coordinate values.

The names for the subregions are automatically generated from the lower left corner coordinates of each block.

- **Selected shapes** - subregions are defined by selected shapes. This is useful for creating output values for distributed computation areas that are not adjacent to each other. The names for the subregions can be defined by text elements that is placed inside the shapes and selected together with the shapes. The software uses the selected text element whose origin point is inside a selected shape for naming the respective subregion in the report. In addition, if the text element ends with a numerical value, this can be used as a multiplier for the calculated quantities. This may be useful, for example, if a material has a certain density which effects the volume of this material. If no text element is used, the subregions are named automatically with "Region 1", "Region 2", etc.
- **Elevation range** - the computation is performed for a given elevation or depth range. A subregion is defined by a given elevation or depth value interval. The subregions are named automatically by the range values.

A report can include both, results for horizontal regions and for vertical ranges.

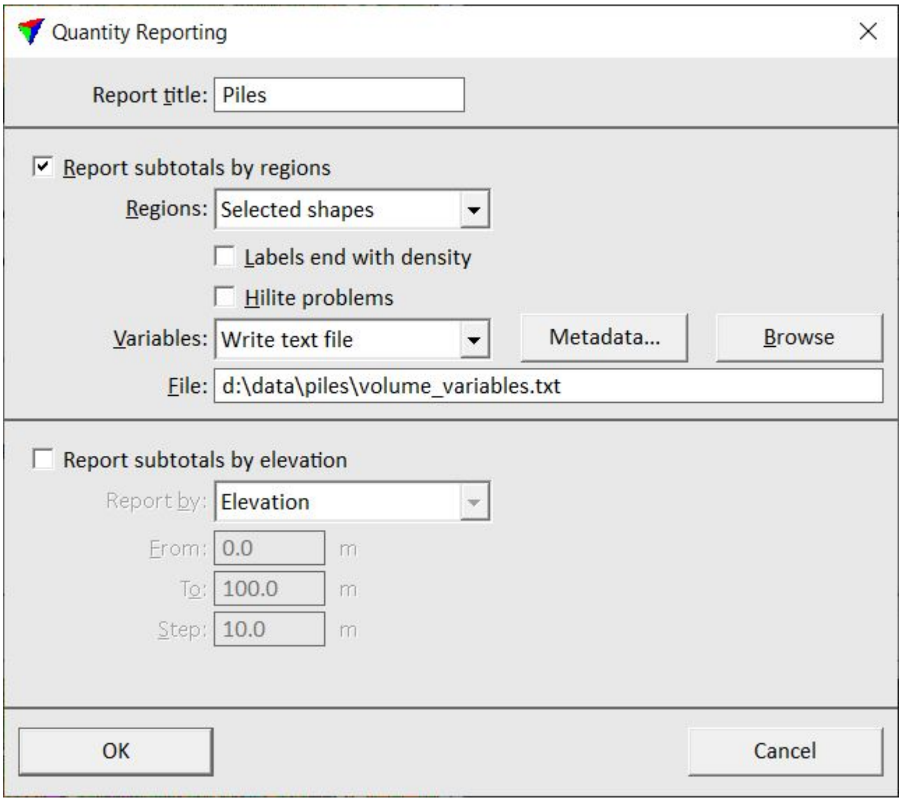
A variable text file can be created for horizontal regions only. The file saves a variable name and its value into a comma-separated text file. Each variable - value pair is written into a new line which starts with the # character. The file starts with the **Metadata** variables, followed by variables for each region, and finally the total values for the whole computation area.

The variable names for each region are a combination of the subregions name and the variable name separated by a underline character, for example, Region 1_CUT for the cut volume of a subregion, or Example1_FILL for the fill volume of a region named Example1 .

To setup quantity reports for subregions:

1. Click on the **Reporting** button in the **Compute Quantity** dialog.

   The **Quantity reporting** dialog opens:



2. Define settings according to your needs for the output report.

3. Click OK.

4. You can continue with **Calculate** in the **Compute Quantity** dialog.

If one of the options in the **Quantity reporting** dialog is switched on, an **Output report** window is displayed. Use the commands from the **File** pulldown menu in the report window to save the report as a text file or to print it directly.

If **Variables** is set to **Write text file** in the **Quantity reporting** dialog, the file storing variable - value pairs is created and saved at the given location on a hard disk.

SETTING	EFFECT
Report title	Free text that is used as title in the report.
Report subtotals by regions	If on, the quantity calculation results are reported for each region separately.
Regions	Method of how regions are defined: • **Coordinate blocks** - regions are defined by squared blocks with an edge length of the given **Block size** value. • **Selected shapes** - regions are defined by selected shape elements
Labels end with density	If on, a numeric value at the end of a text element inside a region is used as a multiplier to all volume values calculated for this region.

SETTING	EFFECT
	This is only active if **Regions** is set to **Selected shapes** .
Hilite problems	If on, region shapes without a text element inside are highlighted.
Variables	Determines if a variable text file is written and saved: • **Do not write** - no text file is written. • **Write text file** - a text file written and saved at the given **File** location.
Metadata	The button opens the **Quantity metadata** dialog. The metadata is included at the beginning of the variable text file.
Report subtotals by elevation	If on, the quantity calculation results are reported for defined elevation or depth ranges.
Report by	Defines the type of range for the report: • **Elevation** - quantities are reported for defined elevation ranges. • **Depth** - quantities are reported for defined depth ranges.
From	Start value of the elevation/depth range for the report.
To	End value of the elevation/depth range for the report.
Step	Elevation or depth interval that defines the vertical size of the subregions.
Order	Defines what is stored in the rows and columns of the report: **Regions as rows** or **Subtotals in rows**.

Compute Section Quantity

Not Lite

Compute Section Quantity tool lets you attach section templates to station ranges along an alignment element. This template list can be used for computing quantities and for generating an excavation based on the cross section templates. The section templates must be defined with the Define Section Templates tool before they can be used for section quantity computation.

The alignment can be defined by a linear element, such as a 3D line string or complex chain. The area of calculation is limited by the extensions of the alignment element and the section template(s).

The excavation is computed related to one or two surfaces. The surfaces can be defined by surface models loaded in TerraModeler.

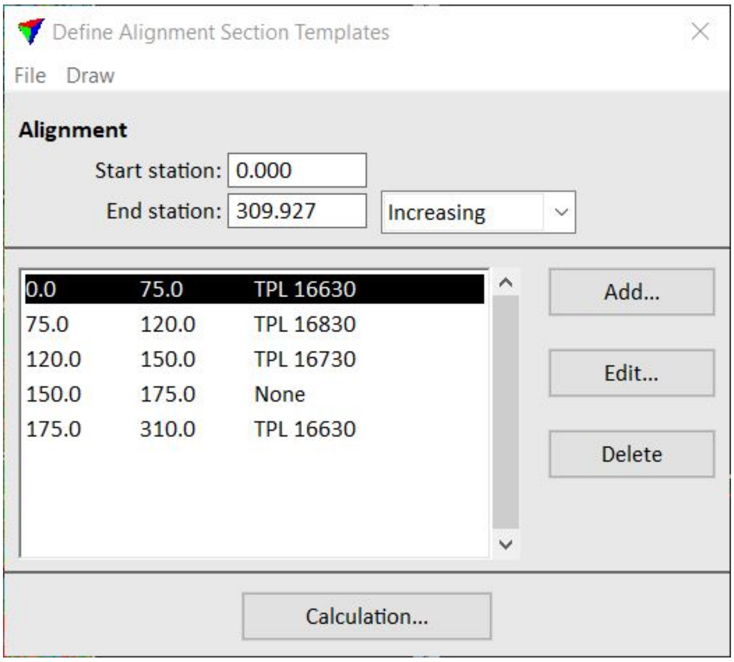
The results of the computation are shown in the tool’s dialog. They can be presented graphically in the CAD file and a text report can be stored as well.

To attach templates to an alignment:

1. Select an alignment element.

2. Select the **Compute Section Quantity** tool.

   The **Define Alignment Section Templates** dialog opens:



   The template list contains rows with a station value range and the name of an assigned section template. A gap between station ranges of successive rows is interpreted as a linear transformation between two template types. You can skip a station range by selecting template type **None**. Such a station range is not included in the calculation.

SETTING	EFFECT
Start station	Station value at the start point of the alignment element.
End station	End station value defined by the length of the alignment element: • **Increasing** - station values are increasing from start to end station. • **Decreasing** - station values are decreasing from start to end station

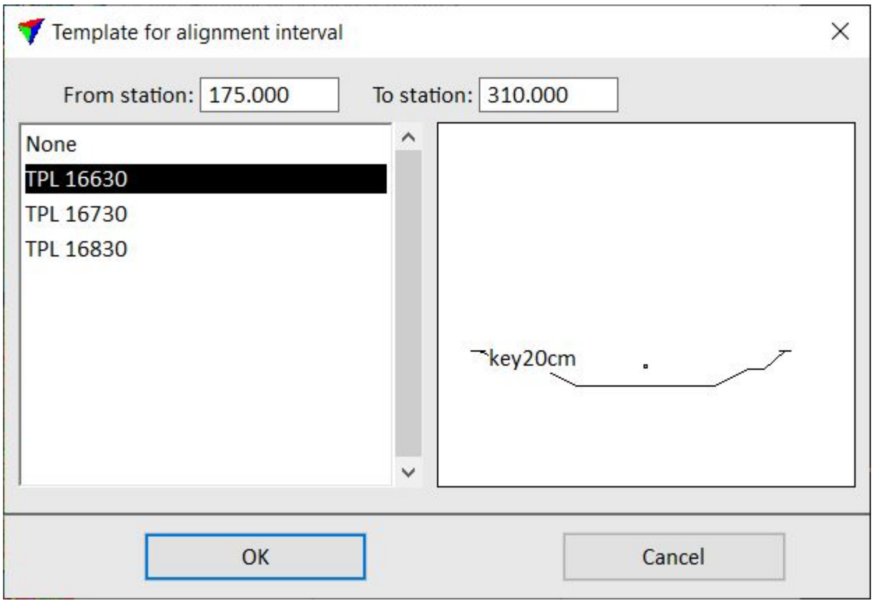
TO	CHOOSE BUTTON/MENU COMMAND
Add a new station range to the list.	Add
Modify the template for a selected station range.	Edit
Delete a selected station range from the list.	Delete
Start the quantity calculation based on the template list.	Calculation
Load a previously saved template list file from a hard disk.	File / Load
Save a template list into a text file on a hard disk.	File / Save as
Draw the section templates into cross sections. The cross sections must be created in advance along the same alignment elements using the Draw Alignment Sections tool.	Draw / In sections

3. Use the **Add** button to add a new template to the list.

OR

3. Use the **Edit** button to edit a selected template.

This opens the **Template for alignment interval** dialog:



4. Define **From station** and **To station** values in order to define the station range for using this template.

5. Select a section template form the list of **Section templates** that is used for the given section range.

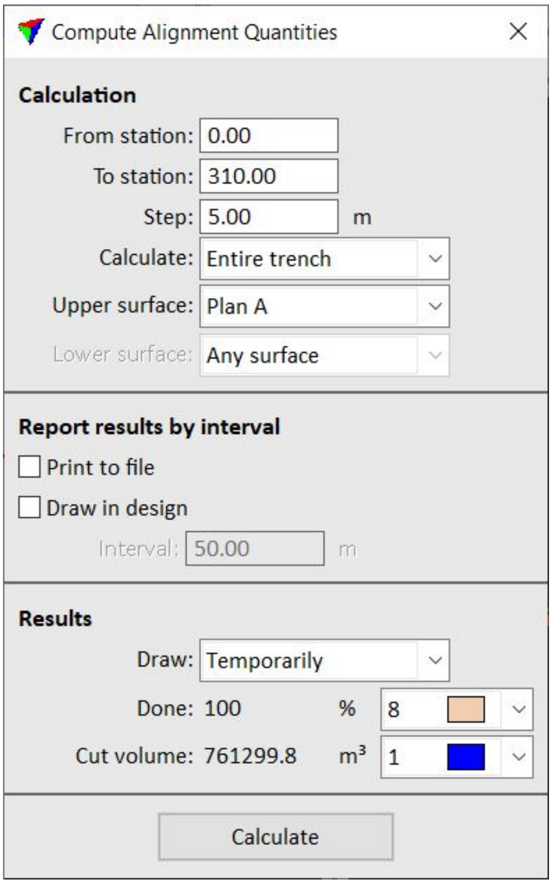
6. Click OK.

   The template is added to the list in the **Define alignment section templates** dialog. You can continue with step 3 until templates for all station ranges are defined.

   It is recommended to save the template list into a file that can be reloaded into the dialog. Select **Save as** command from the **File** menu to create a simple text file with the extension *.KLS.

7. Click on the **Calculation** button in the **Define alignment section templates** dialog.

   This closes the list and opens the **Compute Alignment Quantities** dialog:



8. Define settings and click **Calculate**.

   This starts the quantity computation. The results are displayed in the lower part of the **Compute alignment** dialog and possibly in the CAD file. If **Print to file** was selected in the first dialog, a standard Windows dialog opens to define the location for storing the text file on a hard disk.

SETTING	EFFECT
From station	Start station value for the quantity computation.
To station	End station value for the quantity computation.
Step	Defines the distance between consecutive section templates for the calculation. A small step produces more accurate results but the calculation process takes longer.
Calculate	Defines how to calculate the excavation related to existing surface models: • **Entire trench** - the quantity of excavation is computed for the area enclosed by a surface model and the trench boundaries defined by the section templates.

SETTING	EFFECT
	• **Layer** - the quantities of excavation is computed for the area between two surface models and the trench boundaries defined by the section templates.
Upper surface	Name of the upper surface for quantity calculation.
Lower surface	Name of the lower surface for quantity calculation. This is only active if **Calculate** is set to **Layer**.
Print to file	If on, the results of the computation are saved into a simple text file.
Draw into design	If on, a linear scale of the alignment is drawn into the CAD file. The scale is divided into sections of the length defined by the Interval value. For each section, the rounded calculated cut volume value is displayed.
Draw	Display method for presenting the calculation results graphically in the CAD file: • **Temporarily** - temporary display of elements. Disappears if the tool is deactivated. • **Permanently** - draws permanent elements into the CAD file on the active level.
Color list next to calculation result line	Defines the display color for drawing the results in the CAD file. Uses the active color table of the CAD file.

Compute Stockpile Volume

Compute Stockpile Volume tool computes stockpile quantities and report volumes separately for stockpile regions. The calculation is based on a grid method. You can control the accuracy and speed of calculation by defining the grid size. The tool suits for reporting volume of stockpiles on smooth surfaces, as the reference surface cannot represent a rough, complex surface. Similar results applying more complex reference surface can be computed with Compute Quantity tool.

The surface for volume calculation can be defined by surface models loaded in TerraModeler or by points loaded in TerraScan.

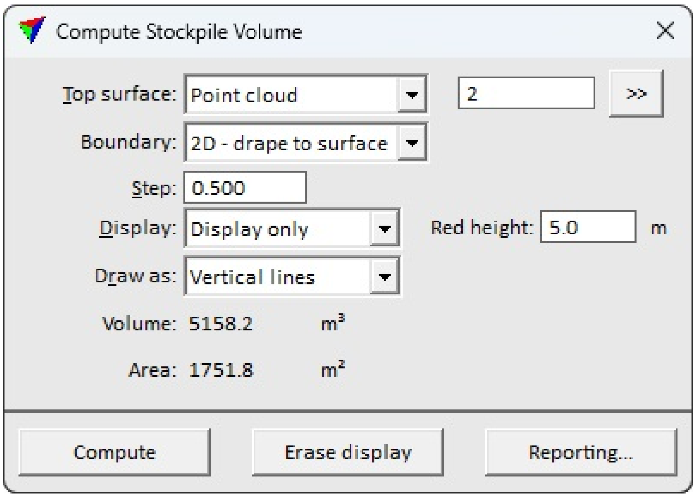
In addition to the surface, this tool requires one or several stockpile region boundaries limiting stockpiles. A region boundary is used to bound volume computation and reporting, and to represent the reference surface.

To compute stockpile volumes:

1. Select polygon elements defining stockpile boundaries and (Optionally) region name text labels.

2. Select the **Compute Stockpile Volume** tool.

   The **Compute Stockpile Volume** dialog opens:



2. Define settings, and optionally Reporting options. Click **Compute**.

   The **Stockpile Volume Report** dialog opens and the summary of computation results is displayed in the **Compute Stockpile Volume** dialog. If a **Display** option was selected, results are displayed graphically.

SETTING	EFFECT
Top surface	Name of the upper surface model for the quantity calculation.

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. Available only if **Top surface** is set to **Point cloud**.
Boundary	Method for defining the reference surface: • **2D, drape to surface** - stockpile boundaries only define the computation boundary. Reference surface is generated on the fly draping the boundary to the **Top surface** level. • **3D, use directly** - the polygons define both the computation boundary and the reference surface. The reference surface is triangulated directly from polygon vertices. Use only if polygon boundaries share the elevation system with the **Top surface**.
Step	Defines the grid size for the calculation. A small step produces more accurate results but the calculation process takes longer.
Display	Display method for presenting the calculation results graphically in the CAD file: • **No display** - nothing is drawn into the CAD file. • **Draw temporarily** - temporary display of vertical line elements. Disappears if the tool is deactivated or display cleared with **Erase display**. • **Write to file** - draws triangles as permanent elements into the CAD file on the active level.
Red height	Upper limit for the display color scale. Available only if **Display** is active.
Draw as	Elements used to present the results graphically: • **Vertical lines** - a vertical line is drawn at the center of each calculated grid cell. • **Pilars** - a 3D block is drawn for each calculated grid cell. Each block is build from 4 shape elements which may end up with a big number of elements to draw.

SETTING	EFFECT
	Available only if **Display** is set to **Write to file**.

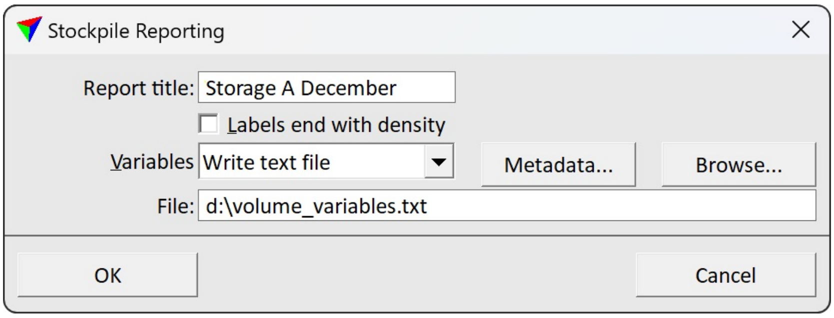
Reporting options

The **Stockpile reporting** dialog provides an option to create text file reports storing variable - value pairs. These reports can be further used to form efficiently pre-formatted report documents filling calculated values.

To setup stockpile reporting:

1. Click on the **Reporting** button in the **Compute Stockpile Volume** dialog.

   The **Stockpile Reporting** dialog opens:



2. Define settings according to your needs for the output report.

3. Click OK.

4. You can continue with **Calculate** in the **Compute Stockpile Volume** dialog.

   If **Variables** is set to **Write text file** in the **Quantity reporting** dialog, the file storing variable - value pairs is created and saved at the given location on a hard disk.

SETTING	EFFECT
Report title	Free text that is used as title in the report.
Labels end with density	If on, a numeric value at the end of a text element inside a region is used as a multiplier to all volume values calculated for this region.
Variables	Determines if a variable text file is written and saved: • **Do not write** - no text file is written. • **Write text file** - a text file written and saved at the given **File** location.
Metadata	The button opens the **Quantity Metadata** dialog. The metadata is included at the

SETTING	EFFECT
	beginning of the variable text file.
File	If on, region shapes without a text element inside are highlighted.

Compute Tunnel Volume

Not Lite

Compute Tunnel Volume tool computes the volume of a tunnel defined by cross section elements drawn into their true 3D positions. The cross section elements may represent the design of a new tunnel or may be created based on the point cloud from a terrestrial or mobile laser scanner. The tool does not rely on surface models loaded in TerraModeler.

The cross section elements must be closed vector elements, such as shapes or complex shapes, drawn perpendicular to an alignment element that runs along the tunnel. The interval between consecutive cross sections may vary.

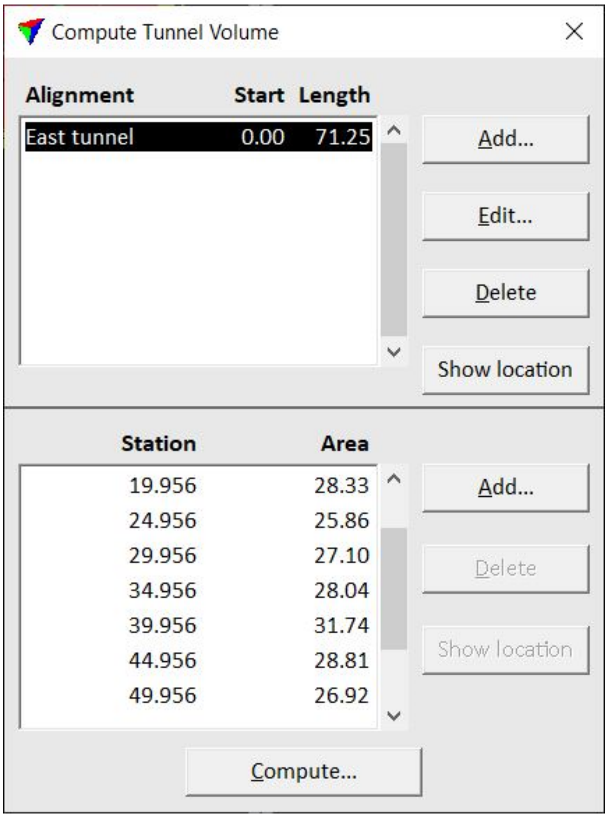
The volume can be computed for the whole tunnel area defined by the cross sections. Upper and lower elevation values can be defined for limiting the calculation to the area between the limit(s) and the cross section boundaries. The limits are given in absolute elevation values.

The tool can also compute multiple volumes for several elevation intervals within the cross sections. Then, the upper or lower elevation limit is moved within the tunnel area between two absolute elevation values in steps of a given size. The volumes are reported for each elevation interval defined by the lowest or highest elevation value and the moving elevation limit.

To compute tunnel volume:

1. Select the **Compute Tunnel Volume** tool.

   This opens the **Compute Tunnel Volume** dialog:

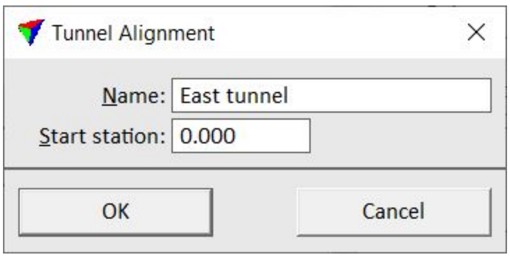


The upper list contains a list of alignments along which tunnel sections are used for the computation. The lower list displays all the tunnel cross sections defined for the alignment selected in the upper list.

2. Select an alignment element running along a tunnel.

3. Click **Add** next to the upper list to add the selected alignment.

   This opens the **Tunnel Alignment** dialog:



4. Type a **Name** for the alignment, define a **Start station** value, and click OK.

5. Select the row of the alignment in the upper list box of the **Compute Tunnel Volume** dialog.

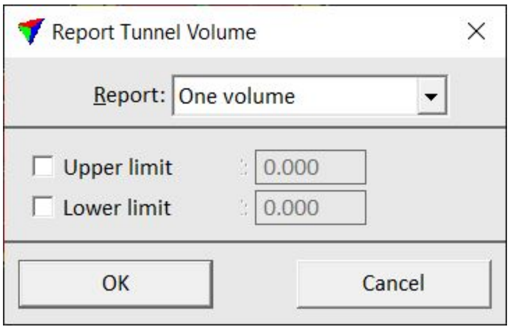
6. Select cross section elements for the selected alignment.

7. Click **Add** next to the lower list to add the selected cross sections.

8. Validate the created list of cross sections. Make sure that station positions and cross section areas make sense.

9. Click **Compute** to start the volume computation.

   This opens the **Report Tunnel Volume** dialog:



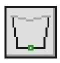
10. Define settings and click OK.

    This computes the volume and displays a report. You can save the report as a text file or print it directly by using the corresponding commands from the **File** menu in the **Output** window.

SETTING	EFFECT
Report	Method of volume reporting: • **One volume** - calculates one volume for the complete tunnel area defined by the cross sections. • **Multiple volumes** - calculates volumes for multiple elevation steps within the tunnel and the cross sections.
Upper limit	If on, the volume is calculated from the tunnel floor up to the given elevation value. This is only active if **Report** is set to **One volumes**.
Lower limit	If on, the volume is calculated from the given elevation value up to the tunnel roof. This is only active if **Report** is set to **One volumes**.
Move	Limit that is moved to define the elevation steps for multiple volume calculation: **Upper limit** or **Lower limit**. This is only active if **Report** is set to **Multiple volumes**.
First	First elevation value included in volume calculation. This is only active if **Report** is set to **Multiple volumes**.
Last	Last elevation value included in volume calculation. This is only active if **Report** is set to **Multiple volumes**.
Step	Size of an elevation interval for volume calculation. This is only active if **Report** is set to **Multiple volumes**.

Define Section Templates

Not Lite



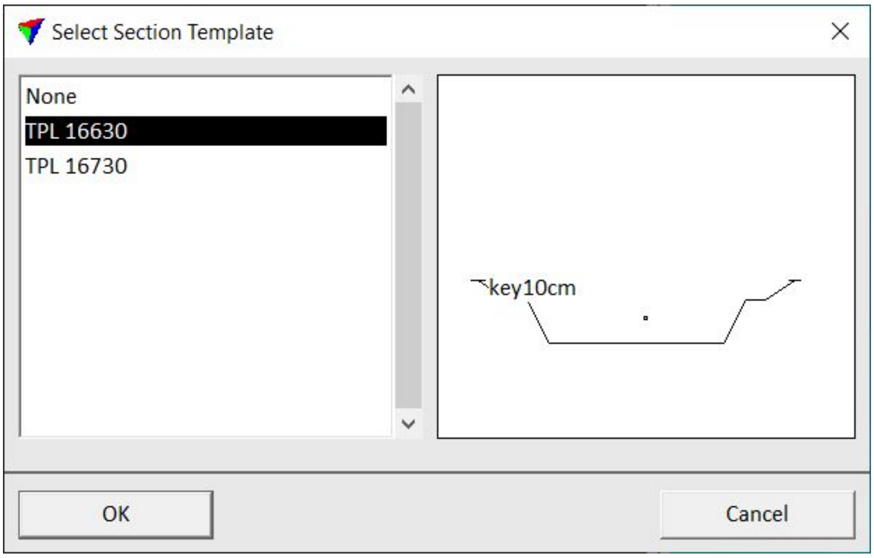
Define Section Templates tool lets you create a list section templates, which are, for example, definitions of typical trench cross sections.

TerraModeler cross sections templates are meant to be used for computing trench excavation quantities with the Compute Section Quantity tool. The cross section templates are automatically saved into a file using the same directory and name as the active CAD file. The file extension is *.KDB. For example, if the active CAD file is D:\DGN\PROJ12.DGN, the section templates are saved to and loaded from the file D:\DGN\PROJ12.KDB.

To add or edit section templates:

1. Select the **Define Section Templates** tool.

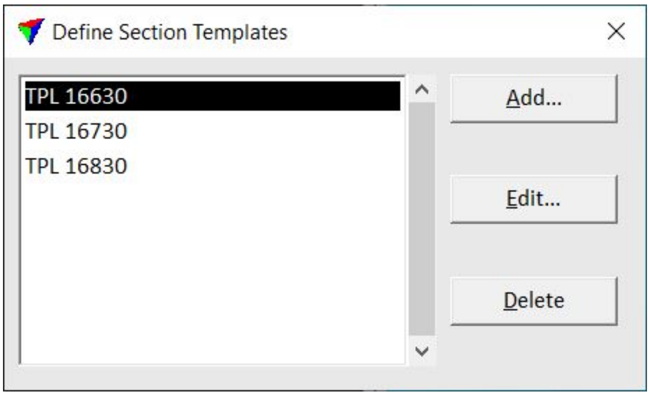
   This opens the **Select Section Template** dialog:



   The dialog shows section templates that have been defined earlier and saved into a .KDB file.

   Click OK.

   The **Define Section Templates** dialog opens:

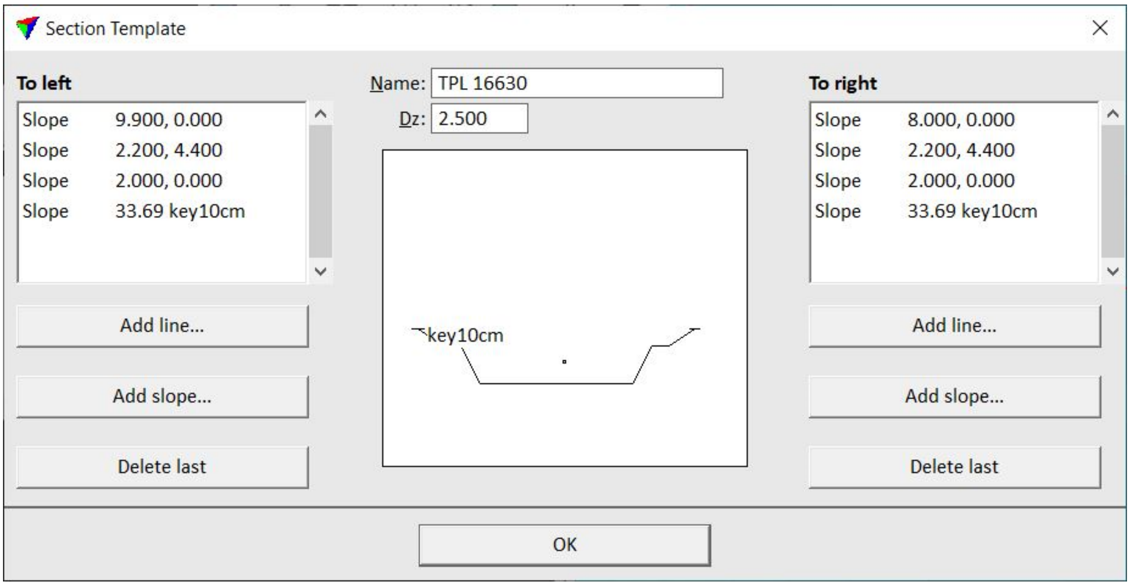


2. Use the **Add** button to add new template definitions to the list.

   OR

2. Use the **Edit** button to edit a selected section template.

   This opens the **Section Template** dialog which is used to define a section template:



   A small circle in the center of the template preview symbolizes the spot where an alignment element goes through. **Dz** specifies an initial depth below that spot.

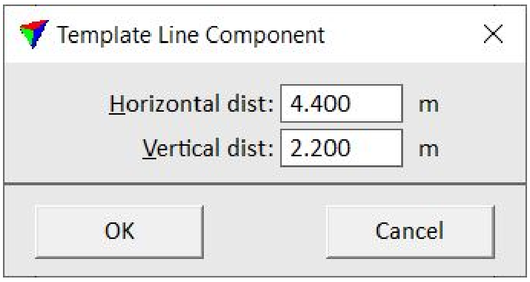
   The shape of the cross section is defined as component lists that extend to the left and to the right from the initial **Dz** point. A component can be a line or a slope. A line has a vertical and a horizontal distance. A slope has a fixed angle at which it extends until it reaches a specified surface.

3. Enter a **Name** for the section template and a **Dz** value.

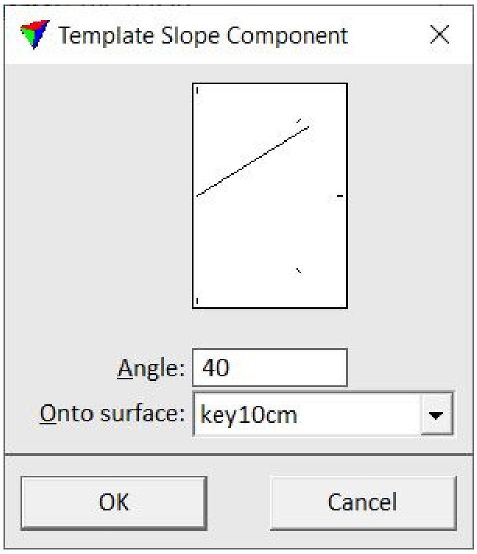
4. Define components for the left and right side of the section by clicking on the respective **Add line** or **Add slope** buttons in the **Section Template** dialog. You can delete the last added component using the **Delete last** buttons.

   Add line opens the **Template Line Component** dialog:

   Add slope opens the **Template Slope Component** dialog:



- **Horizontal dist** - XY distance between the initial **dz** point or the end point of the last section component and the next point.
- **Vertical dist** - Z distance between the initial **dz** point or the end point of the last section component and the next point.



- **Angle** - gradient of the slope. Positiv or negativ values between 0 and 90 degree are possible.
- **Onto surface** - name of the surface model to which the slope is extended. The list contains surface models loaded in TerraModeler.

5. After a section has been defined, click OK in the **Section Template** dialog.

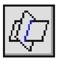
   This adds the new template to the list or applies the changes to an existing template.

6. Close the **Define Section Templates** dialog.

   This saves the *.KDB file to the hard disk.

   You can use the **Delete** button in the **Define Section Templates** dialog in order to delete a selected section template.

Draw Surface Intersection



Draw Surface Intersection calculates the intersection of two surfaces and draws linear elements along the intersection. These linear elements represent the line of intersection along which the two surfaces have exactly the same elevation.

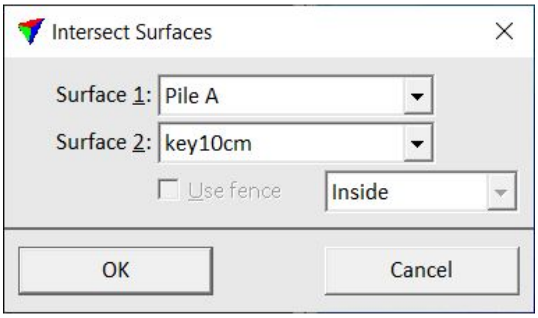
When calculating the line of intersection, TerraModeler compares individual triangles from the two surfaces with each other. If the planes of two triangles intersect each other, the intersection is drawn as a line segment. Thus, the intersection line might be represented by quite complex line strings or even by several line elements.

This tool can be used, for example, when computing cut and fill volumes as it creates a linear element showing the location where two surfaces have the same elevation. The line of intersection is the boundary between cut and fill areas of the two surfaces.

To draw intersection of two surfaces:

1. Select the **Draw Surface Intersection** tool.

   The **Intersect Surfaces** dialog opens:



2. Select the two surfaces and click OK.

   This starts the calculation of the intersection line. The line is drawn into the CAD file using active level and symbology settings in the CAD file.

SETTING	EFFECT
Use fence	If on, the intersection is calculated for an area that is defined by a fence: **Inside** or **Outside** the fence.

Regions toolbox

Not Lite, Not UAV

The tools in the **Regions** toolbox are used to create, manage, delete, and update regions.

Regions are another method to divide an area into thematic subareas, the regions. From linear region boundaries the tools can create and update closed shapes. A center point identifies the region and links it to thematic information.

Region definitions can be used for:

- labeling surface areas
- using color-filled shapes for visualization purposes
- create pattern for boundaries and region areas for visualization purposes

Settings related to regions can be found in Regions / Region design category, Regions / Region levels category, Regions / Region types category and Regions / Title formats category of the TerraModeler **Settings**.

In contrast to tools in the Domains toolbox, region tools do not rely on surface models or relate to the triangle structure of surfaces.

TO	USE TOOL	
Open regions window		Regions *Not Lite, Not UAV*
Create a new region definition		Create Region *Not Lite, Not UAV*
Delete existing region definition		Delete Region *Not Lite, Not UAV*
Edit region information		Edit Region Info *Not Lite, Not UAV*
Move region center point and title		Move Region Center *Not Lite, Not UAV*
Update region definitions (modified boundaries)		Update Regions *Not Lite, Not UAV*

Create Region

Not Lite, Not UAV

Create Region tool creates a new region definition. A region is defined by boundary lines that form closed areas. When a region is created, the region definition is added to the list in the **Regions** window and a center marker is placed inside the boundary lines.

The software creates a region from lines that are visible in a top view and form a closed area around the region center point. The elevation of the lines also effect the elevation of the region shapes. It is recommended to work with 2D elements or at a single elevation level when working with regions. If necessary, the correct elevation values can be assigned to the final region shapes by using, for example, Linear Elevation tool or Drop Element On Surface tool in TerraModeler, or the Drape Linear Element tool together with point cloud data in TerraScan.

The boundary line network should form closed areas in order to ensure that regions can be created. However, the lines can overlap each other or be located on different CAD file levels. There are some tolerances regarding gaps in the line network that are defined in Regions / Region design category of the TerraModeler **Settings**.

Each region belongs to a region types. A region type can refer to, for example, a category of land use. The region type determines the symbology used for visualizing the regions. Region types are defined in Regions / Region types category of the TerraModeler **Settings**.

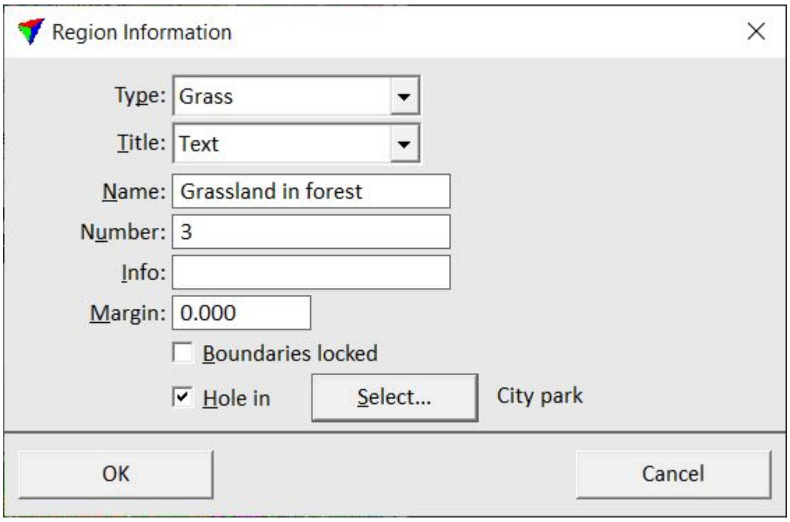
When a region is created, a center marker is placed inside the region area. The center marker can be labeled with a so-called title. The title can include some information about the region. It is drawn as text element into the CAD file. Title formats are defined in Regions / Title formats category of the TerraModeler **Settings**.

To create a new region definition:

1. Select the **Create Region** tool.

2. Click inside boundary lines that define the region area.

   This opens the **Region Information** dialog:



3. Define settings for the new region.

4. Click OK.

This adds the region definition to the list. You can continue with step 2.

SETTING	EFFECT
Type	Type of the new region. See Regions / Region types category for instruction how to define region types.
Title	Title format for labeling the new region. See Regions / Title formats category for instruction how to define region titles.
Name	Free text that is used a name for the new region.
Number	Number of the region. Counts up automatically.
Info	Free text that is used as additional information about the new region.
Margin	Area added around a region as a placeholder, for example, for line pattern that need more space than a simple line. The region is shrunk by this value in order to avoid overlap between adjacent regions when using different line pattern for the display.
Boundaries locked	If on, the region is not recomputed when using the Update Regions tool after the boundary line elements were modified.
Hole in	If on, the new region is defined as a hole. If the area of the new region is completely

SETTING	EFFECT
	surrounded by another region, the option is switched on automatically. The name of the surrounding region is displayed next to the **Select** button. If the region should not form a hole, click on the **Select** button and select **Not a hole** in the **Select parent region** dialog.

Create region tool does not draw the region shape or any other display into the CAD file. Use Update Regions tool in order to display the region.

Delete Region

Not Lite, Not UAV

Delete Region tool deletes a region definition. The region must be drawn in the CAD file by the Update Regions tool in order to be deleted with this tool.

You may use the Region / Delete command to perform the same action for selected regions in the **Regions** window. The command can also remove regions that are not drawn into the CAD file.

To delete a region:

1. Select the **Delete Region** tool.
2. Move the mouse pointer inside a top view.

   The region at the mouse pointer location is highlighted.
3. Identify the region to be deleted with a data click inside the region.

   The software asks you to confirm the deletion process.
4. Click **Yes** in the alert dialog.

   This deletes the region definition and all related elements. You can continue with step 2.

The region display is always linked to the region definition. If a region definition is deleted, all display elements are deleted as well.

Edit Region Info

Not Lite, Not UAV

Edit Region Info tool enables the modification of thematic information of a region.

To change region information:

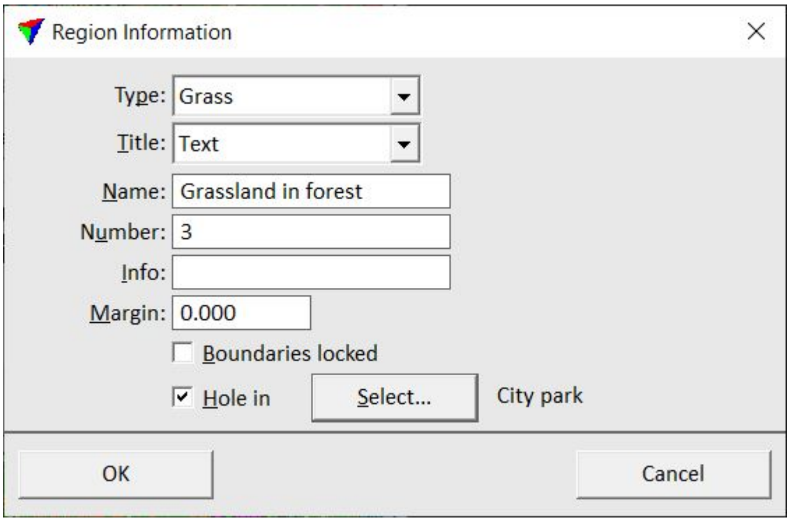
1. Select the **Edit Region Info** tool.

2. Move the mouse pointer inside a top view.

   The region at the mouse pointer location is highlighted.

3. Identify the region to be modified with a data click.

   This opens the **Region Information** dialog:



4. Modify the information for the region.

5. Click OK in order to apply the changes.

   You can continue with step 2. Use the Update Regions tool in order to apply changes to the display.

SETTING	EFFECT
Type	Type of the region. See Regions / Region types category for instruction how to define region types.
Title	Title format for labeling the region. See Regions / Title formats category for instruction how to define region titles.
Name	Free text that is used a name for the region.

SETTING	EFFECT
Number	Number of the region.
Info	Free text that is used as additional information about the region.
Margin	Area added around a region as a placeholder, for example, for line pattern that need more space than a simple line. The region is shrunk by this value in order to avoid overlap between adjacent regions when using different line pattern for the display.
Boundaries locked	If on, the region is not recomputed when using the Update Regions after the boundary line elements were modified.
Hole in	If on, the region is defined as a hole. If the area of the region is completely surrounded by another area, the option is switched on automatically. The name of the surrounding area is displayed next to the **Select** button. If the region should not form a hole, click on the **Select** button and select **Not a hole** in the **Select parent region** dialog.

Move Region Center

Not Lite, Not UAV

Move Region Center tool moves the center marker of a region to another location. The center marker may be linked with a text element which has been defined as region title.

To move the region center marker:

1. Select the **Move Region Center** tool.

2. Move the mouse pointer inside a top view.

   The region center closest to the mouse pointer location is highlighted.

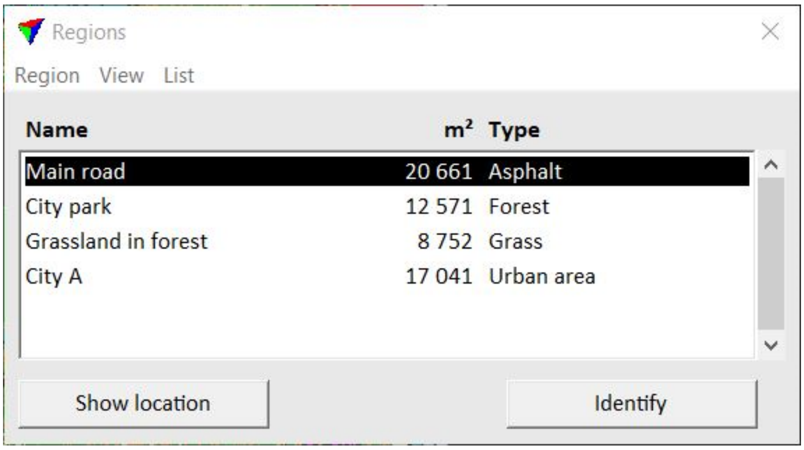
3. Identify the region center to be moved with a data click.

4. Define the new location of the center marker with another data click.

Regions

Not Lite, Not UAV

Regions tool opens the **Regions** window. The window contains a list of regions that are currently defined in the CAD file. The window's menu includes commands for creating, modifying, and deleting regions, and for creating output files of the region list.



TO	CHOOSE MENU COMMAND
Create a new region	Region / Create
Modify the information for a region	Region / Edit information
Move the center point of a region	Region / Move center point
Assign a new boundary to a region	Region / Assign new boundary
Delete a region	Region / Delete
Modify the size of the window	View/Small dialog View/Medium dialog View/Large dialog
Define fields that are displayed in the list	View / Fields
Sort the list of regions	View / Sort by
Create output files for the list of regions	List / Regions

Region / Create

Create command performs the same action as the Create Region tool.

Region / Edit information

Edit information command performs the same action as the Edit Region Info tool.

Region / Move center point

Move center point command performs the same action as the Move Region Center tool.

Region / Assign new boundary

Assign new boundary command defines a new boundary to a region. The new boundary must be a closed shape element.

To assign a new boundary to a region:

1. Place the new boundary as shape element.
2. Select the region in the **Regions** window.
3. Select **Assign new boundary** command from the **Region** pulldown menu.
4. Identify the new boundary shape with a data click.
5. Accept the selected boundary.

   This assigns the new boundary to the selected region. The boundary type changes to Shape for this region. Use Update Regions tool in order to see the changes in the CAD file.

Region / Delete

Delete command deletes a region definition. The region definition must be selected in the list of regions.

To delete a region:

1. Select the region in the **Regions** window.
2. Select the **Delete** command from the **Region** pulldown menu.

   The software asks you to confirm the deletion process.

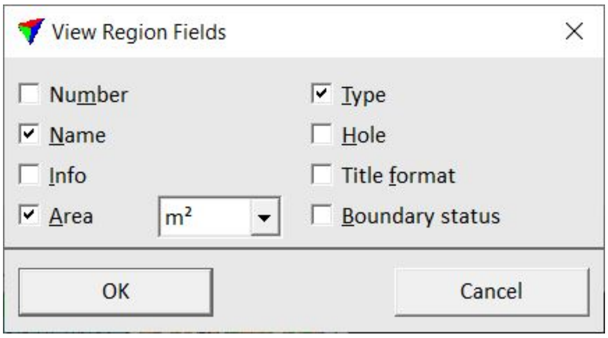
3. Click **Yes** in the alert dialog.

   This deletes the region definition and all related elements.

The **Delete** command can perform the same action for all regions as the Delete Region tool.

View / Fields

Fields command lets you choose the fields that are displayed in the **Regions** window. The fields show the information stored for each region, such as number, name, information, size, type, if a region forms a hole within another region, title format and boundary status.



If the information for a field is not displayed completely in the list, you may enlarge the **Regions** window by using the commands from the **View** pulldown menu.

View / Sort by

Sort by command sorts the list of regions according to the selected attribute:

- **Name** - ascending alphabetical order of region names.
- **Number** - ascending by region number, the smallest number is displayed first in the list.
- **Type** - ascending alphabetical order of region types.
- **Area** - descending by region size, the largest region is displayed first in the list.

List / Regions

Regions command provides several options for the output of the list. The output options include:

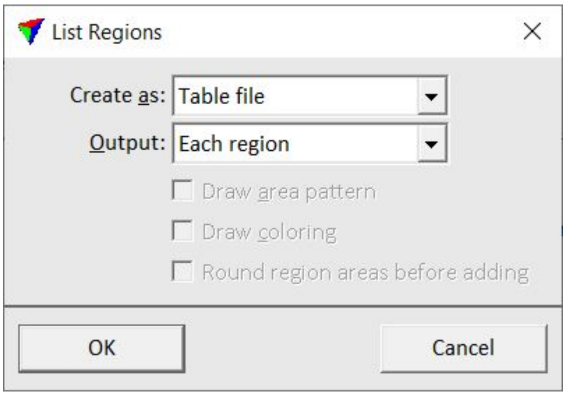
- **Picture** - the list is drawn as cell element into the CAD file. The list can include a color field and/or an area pattern field which illustrate the symbology of the different regions. *(Bentley CAD only)*
- **Text file** - a simple text file is created where fields are delimited by spaces.
- **Table file** - a text file is created where fields are delimited by a tab.

The output can either contain each region of the list or summarize regions of the same type. Depending on the output settings, the list can include information such as type, name, area, count of regions per type, total area per type, color field, area pattern field.

To create output of the region list:

1. Select the **Regions** command from the **List** pulldown menu.

   The **List regions** dialog opens:



2. Define settings and click OK.

3. If **Create as** is set to **Picture**, move the mouse pointer inside a top view. Define the location of the region list by a data click.

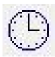
   OR

4. If **Create as** is set to **Text file** or **Table file**, a standard Windows dialog opens for saving a file. Define a location on a hard disk in order to store the text file.

SETTING	EFFECT
Create as	Type of the output: • **Picture** - draws the list of regions as a cell element into the CAD file. • **Text file** - saves a space-delimited text file. • **Table file** - saves a tab-delimited text file.
Output	Defines how regions of the same type are included in the output: • **Each region** - the output includes a line for each region. • **Region type totals** - the output includes a line for each region type, the areas of regions of the same type are summarized.
Draw area pattern	If on, the list output includes an area pattern field for each region (type). This is only active if **Create as** is set to **Picture**.
Draw coloring	If on, the list output includes a color field for each region (type). This is only active if **Create as** is set to **Picture**.
Round regions areas before adding	If on, the region sizes are rounded before they are summed up for the total size. This is only active if **Output** is set to **Region type totals**.

Update Regions

Not Lite, Not UAV



Update Regions tool redraws the regions in the CAD file. If new region definitions were created or existing regions were updated, the tool draws all display elements into the CAD file that are defined for a region. This includes a center marker, a so-called solution shape, and optionally a color-filled shape, area pattern, and a title.

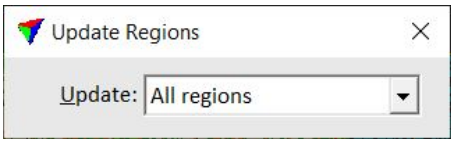
A region is displayed by the center marker and the solution shape. The additional elements drawn for a region depend on the region type and are defined in Regions / Region types category of the TerraModeler **Settings**. The title format and content are defined in Regions / Title formats category of the TerraModeler **Settings**.

The different region elements can be drawn on several CAD file levels which supports their modification with CAD tools. The levels are defined in Regions / Region levels category of the TerraModeler **Settings**.

To redraw regions:

1. Select the **Update Regions** tool.

   The **Update Regions** dialog opens:

2. Select in the **Update** list what is updated: **All regions** or regions in the current **View contents only**.

3. Click inside a top view.

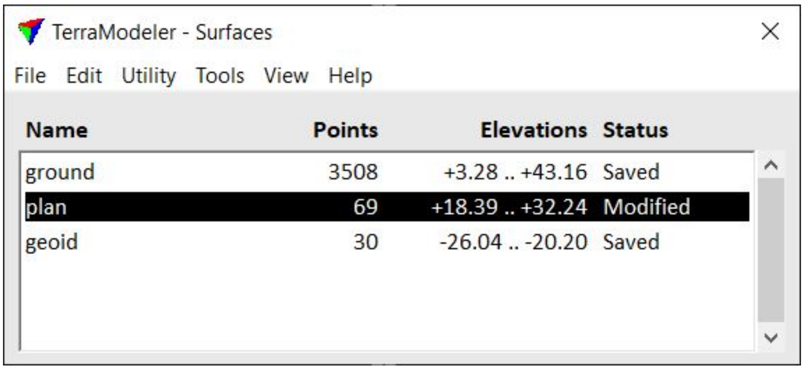
   The update starts and regions are redrawn.

If a region can not be drawn, an error message is displayed. This can be caused, for example if region boundaries are not closed or if they are deleted. There are some tolerances related to gaps in the boundary line network which are defined in Regions / Region design category of the TerraModeler **Settings**.

Managing Surfaces

TerraModeler supports theoretically an unlimited number of surfaces during the same session. In practice, the performance of the software and the ability to create, process, and display surface models depends on the size of the models and the amount of RAM available on the computer.

The Surfaces tool opens a window that contains a list of surfaces that are currently loaded into TerraModeler.



The list of loaded surfaces in the **Surfaces** window shows information about each surface. The information that can be displayed includes surface attributes, storage information, and status information. The file **Status** determines, whether a surface model has been modified (**Status = Modified**), saved (**Status = Saved**) or opened as a reference surface (**Status = Reference**).

The window's menu provides commands for managing the surfaces, such as open, save, rename and close a surface, view surface statistics, modify the triangles of a surface, import and export data, etc. Further, there are options to derive new surfaces by applying calculations to existing surface models. The menu also contains commands for opening all toolbars that are part of the **TerraModeler** toolbox.

There are three methods included in the **Surfaces** window that produce surface representation files in batch processes:

- Produce contours - creates contour CAD files that can store contour lines, labels and ticks.
- Produce lattice models - creates lattice text files that store a gridded DEM of format XYZ.
- Produce triangles - creates triangle CAD files that store the triangle shapes.

The batch processes can include data from several sources, such as point cloud data from TerraScan projects, survey elements from TerraSurvey or other applications, and breaklines filtered by breakline rules.

Edit pulldown menu

Menu commands from **Edit** pulldown menu in the **Surfaces** window are used to modify surfaces and to delete surface files from the hard disk.

TO	USE COMMAND
Modify surface information	Modify settings
Modify display settings for surfaces in profiles	Profile settings
Exclude long triangles from a surface	Exclude long triangles
Change elevations of a surface by a mathematical equation	Modify elevations
Remove unnecessary points from a surface	Thin
Remove previously deleted points and triangles from a surface	Compress
Add intermediate points to a surface	Add inferred points
Delete a surface file from the hard disk	Delete surface

Add inferred points

Add inferred points command inserts points on hill tops and in depressions in a surface model that is created from contour point types. The added points result in a rounder shape for hill tops and depressions.

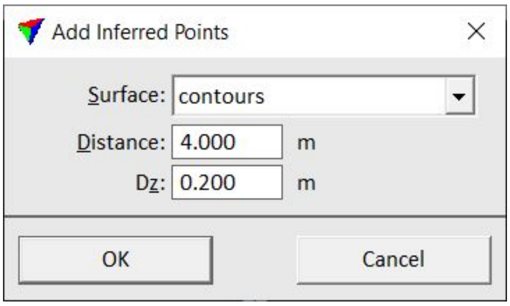
The points are added as points of type inferred points to the surface model. See Point Types for more information about surface model point types. The surface model can be created, for example, from contour line elements drawn in the CAD file using the Triangulate View or Triangulate Elements tools. The element type for inserting the line elements into the surface must be set to **Contour**.

To add inferred points to a surface model:

1. (Optional) Select a surface model from the list in the **Surfaces** window.

2. Select **Add inferred points** command from the **Edit** pulldown menu.

   This opens the **Add Inferred Points** dialog:



3. Select a **Surface** to which points are added. If a surface was selected in step 1, this surface is selected in the dialog.

4. Define additional settings and click OK.

   This adds the inferred points to the surface model. An information dialog is displayed that shows the number of added points.

SETTING	EFFECT
Surface	Name of the effected surface model.
Distance	Horizontal spacing between the highest/lowest closed contour line and inferred points as well as between the points.
Dz	Vertical distance between the highest/lowest closed contour line and inferred points as well as between the points.

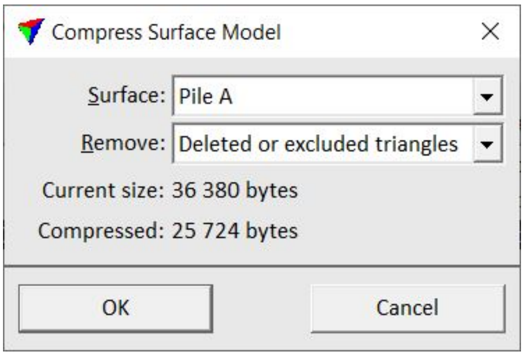
Compress

Compress command deletes previously removed points and triangles from a surface model. It can also delete excluded triangles from the model. Thus, the tool is used to minimize the amount of memory for storing the surface model.

To compress a surface model:

1. Select **Compress** command from the **Edit** pulldown menu.

   This opens the **Compress Surface Model** dialog:



2. Select a **Surface** to compress.

3. Define what triangles are removed: **Deleted triangles only** or **Deleted or excluded triangles**.

4. Click OK.

   This starts the compression process. When the process is finished, a information dialog appears that informs about the file size after the compression.

The process is applied to the surface model in the RAM. Therefore, it is still possible to restore the uncompressed model from the file on the hard disk using the File / Restore from file command in the **Surface** window. To save the compressed model, use one of the options from File / Save surface.

Delete surface

Delete surface command deletes a surface model completely. It removes the surface model from the RAM and deletes the surface model file from the hard disk.

To delete a surface model completely:

1. Select a surface model in the list of loaded surfaces.

2. Select **Delete surface** command from the **Edit** pulldown menu.

3. Confirm the process with **Yes** in the alert dialog.

   This deletes a surface from the memory and the related file on the hard disk.

To remove a surface from RAM but keep the file on the hard disk, use the Close surface commands of the **File** pulldown menu.

Exclude long triangles

Exclude long triangles command excludes narrow, long triangles from a surface model. There can be different exclusion settings for triangles at the outer boundary of a surface model or for any long triangles.

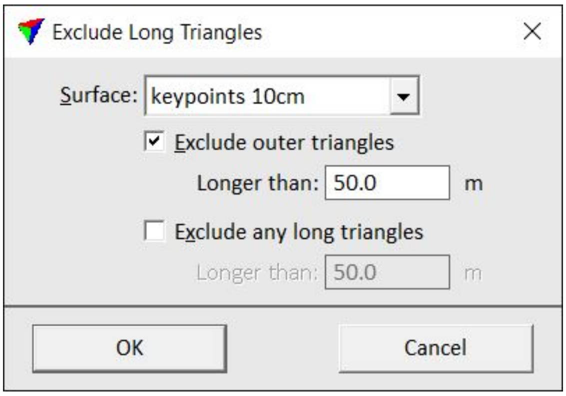
Excluded triangles are neither displayed nor included in surface-based computation processes. They can be included again, for example, by using the Exclude Triangle tool or the Exclude Area tool. To delete excluded triangles completely from a surface model, use Compress command from the **Surfaces** window.

To exclude long triangles:

1. (Optional) Select a surface model from the list in the **Surfaces** window.

2. Select **Exclude long triangles** command from the **Edit** pulldown menu.

   This opens the **Exclude Long Triangles** dialog:



3. Select a **Surface** to exclude triangles from. If a surface was selected in step 1, this surface is selected in the dialog.

4. Define settings for triangle exclusion.

5. Click OK.

   This excludes triangles from the selected surface.

SETTING	EFFECT
Exclude outer triangles	If on, only triangles along the outer boundary of the surface model with an edge length **Longer than** the given value are excluded.
Exclude any long triangles	If on, any triangles of the surface model with an edge length **Longer than** the given value are excluded.

You can use this command several times with different maximum lengths. If the results of the first exclusion is not satisfying, you can re-run the process with a new maximum length.

Modify elevations

Modify elevations command changes the elevation values of a surface model by using a mathematical equation. The process computes a new elevation value for each point of a surface model.

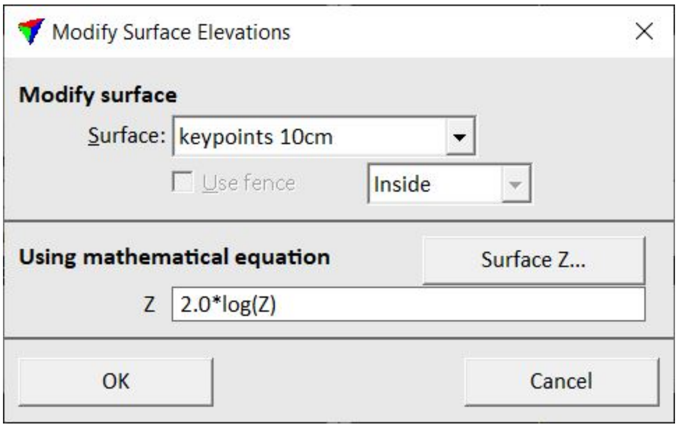
The old elevation of a point can be used in the equation as variable Z. The equation is actually a C language expression. You can use any mathematical functions that MDL recognizes. Valid expressions include functions, such as **sin(a)**, **cos(a)**, **tan(a)**, **exp(a)**, **log(a)**, **log10(a)**, **pow(a,b)**, **sqrt(a)**, **ceil(a)**, **fabs(a)**, and **floor(a)**, where **a** and **b** are floating point values.

To modify elevation values:

1. (Optional) Select a surface model from the list in the Surfaces window.

2. Select **Modify elevations** command from the **Edit** pulldown menu.

   This opens the **Modify Surface Elevations** dialog:



3. Select a Surface to modify elevations. If a surface was selected in step 1, this surface is selected in the dialog.

4. Define the mathematical equation.

5. Click OK.

   This starts the calculation of the new elevation values. Depending on the size of the surface the process may take some time. Use the Update Displays tool or another CAD tool for updating a view in order to visualize changes in visible surface displays.

SETTING	EFFECT
Surface	Name of the effected surface model.
Use fence	If on, only points **Inside** or **Outside** a fence are effected.
Z =	Mathematical expression that is applied to surface model points in order to calculated

SETTING	EFFECT
	new elevation values.
Surface Z	The button opens the **Append surface** dialog which lets you select a surface model. The elevation values of this surface are then used in the mathematical expression.

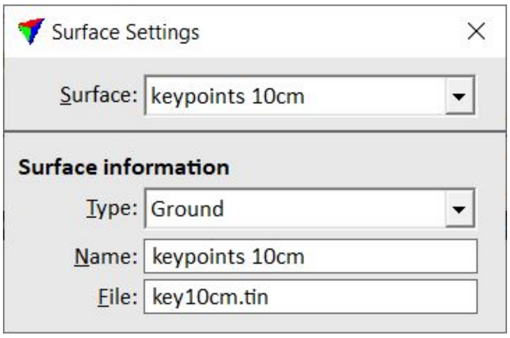
Modify settings

Modify settings command is used to modify the basic information of a surface model, such as type, name and file for storing the model.

To modify the basic information of a surface:

1. (Optional) Select a surface model from the list in the **Surfaces** window.
2. Select **Modify settings** command from the **Edit** pulldown menu.

   This opens the **Surface Settings** dialog:

3. Select a **Surface** to modify. If a surface was selected in step 1, this surface is selected in the dialog.
4. Modify the surface **Type**, enter a new **Name** for the surface, and/or assign a new **File** name for the surface.
5. Close the dialog.

   This saves the new settings to the surface model file. If the **File** name was modified, a new copy of the surface model file is created at the same location where the original file is stored.

Profile settings

Profile settings command opens the **Profile surfaces** dialog. The dialog contains settings for drawing surfaces into profiles. See Profile surfaces for a description of the settings in the dialog.

Thin

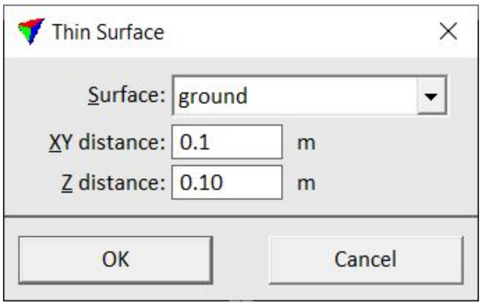
Thin command removes points from a surface model if they are too close to each other. The allowed minimum distances between points are defined in the tool’s dialog. A point is removed if it is within both, the minimum horizontal distance and the minimum vertical distance from the closest adjacent point.

To delete points close to each other:

1. (Optional) Select a surface model from the list in the **Surfaces** window.

2. Select **Thin** command from the **Edit** pulldown menu.

   This opens the **Thin Surface** dialog:



3. Select a **Surface** to thin. If a surface was selected in step 1, this surface is selected in the dialog.

4. Define minimum distances.

5. Click OK.

   This starts the thinning process. Depending on the size of the surface the process may take some time.

SETTING	EFFECT
Surface	Name of the effected surface model.
XY distance	Minimum horizontal distance between two points.
Z distance	Minimum vertical distance between two points.

Removed points and triangles remain as inactive data in a surface model. Use the Compress command in order to delete the inactive data completely from a surface model.

File pulldown menu

Menu commands from **File** pulldown menu in the **Surfaces** window are used to open, save, close, and restore surfaces.

TO	USE COMMAND
Create a new surface file	New surface
Open a surface file stored on a hard disk	Open surface
Close a surface file	Close surface
Close all surfaces	Close all
Save a surface file on a hard disk	Save surface
Save a surface to a new file on a hard disk	Save as
Save all surface files on a hard disk	Save all
Open a reference surface	Open reference
Import surface files	Import
Export to surface files	Export
Restore a surface after modifications	Restore from file
Open the TerraModeler **Settings** dialog	User settings
Unload TerraModeler	Exit

Close surface

Close surface command removes a surface model from the list of loaded surfaces. It does not delete the surface model file from the hard disk.

You can close surface models using one of the two options in the **File** menu of the **Surfaces** window:

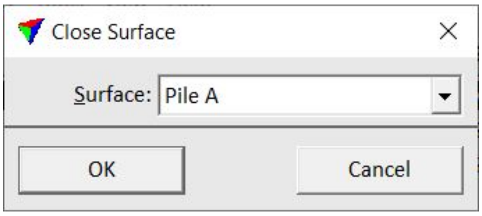
- **Close surface** - close a single surface model.
- **Close all** - close all surface models that are loaded in TerraModeler. If one of the surface models has been modified, you need to confirm whether you want to save the changes to the surface model file or not.

To close a single surface model:

1. Select a surface model in the list of loaded surfaces.

2. Select **Close surface** command from the **File** pulldown menu.

   The **Close Surface** dialog opens:



3. Select the **Surface** to be closed.

4. Click OK.

   This removes the selected surface model from the list of loaded surfaces. If the surface has been modified, you need to confirm whether you want to save the changes to the surface model file or not.

To remove the surface model from RAM and delete the file on the hard disk, use the Delete surface command from the **Edit** pulldown menu.

Exit

Exit command closes TerraModeler, it unloads the application.

If there are modified and unsaved surface models, the software asks you if you want to save changes to the surface model files. Confirm with **Yes** if you want to save the changes or with **No** if you want to close the application without saving changes to surface model files.

Export

Export commands can be used for exporting surface data into files of different formats. This is described in detail in Chapter Importing and Exporting Data.

Import

Import commands can be used for importing surface data from files of different formats. This is described in detail in Chapter Importing and Exporting Data.

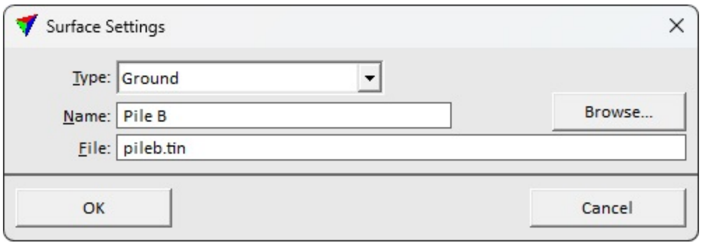
New surface

New surface command creates a new empty surface model. The command adds the new surface to the list. Optionally, the model can be saved to the hard disk automatically after it is created. This is defined in Saving surfaces category of the TerraModeler **Settings**.

To create a new surface:

1. Select **New surface** command from the File menu.

   This opens the **Surface Settings** dialog:



2. Select a surface **Type**.

3. Enter a **Name** for the surface.

4. (Optional) Rename the **File** for storing the model on the hard disk.

5. Click OK.

   A new empty surface is added to the list in the Surfaces window. You can continue with adding elements to the surface by using tools from the TerraModeler Create Surfaces toolbox.

Open reference

Open reference command opens a surface model file for read-only access. A reference surface can be viewed and displayed using display options for surfaces, and it can be included in calculation processes. A reference surface can not by modified.

To open a reference surface:

1. Select **Open reference** command from the **File** pulldown menu.

   The **Open reference surface** dialog box opens, a standard dialog for opening files.

2. Locate and select the file to be opened.

3. Click OK.

   This reads in the surface model file and adds it to the list of loaded surfaces. You can use the reference surface in all tools which do not modify a surface.

When a reference surface is opened, the application reads a copy of the model in the RAM memory. It uses this copy during the session and does not detect if somebody else makes changes to the original surface model file.

Open surface

TerraModeler surface models are stored in binary files. Open surface command opens a previously stored surface model file from a hard disk.

To open a surface file:

1. Select **Open** command from the **File** pulldown menu.

2. The **Open surface** dialog opens, a standard dialog for opening files.

   Locate and select the surface file to be opened.

3. Click OK.

   This reads in the surface model file and adds it to the list of loaded surfaces.

Restore from file

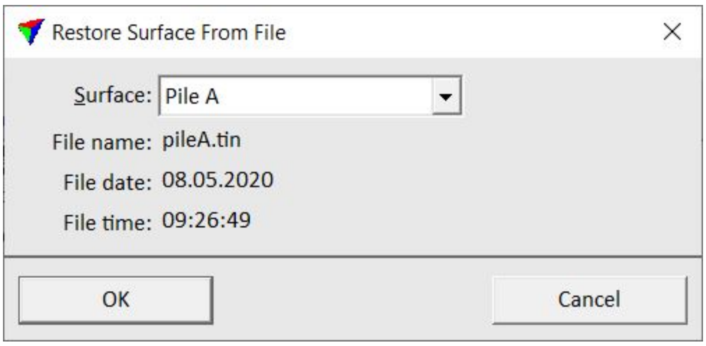
Restore from file command reloads the surface model file from the hard disk. This can be used after modifying a surface in order to restore the previously saved situation, for example if it turns out that the modifications were a mistake.

The restore process can only be done for surfaces that have a status "Modified" which means that the surface model in the RAM memory has been changed but not yet saved to the hard disk. Once the status is "Saved", all modifications are written to the file on the hard disk and the restore process would not change anything to the surface model.

To restore a surface model from a file:

1. Select **Restore from file** command from the **File** pulldown menu.

   This opens the **Restore Surface From File** dialog:



2. Select a **Surface** to restore.

3. Click OK.

   The surface file is reloaded from the file displayed as **File name** in the **Restore Surface From File** dialog.

The list of surfaces in the **Restore Surface From File** dialog contains only surfaces that can be reloaded from file. If there is no such surface available, a dialog is displayed with the information that none of the surface models can be restored.

Save surface

TerraModeler keeps surface models in the RAM memory of the computer. Modifications to the surface model effect only the copy in memory until the model is saved to the hard disk.

TerraModeler surface models are saved as binary files. When a new surface model is created, the application assigns a default file name by combining the CAD file name with an extension such as T00, T01, T02, etc. You can change the file name and store it on any hard disk or other storage device. See Chapter TIN File Format Specification for more information about the surface model file format.

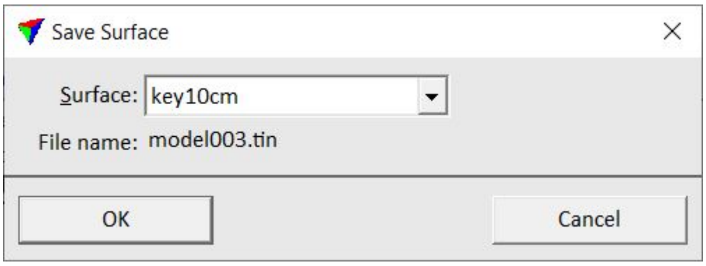
You can save surface models to files using one of the three options in the **File** menu of the **Surfaces** window:

- **Save surface** - save a surface model to the file that is assigned to this model.
- **Save as** - save a new copy of a surface model file.
- **Save all** - save all modified surface models to the files that are assigned to the models.

To save a single surface model:

1. Select the **Save surface** command from the **File** pulldown menu.

The **Save Surface** dialog opens:



2. Select a **Surface** to save.

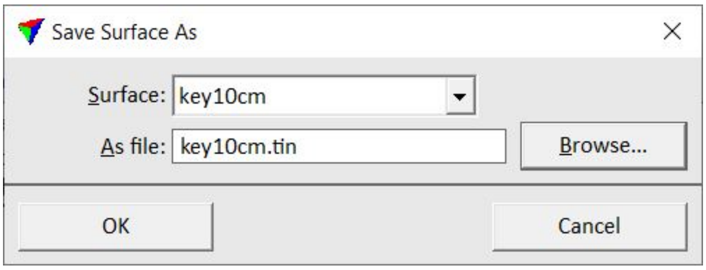
3. Click OK.

The surface is saved to the file displayed as **File name** in the **Save Surface** dialog.

To save a new copy of a surface model file:

1. Select the **Save As** command from the **File** pulldown menu.

The **Save Surface As** dialog opens:



2. Select a **Surface** to save.

3. Define a location and file name in the **As file** field. The **Browse** button lets you select a location on the PC for storing the file.

4. Click OK.

The surface is saved to the given file.

TerraModeler surface models can be saved automatically when the CAD file is saved or after a model is created. This is defined in Saving surfaces category of the TerraModeler **Settings**.

User settings

User settings command opens the TerraModeler **Settings** window. It performs the same action as the Settings tool in TerraModeler General toolbox.

Help pulldown menu

Menu commands from **Help** pulldown menu in **Surfaces** window are used to open the online User Guide and to display license information for TerraModeler.

TO	USE COMMAND
Open the TerraModeler User Guide	Contents
Display the **License information** dialog	About TerraModeler

Tools pulldown menu

Menu commands from **Tools** pulldown menu in **Surfaces** window can be used to open toolbars of TerraModeler. The toolbars are the same as in the **TerraModeler** toolbox.

TO	USE COMMAND
Open the **TerraModeler** toolbox	Main
Open the **General** toolbox	General
Open the **Create Surfaces** toolbox	Create Surfaces
Open the **Draw using Surface** toolbox	Draw using Surface
Open the **Drawing Utilities** toolbox	Drawing Utilities
Open the **Edit Point** toolbox	Edit Point
Open the **Edit Area** toolbox	Edit Area
Open the **Display Surface** toolbox	Display Surface
Open the **Display Regions** toolbox	Display Regions *Not Lite*
Open the **Profiles** toolbox	Profiles
Open the **Quantity** toolbox	Quantity
Open the **Domains** toolbox	Domains *Not Lite, Not UAV*
Open the **Display Single** toolbox	Display Single *Not Lite*
Open the **Regions** toolbox	Regions *Not Lite, Not UAV*
Open the **2D Contours** toolbox	2D Contours

Utility pulldown menu

Menu commands from **Utility** pulldown menu in the **Surfaces** window are used to create derived surface models from existing surfaces, to validate the elevation accuracy of a surface model, and to produce output files in batch mode.

TO	USE COMMAND
View surface statistics	View statistics
Create a copy of a surface	Copy surface
Create a combined surface from two surfaces	Merge surfaces
Subtract elevations of two surfaces	Subtract surfaces
Create a statistical grid surface model	Statistical model
Check surface model elevations against ground control points	Output control report
Create contour lines in batch mode	Produce contours *Not Lite, Not UAV*
Create lattice files in batch mode	Produce lattice models *Not Lite, Not UAV*
Create triangles in batch mode	Produce triangles *Not Lite, Not UAV*
Work with lattice databases	Lattice database *Not Lite, Not UAV*

Copy surface

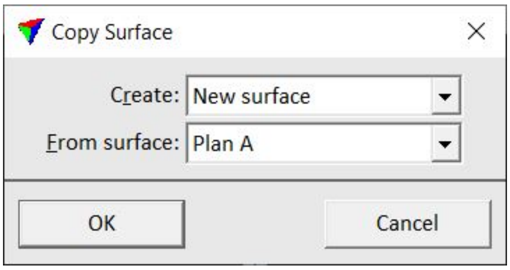
Copy surface command creates an identical copy of a surface model. The copy is created for the surface model in the RAM and for the surface model file stored on a hard disk. The new surface model file is saved at the same location where the original surface model is stored.

To create a copy of a surface model:

1. (Optional) Select a surface model from the list in the **Surfaces** window.

2. Select **Copy surface** command from the **Utility** pulldown menu.

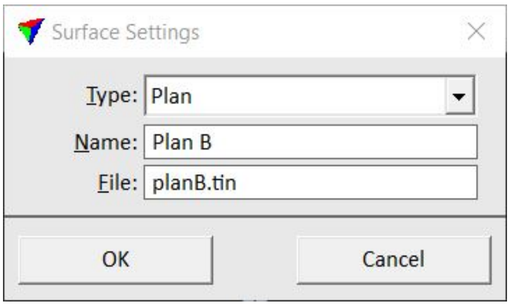
   This opens the **Copy Surface** dialog:



3. Select a **Surface** to copy from. If a surface was selected in step 1, this surface is selected in the dialog.

4. Click **Copy**.

   This opens the **Surface Settings** dialog:



5. Select a **Type** for the new surface.

6. Define a **Name** for the surface.

7. (Optional) Define a new **File** name for the surface model file stored on the hard disk.

8. Click OK.

   This creates a copy of the surface model.

Lattice database

Not Lite, Not UAV

Lattice database commands can be used for working with lattice databases, a collection of grid files stored in the same directory. This is described in detail in Chapter Lattice Database.

Merge surfaces

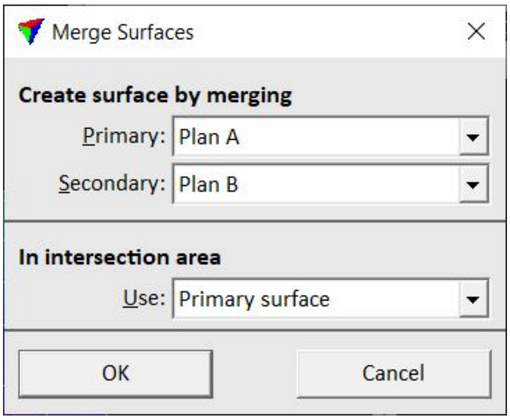
Merge surfaces command combines two surfaces. It creates a new surface model in the RAM and as surface model file on a hard disk. The new surface model file is saved at the same location where the original surface models are stored.

The command can be used, for example, to create a surface representing a new ground level by combining a design surface and an existing ground surface. The design surface, such as an excavation or a road, is set to be the primary surface. The merging operation copies all the information from the primary surface to the new surface. The secondary surface, existing ground, enlarges the area of the new surface. The points from the secondary surface are only used if they are located outside the primary surface.

To merge two surface models:

1. Select **Merge surfaces** command from the **Utility** pulldown menu.

This opens the **Merge Surfaces** dialog:



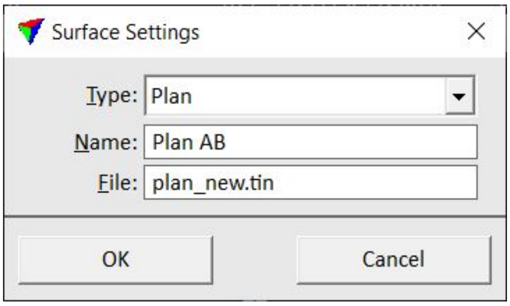
2. Select **Primary** and **Secondary** surfaces to merge.

3. Select a surface that is used in areas where the two surfaces overlap.

4. Click OK.

SETTING	EFFECT
Primary	All the information from the primary surface is copied to the new surface.
Secondary	Information from the secondary surface is used only outside the primary surface.
Use	Defines which surface to use in places where the surfaces overlap: • **Primary surface** - use the primary surface. • **Highest surface** - use the higher of the two surfaces. • **Lowest surface** - use the lower of the two surfaces.

This opens the **Surface Settings** dialog:



5. Select a **Type** for the combined surface.

6. Define a **Name** for the surface.

7. (Optional) Define a new **File** name for the surface model file stored on the hard disk.

8. Click OK.

   This creates a combined surface model.

Output control report

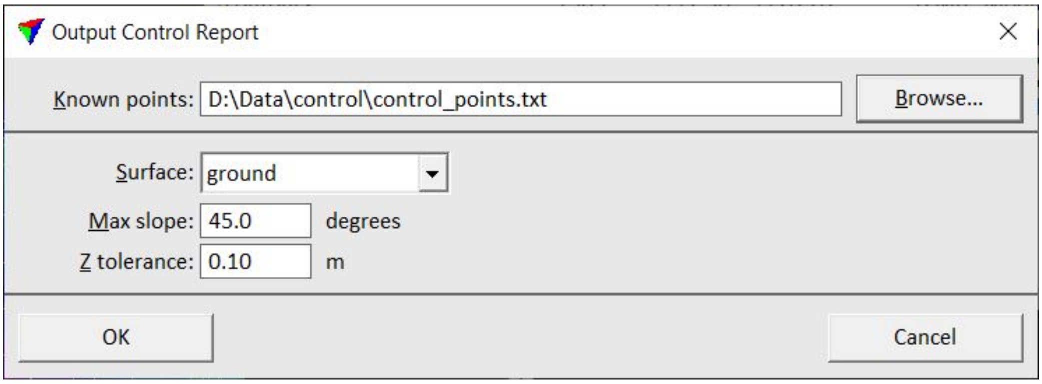
Output control report command creates a report of elevation differences between a surface model and control points. This can be used to check the elevation accuracy of the surface model and to calculate a correction value for improving the elevation accuracy of the model.

The control points have to be stored in a space delimited text file in which each row has four fields: identifier, easting, northing, and elevation. The identifier field is normally a number but it may include non-numeric characters as well.

To create a control report:

1. Select **Output control report** command from the **Utility** pulldown menu.

   This opens the **Output Control Report** dialog:



2. Select the text file that stores the control points.

3. Define settings and click OK.

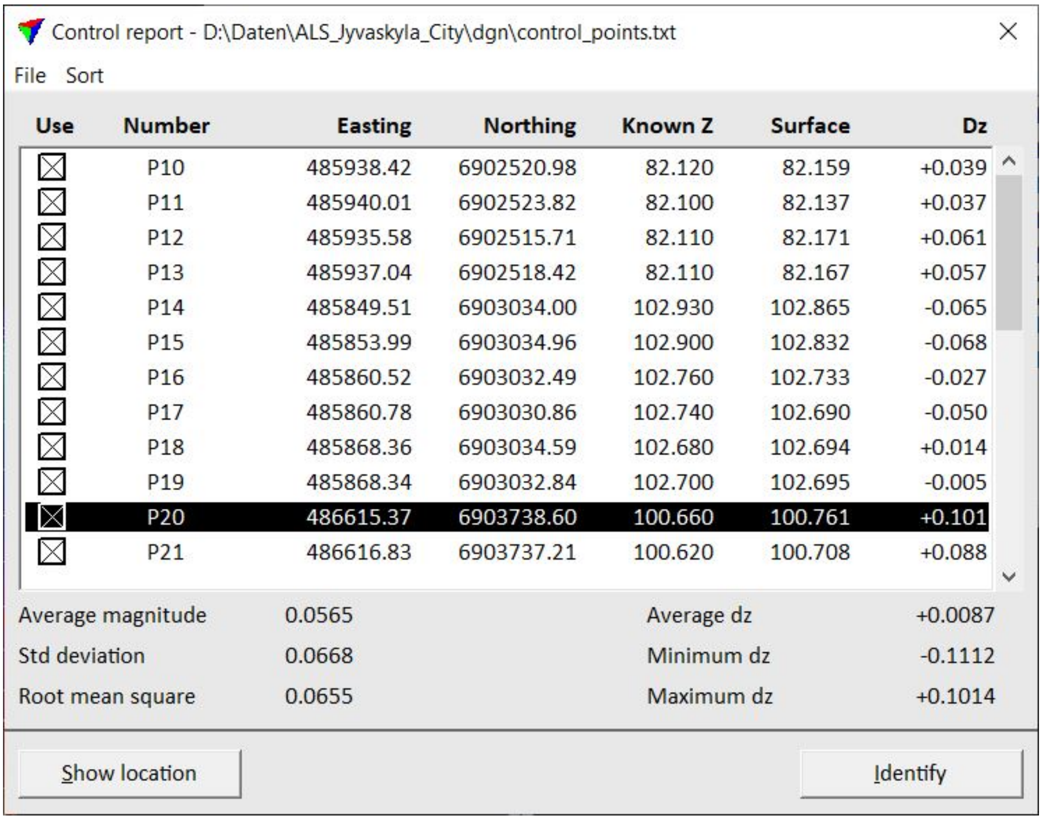
   This calculates the elevation differences and opens the Control report window.

SETTING	EFFECT
Known points	Name and location of the file that stores the coordinates of the control points.
Surface	Surface model used for comparison with the control points.
Max slope	Maximum terrain slope for which an elevation difference is computed.
Z tolerance	Elevation variation in the surface model points. This value is used only when

SETTING	EFFECT
	computing the terrain slope so that small triangles do not exceed the **Max slope** value.

Control report

The **Control report** window shows the coordinates of the control points, the surface elevation values at control point locations and the elevation difference at control point locations. Below the list of control points, the statistical values for the control report are displayed.



The report can be saved or printed directly by using the **Save as text** or **Print** command from the **File** pulldown menu. Outliers in **Dz** can be highlighted using the **Settings** command from the **File** menu. The command opens the **Control report settings** dialog where a limit for highlighting **Dz** values related to the standard deviation can be defined.

The report can be sorted by **deviation**, **Dz**, **magnitude**, **number** (identifier), **easting** or **northing** coordinates using the corresponding commands from the **Sort** pulldown menu.

The **Use** field in the report determines whether a control point is used in the report or not. You can exclude a control point from the report by clicking in the square of the respective control point.

The **Show location** button can be used to identify the location of a control point in a CAD file view. Click the button and move the mouse pointer into a view in order to show the control point location. A data click centers the view at the control point location.

The **Identify** button can be used to select a specific control point in the report. Click the button and move the mouse pointer inside a CAD file view. The control point closest to the mouse pointer is dynamically highlighted. A data click selects the control point in the report.

The **Average dz** value displayed on the lower right side of the report window can be used to apply a linear transformation to the surface model. The value indicates whether a surface is systematically too high or too low compared with the control points. Thus, transforming the elevation of the surface model using this value with inverse sign results in a better match between surface model and control points. You can use, for example, the Modify elevations command in the **Surfaces** window to apply the elevation correction to the surface model.

Produce contours

Not Lite, Not UAV

Produce contours command creates contours in an automated process. This is useful for contour production on project level, when contours have to be created for a large area.

The automatic contour production does not rely on surface models loaded in TerraModeler. The ground model can be defined by:

- **Laser points in a TerraScan project** - usually contour keypoints (for rather smooth, cartographic contours) or model keypoints (for accurate, mathematical contours) in separate files referenced by a TerraScan project.
- **Feature coded breakline elements** - in reference CAD files, vector elements created with TerraSurvey or any other application enabled in Triangulate Survey category in the TerraModeler **Settings**. *References are applicable in Bentley CAD only.*
- **Breakline elements** - in reference CAD files, vector elements properly organized by level and symbology or breaklines filtered by rules using the Triangulate Elements tool. *References are applicable in Bentley CAD only.*

The settings for contour lines, labels, ticks, and peaks and pits labels are stored in text files which are then used by the batch process.

The automatic process writes the contours into multiple CAD files. A seed file is used as starting point for creating the contour CAD files.

Workflow for automatic contour production:

1. Prepare the laser data in TerraScan. Normally, this includes the classification of model/contour keypoints into separate files. Create a TerraScan project that references the keypoint files.

2. If required, create breakline vector elements using CAD and TerraScan tools.

3. Define and save a contour settings file using the Display Contours tool.

Use Write to file as Mode. Contours should be drawn preferable as Soft line strings. The contour settings file stores settings from Display Contours tool, Contour options, Contour label options, and Contour tick options.

4. (Optional) Define and save a settings file for peaks and pits using the Label Peaks and Pits tool.

5. Create a seed file for storing the contour lines.

   The seed file is usually an empty 3D CAD file possibly with a reference file attached. The reference file contains the breaklines that are included in the contour line computation. Alternatively, the breaklines can be stored in the seed file directly.

   Reference files are only available in Bentley CAD.

6. If breakline vectors are used, create filtering rules for breaklines using the Triangulate Elements tool.

7. Create shapes that divide the project area into smaller parts.

   Each shape results in a separate CAD file for contours. The shape boundary is used by the software as fence in order to clip the contour lines to the shape dimensions.

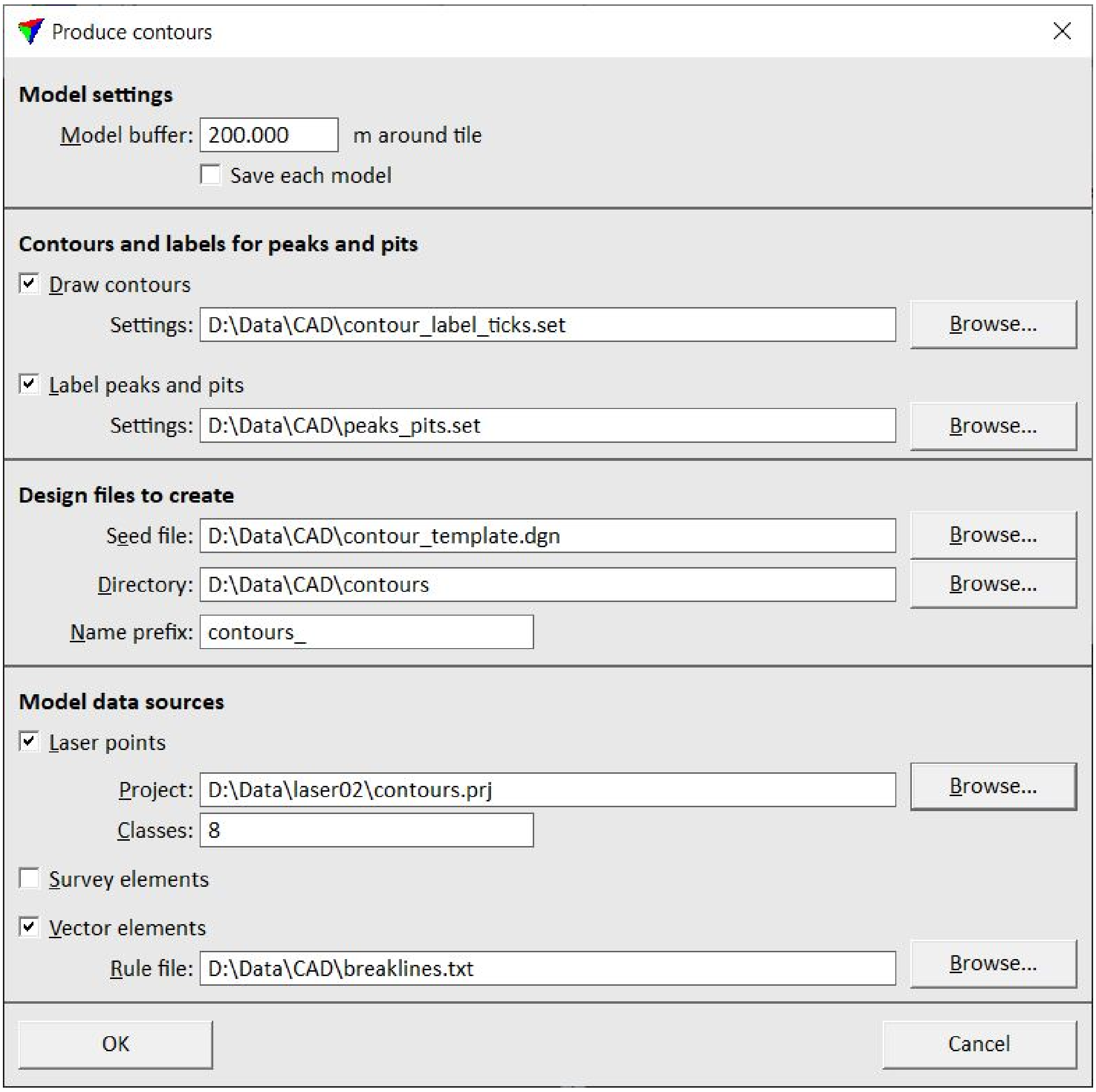
8. (Optional) Place text elements inside the shapes.

   The texts are used for naming the contour CAD files. If no texts are selected, the files are named by automatic numbering.

9. Select the shapes and (optional) the texts.

10. Select **Produce contours** command from the **Utility** pulldown menu in TerraModeler **Surfaces** window.

    This opens the **Produce contours** dialog:



11. Define settings and click OK.

    This opens the **Triangulate surface** dialog. Follow the common steps for Creating a surface model.

    The software starts the contour line production. First, it closes the original CAD file from which the process was started. Next, it opens the seed file, saves a copy of it, computes the contours for the first shape, writes them into the CAD file, and closes the CAD file. Then the process continues with the next shape. After all contour CAD files are created, the original CAD file is opened again.

SETTING	EFFECT
Model buffer	Area around each shape for which the surface model is created in addition to the area of the active shape itself. The buffer should be big enough to ensure smooth contour line transitions on shape boundaries.

SETTING	EFFECT
Save each model	If on, a surface model file is saved for each shape.
Draw contours	If on, contours are drawn in the CAD files. The given **Settings** are used for contour computation and display.
Label peaks and pits	If on, labels for peaks and pits are drawn in the CAD file. The given **Settings** are used for label display.
Seed file	Location of the CAD file that serves as seed file for contour line production.
Directory	Location on a hard disk where the contour CAD files are stored.
Name prefix	Text string that is added at the beginning of the file name of contour CAD files. If text elements are selected for each shape, the contour CAD file names are a combination of the prefix and the selected text element. If no text elements are selected, the contour CAD files are named with the prefix and an automatically increasing number.
Laser points	If on, laser points from the given **Project** and **Classes** are used for surface model creation.
Survey elements	If on, feature coded elements created by TerraSurvey or any other application enabled in Triangulate Survey category in the TerraModeler **Settings** are used for surface model creation.
Vector elements	If on, vector elements filtered as breaklines according to the given **Rule** file are used for surface model creation.

Produce lattice models

Not Lite, Not UAV

Produce lattice models command creates lattice model files in an automated process. This is useful for lattice model production on project level, when lattice model files have to be created for a large area.

Lattice models are representations of a surface using a grid structure of points. The lattice model files include regularly distributed points at constant intervals in both, X and Y directions.

Common storage formats are text files, image files, or program-specific grid file formats. The automatic process writes the lattice model points into multiple output files of a defined format.

The automatic lattice model production does not rely on surface models loaded in TerraModeler. It is more closely connected with point cloud processing functionality in TerraScan. The source data for the lattice model can be defined by:

- **Laser points in a TerraScan project** - usually ground points referenced by a TerraScan project.
- **Feature coded breakline elements** - in reference CAD files, vector elements created with TerraSurvey or any other application enabled in Triangulate Survey category in the TerraModeler **Settings**. *References are applicable in Bentley CAD only.*
- **Breakline elements** - in reference CAD files, vector elements properly organized by level and symbology or breaklines filtered by rules using the Triangulate Elements tool. *References are applicable in Bentley CAD only.*

To create a lattice model file from a surface model, see Export / Lattice file command.

Workflow for automatic lattice model production:

1. Prepare the laser data in TerraScan. Normally, this includes the classification of ground points. Create a TerraScan project that references the laser point files.

2. If required, create breakline vector elements using CAD and TerraScan tools.

3. If breakline vectors are used, create filtering rules for breaklines using the Triangulate Elements tool.

4. Create shapes that divide the project area into smaller parts.

   Each shape results in a separate lattice model file. The shape boundary is used by the software as fence in order to clip the lattice model to the shape dimensions.

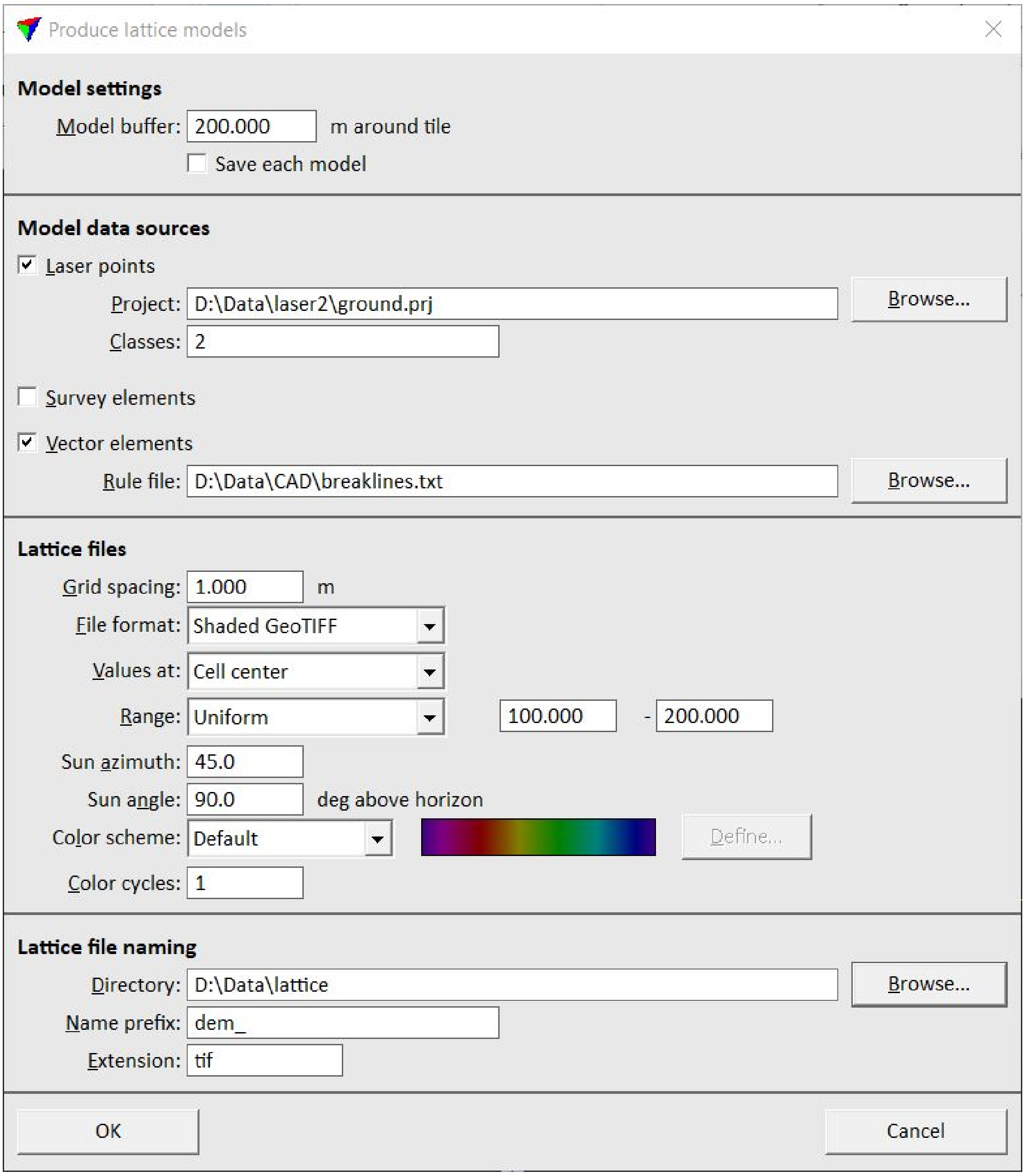
5. (Optional) Place text elements inside the shapes.

   The texts are used for naming the lattice model files. If no texts are selected, the files are named by automatic numbering.

6. Select the shapes and (optional) the texts.

7. Select **Produce lattice models** command from the **Utility** pulldown menu in TerraModeler **Surfaces** window.

   This opens the **Produce lattice models** dialog:



8. Define settings and click OK.

The software starts the lattice model production. It computes the lattice model for the first shape and writes it into the output file. Then the process continues with the next shape until all lattice models are created.

SETTING	EFFECT
Model buffer	Area around each shape for which the surface model is created in addition to the area of the active shape itself. The buffer should be big enough to ensure smooth transitions between lattice models on shape boundaries.

SETTING	EFFECT
Save each model	If on, a surface model file is saved for each shape.
Laser points	If on, laser points from the given **Project** and **Classes** are used as data source.
Survey elements	If on, feature coded elements created by TerraSurvey or any other application enabled in Triangulate Survey category in the TerraModeler **Settings** are used as data source.
Vector elements	If on, vector elements filtered as breaklines according to the given Rule file are used as data source.
Grid spacing	Constant distance between points in the grid structure of the lattice model files.
File format	Format of the output files. Available formats are: • **ArcInfo Grid**, **Intergraph GRD** • **GeoTIFF** formats, **Shaded GeoTIFF** • **Japanese DMF** and **LEM** • **Raw image** formats • **Surfer ASCII** and **binary** • **XYZ text** Many of the additional settings for the lattice files depend on the selected format for the output files.
Values at	Defines the exact location of the lattice point coordinates: • **Cell center** - the coordinates of the grid cell center are stored. • **Cell corner** - the coordinates of the grid cell corner are stored. For some output file formats only one option is available.
Outside Z	Defines a constant value for grid cells outside the lattice model area where no elevation value can be derived from the source data. This can be, for example, a software-specific value.
Z unit	Defines the elevation value unit for several raster formats.
Create TFW files	If on, external georeferencing files are created for **GeoTIFFs** .

SETTING	EFFECT
Write coordinate block	If on, a coordinate block file is written for **Intergraph GRD** files.
Byte order	Defines the byte order for **Raw integer** image formats related to the processor type: **Intel** or **Motorola**.
Range	Determines how the color scheme is fitted to the elevation values for **Shaded GeoTIFF** files: • **Uniform** - the range should include all elevation values of the area for which GeoTIFFs are produced in order to ensure seamless coloring over all images. • **Fit to tile** - the colors are fitted to elevation values of each tile individually.
Sun azimuth	Direction of the light for **Shaded GeoTIFFs**. The direction from the north is defined with zero and angle values increase clockwise.
Sun angle	Height of the light source above the horizon.
Color scheme	Color scheme to use for **Shaded GeoTIFFs**: • **Default** - default color scheme containing magenta, red, yellow, green, cyan, blue, and purple. • **Selected colors** - use **Define** button to create a customized color scheme. Define a customized color scheme for more information.
Color cycles	Number of color cycles used for **Shaded GeoTIFFs**. Use zero to create a gray scale display showing triangle slope only.
Outside points	Defines for **Xyz text** files how to handle grid points outside the lattice model area where no elevation value can be derived from the source data: • **Skip** - outside grid points are not written into the text file. • **Output** - outside grid points are written into the text file using the constant value given in the **Outside Z** field.
Directory	Location on a hard disk where the lattice model files are stored.
Name prefix	Text string that is added at the beginning of the file name of lattice model files. If text elements are selected for each shape, the lattice model file names are a

SETTING	EFFECT
	combination of the prefix and the selected text element. If no text elements are selected, the lattice model files are named with the prefix and an automatically increasing number.
Extension	Extension for the lattice model files. It is recommended to use the standard extensions for specific file formats, for example, .TIF for GeoTIFFs, .GRD for ArcInfo Grids, .TXT, .XYZ or .ASC for text files.

Produce lattice models command can be used to create GeoTIFFs of shaded surface model representations in a batch process. Select **Shaded GeoTIFF** as **File format** and respective settings for elevation range, sun azimuth, sun angle, and color scheme. The output result is similar to the display created by the Display Shaded Surface tool.

Produce triangles

Not Lite, Not UAV

Produce triangles command creates triangles in an automated process. This is useful for triangle CAD file production on project level, when triangles have to be drawn for a large area.

The automatic triangle production does not rely on surface models loaded in TerraModeler. The ground model can be defined by:

- **Laser points in a TerraScan project** - usually ground point or model keypoints referenced by a TerraScan project.
- **Feature coded breakline elements** - in reference CAD files, vector elements created with TerraSurvey or any other application enabled in Triangulate Survey category in the TerraModeler **Settings**. *References are applicable in Bentley CAD only.*
- **Breakline elements** - in reference CAD files, vector elements properly organized by level and symbology or breaklines filtered by rules using the Triangulate Elements tool. *References are applicable in Bentley CAD only.*

The automatic process writes the triangles as shape elements into multiple CAD files. A seed file is used as starting point for creating the triangle CAD files.

To produce triangle CAD files automatically in a batch process:

1. Prepare the laser data in TerraScan. Normally, this includes the classification of ground or model keypoints. Create a TerraScan project that references the laser point files.

2. If required, create breakline vector elements using CAD and TerraScan tools.

3. Create a seed file for storing the triangles.

The seed file is usually an empty 3D CAD file possibly with a reference file attached. The reference file contains the breaklines that are included in the triangle computation. Alternatively, the breaklines can be stored in the seed file directly.

Reference files are available in Bentley CAD only.

4. If breakline vectors are used, create filtering rules for breaklines using the Triangulate Elements tool.

5. Create shapes that divide the project area into smaller parts.

   Each shape results in a separate CAD file for triangles. The shape boundary is used by the software as fence in order to decide whether triangles are inside or outside the shape and to clip the triangles to the shape dimensions.

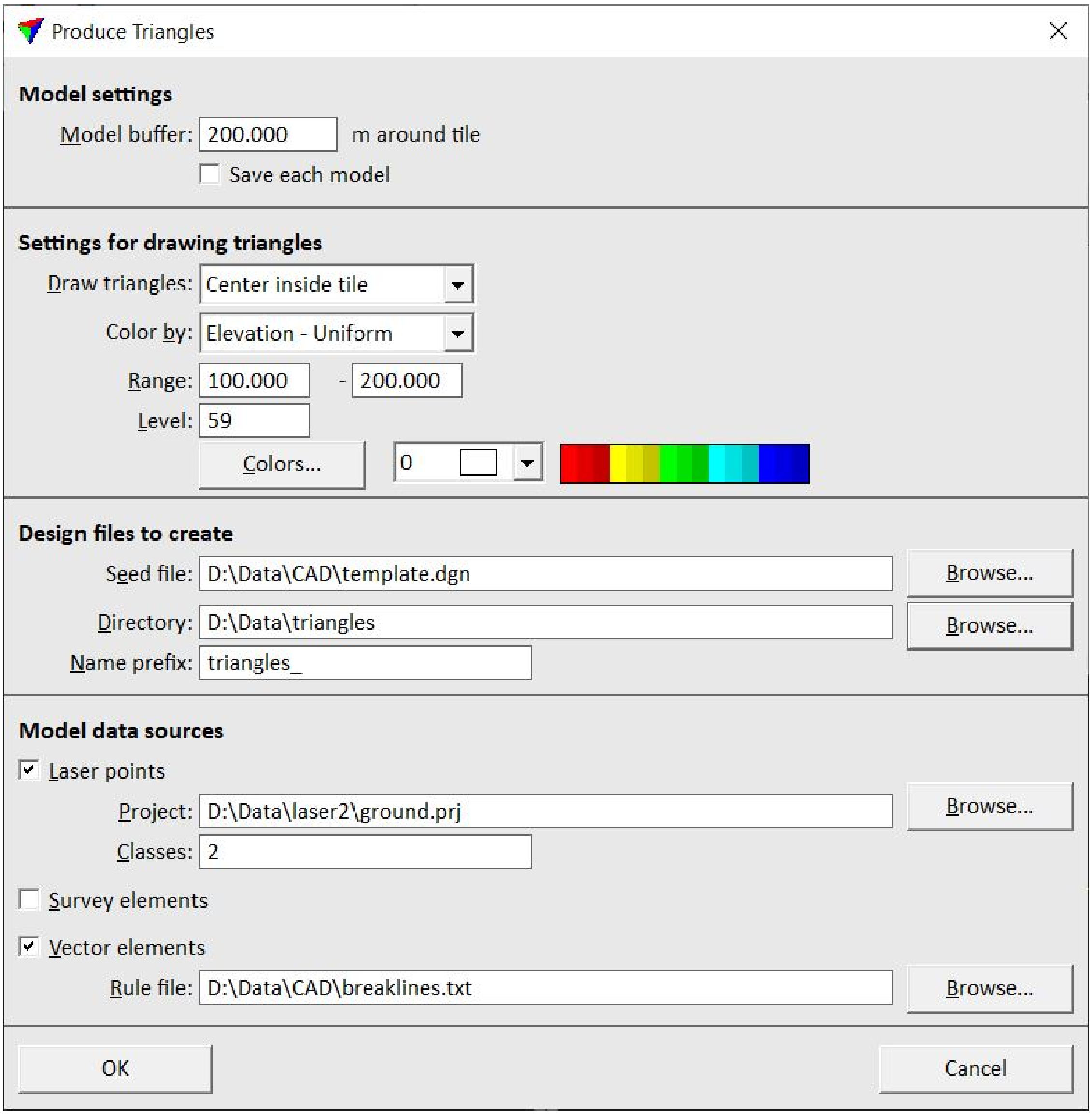
6. (Optional) Place text elements inside the shapes.

   The texts are used for naming the triangle CAD files. If no texts are selected, the files are named by automatic numbering.

7. Select the shapes and (optional) the texts.

8. Select **Produce triangles** command from the **Utility** pulldown menu in TerraModeler **Surfaces** window.

   This opens the **Produce Triangles** dialog:



9. Define settings and click OK.

   This opens the **Triangulate surface** dialog. Follow the common steps for Creating a surface model.

   The software starts the triangle production. First, it closes the original CAD file from which the process was started. Next, it opens the seed file, computes the triangles for the first shape, writes them into the CAD file, and closes the CAD file. Then the process continues with the next shape. After all triangle CAD file are created, the original CAD file is opened again.

SETTING	EFFECT
Model buffer	Area around each shape for which the surface model is created in addition to the area of the active shape itself. The buffer should be big enough to ensure smooth contour line transitions on shape boundaries.

SETTING	EFFECT
Save each model	If on, a surface model file is saved for each shape.
Draw triangles	Defines how triangles are drawn on tile boundaries: • **All** - all triangles are drawn. • **Center inside tile** - triangles are drawn completely if the center point is inside the tile. • **Clipped to tile boundary** - triangles are clipped to the tile boundaries. This results in rectangles along tile boundaries.
Color by	Defines the coloring mode: • **Elevation - Uniform** - triangles are colored by elevation value. The colors are fitted to the given **Range** of elevation values. • **Elevation - Fit to tile** - triangles are colored by elevation value. The colors are fitted to elevation values of each tile individually. • **Slope** - triangles are colored by slope gradient.
Range	Elevation range for all triangle CAD files. The range should include all elevation values of the area for which triangles are produced in order to ensure seamless coloring over all triangle CAD files. This is only active if **Color by** is set to **Elevation - Uniform**.
Level	Level on which the triangles are drawn into the CAD file.
Colors	Opens the **Color scheme** dialog for the definition of a color scheme for triangle coloring. Creating a color scheme of discrete colors for more information.
Color list	Opens the Bentley CAD color table for single color selection.
Seed file	Location of the CAD file that serves as seed file for triangle production.
Directory	Location on a hard disk where the triangle CAD files are stored.
Name prefix	Text string that is added at the beginning of the file name of triangle CAD files. If text elements are selected for each shape, the triangle CAD file names are a combination of the prefix and the selected text element. If

SETTING	EFFECT
	no text elements are selected, the triangle CAD files are named with the prefix and an automatically increasing number.
Laser points	If on, laser points from the given **Project** and **Classes** are used as data source.
Survey elements	If on, feature coded elements created by TerraSurvey or any other application enabled in Triangulate Survey category in the TerraModeler **Settings** are used as data source.
Vector elements	If on, vector elements filtered as breaklines according to the given **Rule** file are used as data source.

Statistical model

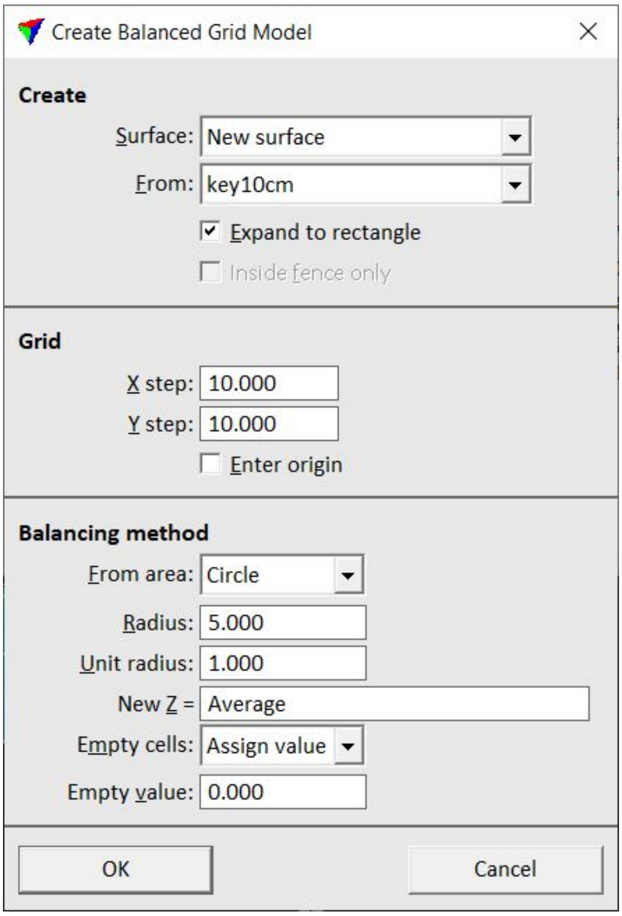
Statistical model command calculates statistical values based on an existing surface model. It creates a new grid model in the RAM and as surface model file on a hard disk. The new grid model file is saved at the same location where the original surface model is stored.

The new surface is a grid type surface. It contains evenly-spaced points at constant intervals in both, X and Y directions. The statistical values are mathematically derived from the existing surface within a circular or squared calculation area. When calculating a value for a grid point, the application searches for points in the existing model within a given radius from the grid point location. The values in the grid model can represent the number of points; the average, sum, or gravitational sum of elevation values; the minimum or maximum elevation values.

To create a statistical grid model:

1. Select **Statistical model** command from the **Utility** pulldown menu.

   This opens the **Create Balanced Grid Model** dialog:

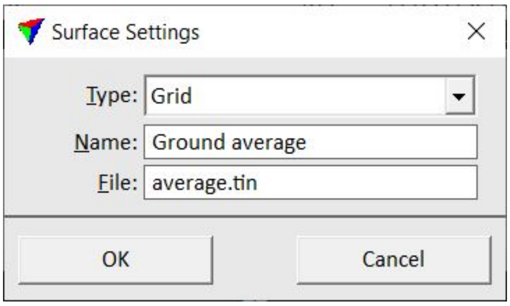


2. Define settings and click OK.

SETTING	EFFECT
Surface	Surface to create.
From	Existing surface from which to calculate values.
Expand to rectangle	If on, the grid surface is expanded to a rectangular area that encloses the original surface completely.
Fence contents	If on, use only points from the original surface that are inside a fence.
X step	Interval in X direction between points in the grid model.
Y step	Interval in Y direction between points in the grid model.
Enter origin	If on, you can enter the origin of the new grid model with a data click. If the data click is located inside the model area, a grid point of the model is placed at the location of the mouse click.

SETTING	EFFECT
From area	Shape of the calculation area around a grid point: • **Circle** - circle with a given **Radius**. • **Square** - square with **Radius** value equal to half the side length.
Radius	Determines the size of the calculation area around a grid point.
Unit radius	Radius at which points get a weight factor of 1 when computing the gravitational sum of points.
New Z	Mathematical expression to calculate for each grid point inside the calculation area. This may include variables: • **Count** - the number of points. • **Average** - the average of elevation values. • **Sum** - the sum of elevation values. • **Gravsum** - the gravitational sum of elevation values. • **Minimum** - smallest elevation value. • **Maximum** - highest elevation value.
Empty cells	Determines how cells without a calculated value are handled in the grid model: • **Assign value** - a constant value is assigned. • **Skip** - empty cells are skipped in the model.
Empty value	Constant value that is assigned to cells without a calculated value in the grid model. This is only active if **Empty cells** is set to **Assign value**.

This opens the **Surface Settings** dialog:



3. Select a **Type** for the new surface.

4. Define a **Name** for the grid model.

5. (Optional) Define a new **File** name for the surface model file stored on the hard disk.

6. Click OK.

   This creates a grid model that represents the derived statistical values of the original surface model.

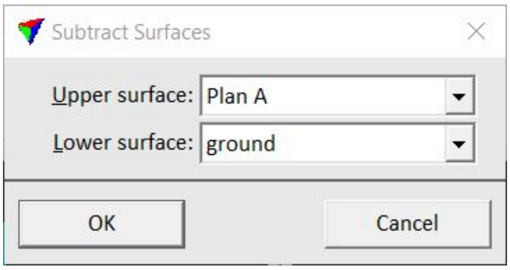
Subtract surfaces

Subtract surfaces command calculates the elevation difference of all surface points between two existing surface models. It creates a new surface model in the RAM but does not save the surface model file on a hard disk. Use an option from Save surface command in order to save the new surface model.

To subtract two surface models:

1. Select **Subtract surfaces** command from the **Utility** pulldown menu.

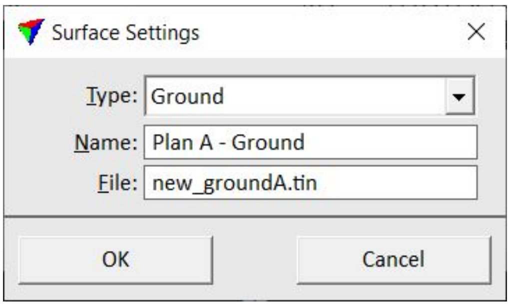
   This opens the **Subtract surfaces** dialog:



2. Select **Upper surface** and **Lower surface** to be subtracted.

3. Click OK.

   This opens the **Surface Settings** dialog:



4. Select a **Type** for the new surface.

5. Define a **Name** for the surface.

6. (Optional) Define a new **File** name for the surface model file stored on the hard disk.

7. Click OK.

   This creates a surface model that represents the elevation differences between the two original surface models.

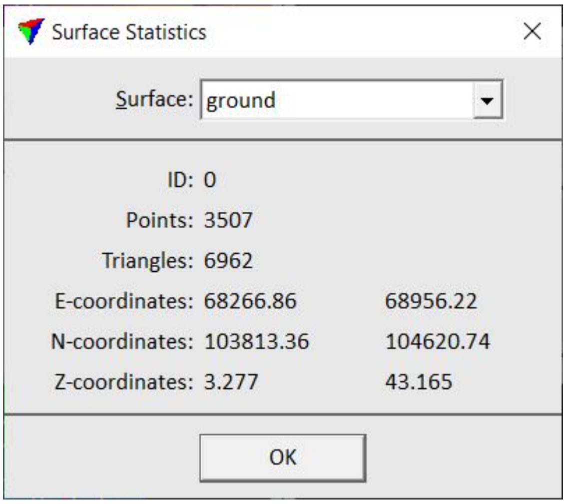
View statistics

View statistics command shows statistical information for a selected surface model. This includes the numbers of points and triangles, and the coordinate ranges for easting, northing, and elevation.

To view surface statistics:

1. (Optional) Select a surface model from the list in the **Surfaces** window.
2. Select **View statistics** command from the **Utility** pulldown menu.

   This opens the **Surface Statistics** dialog:



3. Select a **Surface** to view statistics. If a surface was selected in step 1, this surface is selected in the dialog.

   This displays the statistics for the selected surface model.

View pulldown menu

Menu commands from **View** pulldown menu in **Surfaces** window are used to change the appearance of the **Surfaces** window.

TO	USE COMMAND
Display a minimized **Surface** window	Minimal dialog
Display a small size **Surfaces** window	Small dialog
Display a medium size **Surfaces** window	Medium dialog
Display a large size **Surfaces** window	Large dialog
Change the display of fields in the **Surfaces** window	Fields
Sort the list of surfaces in the **Surfaces** window	Sort by

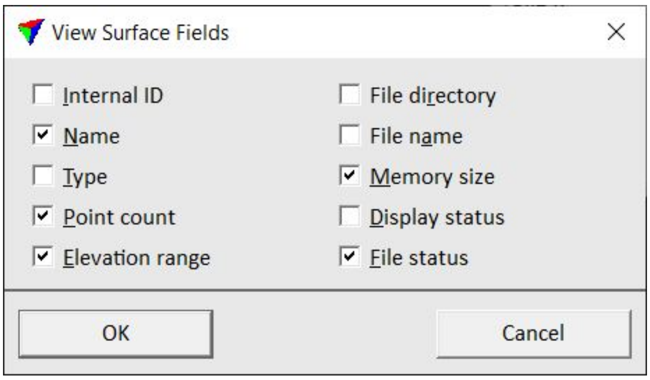
Fields

Fields command lets you select which information columns are displayed in the **Surface** window.

To select visible fields:

1. Select **Fields** command from the **View** pulldown menu.

   This opens the **View Surface Fields** dialog:



2. Switch on fields that you want to display in the **Surface** window and click OK.

Sort by

Sort by command sorts the list of loaded surface models according to the selected attribute. The sub-menu includes the following options:

- **Internal ID** - sorts increasing by the internal number of the surface model.
- **Name** - sorts alphabetically increasing by the name of the surface model.
- **Type** - sorts alphabetically increasing by the type of the surface model.
- **File name** - sorts alphabetically increasing by the file name of the surface model.
- **Point count** - sorts decreasing by the amount of points contained in the surface models.

The order of surface models in the list determines the drawing order of the models in the CAD file. Models that appear first in the list are drawn before models that appear further down in the list. Depending on the display method, this may result in only the last model in the list being visible in top views if surface models overlap.

To sort surface models:

1. Select an option from **Sort by** command from the **View** pulldown menu.

   This sort the surface model list according to the selected attribute.

Importing and Exporting Data

TerraModeler can import data from a variety of sources and output data to several file formats. Most of the import and export functions can be found in the **File** pulldown menu of the Surfaces window.

FOR DATA TYPE	USE FOR IMPORT	USE FOR EXPORT
Xyz text file	Import / XYZ text file	Export / Xyz text file
Xyz binary file (LAS, BIN, FBI)	Import / XYZ binary file	-
Disimp grid file	Import / Lattice file *Not Lite*	Export / Lattice file *Not Lite*
Intergraph grid file (GRD)	Import / Lattice file *Not Lite*	Export / Lattice file *Not Lite*
Ordnance Survey NTF grid file	Import / Lattice file *Not Lite*	-
Triangles in CAD file	Import / Triangles	Display Triangles tool
Triangle text file	-	Export / Triangle text file
InfraModel 4.0 file	Import / InfraModel file	Export / Triangle text file
LandXML 1.0/1.2 file	Import / LandXML file	Export / Triangle text file
ArcInfo grid file (GRD)	-	Export / Lattice file *Not Lite*
GeoTIFF grid file	-	Export / Lattice file *Not Lite*
Japanese grid file	-	Export / Lattice file *Not Lite*
Raw binary grid file	-	Export / Lattice file *Not Lite*
Surfer grid file	-	Export / Lattice file *Not Lite*
Xyz text grid file	Import / XYZ text file	Export / Lattice file *Not Lite*
Elevation-colored raster image	-	Export / Raster image *Not Lite*
Graphical elements	Triangulate Elements tool	Export / Graphical elements
SiteWorks surface model	Open surface	-

Export / Graphical elements

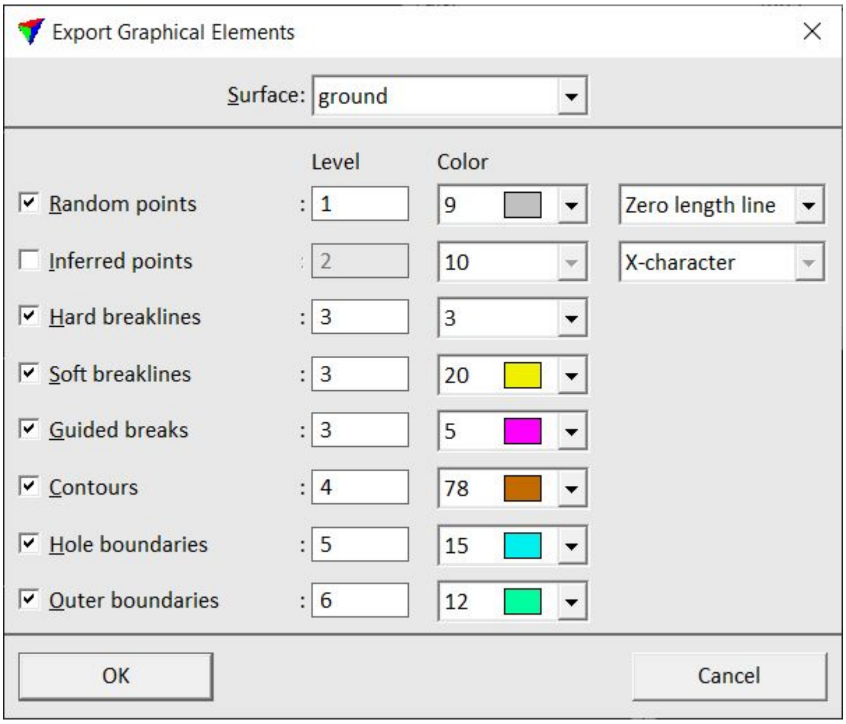
Graphical elements command draws different types of surface model points and lines on given levels into the CAD file. Random and inferred points can be drawn as leveling text elements, as zero length lines (points), or as X character text elements. Breaklines, contours, hole boundaries, and outer boundaries are drawn as linear elements.

The text design of the leveling text elements for random and inferred points are defined in Elevation labels category of the TerraModeler **Settings**.

To draw surface model elements into the CAD file

1. Select **Graphical elements** command from the **File / Export** pulldown menu.

   This opens the **Export Graphical Elements** dialog:



2. Select a **Surface** model for which to draw elements into the CAD file.

3. Select the surface element to be drawn and define **Level** and **Color** settings.

4. Define the element type for drawing random and inferred points.

5. Click OK.

   This draws the selected elements into the CAD file.

The command may be useful if you want to modify the surface in a way TerraModeler does not provide any suitable tools for. For example, TerraModeler does not include a tool for shifting a group of surface model elements horizontally. You can accomplish this by:

1. Export the surface model elements as graphical elements into the CAD file.
2. Switch off all other levels in a view.
3. Move the graphical elements using CAD tools.
4. Delete the old surface model.
5. Use Triangulate View tool to create a new surface model from the shifted graphical elements.

Export / Lattice file

Not Lite

Lattice file command creates a lattice model file from a surface model. Lattice models are representations of a surface using a grid structure of points. The lattice model files include regularly distributed points at constant intervals in both, X and Y directions. Common storage formats are text files, image files, or software-specific grid file formats.

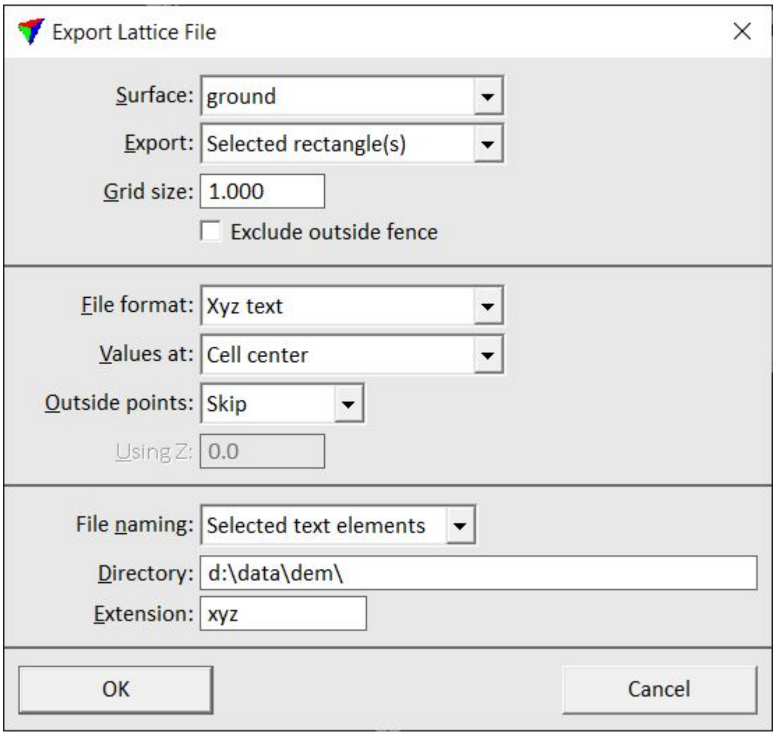
The command can create lattice files for multiple selected rectangles in a batch process. This requires rectangles drawn in the CAD file that define the lattice file boundaries. In order to automate the file naming for the output files as well, a text element can be placed inside the rectangles. Both, the rectangles and the text elements must be selected in order to use them in the export process.

To create lattice model file(s) from a surface model:

1. (Optional) Draw and select rectangle(s) and text element(s) for defining the area of the output file(s).

2. Select **Lattice file** command from the **File / Export** pulldown menu.

   This opens the **Export Lattice File** dialog:



3. Define settings and click OK.

4. If the area of the output file is not selected in step 1, define the export area by drawing a rectangle with two data clicks in a top view.

5. If the output file names are not defined by selected text elements, the **Export grid file** dialog opens, a standard dialog for saving file.

6. Enter a name including the file extension for the output file and click OK.

   The lattice file is created.

SETTING	EFFECT
Surface	Surface model for which to create the lattice file.
Export	Area covered by the output file: • **Whole surface** - the complete surface model is exported. • **Rectangle** - the surface model area for export is defined by a orthonormal rectangle. • **Rotated rectangle** - the surface model area for export is defined by a rotated rectangle. This is only active if **File format** is set to **Intergraph GRD**. • **Selected rectangle(s)** - the area defined by selected rectangle(s) is exported. This is only active if at least one rectangular shape is selected.
Grid size	Constant distance between points in the grid structure of the lattice model file.
Exclude outside fence	If on, the surface points outside a selected polygon or fence are treated as outside points.
File format	Format of the output file. Available formats are: • **ArcInfo Grid**, **Intergraph GRD** • **Disimap** raster format • **GeoTIFF** formats, **Shaded GeoTIFF** • **Japanese DMF** and **LEM** • **Raw image** formats • **Surfer ASCII** and **binary** • **XYZ text** Many of the additional settings for the lattice files depend on the selected format for the output files.
Values at	Defines the exact location of the lattice point coordinates: • **Cell center** - the coordinates of the grid cell center are stored.

SETTING	EFFECT
	• **Cell corner** - the coordinates of the grid cell corner are stored. For some output file formats only one option is available.
No data value	Defines a constant value for grid cells inside the lattice model area where no elevation value can be derived from the source data. This can be, for example, a software-specific value.
Z unit	Defines the elevation value unit for several raster formats.
Create TFW files	If on, external georeferencing files are created for **GeoTIFFs** .
Write coordinate block	If on, positional information is written into the structure of **Intergraph GRD** files.
Byte order	Defines the byte order for **Raw image** formats related to the processor type: **Intel** or **Motorola**.
Outside points	Defines for **Xyz text** files how to handle grid points outside the lattice model area where no elevation value can be derived from the source data: • **Skip** - outside grid points are not written into the text file. • **Output** - outside grid points are written into the text file using the constant value given in the **Using Z** field.
File naming	Defines the file naming method if several output files are created: • **Enter name for each** - the software asks you to enter a name for each lattice file that is created. • **Selected text elements** - the software uses a text element placed and selected inside the rectangle(s) as file name. This is only active if **Export** is set to **Selected rectangle(s)**.
Directory	Directory for storing the output file(s). This is only active if **File naming** is set to **Selected text elements**.
Extension	File extension for the output file(s). This is only active if **File naming** is set to **Selected text elements**.

Export / Raster image

Not Lite

Raster image command creates image files from a surface model. It can either create an output file for the whole surface model or for the area defined by a selected rectangle. The coloring can be based on the elevation values of a surface model or on elevation differences between two surface models. The images can be created using color depths of 24 bit (true-color), 256 colors, or 8 bit (gray-scale).

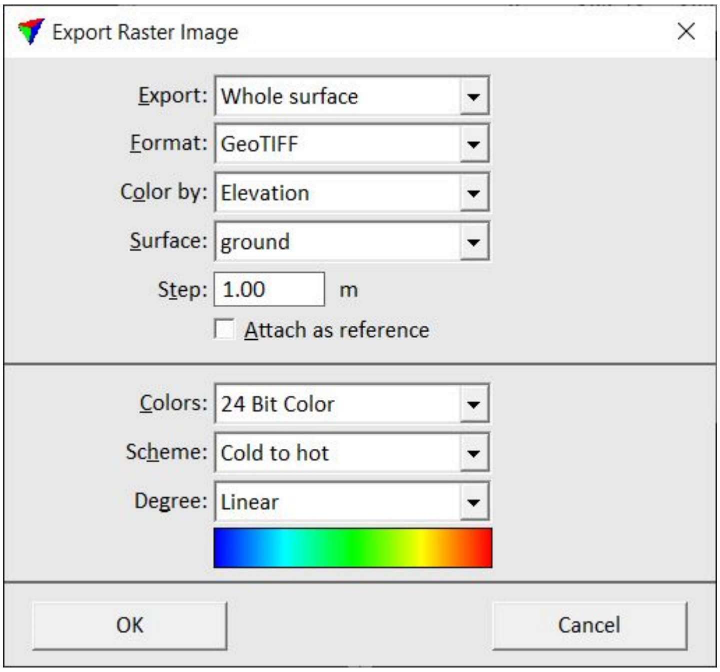
To export a raster image:

1. (Optional) Draw and select a rectangle that defines the area of the output file.

2. (Optional) Select a surface model from the list in the **Surfaces** window.

3. Select **Raster image** command from the **File / Export** menu.

   This opens the **Export Raster Image** dialog:



4. Define settings and click OK.

   TerraModeler computes the pixel colors and then opens another **Export Raster Image** dialog, a standard dialog for saving files.

5. Enter a name for the output file and click OK.

   A raster image is created.

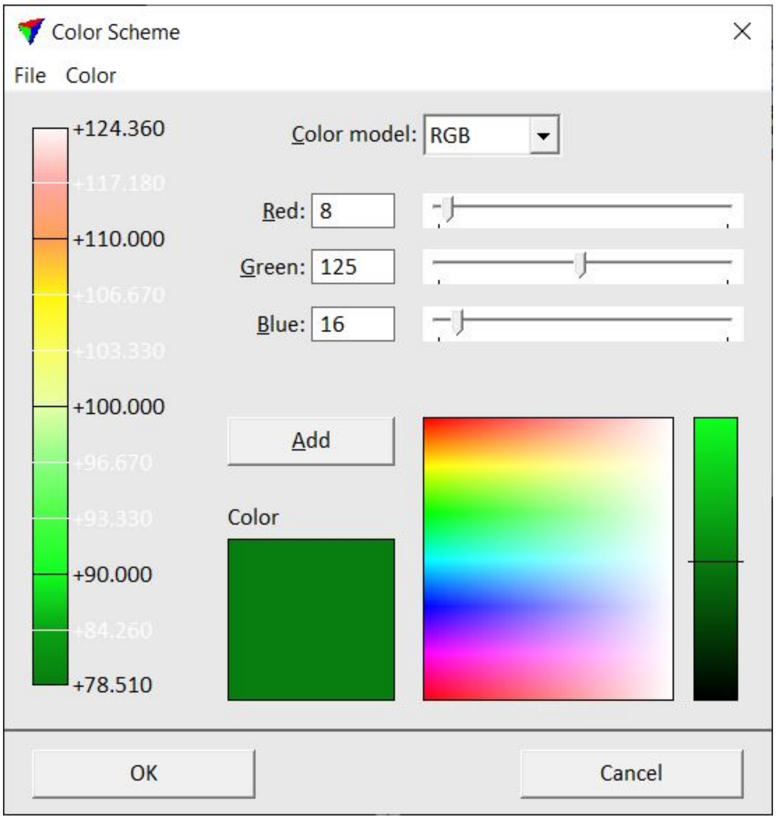
SETTING	EFFECT
Export	Area covered by the output file:

SETTING	EFFECT
	• **Whole surface** - the complete surface model is exported. • **Selected rectangle** - the area defined by a selected rectangle is exported. This is only active if a rectangular shape is selected.
Format	File format of the output file: **Windows BMP** or **GeoTIFF**.
Color by	Attribute used for coloring the output file: • **Elevation** - coloring based on the elevation values of a surface model. • **Vertical difference** - coloring based on the vertical difference between two surface models. • **Shortest distance** - coloring based on the shortest distance between two surface models. • **Shaded surface** - coloring based on lightning conditions.
Surface(s)	Name of the surface(s) to be used for output file creation. If color by is set to **Vertical difference** or **Shortest distance**, two surfaces must be selected.
Step	Size of each pixel in the output file.
Attach as reference	If on, the output file is attached as reference in the CAD file. *(Bentley CAD only)*
Colors	Color depth of the output raster image: • **24 Bit Color** - true-color image. • **256 Colors** - 256 colors image. • **Grey scale** - 8 bit gray scale image.
Sun azimuth	Direction from which the sun illuminates the model. Zero is north and angles increase clockwise. Visible only when **Color by** is set to **Shaded surface**.
Sun angle	Sun angle above the horizon. Visible only when **Color by** is set to **Shaded surface**.
Scheme	Type of color scheme: • **Cold to hot** - varies from blue color hues for low values to red color hues for high values. • **Hot to cold** - varies from red color hues for low values to blue color hues for high values.

SETTING	EFFECT
	• **Black to white** - varies from black for low values to white for high values. Visible only if **Colors** is set to **Gray Scale**. • **White to black** - varies from white for low values to black for high values. Visible only if **Colors** is set to **Gray Scale**. • **Earth tones** - varies between green, yellow, brown, and gray hues with low saturation. Visible only when **Color by** is set to **Shaded surface**. • **Selected colors** - you can define an own coloring scheme. Click on the **Define** button in order to define a Color scheme for 8 bit color images or a Color scheme for 24 bit color images.
Degree	Determines how the color hues are distributed in Cold to hot and Hot to cold color schemes: • **Hot**, **Warm** - color hues are shifted to the red-yellow color range. • **Very light, Light** - color values are shifted to the white color range. Visible only when **Colors** is set to **Gray Scale**. • **Linear** - color hues are linearly distributed. • **Dark, Very dark** - color values are shifted to the black color range. Visible only when **Colors** is set to **Gray Scale**. • **Cool**, **Cold** - color hues are shifted to the blue-green color range.
Color cycles	Number of color cycles. Use zero to create a gray scale display showing triangle slope only. Visible only when **Color by** is set to **Shaded surface**.

Color scheme for raster image files

The **Color Scheme** dialog lets you define a color scheme for the raster image export files. The dialog uses the RGB or the HSV color models. The software creates smooth transitions between the colors assigned to the color scheme.



The vertical bar on the left shows all the colors currently assigned in the scheme. Each color boundary has an elevation label displaying the elevation where a smooth color change occurs. The text label is black, if the elevation boundary is fixed. A white text indicates that the elevation boundary has not been fixed and is recalculated if you change the number of colors in the scheme.

From the coloring scheme dialog, you can save a coloring scheme file to a hard disk.

USE MENU ITEM	TO
File / Open	Open a previously saved color scheme file.
File / Save As	Save the color scheme to a file.
Color / Remove last	Remove last color from the color scheme.
Color / Remove all	Remove all colors from the color scheme.

To assign a new color:

1. Define the **Color model** you want to use.

2. Click in the color field on the right at the location of the color you want to add to the color scheme.

   OR

2. Define **Red Green Blue** or **Hue Saturation Value** values by typing a number or moving the slider. The **RGB** values can range from 0 to 255, the **H** value can range from 0 to 359, the **SV** values from 0 to 100.

   This selects the color and adjusts the values in the **RGB** or **HSV** fields. The selected color is displayed in the **Color** field.

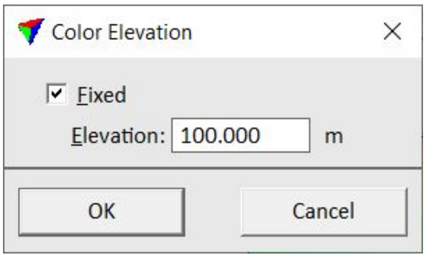
3. Click **Add** in order to add the selected color to the color scheme.

   The color is added to the lower end of the bar, thus, the first selected color is assigned to the highest elevation values.

To fix an elevation boundary:

1. Click on the text label of the elevation boundary.

   This opens the **Color Elevation** dialog



2. Switch **Fixed** on and enter an **Elevation** value on which to fix the color transition.

3. Click OK.

   This recalculates the other elevation boundaries according to the fixed elevation value.

To create a new color scheme:

1. Select **Remove all** command from the **Color** pulldown menu in the **Coloring scheme** dialog.

   This removes all assigned colors from the current color scheme.

2. Assign new colors to the color scheme as described above.

3. (Optional) Save the color scheme as a file on a hard disk using the **Save as** command from the **File** pulldown menu in the **Color Scheme** dialog.

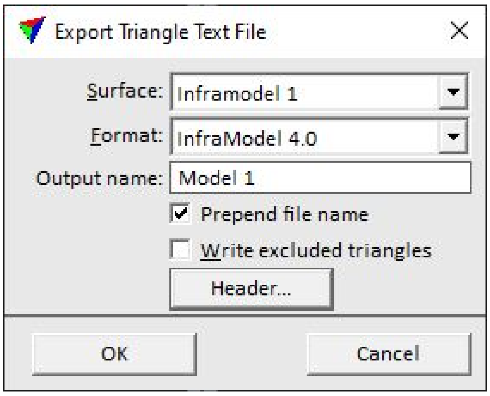
Export / Triangle text file

Triangle text file command creates a text file from surface model triangles. The triangles can be written into simple text files for points and edges, into exchange format text files, such as LandXML, or into survey application-specific text files. Supported triangle text file formats are **4ce DOT**, **GMS**, **InfraModel 4.0**, **ICC**, **Moss triangulation**, **LandXML 1.0/1.2**, **Point & edges text file**, **SMS / WMS** and **WorldToolKit NFF**.

To create a text file from surface model triangles

1. Select **Triangle text file** command from the **File / Export** pulldown menu.

   The **Export Triangle Text File** dialog opens:



2. Select a **Surface** for which to export the triangles.

3. Select a **Format** for the output file.

4. If required by the format, define additional settings.

5. Click OK.

   This opens another **Export triangle text file** dialog, a standard dialog for saving files. If triangle points and edges are saved into separate files, the dialog opens for a second time after saving the first file.

6. Enter a name for the output file and click OK.

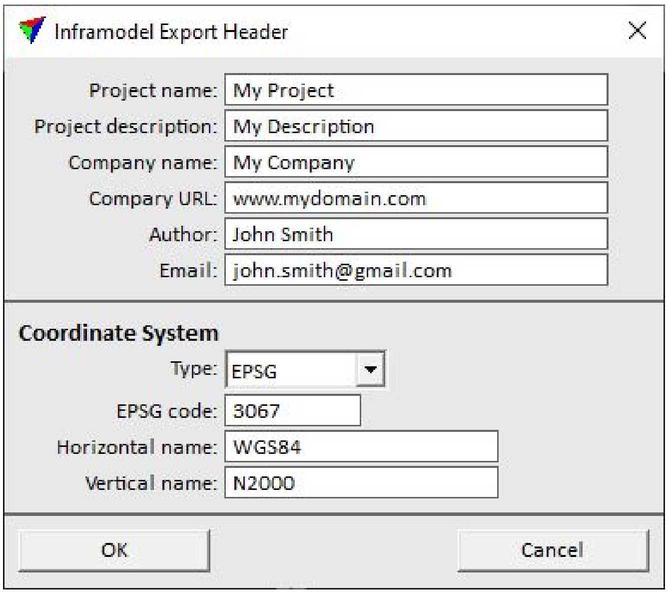
   This saves the triangles into the text file(s).

SETTING	EFFECT
Surface	Name of the surface model for which to export the triangles. The list includes all surface models that are open in TerraModeler.
Format	Format of the output file. Any additional settings depend on the selected format for the output file.

SETTING	EFFECT
ID	Surface identifier for **GMS** triangle files.
Output name	Name of the surface model written into **LandXML 1.0**, **LandXML 1.2** and **Inframodel** files.
Prepend file name	If on, the model's **Output name** is added to the ***Surface name*** tag in **LandXML 1.2** and **InfraModel** files. Example: *<Surface name="output file name - model name">*
Write excluded triangles	If on, excluded triangles are written into **LandXML 1.2** and **InfraModel** files.
Header	Opens the Inframodel Export Header dialog which allows the input of metadata for **InfraModel** files.
Y Axis up	If on, the coordinate system is rotated in a way that the Y axis points up for exporting triangle coordinates into **WorldToolKit NFF** files.
Output textures	If on, domain-related texture information is written into **WorldToolKit NFF** files.

Inframodel Export Header

The **Inframodel Export Header** dialog lets you define metadata for InfraModel ouput files.



SETTING	EFFECT
Project name	Free text for the ***Project name*** tag.
Project description	Free text for the ***Project desc*** tag.
Company name	Free text for the ***Author company*** tag.
Company URL	Free text for the ***Author companyURL*** tag.
Author	Free text for the ***Author createdBy*** tag.
Email	Free text for the ***Author createdByEmail*** tag.
Type	Type of coordinate system definition: **EPSG** or **Other**.
EPSG code	Code number of an EPSG coordinate system definition. This is only active if **Type** is set to **EPSG**.
Horizontal name	Name of the ellipsoid definition.
Vertical name	Name of the height system definition.

Export / Xyz text file

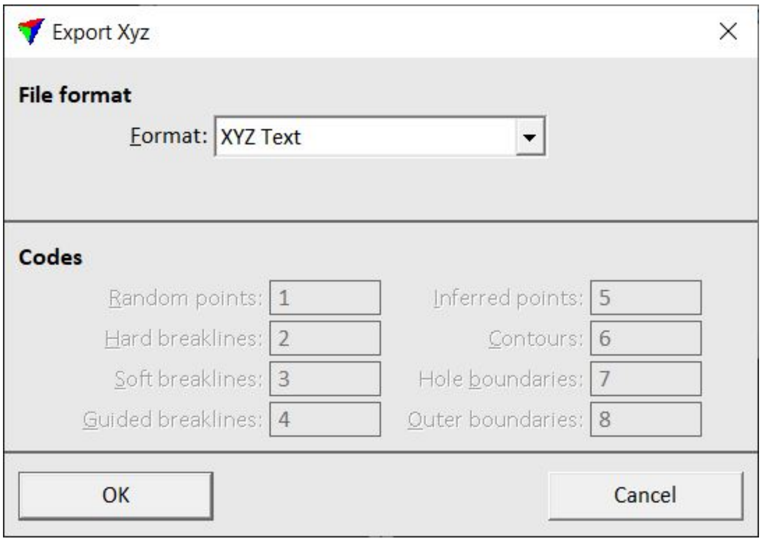
Xyz text file command creates a text file from surface model points. The points can be written into a simple text file where the easting, northing, and elevation coordinates of each point are stored in one line, or into survey application-specific text files. Supported text file formats are **Geodimeter**, **Gemini**, **Tekla fixed length and Tekla space delimited**, **Wild GRE**, **Wild COGO**, and **XYZ Text**.

To create a text file from surface model points

1. Select a surface in the **Surfaces** window.

2. Select **Xyz text file** command from the **File / Export** menu.

   The **Export Xyz** dialog opens:



3. Select a **Format** for the output file.

4. If another format than XYZ Text is selected, define additional settings.

5. Click OK.

   This opens the **File for point output** dialog, a standard dialog for saving files.

6. Enter a name for the output file and click OK.

   This creates the output file.

SETTING	EFFECT
Format	Format of the output file.
Tekla	Order of fields in Tekla text files. This is only active when **Format** is set to **Tekla fixed length** or **Tekla space delimited**.

SETTING	EFFECT
Codes	Feature codes for different surface point types in the output file. This is active for all **Format** except **XYZ Text**.

Import / InfraModel file

InfraModel file command creates a new surface model from InfraModel 4.0 files.

To import InfraModel files:

1. Select **InfraModel file** command from the **File / Import** pulldown menu.

   A new surface model is created from the input file. After the import is finished, a message shows the amount of points that are included in the surface model.

Import / LandXML file

LandXML file command creates a new surface model from LandXML files.

To import LandXML files:

1. Select **LandXML file** command from the **File / Import** pulldown menu.

   A new surface model is created from the input file. After the import is finished, a message shows the amount of points that are included in the surface model.

Import / Lattice file

Not Lite

Lattice file command imports lattice files and create a triangulated surface model using the grid-based points. Accepted lattice file formats are **ArcInfo Grid**, **Intergraph GRD**, **Finnish National Land Survey Disimp grid** files and **Ordnance Survey NTF grid** files.

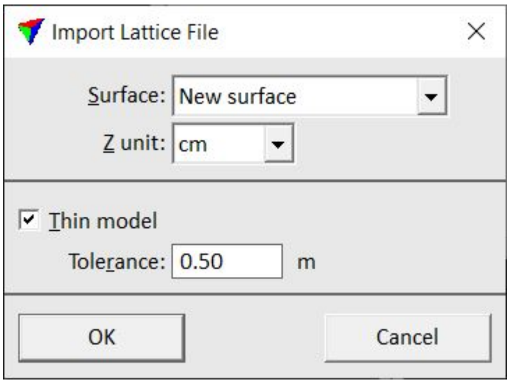
To import a lattice file:

1. Select **Lattice file** command from the **File / Import** pulldown menu.

   This opens the **Import grid file** dialog, a standard dialog for opening files.

2. Locate the file that stores the surface model points.

   This opens the **Import Lattice File** dialog:



3. Define settings and click OK.

   This opens the **Surface settings** dialog. Continue as described for File / New surface command.

   A new surface model is created from the lattice file.

SETTING	EFFECT
Surface	Surface to be created from the lattice file. Always **New surface**.
Z unit	Unit of the elevation values in the lattice file.
Thin model	If on, the software leaves out less relevant points from the lattice file in order to reduce the file size of the surface model file.
Tolerance	Elevation tolerance for creating the model from the original lattice file. The surface model may differ in elevation up to the given value from the original. This is only active if **Thin model** is switched on.

A triangulated surface model requires a lot more memory than a lattice model file, if all the lattice points are used in the triangulation. Therefore, it might be necessary to leave out some of the lattice points. TerraModeler is capable of leaving out less relevant points and still building a triangulated network within a given tolerance of the original grid points.

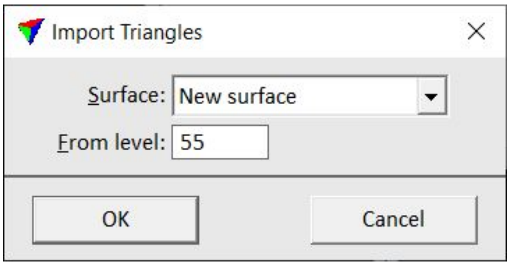
Import / Triangles

Triangles command creates a new surface model from triangle elements drawn in the CAD file. This can be used, for example, to import a triangulated surface model created by another application.

To import triangles from the CAD file:

1. Select **Triangles** command from the **File / Import** pulldown menu.

   This opens the **Import Triangles** dialog:



2. Define settings and click OK.

   This opens the **Surface settings** dialog. Follow the common steps for Creating a surface model.

   A new surface model is created from the triangle elements. The triangle structure of the surface model is identical with the original triangle elements.

SETTING	EFFECT
Surface	Surface to be created from the triangles. Always **New surface**.
From level	Level in the CAD file on which the triangle elements are located.

Import / XYZ binary file

Xyz binary file command is used to create a new surface model or to add points to an existing model from a TerraScan binary file. The binary file stores at least the easting, northing and elevation coordinates for each point of the model.

The binary file may have additional fields, such as the point code, which determines the type of the point. See Point Types for the code definitions.

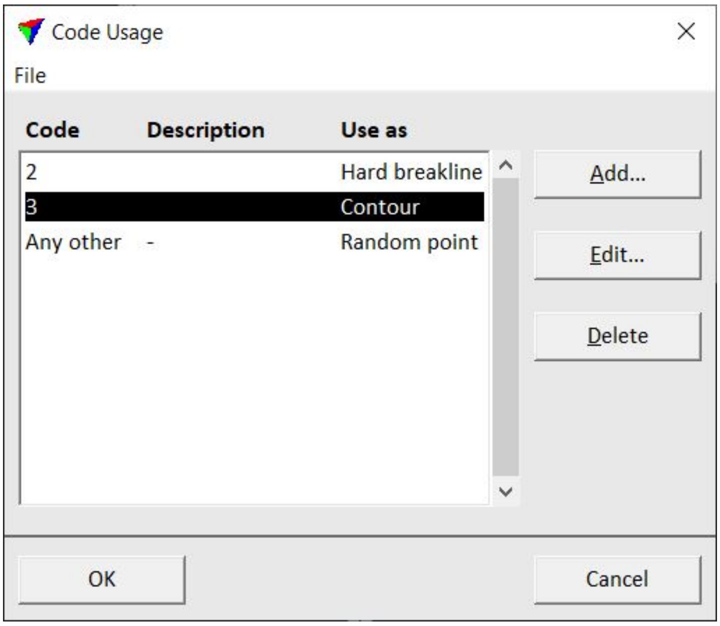
To import points from a binary file

1. Select **Xyz binary file** command from the **File / Import** pulldown menu.

   This opens the **Import Xyz binary file** dialog, a standard dialog for opening files.

2. Locate the file that stores the surface model points.

   This opens the **Code Usage** dialog which defines point codes and their usage for the surface model creation:



   By default, a new code definition file includes a line for **Any other** code used as random points in a surface model. You can add specific code definitions by using the **Add** button in the **Code usage** dialog.

   The **Code definition** dialog opens where you can enter a **Code**, (optional) a **Description**, and define how points with this code are used. Available surface model elements are **Random point**, **Soft breakline**, **Hard breakline**, **Contour**, **2d hole**, **3d hole**. Points with certain codes may also be ignored for the import.

   You can modify or delete a code definition by using the Edit or Delete buttons respectively in the Code usage dialog. Use commands from the File pulldown menu in order to save the code definitions into a file, open an existing code definition file, or create a new file.

3. If required, create additional code definitions in the **Code usage** dialog.

4. (Optional) Save the code definitions into a text file using the **Save as** command from the **File** pulldown menu in the **Code usage** dialog.

   The code definitions are saved into a text file with the ending .COD.

5. Click OK.

   This opens the **Triangulate surface** dialog. Follow the common steps for Creating a surface model.

   A new surface model is created from the Xyz binary file or the points are added to an existing surface model.

Import / XYZ text file

Xyz text file command in used to create a new surface model or to add points to an existing model from a text file. The text file stores at least the easting, northing, and elevation coordinates for each point of the model.

TerraModeler requires that one row in a text file represents one point. The coordinates have to be stored in fields separated by white space, comma, or semicolon. The text file may have additional fields, such as a point number, code, a text string, pen up field, or any other fields that can be ignored for the import.

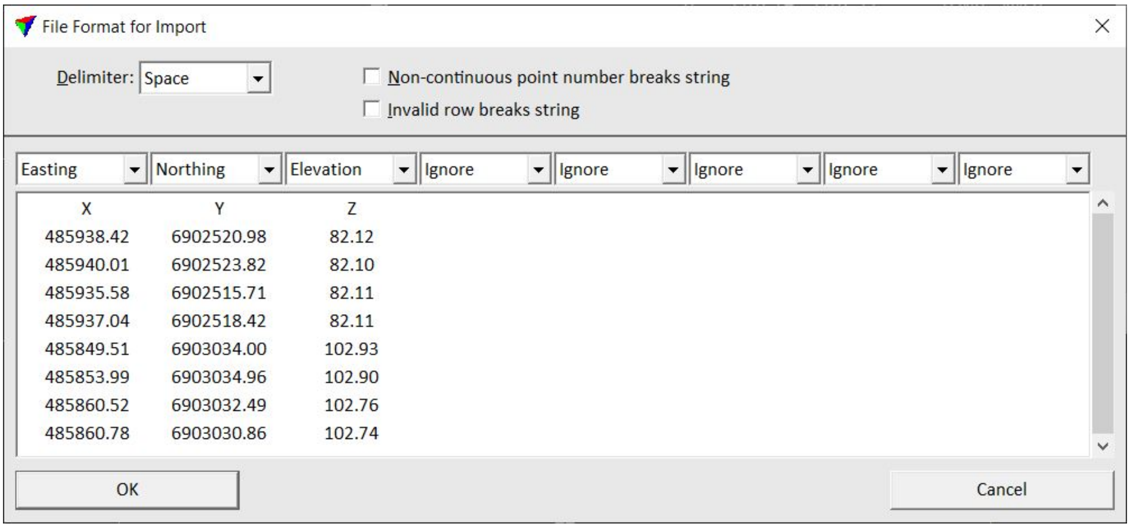
To import points from a text file

1. Select **Xyz text file** command from the **File / Import** pulldown menu.

   This opens the **Import Xyz text file** dialog, a standard dialog for opening files.

2. Locate the text file that stores the surface model points.

   This opens the **File Format for Import** dialog:



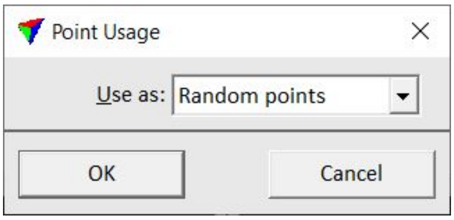
   TerraModeler reads the first 25 rows from the text file and displays them in the dialog. The **Delimiter** is detected automatically.

3. Check the **Delimiter** and select the fields where **E**, **N**, and **Z** coordinates are stored.

4. If required, define additional fields and settings.

5. Click OK.

   This opens the **Point Usage** dialog:



6. Define whether to use the points as **Random points** or as **Breakline points**.

7. Click OK.

   This opens the **Triangulate surface** dialog. Follow the common steps for [Creating a surface model](#).

   A new surface model is created from the Xyz text file or the points are added to an existing surface model.

SETTING	EFFECT
Delimiter	Character that separates the fields in a line of one point from each other: **Space**, **Comma**, or **Semicolon**.
Non-continuous point number breaks string	If on, breaklines are split into separate chains if the point number increment is bigger than 1.
Invalid row breaks the string	If on, breaklines are split into separate chains if there is an empty or invalid line in the input text file.
Attributes for fields in the text file	Attribute that is stored for a point in a specific field: • **Ignore** - the field is ignored during import. • **E**, **N**, **Z** - easting, northing, elevation coordinates. • **Code** - text string. Defines, for example, the type of a breakline element. • **Number** - point number. • **String** - chain number. Defines, for example, continuous breakline elements. • **Pen up** - marks the end of a chain if it is not 0.

Lattice Database

Not Lite, Not UAV

A lattice database in TerraModeler is a collection of grid-based files placed in the same directory on a hard disk. The lattice files are typically produced by a national land survey organization that manages elevation data covering the whole country. This type of data provides an inexpensive way to create a small-scale surface model of large areas or a quick way to derive approximate elevation values for any location in the country.

TerraModeler offers some dedicated tools which can extract elevation values from a collection of lattice files. The tools search for lattice files in a directory that is specified in Lattice database category of the TerraModeler **Settings**. All tools require that the CAD file uses the same coordinate system as the lattice files were created with.

Supported lattice file formats include:

- **Disimp** - used by the National Land Survey of Finland.
- **Ordnance Survey NTF** - used by the UK Ordnance Survey.
- **Intergraph GRD** - has to be orthonormal.

TO	USE COMMAND
Draw boundaries for the lattice database area	Lattice database / Draw boundaries
View lattice database elevation values	View Elevation tool
Draw contours for lattice database files into the CAD file	Lattice database / Draw contours
Triangulate lattice database points	Lattice database / Import fence contents

Lattice database / Draw boundaries

Not Lite, Not UAV

Draw boundaries command draws the boundaries of all lattice files in the lattice file directory into the CAD file. The command can be used, for example, to check what areas are covered by the lattice database files. This may be the first thing you do before working with lattice database files.

The boundaries are drawn as shape elements using the active level and symbology settings in the CAD file. The name of each lattice file is placed as text element at the center of the corresponding rectangle using active text setting in the CAD file.

To draw lattice file boundaries into the CAD file:

1. Select **Draw boundaries** command in the **Utility / Lattice database** pulldown menu.

   This scans through the lattice database and draws rectangular shapes and text elements into the CAD file showing the area covered by each file. An information dialog is displayed that shows for how many lattice files the boundaries are drawn.

Lattice database / Draw contours

Not Lite, Not UAV

Draw contours command draws contour lines, labels, and ticks for lattice database files.

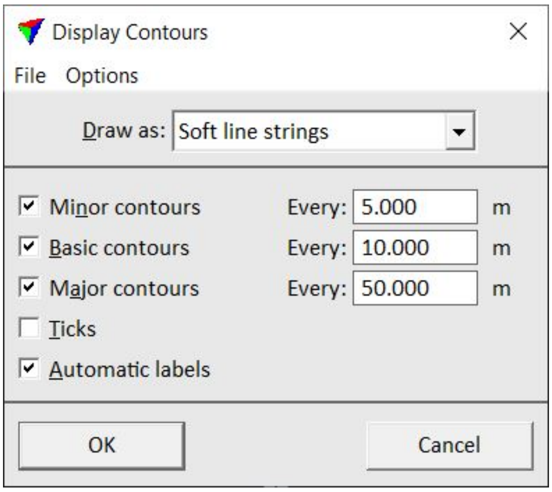
To draw contours for lattice database files:

1. Select **Draw contours** command from the **Utility / Lattice database** pulldown menu.

   This opens the **Draw Grid Contours** dialog, a standard dialog for opening files.

2. Select the lattice file(s) for drawing contours.

   The **Display Contours** dialog opens:

   The dialog is a reduced version of the **Display Contours** dialog described for the Display Contours tool. It provides the same options for drawing contour lines, labels and ticks, and for saving and loading contour settings files.

3. Define settings and click OK.

   This draws the contours into the CAD file.

Lattice database / Import fence contents

Not Lite, Not UAV, Not Spatix

Import fence contents command provides an option for creating a triangulated surface model from lattice database points inside a fence. The fence may overlap several individual lattice files. A triangulated surface model enables the addition of new elevation data, such as random points or breaklines. However, changes to the surface model can not be saved into the lattice database files.

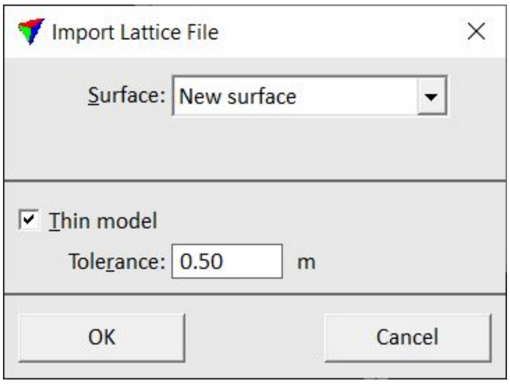
A triangulated surface model requires a lot more memory than a lattice model file, if all the lattice points are used in the triangulation. Therefore, it might be necessary to leave out some of the lattice points. TerraModeler is capable of leaving out less relevant points and still building a triangulated network within a given tolerance of the original grid points.

To triangulate lattice database points:

1. Place a rectangular fence.

2. Select **Import fence** contents command from the **Utility / Lattice database** pulldown menu.

   This opens the **Import Lattice File** dialog:



3. Define settings and click OK.

   This opens the **Surface Settings** dialog. Continue as described for the File / New surface command.

   The software processes all lattice files inside the fence and creates a triangulated surface model from the points.

SETTING	EFFECT
Surface	Surface to be created from the lattice file. Always **New surface**.
Thin model	If on, the software leaves out less relevant points from the lattice file in order to reduce the file size of the surface model file.

SETTING	EFFECT
Tolerance	Elevation tolerance for creating the model from the original lattice file. The surface model may differ in elevation up to the given value from the original. This is only active if **Thin model** is switched on.

Viewing lattice database elevation

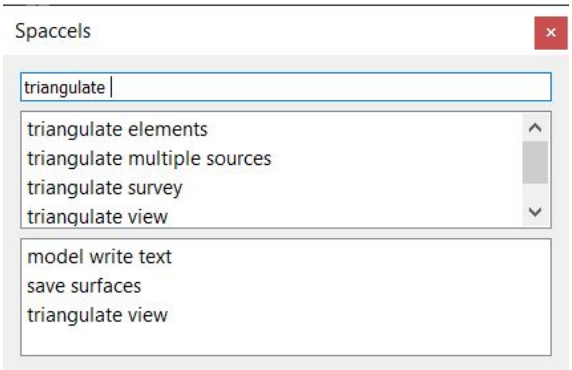
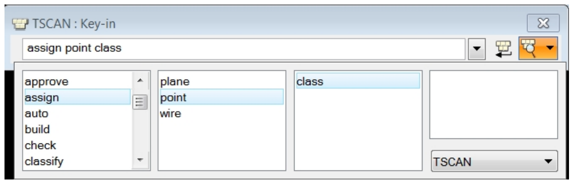
Not Lite, Not UAV

View Elevation tool can display lattice database elevation values. In the tool’s dialog, select **Lattice database** as the **Surface** to view. If you move the mouse pointer inside the lattice database area, TerraModeler determines the correct lattice file and displays the elevation value closest to the mouse pointer location.

Additionally, you can use the **Point on surface** lock to place elements on the elevation level of the lattice database files. This offers a simple way to place an element on the approximate ground elevation anywhere in the area covered by the lattice database files.

Key-in commands/Spaccels

Key-in commands (Bentley CAD) or **Spaccels** (**Sp**atix **accel**erates) 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 TerraModeler 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 TMODEL in the list at the lower right corner of the **Browse Key-in** dialog, the selection of commands is limited to TerraModeler 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

SPATIX	MICROSTATION
	or performs another action called by the command.
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
Model app main
Model close surfaces
Model write text
Save surfaces

Model App Main

Model App Main opens the **TerraModeler** toolbox. By default, the toolbox is opened when TerraModeler is loaded. The command can be used to re-open the toolbox after it was accidentally closed.

Syntax:

model app main

Model Close Surfaces

Model Close Surfaces key-in command closes all surfaces that are loaded in TerraModeler.

Syntax:

model close surfaces

Corresponding command: Close all

Model Write Text

Model Write Text key-in command writes LandXML files. It requires a surface model loaded in TerraModeler. The command may include a variable for the LandXML file name in order to avoid that existing files are overwritten. The variable may be related to TerraScan project blocks from which the surface model has been created.

Syntax:

model write text landxml12 d:\myproject\output\#bnumber.xml

PARAMETER	EFFECT
landxml12	LandXML file format in version 1.2.
d:\myproject\output\#bnumber.xml	Location and name of the output file. #bnumber is a variable that will be replaced by the number of a block in a TerraScan project. The same variable may be used for creating the surface model to export.

Corresponding command: Export Triangle Text File

Save Surfaces

Save Surfaces key-in command saves all surface models loaded in TerraModeler to their files on the hard disk.

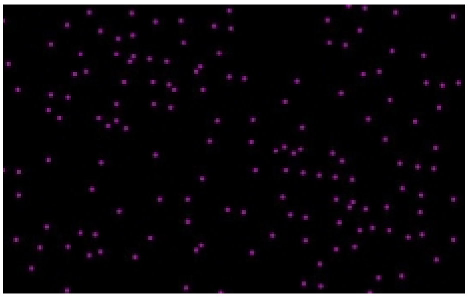
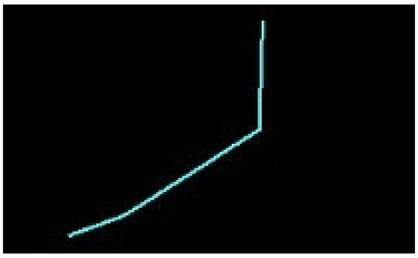
Syntax:

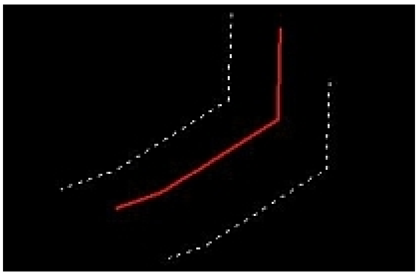
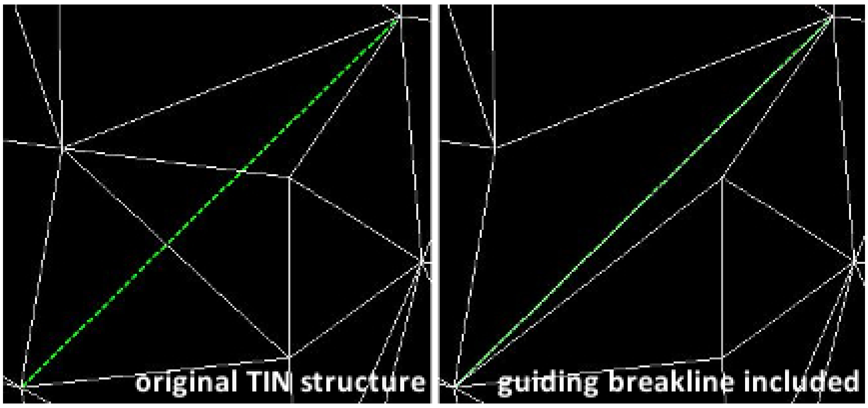
save surfaces

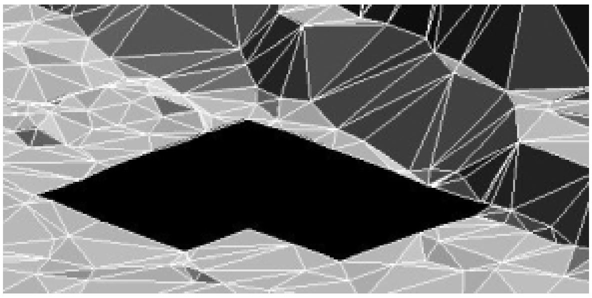
Corresponding menu command: Save all

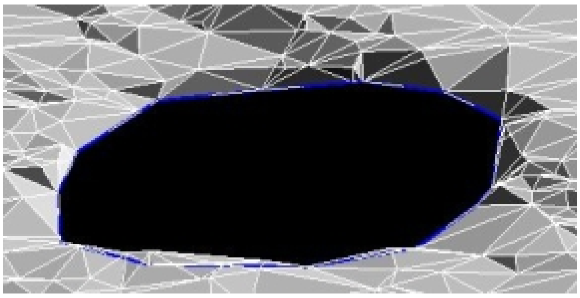
Breakline types

TerraModeler knows the following breakline types which are closely related to the Point Types of the TIN format:

BREAKLINE TYPE	DESCRIPTION	TYPICAL EXAMPLES
Random point	Randomly distributed point elements.	Surveyed point features, thinned ground points from a point cloud, model keypoints. 
Hard breakline	3D line element that represents a sharp edge in a surface. Vertices of the line element are added to the TIN and thus, effect the TIN structure.	Edges of hard surfaces, sharp terrain slope edges. 
Soft	3D line eleme	Smooth, "round" terrain slope edges.

breakline	nt that represents a smooth edge in a surface. Vertices of the line element are added to the TIN and thus, effect the TIN structure.	
Guiding breakline	2D line element that forces triangle edges to follow its direction. No additional vertices are added to the TIN from this type of breakline.	Quick solution for approximate terrain slope edges. 
Cont	3D contour line	Common contour lines that represent the ground surface.

our	element.	
2D hole	2D shape element that represents void areas ("gaps") in a surface model. The elevation along the boundary is determined by the closest elevation next to the polygon boundary. In the TIN file format specification, this is referred to as "inner	Building footprints. 

	boundary".	
3D hole	3D shape element that represents void areas ("gaps") in a surface model. The elevation along the boundary is determined by the elevation of the polygon boundary. In the [TIN file format specification](), this is referred to as "inner boundary".	Water boundaries. 
2D boundary	2D shape element representing the	A surface model should have only one outer boundary.

	outer boundary of a surface model. Areas outside of the boundary are invalid and not part of the surface model. The elevation along the boundary is determined by the closest elevation next to the shape boundary.	
3D boundary	3D shape element representing the outer boundary of a surface model. Areas	A surface model should have only one outer boundary.

	outside of the boundary are invalid and not part of the surface model. The elevation along the boundary is determined by the elevation of the shape boundary.	

TIN File Format Specification

This chapter describes a triangulated surface model file format of TerraModeler. The file format is free for anyone to read and write. It is recommended to use extension .tin for TerraModeler TIN files but this is not mandatory.

The file format is meant for storing triangulated terrain models. The file format can store both xyz points and triangles which connect the points.

The triangle records are optional in the file. They may be left out of the file to save disc space if no manual editing of the surface has been done. The point records store all the relevant information from which the triangulation can be calculated as needed. However, if a user modifies the surface model (for example, assign additional breakline connections between points, mark areas as excluded, etc.), the triangle information must be stored and the triangle records contain the up-to-date edges of the surface.

File Organization

The file contains binary data consisting of a file header, a number of point records and a number of triangle records. The header must appear first in the file. It is recommended that point records start immediately after the header and triangle records immediately after point records.

Triangle records are optional. The file format may be used to store surface model points only with the intention of triangulating those when actively used.

Data Types

The format definition used the following data types:

C TYPE	SIZE	ALIAS
char	1 byte	
unsigned char	1 byte	BYTE
long	4 bytes	
unsigned long	4 bytes	UINT
unsigned __int64	8 bytes	UINT64
double	8 bytes IEEE floating point	

All data is in big-endian format. For example, the long integer value of 16909060 (or 0x01020304 in hexadecimal) is stored in a way that the first byte has value 4, the second byte value 3, the third byte value 2, and the fourth byte value 1.

All strings must be null-terminated (have value zero to mark end of string).

File Header

The file header is at the beginning of the file and contains 160 bytes. The header fields which are not required and not used must be filled with zero.

DATA TYPE	ITEM	SIZE	REQUIRED
char[4]	Format Recognition String	4 bytes	Yes
UINT	Format Recognition Value	4 bytes	Yes
UINT	Version	4 bytes	Yes
UINT	Header Size	4 bytes	Yes

UINT	Number of Points	4 bytes	Yes
UINT	Point Size	4 bytes	Yes
UINT	Number of Triangles	4 bytes	Yes
UINT	Triangle Size	4 bytes	Yes
char[40]	Surface Name	40 bytes	No
char[40]	Software	40 bytes	No
UINT	Surface Type	4 bytes	No
UINT	Coordinate Resolution	4 bytes	Yes
double	X Origin	8 bytes	Yes
double	Y Origin	8 bytes	Yes
double	Z Origin	8 bytes	Yes
UINT64	Point Data Position	8 bytes	Yes
UINT64	Triangle Data Position	8 bytes	Yes

Header fields:

- **Format Recognition String** - must contain “TTIN”. Used for file recognition.
- **Format Recognition Value** - must contain value 20101221. Used for file recognition.
- **Version** - must contain value 1.
- **Header Size** - normally set to value 160.
- **Number of Points** - total number of point records in the file.
- **Point Size** - size of each point record. Normally set to value 14.
- **Number of Triangles** - total number of triangle records in the file.
- **Triangle Size** - size of each triangle record. Normally set to value 26.
- **Surface Name** - descriptive name for the surface such as “Ground”.
- **Software** - software which generated the file. For example “TerraModeler”.
- **Surface Type** - surface type which may be used to categorize models of sub soil layers or other types of terrain surfaces. This is most often zero (= ground).
- **Coordinate Resolution** - number of integer steps in a real world unit. If the real world unit is meter and the coordinate resolution is 100, each integer step in point record x, y, and z fields corresponds to one centimeter.
- **X**, **Y** and **Z Origin** - center of the coordinate system in real world units.
- **Point Data Position** - file position where the point records start. File position is relative to the beginning of the file. This should normally be 160.
- **Triangle Data Position** - file position where triangle records start.

Point Record

Point records store information about the points from which the model is built.

DATA TYPE	ITEM	SIZE
long	X	4 bytes
long	Y	4 bytes
long	Z	4 bytes
BYTE	Break	1 byte
BYTE	Type	1 byte

Point record fields:

- **X** - X coordinate of the point.
- **Y** - Y coordinate of the point.
- **Z** - Z coordinate of the point.
- **Break** - 0 if the point is a random point or the first point in a breakline sequence. 1 if point is meant to be connected by a breakline edge with the previous point record.
- **Type** - point type. See Point Types for more information.

Each linear breakline feature appears as consecutive point records. The first point of a breakline should have **Break** = 0 and **Type** should be non-zero. All additional points of the same breakline should have **Break** = 1 and **Type** should be the same as for the first point.

Triangle Record

Triangle records store the actual triangulation and the neighbor relation of the triangles.

DATA TYPE	ITEM	SIZE
UINT[3]	Vertex	12 bytes
UINT[3]	Neighbor	12 bytes
BYTE	Flags	1 byte
BYTE	Domain	1 byte

Triangle record fields:

- **Vertex** - three points in clockwise order. Value 0 refers to the first point record.
- **Neighbor** - neighboring triangles in clockwise order. Value 0 means no neighboring triangle. Value 1 refers to first triangle record. Neighbor[0] is the triangle on the other side of edge Vertex[0] - Vertex[1]. Neighbor[1] is the triangle on the other side of edge Vertex[1] - Vertex[2]. Neighbor[2] is the triangle on the other side of edge Vertex[2] - Vertex[0].
- **Flags** - active/excluded/deleted state of the triangle and edge types.

- **Domain** - region or land type. 0 for undefined or default value.

The different bits in the **Flags** field have the following meaning:

- Bits 0-1 - triangle state: 0 = active, 1 = auto excluded, 2 = user excluded, 3 = deleted
- Bits 2-3 - type of edge 0: 0 = normal, 1 = soft break, 2 = hard break, 3 = other break
- Bits 4-5 - type of edge 1
- Bits 6-7 - type of edge 2

Coordinate Values

The X coordinate value is normally the same as easting.

The Y coordinate value is normally the same as northing.

The Z coordinate value is normally the same as elevation.

Point records store coordinate values as 32 bit integers to save disc space. To translate integer values into normal projection system coordinate values in meters or feet, the following equations have to be used:

```
WorldX = XOrigin + X / CoordinateResolution
WorldY = YOrigin + Y / CoordinateResolution
WorldZ = ZOrigin + Z / CoordinateResolution
```

Point Types

The **Type** field in the point record indicates the point type. Valid point types are:

- 0 - random point.
- 1 - point is part of a soft breakline element.
- 2 - point is part of a hard breakline element.
- 3 - point is part of a contour line.
- 4 - point is an inferred point at the top of a hill or at the bottom of a depression. These points may be automatically generated for hills and depressions when generating a surface from contours.
- 5 - point is part of an outer boundary. An outer boundary is defined by a clockwise polygon where the area outside the polygon is invalid. The last point record in the sequence has the same coordinates as the first point record in order to close the polygon. A surface model should not have more than one outer boundary.
- 6 - point is part of an inner boundary. An inner boundary is defined by a clockwise polygon where the area inside the polygon is invalid (“hole”). The last point record in the sequence has the same coordinates as the first point record in order to close the polygon.

C Structure Definitions

```
// TIN file header

typedef struct {
     char RecogStr[4] ;    // Recognition "TTIN"
     UINT RecogVal ;       // Recognition 20101221
     UINT Version ;        // Version 1
     UINT HdrSize ;        // Header size = sizeof(SurfHdr)
     UINT PntCnt ;         // Number of points
     UINT PntSize ;        // Size of each point 12
     UINT TriCnt ;         // Number of triangles
     UINT TriSize ;        // Size of each triangle 28
     char Desc[40] ;       // Descriptive name for surface
     char Software[40] ;   // Software which generated the file
     UINT Type ;           // Surface type
     (0=ground,1=design,2=bedrock,..)
     UINT CoordSize ;      // Number of integer steps per real world
     unit
     double OrgX ;         // Origin of coordinate system
     double OrgY ;
     double OrgZ ;
     UINT64 PntPos ;       // File position where point data starts
     UINT64 TriPos ;       // File position where triangle data starts
} TinHdr ;

// TIN file point record

typedef struct {
     long X ;
     long Y ;
     long Z ;
     BYTE Break ;          // Break line edge with previous point
     BYTE Type ;           // Point type TINPT_xxxx
} TinPnt ;

// TIN file triangle record

typedef struct {
     UINT Vertex[3] ;      // Triangle vertices in clockwise order
     UINT Neigbour[3];     // Neighbour triangle indexes
     BYTE Flags ;          // Bits 0-1:excluded,2-3:edge0,4-5:edge1,6-
     7:edge2
     BYTE Domain ;         // Region or land type
} TinTri ;

// Point types

#define TINPT_RANDOM 0
#define TINPT_SOFTBRK 1
#define TINPT_HARDBRK 2
#define TINPT_CONTOUR 3
#define TINPT_INFERRED 4
```

```
#define TINPT_OUTBND 5
#define TINPT_INTBND 6
```

Configuration Variables for Bentley CAD platforms

Bentley CAD applications are able to locate TerraModeler with the help of configuration variables. When you install TerraModeler, the installation program creates a configuration file TERRA.CFG which defines the required environment variables. This file is placed in Bentley CAD tool's CONFIG\APPL directory.

For example, C:\...\MICROSTATION\CONFIG\APPL\TERRA.CFG contains:

```
#-------------------------------------------------------
# TERRA.CFG - Configuration for Terra Applications
#-------------------------------------------------------
TERRADIR=c:/terra/
TERRACFG=$(TERRADIR)config/
MS_MDLAPPS < $(TERRADIR)ma/
MS_HELPPATH < $(TERRADIR)docs/
%if exists ($(TERRACFG)*.cfg)
% include $(TERRACFG)*.cfg
%endif
```

This configuration file includes the application-specific configuration files in C:\TERRA\CONFIG directory. TerraModeler's configuration file TMODEL.CFG contains:

```
#-------------------------------------------------------
# TMODEL.CFG - TerraModeler Configuration File
#-------------------------------------------------------
# Directory for storing the user license file (TMODEL.LIC)
TMODEL_LICENSE=$(TERRADIR)license/
# Directory for application settings (TMODEL.INF)
TMODEL_SET=$(TERRADIR)/tmodel/
```

In a default configuration, Bentley CAD automatically includes these settings as configuration variables. You can check the values for these variables in the **Configuration Variables** dialog of Bentley CAD. In case these variables have not been defined correctly, you should define them manually.

Installation Directories

TerraModeler 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 TerraModeler has been installed in path C:\TERRA64.

C:\TERRA64	installation directory for Terra applications
app	application files for Spatix
tscan.ix	application
config	application configuration files
tmodel.cfg	defines environment variables for the CAD platform
tmodel.ini	defines environment variables for Spatix
docs	documentation, such as user guides in PDF format
example	sample files
tmodel.dgn	example MicroStation design file
tmodel.clr	example color scheme
geoid	geoid model files
include	C/C++ header files with prototypes of public functions in TerraModeler
model_functions.h	general public functions
model_types.h	data types used
license	user license files
tmodel.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
tmodel	user settings and configuration files

Scripting interface

TerraModeler implements a scripting interface which you can use to send key-in commands to the CAD system. Any programming method provided by the CAD system is applicable, such as MDL, Bentley CAD Visual Basic or simply function key definition. In your own code, you may launch any action available in TerraModeler as toolbox icon, pulldown menu command or dialog push button, and set values of any dialog variable. Actions and dialog variables are only available when the corresponding dialog is created. This ensures that the application state is ready for the action or for setting a variable.

Scripting key-in commands

TerraScan, TerraPhoto, TerraMatch and TerraModeler implement the same scripting interface and thus, work with the same key-in command logic. The key-in commands below show the syntax for TerraModeler commands while examples are given from TerraScan. The syntax for the other applications follow logical rules:

APPLICATION	SET LOGGING	LAUNCH ACTION	SET VARIABLE
TerraScan	scan log	scan action	scan set
TerraPhoto	photo log	photo action	photo set
TerraMatch	match log	match action	match set
TerraModeler	model log	model action	model set

Model Log Dialogs *logfile*

Model Log Dialogs *logfile* starts to write actions and dialog variables to a text file at a given location on the hard disc. E.g., "scan log dialogs d:\logs\dialogs.txt" writes the logfile "dialogs.txt" in directory d:\logs\ on the hard disc. If the file does not exist, it is created when the first action to log is performed.

The command is most useful to find out action, dialog and variable names. Start the logfile creation by using the key-in command in the Key-in line[Bentley CAD] or Spaccels window[Spatix]. Call the tool or command in the software and define settings in the dialog. Close the dialog with OK or Cancel. The application writes the dialog name, variable names and setting values into the logfile.

Use the key-in command without the *logfile* variable in order to stop writing actions to the logfile.

Model Action *action*

Model action *action* starts the given action. E.g. "scan action trajectviewfields" launches the command **View fields** in the **Trajectory** window which means it opens the **View Trajectory Fields** dialog. Actions are linked to dialog push buttons or dialog pulldown menus in the software. An action can be started only when the dialog is open.

Model Action HideDialog

Model action hidedialog instructs the software to keep the next modal dialog hidden and close it with OK.

Model Set *variable = value*

Model set *variable = value* assigns the given value to the given variable. E.g. "scan set VtfNumber=1" sets the **Number** attribute to be visible in the **View Trajectory Fields** dialog. If there is no matching variable in an open dialog, the application stores the variable-value-assignment in a buffer. When the next dialog is created, it will apply the assignment and clear the buffer.

Model Close Dialog *dialog*

Model close dialog *dialog* closes a modeless dialog with the given name. E.g. "scan close dialog ManageTrajectories" closes the Trajectories window.

Scripting examples (Bentley CAD VBA)

TerraModeler setup provides an example script that includes Bentley CAD VBA modules. The script is stored in the \EXAMPLE folder of the Terra installation directory, e.g. C:/TERRA64/EXAMPLE/SCRIPTING_EXAMPLE.MVBA. The example script can be loaded in the VBA Manager of Bentley CAD and used, for example, as starting point for own scripts.

- A -

- C -

- D -

- E -

- F -

- I -

- L -

- M -

- O -

- P -

- R -

- S -

- T -

- U -

- V -